CENTURION 5/6/7 T

LATHE CONTROL

Operation Manual

Version 1.3
January, 2003

MILLTRONICS MANUFACTURING COMPANY
1400 Mill Lane
Waconia, MN 55387
(952) 442-6455

TABLE OF CONTENTS

PREFACE.....i

AXIS DEFINITIONS.....ii

1. INTRODUCTION.....1-1

2. PROGRAM CONFIGURATION.....2-1
   2.1 Block.....2-1
   2.2 Program.....2-2
   2.3 Main program, subprogram and subroutines.....2-3
   2.4 Command format ranges.....2-4
   2.5 Command formats for axes: M and G Codes.....2-5

3. PREPARATORY FUNCTIONS G CODES.....3-1
   3.1 Positioning (G00) rapid traverse (modal).....3-3
   3.2 Linear interpolation (G01).....3-5
      3.2.1 Polar definition of a line.....3-5
   3.3 Circular interpolation (G02, G03).....3-8
      3.3.1 Corner rounding.....3-19
      3.3.2 Angle chamfering.....3-20
      3.3.3 Back line.....3-20
   3.4 Dwell command (G04).....3-22
   3.5 Exact stop (G09).....3-22
   3.6 Set data on/off (G10, G11).....3-23
   3.7 Clear floating zero (G12).....3-23
   3.8 Inch dimensioning mode (modal) (G20).....3-23
   3.9 Metric dimensioning mode (modal) (G21).....3-23
   3.10 Safe zone on/off (G22, G23).....3-24
   3.11 Reference point return (G28, G29, G30).....3-24
   3.12 Threading (G32, G34-G36).....3-26
   3.13 Return to initial level or to R level (G38/G39)...3-27
   3.14 Cutter compensation (G40, G41, G42).....3-27
   3.15 Non-movement (G65).....3-37
   3.16 Z to clearance (G45).....3-40
   3.17 X to Clearance (G46).....3-41
   3.18 Z to tool change (G47).....3-41
   3.19 X to tool change (G48).....3-41
   3.20 Tool length offsets cancel (G49).....3-41
   3.21 Floating zero (G50).....3-42
   3.22 Local coordinate system (G52).....3-44
   3.23 Machine coordinate system (G53).....3-45
   3.24 Work coordinate systems (G54 - G59).....3-46
   3.25 Single direction or one shot rapid positioning (G60).....3-47
   3.26 Exact stop mode (modal) (G61).....3-47
   3.27 Tapping mode (modal) (G63).....3-47
   3.28 Cutting mode (modal) (G64).....3-47

3.29 Finish Cycle (G70).....3-48
3.30 Rough Turning Cycle (G71).....3-50
3.31 Rough Facing Cycle (G72).....3-56
3.32 Pattern Repeat Cycle (G73).....3-61
3.33 Multiple Thread Cutting Cycle (G76).....3-64
3.34 Cycle 1 (G92).....3-67
3.35 Face Grooving/High Speed Peck Drilling Cycle (G74).....3-75
3.36 Diameter Grooving (G75).....3-78
3.37 Drill Cycle (G81).....3-82
3.38 Drill Dwell Cycle (G82).....3-83
3.39 Drill Peck Cycle (G83).....3-84
3.40 Tap Cycle (G84).....3-85
3.41 Bore Cycle (G85).....3-87
3.42 High Speed Boring Cycle (G86).....3-88
3.43 Bore/Dwell Cycle (G89).....3-89
3.44 Cycle A (G90).....3-91
3.45 Cycle B (G94).....3-93
3.46 Incremental mode (G91).....3-96
3.47 Absolute mode (G93).....3-96
3.48 Constant Surface Speed (CSS) (G96).....3-97
3.49 Revolutions Per Minute (RPM) (G97).....3-98
3.50 Feed Per Minute (G98).....3-98
3.51 Feed per Revolution (G99).....3-99
3.52 Cancel mirror image (G501) Set mirror image (G511)3-99
3.53 Cancel scaling (G550) Set scaling (G551).....3-100
3.54 Coordinate system rotation (G568 - G569).....3-101

4. MISCELLANEOUS FUNCTIONS (M Functions).....4-1
4.1 Program stop (M00).....4-3
4.2 Optional stop (M01).....4-3
4.3 Block skip (/).....4-3
4.4 End of program (M02, M30, M99).....4-3
4.5 Spindle on/off (M03, M04, M05).....4-3
4.6 Tool change (M06).....4-3
4.7 Coolants on/off (M08, M09).....4-4
4.8 Wait channel (M32).....4-4
4.9 (M20 - M29).....4-4
4.10 (M60 - M69).....4-4
4.11 (M70 - M79).....4-4
4.12 (M80 - M89).....4-4
4.13 Graphics off/on (M90, M91).....4-5
4.14 Subprogram call (M98) Subprogram terminate (M99)...4-5
4.14.1 Preparation of subprogram.....4-6
4.14.2 Subprogram execution.....4-6

5. FRONT PANEL OPERATION.....5-1
5.1 Front panel description.....5-1
5.2 Home sequence F1 HOME (MAIN-HOME).....5-5

5.2.1 F10 HOME - HERE.....5-6
5.3 Jog F2 JOG (MAIN-JOG).....5-7
5.4 Handwheel F3 HDW (MAIN-HDW).....5-8
5.5 Run F4 RUN (MAIN-RUN).....5-12
5.5.1 F9 RESUM (MAIN-RUN).....5-14
5.5.2 F2 OLD (MAIN-RUN-OLD).....5-15
5.5.3 F3 BLOCK (MAIN-RUN).....5-15
5.5.4 F4 OSTOP (MAIN-RUN).....5-15
5.5.5 F5 BSKIP (MAIN-RUN).....5-15
5.5.6 F6 DISPL (MAIN-RUN-DISPL).....5-16
5.5.6.1 F1 NEXT (MAIN-RUN-DISPL).....5-16
5.5.6.2 F2 DIST (MAIN-RUN-DISPL).....5-17
5.5.6.3 F3 GRAPH (MAIN-RUN-DISPL-GRAPH).....5-17
5.5.6.3.1 F2 PAN.....5-18
5.5.6.3.2 F3 WIND.....5-19
5.5.6.3.3 F4 AUTO.....5-19
5.5.6.3.4 F5 ZOOM-.....5-19
5.5.6.3.5 F6 ZOOM+.....5-20
5.5.6.3.6 F7 LIMIT.....5-20
5.5.6.3.7 F8 ZONE.....5-20
5.5.6.3.8 F9 FRESH.....5-20
5.5.6.3.9 F10 CLEAR.....5-20
5.5.6.4 (F4 DIAG) MAIN-RUN-DISPL-DIAG.....5-20
5.5.6.5 F5 ERROR (MAIN-RUN-DISPL-ERROR).....5-22
5.5.6.6 F7 OBS (MAIN-RUN-DISPL-OBS).....5-22
5.5.6.7 F9 SHELL (MAIN-RUN-DISPL-SHELL).....5-22
5.5.7 F7 MENU (MAIN-RUN-MENU).....5-22
5.5.8 F8 DRY (MAIN-RUN-DRY).....5-23
5.5.9 F10 HDW (MAIN-RUN-HDW).....5-23
5.6 Manual data input F5 MDI (MAIN-MDI).....5-25
5.6.1 Gcode (MAIN-MDI-MCODE).....5-26
5.6.2 Mcode (MAIN-MDI-MCODE).....5-27
5.7 Display F6 DISPL (MAIN-DISPL).....5-27
5.8 Parameters F7 PARMS (MAIN-PARMS).....5-27
5.8.1 F1 SETUP (MAIN-PARMS-SETUP).....5-28
5.8.1.1 F2 PREC.....5-30
5.8.1.2 F3 POWER.....5-31
5.8.1.3 F4 AXIS.....5-34
5.8.1.4 F5 MISC.....5-39
5.8.1.5 F6 FDOVR 5-45
5.8.1.6 F5 HWOVR.....5-46
5.8.1.7 F6 SPOVR.....5-47
5.8.1.8 F7 BSCRW.....5-48
5.8.1.9 F9 DOS.....5-48
5.8.2 F2 COORD (MAIN-PARMS-COORD).....5-49
5.8.3 F3 TOOL (MAIN-PARMS-TOOL).....5-50
5.8.4 F4 WEAR (MAIN-PARMS-LOAD).....5-51
5.8.5 F6 SAVE (MAIN-PARMS-SAVE).....5-52
5.8.6 F7 LOAD (MAIN-PARMS-LOAD).....5-52
5.8.7 F8 PROG (MAIN-PARMS-PROG).....5-52
5.8.8 F9 CTRL (MAIN-PARMS-CTRL).....5-55

5.8.9 F10 USER (MAIN-PARMS-USER).....5-57
5.9 Programming F8 PROG (MAIN-PROG).....5-57
5.9.1 F1 TEXT (MAIN-PROG-TEXT).....5-58
5.9.1.1 F1 EDIT (MAIN-PROG-TEXT-EDIT).....5-59
5.9.1.1.1 F1 BLOCK.....5-61
F1 BEGIN.....5-61
F2 END.....5-61
F3 WORD.....5-61
F5 HIDE.....5-61
F6 DEL.....5-61
F7 COPY.....5-61
F8 MOVE.....5-62
5.9.1.1.2 F2 CURSR.....5-62
F1 BBLOK.....5-62
F2 EBLOK.....5-62
F3 TAB.....5-62
F5 TOF.....5-62
F6 EOF.....5-62
F7 PGUP.....5-62
F8 PGDN.....5-62
F9 LEFT.....5-62
F10 RIGHT.....5-62
5.9.1.1.3 F3 WORDS.....5-62
5.9.1.1.4 F4 MISC.....5-63
F1 UNDO.....5-63
F2 REST.....5-63
F3 HDW.....5-63
F6 LNDEL.....5-63
F7 CHNG.....5-63
F8 FIND.....5-64
F9 FNEXT.....5-65
5.9.1.1.5 F5 INS.....5-65
5.9.1.1.6 F6 DEL.....5-65
5.9.1.2 F2 NEW (MAIN-PROG-TEXT-NEW).....5-66
5.9.1.3 F3 OLD (MAIN-PROG-TEXT-OLD).....5-66
5.9.1.4 F4 ANY (MAIN-PROG-TEST-ANY).....5-66
5.9.1.5 F7 MENU (MAIN-PROG-TEXT-MENU).....5-67
5.9.2 F2 CONV (MAIN-PROG-CONV).....5-68
5.9.2.1 F1 EDIT (MAIN-PROG-CONV-EDIT).....5-68
5.9.2.1.1 EDIT KEYS.....5-69
F1 EDIT.....5-69
F2 VIEW.....5-69
F3 EVENT.....5-70
F4 T##.....5-70
F6 INS.....5-70
F7 DEL.....5-70
F9 PREV.....5-70
F10 NEXT.....5-70
5.9.2.2 STORE/INPUT KEYS.....5-71
F1 STORE.....5-71
F3 TOGL.....5-71

F5 DEL.....5-72
F7 ↑.....5-72
F8 ↓.....5-72
F9 ←.....5-72
F10 →.....5-72
5.9.2.3 F2 NEW (MAIN-PROG-CONV-NEW).....5-72
5.9.2.4 F3 OLD (MAIN-PROG-CONV-OLD).....5-72
5.9.2.5 F4 ANY (MAIN-PROG-CONV-ANY).....5-72
5.9.2.6 MACHINE OPERATION KEYS (MAIN-PROG-CONV).....5-73
5.9.3 F4 EDIT (MAIN-PROG-EDIT).....5-74
5.9.3.1 F7 MENU (MAIN-PROG-CONV-MENU).....5-74
5.10 Verify F9 VERF (MAIN-VERF).....5-75
5.10.1 F3 BLOCK (MAIN-VERF).....5-77
5.10.2 F4 OSTOP (MAIN-VERF).....5-77
5.10.3 F5 BSKIP (MAIN-VERF).....5-78
5.10.4 F6 DISPL (MAIN-VERF-DISPL).....5-78
5.10.5 F8 DRY.....5-78
5.10.6 F9 HALT (MAIN-VERF).....5-78
5.11 Utilities F10 UTIL (MAIN-UTIL).....5-79
5.11.1 F3 FILES (MAIN-UTIL-FILES).....5-79
5.11.1.1 F1 LOAD (MAIN-UTIL-FILES-LOAD).....5-79
5.11.1.2 F2 SAVE (MAIN-UTIL-FILES-SAVE).....5-80
5.11.1.3 F3 NAME (MAIN-UTIL-FILES-NAME).....5-80
5.11.1.4 F4 COPY (MAIN-UTIL-FILES-COPY).....5-81
5.11.1.5 F5 ▸RAM (MAIN-UTIL-FILES-RAM).....5-81
5.11.1.6 F9 ERASE (MAIN-UTIL-FILES-ERASE).....5-81
5.11.2 F4 RS232 (MAIN-UTIL-RS232).....5-81
5.11.2.1 F1 COM1 OR COM2.....5-81
5.11.2.2 F2 COM1 OR COM2.....5-81
5.11.2.3 F5 SEND (MAIN-UTIL-RS232-SEND).....5-81
5.11.2.3.1 F1 TEXT.....5-82
5.11.2.4 F6 RECEV (MAIN-UTIL-RS232-RECEV.....5-82
5.11.3 F6 DNC (MAIN-UTIL-DNC).....5-83
5.11.3.1 F3 FAST (MAIN-UTIL-DNC-FAST).....5-84
5.11.3.1.1 F1 RS-232.....5-85
5.11.3.1.2 F2 FILE.....5-85
5.11.3.1.3 F3 DISK.....5-85
5.11.3.1.4 F4 ANY.....5-85
5.11.3.1.5 F5 OLD.....5-86
5.11.3.1.6 F6 RAM▸.....5-86
5.11.3.2 F9 SKIP.....5-86
5.11.3.3 F4 RUN (MAIN-UTIL-DNC-RUN).....5-86
5.11.3.4 F9 VERF (MAIN-UTIL-DNC-VERF).....5-86
5.11.4 F7 CHART (MAIN-UTIL-CHART).....5-87
5.11.5 F8 INFO (MAIN-UTIL-INFO).....5-88
5.11.5.1 F1 STD (MAIN-UTIL-INFO-STD).....5-88
5.11.5.2 F2 SYS (MAIN-UTIL-INFO-SYS).....5-90
5.11.5.3 F3 FP (MAIN-UTIL-INFO-FP).....5-91
5.11.5.4 F4 PATH (MAIN-UTIL-INFO-PATH).....5-92

5.11.6 F9 BLANK (MAIN-UTIL-BLANK).....5-92

6. PARAMETRIC PROGRAMMING.....6-1
6.1 Parametric reference.....6-1
6.2 Parametric assignment statement.....6-1
6.3 PB50 Byte Parameter.....6-2
6.4 Parametric operators.....6-2
6.4.1 Arithmetic operators.....6-2
6.5 Relational operators.....6-3
6.6 Function operators.....6-3
6.7 Mathematic expressions.....6-4
6.8 Conditional statements.....6-4
6.9 Transfer statements.....6-6
6.10 GOTO statement.....6-6
6.11 CALL statement.....6-6
6.12 GOSUB and RETURN.....6-7
6.13 Computational Functions.....6-8
6.13.1 Tangent Arc [TANA], Tangent Line [TANL] 3 Point Circle Generate [CGEN].....6-8
6.13.2 Back line.....6-11
6.13.3 Miscellaneous Commands.....6-13
6.13.4 Parts Inspection and Digitizing Commands...6-18

7. CONVERSATIONAL PROGRAMMING SCREENS.....7-1
7.1 POSITION.....7-3
7.2 ROUGH TURNING.....7-4
7.3 PATTERN REPEAT CYCLE.....7-9
7.4 FINISH TURNING.....7-11
7.5 CYCLE A.....7-19
7.6 LINE.....7-24
7.7 ARC.....7-26
7.8 TANGS.....7-28
7.8.1 Tangent Line.....7-29
7.8.2 Tangent Arc.....7-30
7.9 CGEN.....7-31
7.10 END.....7-33
7.11 Rough Facing Cycle.....7-34
7.12 Pattern Repeat Cycle.....7-38
7.13 Facing Finish Cycle.....7-39
7.14 Cycle B.....7-44
7.15 Multiple Thread Cutting Cycle.....7-48
7.16 Cycle 1.....7-56
7.17 Thread Lines.....7-61
7.18 Drilling Cycles.....7-62
7.18.1 Drill Cycle.....7-62
7.18.2 Drill Dwell Cycle.....7-64
7.18.3 Drill Peck Cycle.....7-66
7.18.4 Face Grooving Cycle.....7-67
7.18.5 Bore Cycle.....7-72
7.18.6 Bore/Dwell Cycle.....7-73
7.18.7 The Right Tap Cycle.....7-75

7.18.8 The Left Tap Cycle.....7-76
7.18.9 Diameter Grooving Cycle.....7-78
7.19 Tool Change Screen.....7-82
7.20 Miscellaneous Screen.....7-85
7.21 F7 CALL.....7-87
7.22 F8 SPECIAL (SPEC).....7-87
7.22.1 F3 SCALE (SPEC-SCALE).....7-87
7.22.2 F4 ROT (SPEC-ROT).....7-88
7.22.3 F5 MIRR (SPEC-MIRR).....7-89
7.22.4 F7 FLZ (SPEC-FLZ).....7-90
7.22.5 END OF PROGRAM.....7-90

APPENDIX.....7-91

ERROR MESSAGES.....7-92

PREFACE

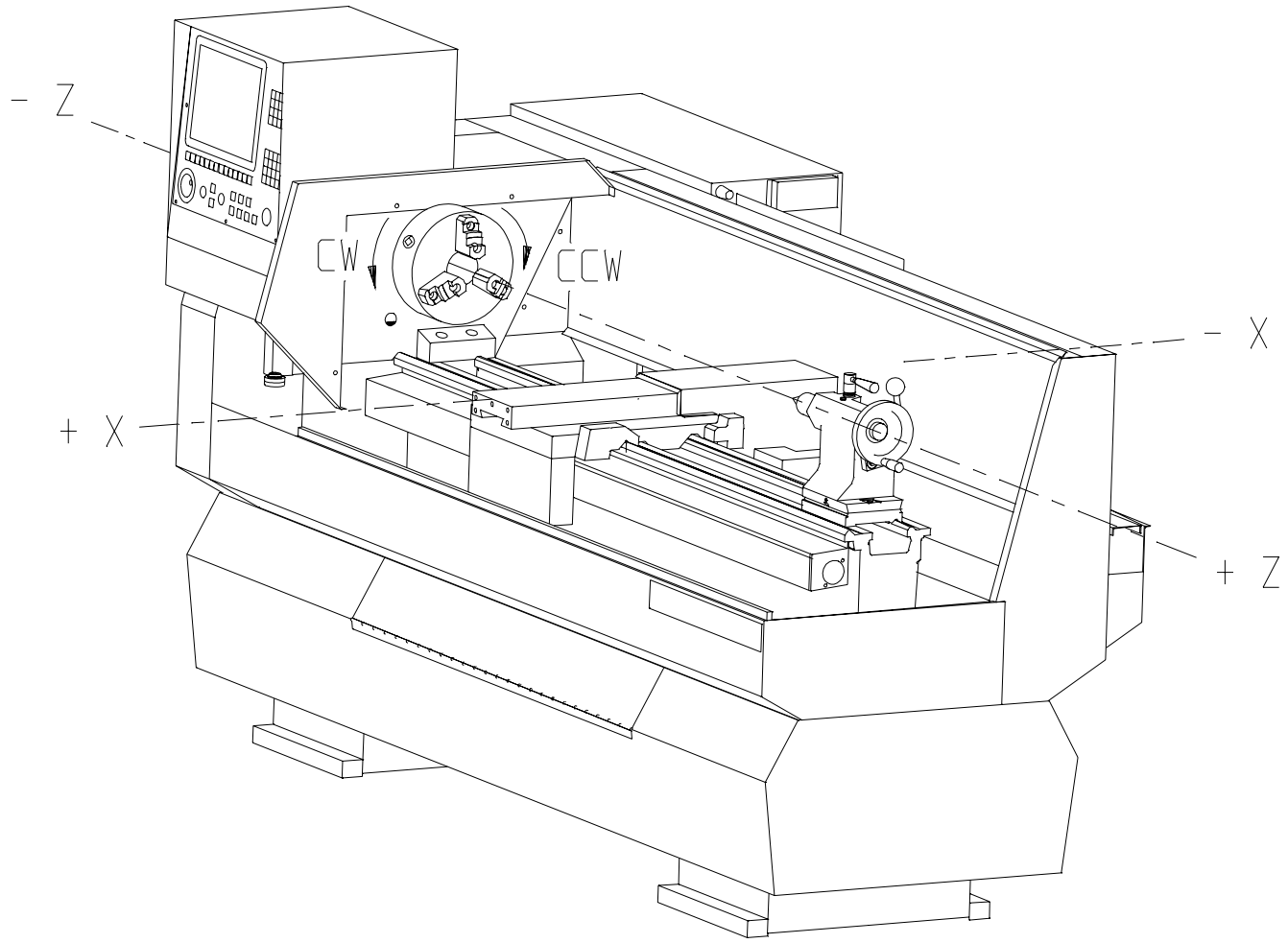
The Centurion 5T and Centurion 6T have two controllable axes in their basic configuration: X and Z. This manual assumes that the tool moves with respect to the work piece.

When this manual makes reference to Centurion 5 it implies Centurion 6 also.

This manual is divided into two sections, M & G code programming and conversational programming. The conversational programming section is designed primarily to explain the various menus, screen entries, and the general flow from one screen to another. It does not go into any detail on how the various functions work. Detailed explanations of each function are covered in the M & G code section and should be referenced there if any questions arise.

AXIS DEFINITIONS

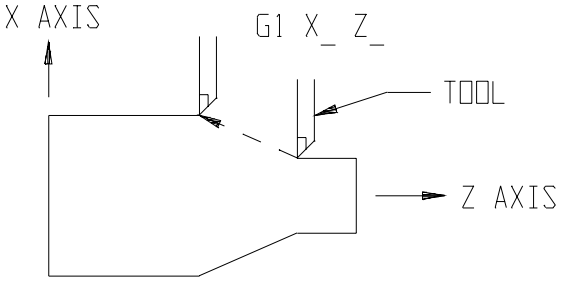
All directions are referenced with respect to the tool. The following illustrates the X and Z directions.



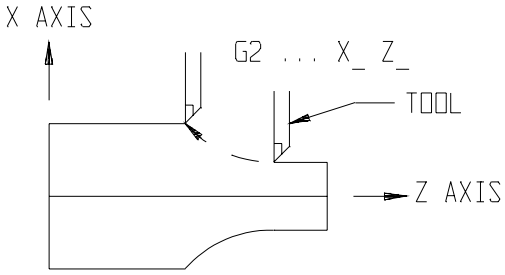
1. INTRODUCTION

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.

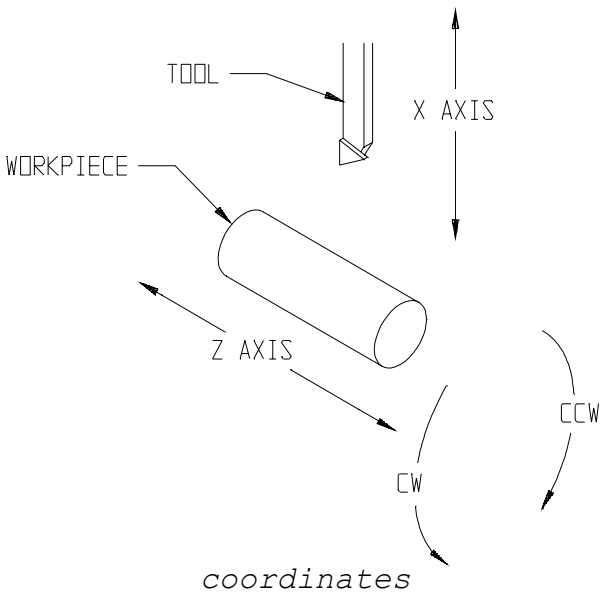


linear interpolation



circular interpolation

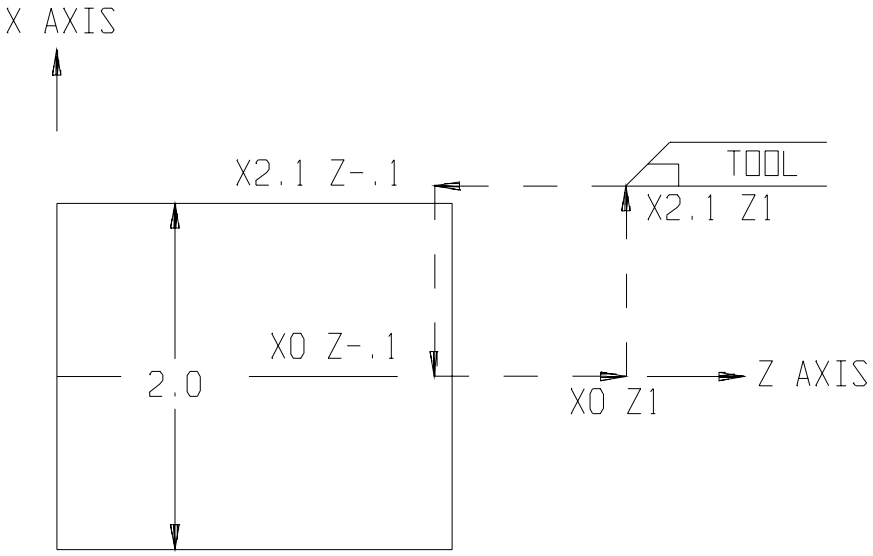
When the commanded position to be reached by the tool is executed, the CNC moves the tool to that position via one of the interpolation modes, circular or linear. The position is given as a coordinate value in a rectangular cartesian coordinate system.



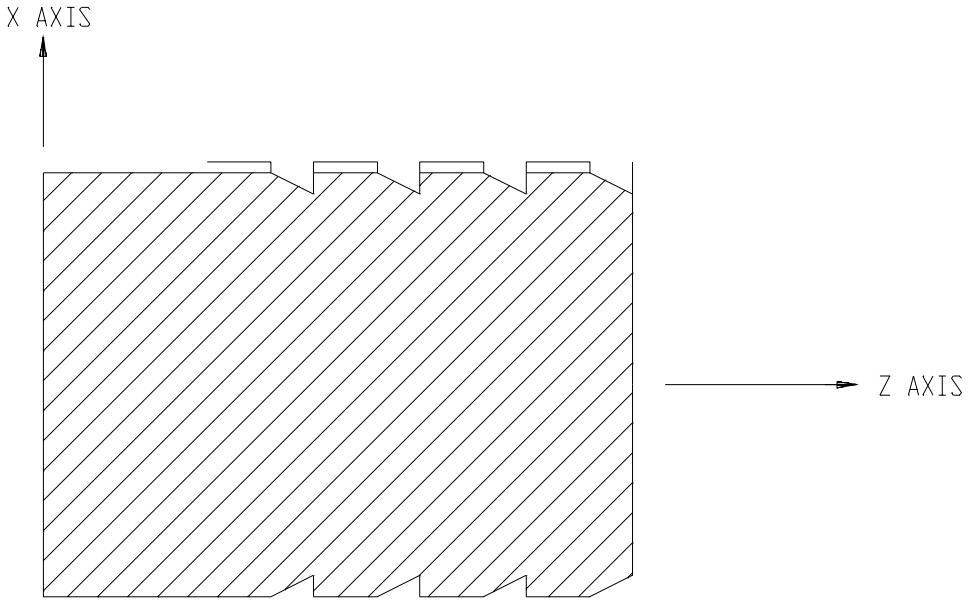
coordinates

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 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.

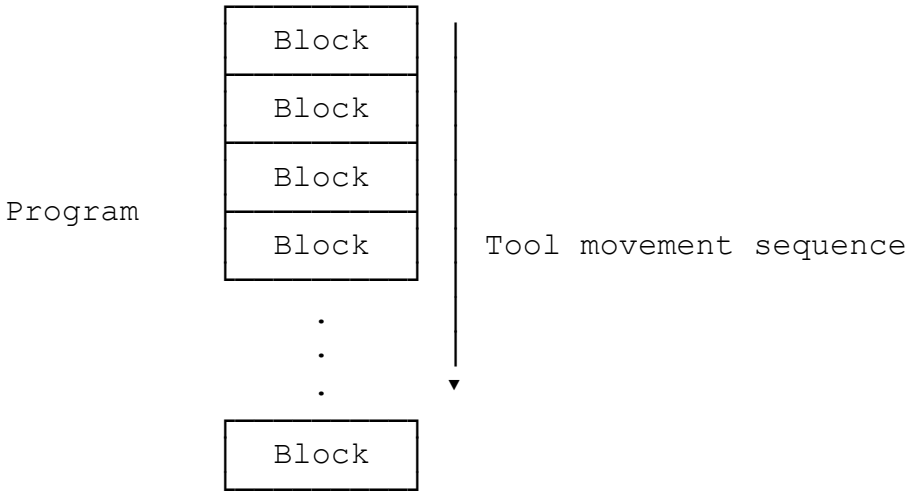


In incremental mode the tool moves to a point the programmed distance from the current tool position.



2. PROGRAM CONFIGURATION

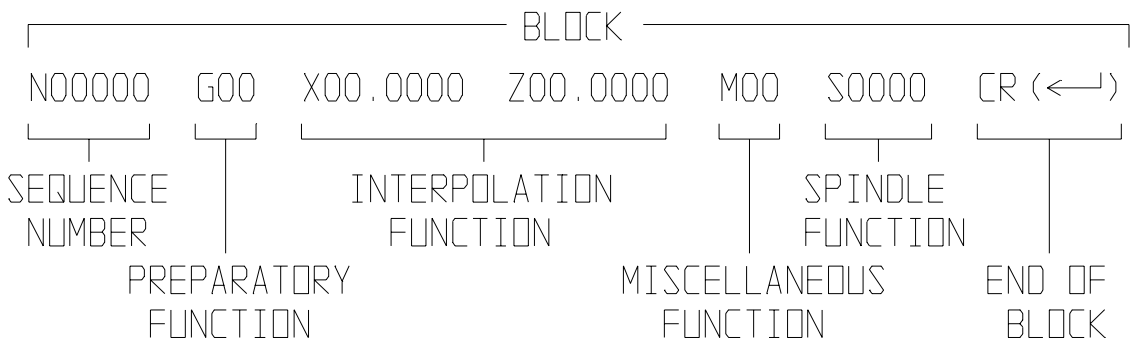
By definition, a program is a group of commands given to the CNC for operating a machine. By specifying commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off. In a program, specify the commands in the sequence of actual tool movements.



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, and the number for naming each program is called the program number.

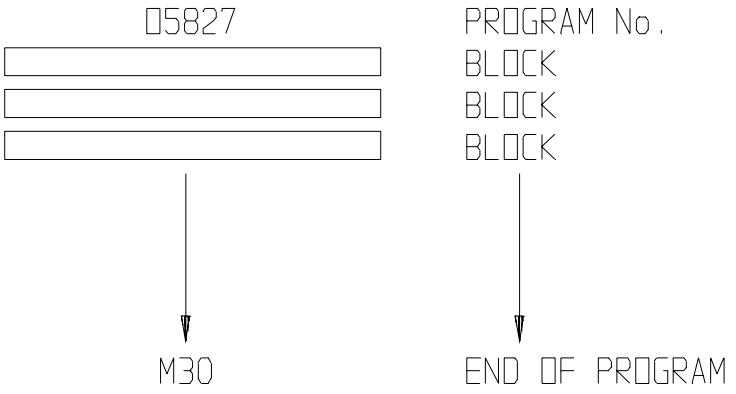
The block and the program have the following configurations:

2.1 Block



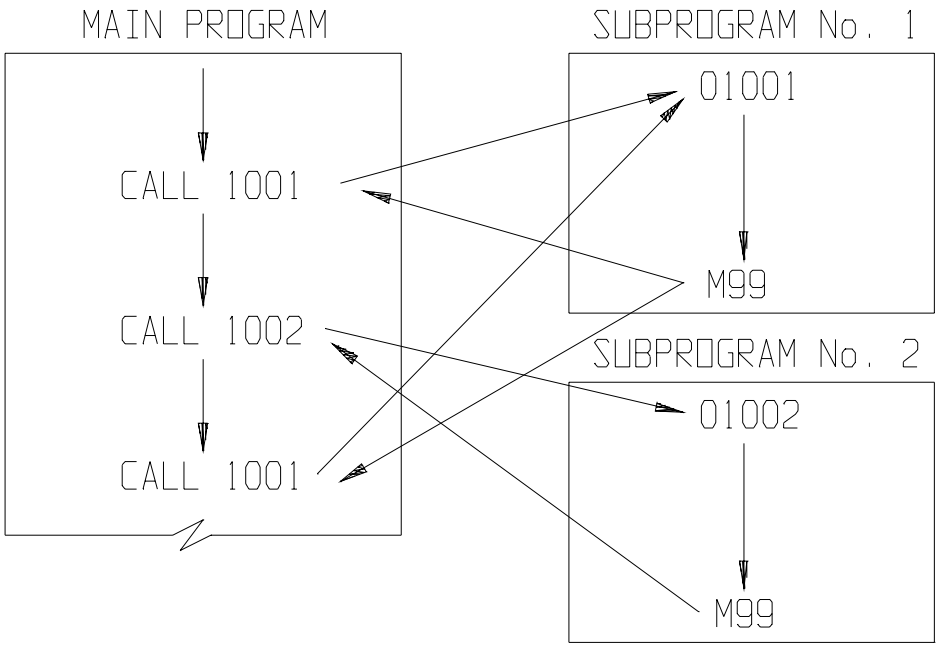
Each block begins with an optional number and ends with a <CR> carriage return.

2.2 Program



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

2.3 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 subprogram "CALL" command appears, the commands of the subprogram are executed before execution of the next block of the main program.

Subprograms can be used to build part libraries of commonly used patterns. Subprograms can reside anywhere in memory.

2.4 Command format ranges

The basic address and command value ranges are listed in Table 1. 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.

FUNCTIONS	COMMAND LETTER	INCH INPUT	METRIC INPUT
Subprogram # and Program #	O	1 - 9999	1 - 9999
Sequence #	N	1 - 99999	1 - 99999
Preparatory function	G	0 - 999	0 - 999
Dimension * words	XYZUVWQ ABCIJKRP	0 ± 999.9999	0 ± 9999.9999
Dwell	P	.01 - 9999.99	.01 - 9999.99
Feedrates *	F	.0001 - 999.9999	.01 - 99999.99
Spindle speed *	S	1 - 9999	1 - 9999
Tools	T	0000 - 2424	0000 - 2424
Misc. function	M	0 - 99	0 - 99
Repeat or loop	L	0 - 9999	0 - 9999

Table 1
Command Format Ranges

* 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.

2.5 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
F10.0     10 mm/min., or 10 inch/min.
G04 P1    Dwell for one second
```

These 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, ZC, E, H, L, N, O, S, 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 greater than 8 digit commands will be accepted.

3. PREPARATORY FUNCTIONS G CODES

These codes are used if the operator is programming the Centurion 5 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 number (up to three digits) 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 in effect only 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.

The following lists G codes accepted by the Centurion 5 control system. Each code has a detailed explanation later in the manual.

G CODES

		Active On Power Up	Modal	One Shot
00	Positioning		X	
01	Linear Interpolation	X	X	
02	Circular/CW		X	
03	Circular/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
20	Inch input	X	X	
21	Metric input		X	
22	Safe zone check on		X	
23	Safe zone check off	X	X	
28-30	Reference point return			X
32	Threading			X
34-36	Threading			X
38	Initial Point Return		X	
39	R-Plane Return		X	

		Active On Power Up	Modal	One Shot
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 system		X	
60	One-Shot Rapid			X
61	Exact stop mode		X	
63	Tapping mode		X	
64	Cutting	X	X	
65	Non-movement			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
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
89	Bore/dwell			X
90	Turning Cycle A			X
91	Incremental dimension		X	

92	Thread Cutting Cycle			X
93	Absolute	X	X	
96	Constant Surface Speed		X	
97	Revs per Minute	*	X	
98	Feed per Minute	*	X	
99	Feed per Rev.		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	
990	Store parameters			X
991	Read parameters			X
995	Read byte parameters			X
996	Set byte parameters			X
997	Force error			X
998	Beep			X
999	Force fatal error			X

** Feed/Min. or Feed/Rev. is selectable as a default via a parameter.*
Note: Unrecognized G codes will cause error #549 to occur.

3.1 Positioning (G00) rapid traverse (modal)

G00 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:

G00 moves the tool at a rapid traverse rate to a position in the work coordinate system for both incremental and absolute commands.

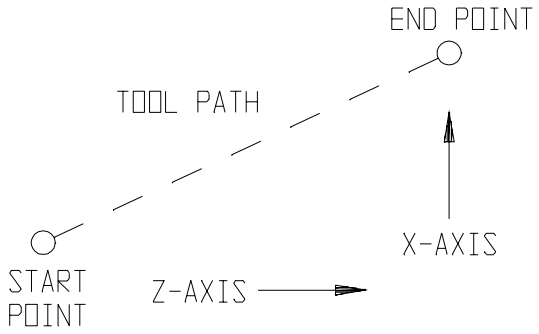
Format: G00 ~-;

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 G00 command with a G01 command. The motions of all axes in G00 mode will be interpolated with all axes reaching the end point simultaneously.



Note 1: The rapid traverse rate in the G00 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 G00, 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.)

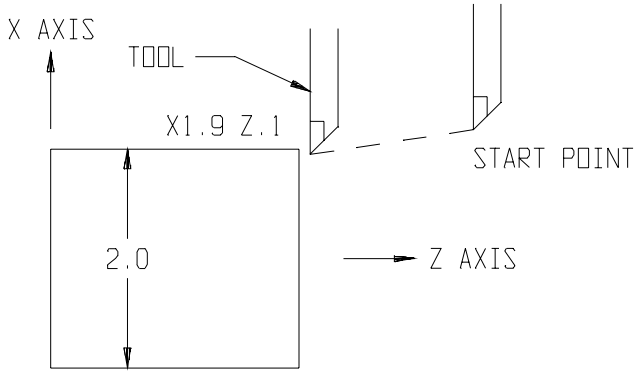
G00 mode automatically accelerates and decelerates in a linear fashion allowing the controlled axis to start and stop smoothly. The rate of accel/decel can be changed by the machine tool builder.

3.2 Linear interpolation (G01)

G01 ~___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:

G01 X1.9 Z.1 F20 ;

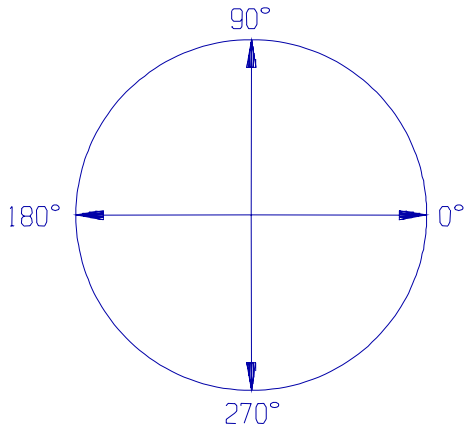


The feedrate specified by the F code is the vector rate along the path, not the rate of each axis.

3.2.1 Polar definition of a line

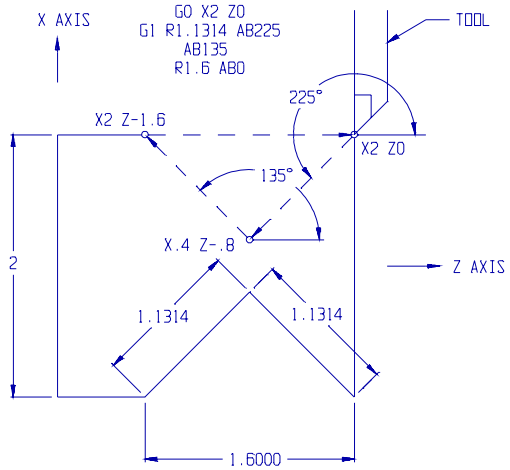
A polar line is specified by a polar radius/length (R), an angle (AB), and a polar center (AA or I, K, or XC, ZC).

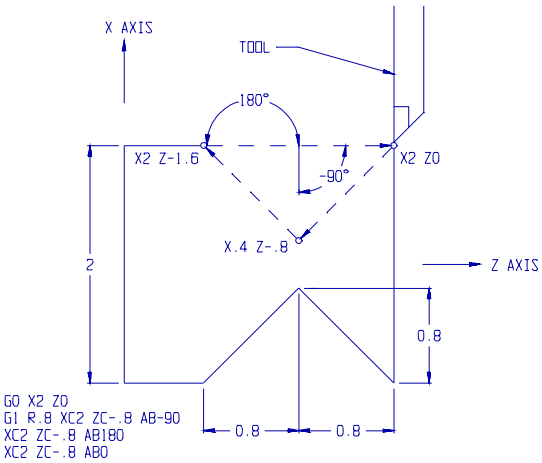
The 3 o'clock position is always 0 degrees. Positive angles result in CCW rotation of the polar radius, while negative angles result in CW rotation of the polar radius. Polar lines can be used when estimating lengths during trig help.

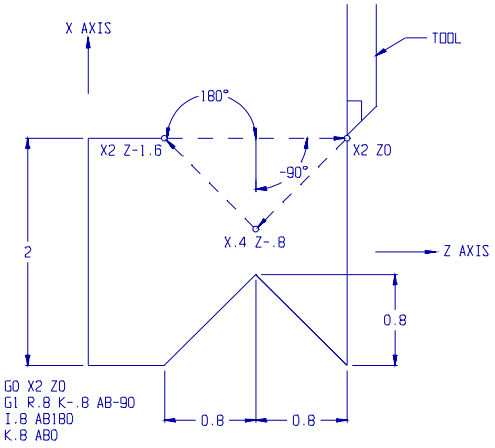


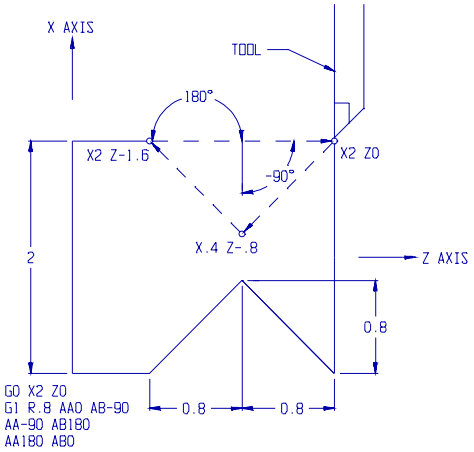
If the polar radius/length (R) or angle (AB) is not specified, then the previous values will be used. If the polar center is not specified then it is taken to be the current machine position.

Some examples of polar programming:









Cutting Feedrate G01, G02, G03 Mode

The feedrate of linear interpolation (G01) and circular interpolation (G02, G03) 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.

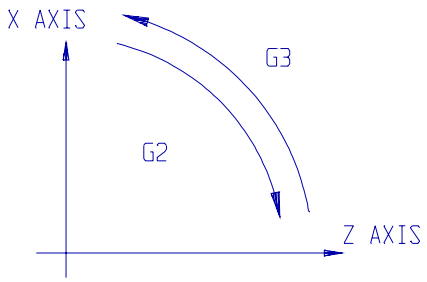
Feedrate Override

The feed can be overridden using this switch on the machine operator's panel by 0 to 140% (per every 10%). Feedrate override cannot be applied to functions in which override is inhibited (e.g. tapping cycle, threading).

3.3 Circular interpolation (G02, G03)

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

	Data to be given		Command	Meaning
1	Direction of rotation		G02	Clockwise (CW)
			G03	Counterclockwise (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

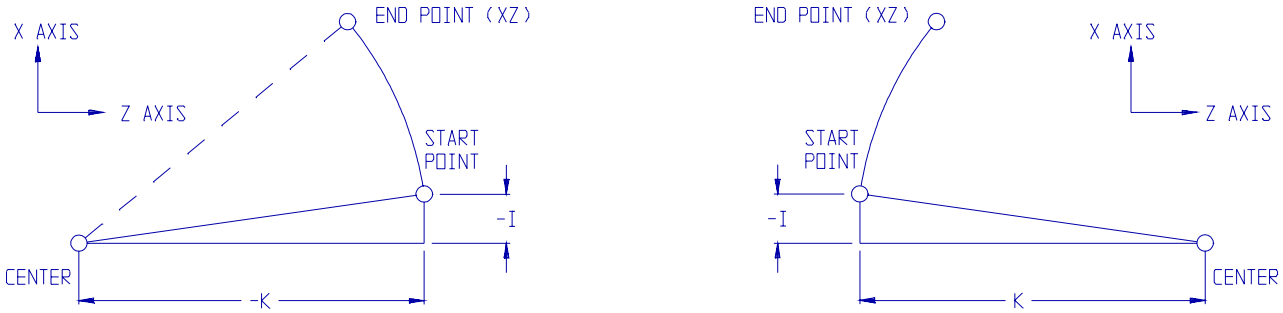


Clockwise and Counterclockwise Directions

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:

G02	X____	Z____	R____
or			
G03	X____	Z____	R____

Arc End Points

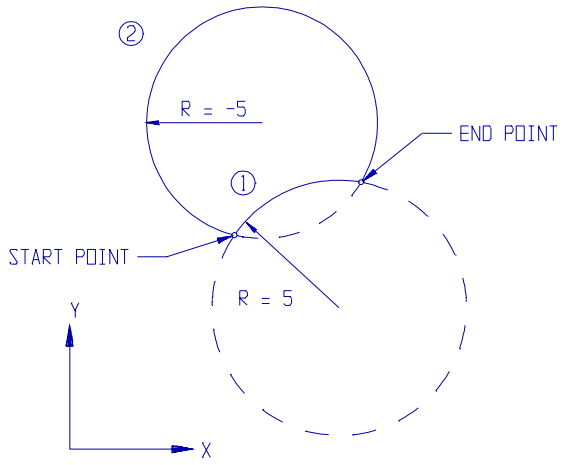
The radius is always specified as its true value. The end points are incremental or absolute depending on G93 and G91. If a radius is used without a center point there are two types of arcs which can be generated. One is less than 180°, and the other is greater than 180°, as shown in the figure that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

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

Examples:

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

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



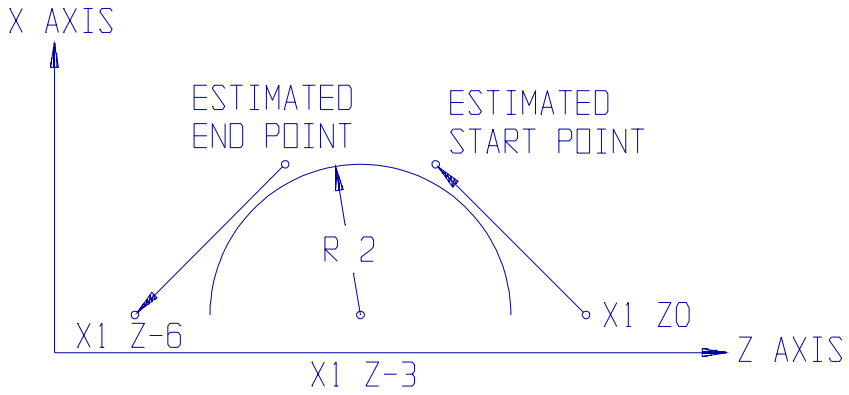
Describing an Arc Using Absolute Center and Trig Help

G02	X___	Z___	XC___	ZC___	R___
or					
G03	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.

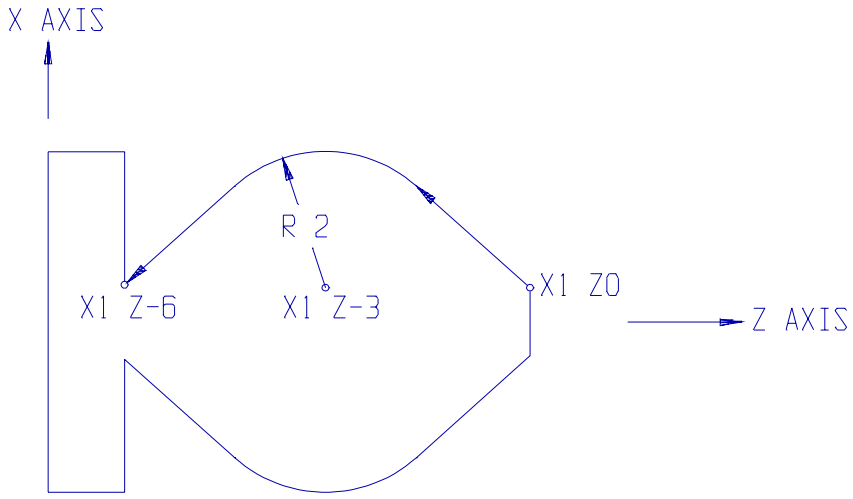
EXAMPLES OF TRIG HELP

Program 1



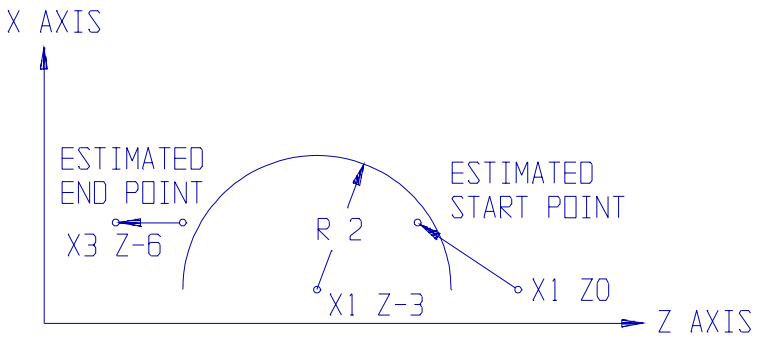
Programmed Path

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



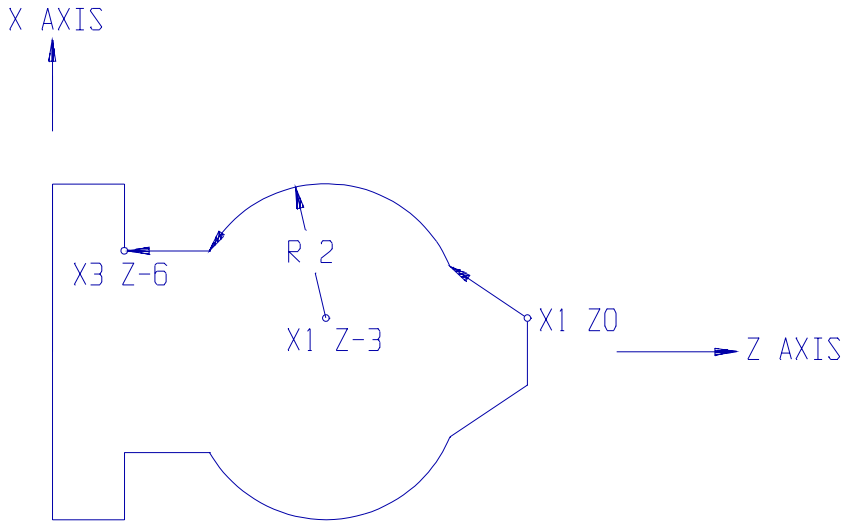
Path Generated by Program 1

Program 2



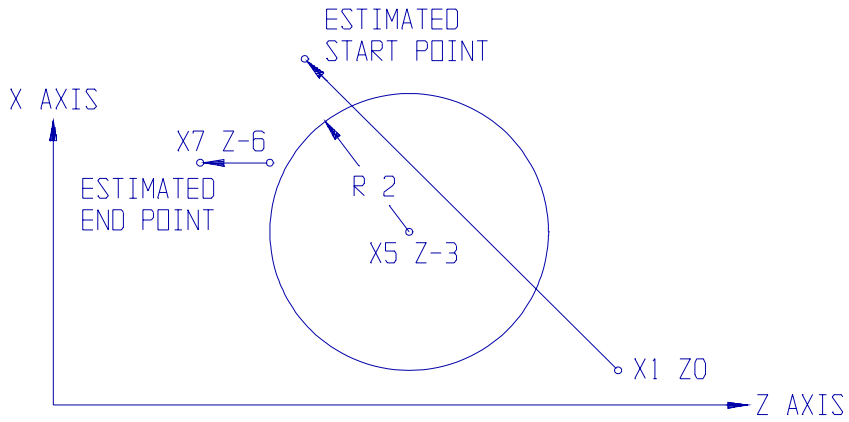
Programmed Path

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



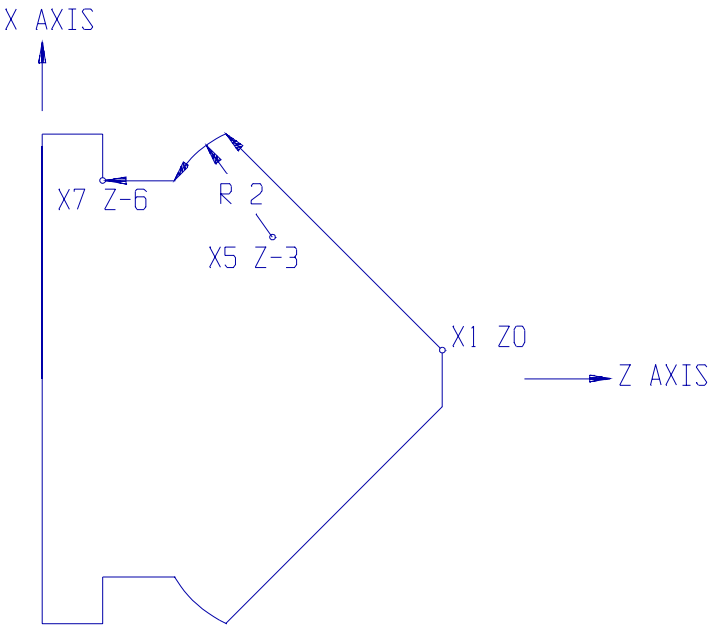
Path Generated by Program 2

Program 3



Programmed Path

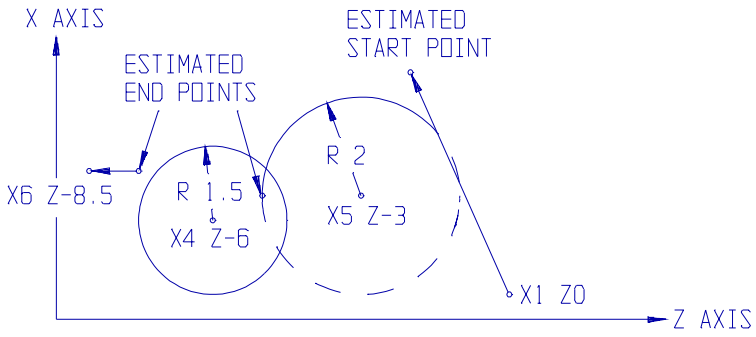
```
G1 X1 Z0
X10 Z-4.5 (estimated start point)
G3 R2 XC5 ZC-3 X7 Z-5 (estimated end point)
G1 X7 Z-6
```



Path Generated by Program 3

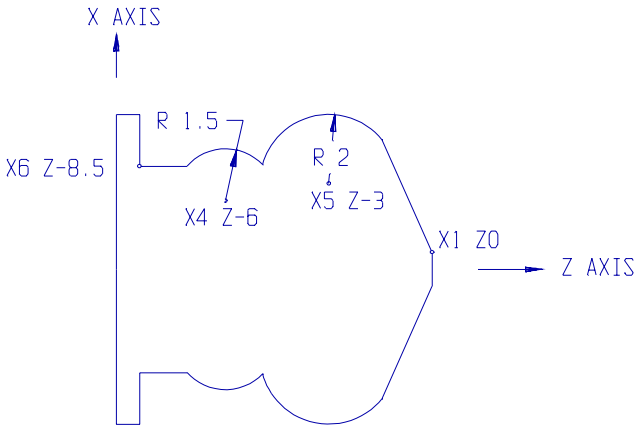
In general, when dealing with lines and arcs, if the line is programmed short of the arc it will be extended to the arc. If the line is programmed past the arc it will be shortened to the arc, and if the line does not intersect the arc it will be made tangent.

Program 4



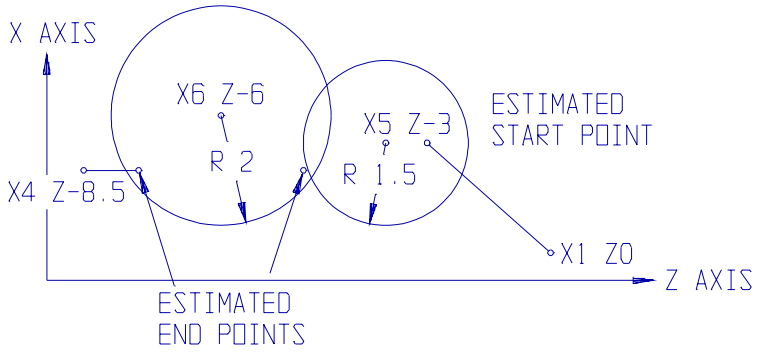
Programmed Path

```
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
```



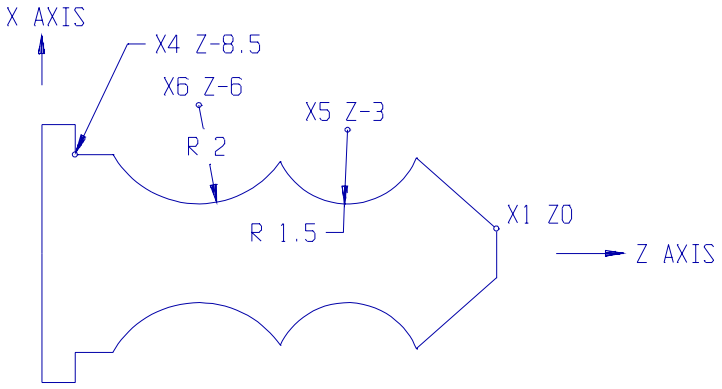
Path Generated By Program 4

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 XC4 ZC-6 X2 Z-7.5 (Estimated End Point)
G1 X2 Z-8.5
```



Path Generated by Program 5

Things To Remember When Estimating Points

** 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 must lie on the line; i.e. the slopes 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 closer 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.

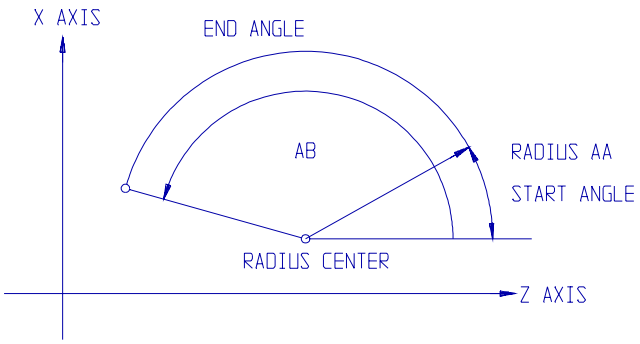
** When estimating the intersection of one arc to another arc, the easiest point to pick on an arc end point is at one of the quadrant points (0°, 90°, 180°, or 270°).

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



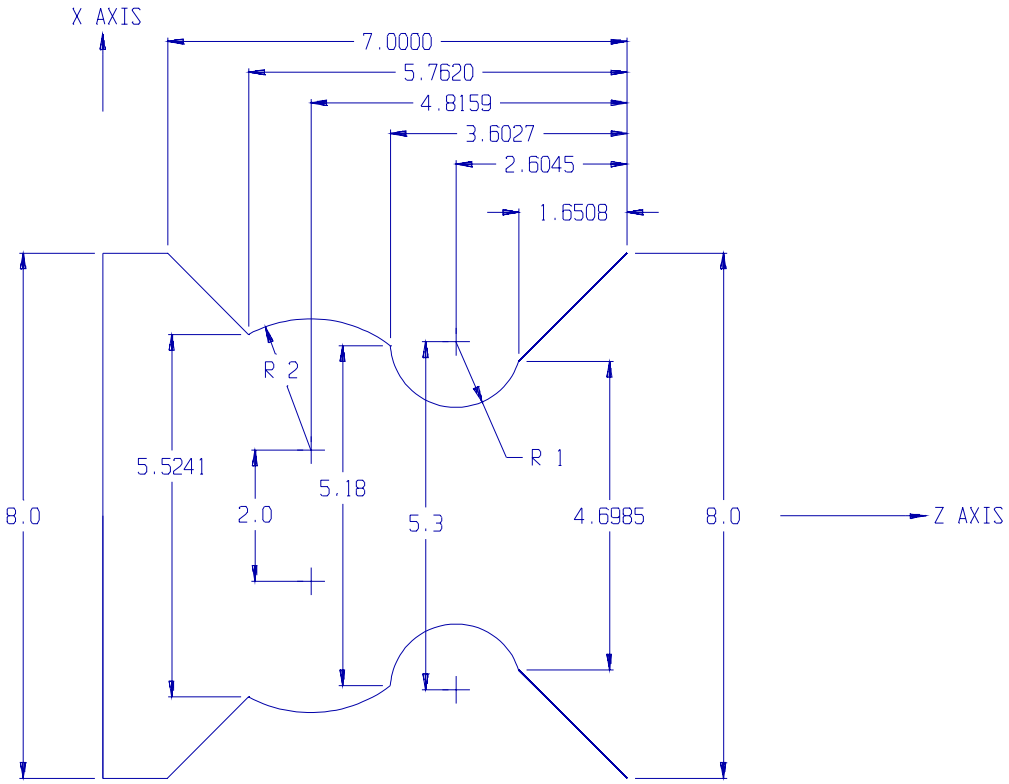
The polar format for arcs can be mixed with the cartesian formats. The following are legal formats.

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

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 W1.9519
G3 I-1.59 K-1.2132 U.3441 W-2.1593
G1 U2.4759 W-1.238
```

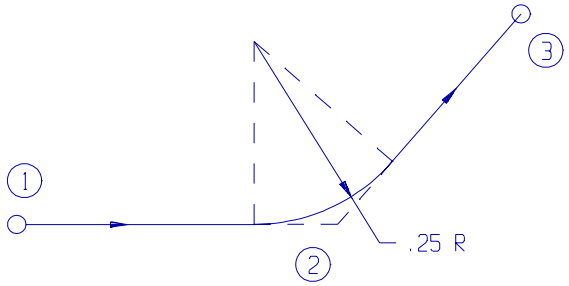
Note: In Incremental, Trig Help cannot be used as each point is related to the current position.

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.

3.3.1 Corner rounding

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

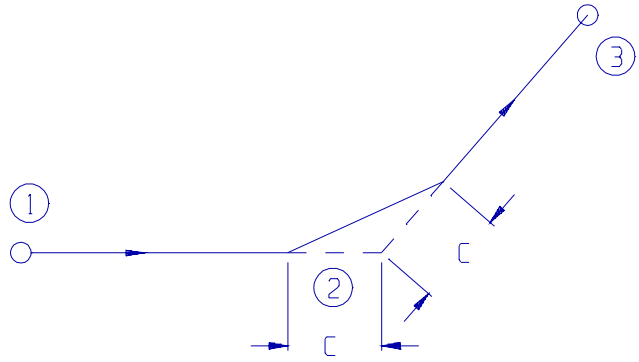
```
(1)  G91 G01 X0 Z0
(2)  Z1, R.25
(3)  X2 Z2
```



3.3.2 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
```

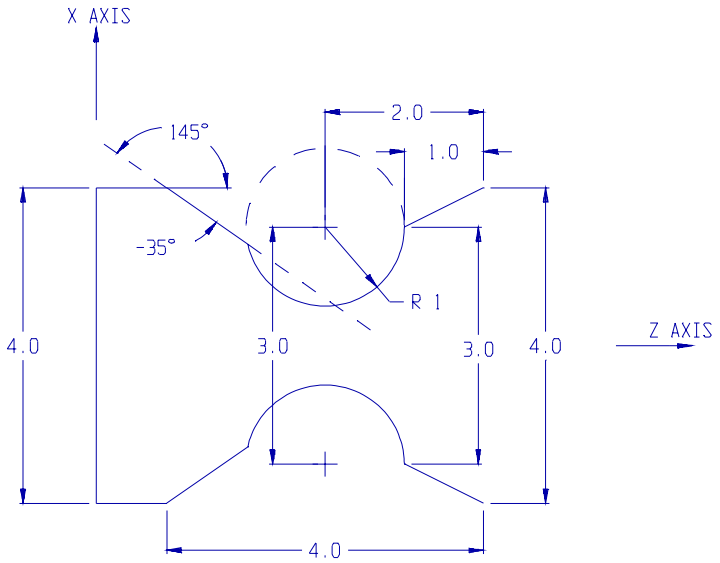


3.3.3 Back line

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 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
```

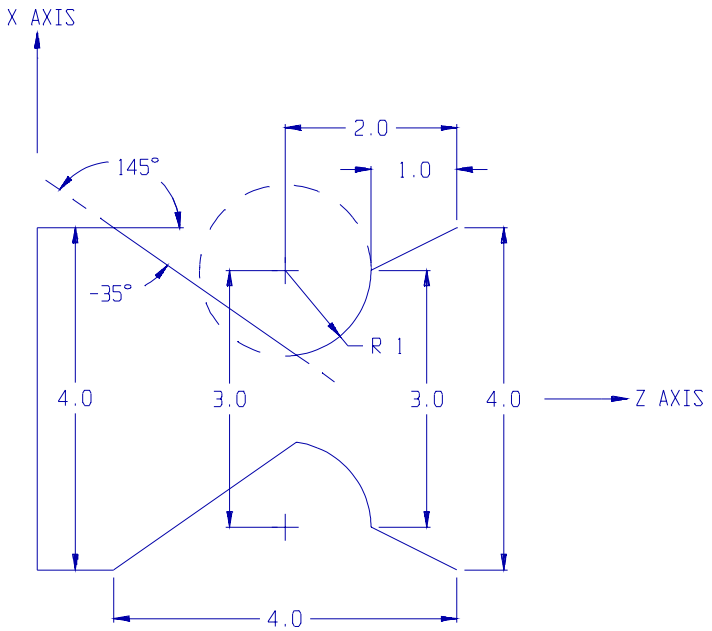


Back	extend line backwards from (4,-4)
C0	use the arc intersection closest to (4,-4)
W145	extend the line from (4,-4) at an angle of 145°

Example 2:

(1) X4 Z0
(2) X3 Z-1
(3) G2 R1 ZC-2 AB270

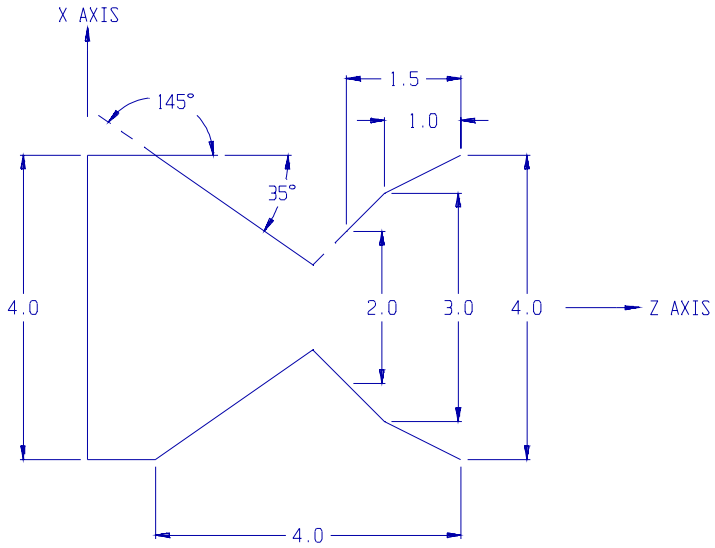
(4) G1 X4 Z-4 BACK C2 W145



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
(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 360° (a complete circle) is assumed.

G02I _____; (a complete circle)

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

G02R _____; (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.

3.4 Dwell command (G04)

The G4 code must be immediately followed by an FXXX.X instruction. This instruction will then cause the program to stop or dwell for XXX.X 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 an X can be used in place of an F following the G4 command.

3.5 Exact stop (G09)

Moves commanded in blocks with G09 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.

3.6 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 499 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 Appendix A on parameter assignments.

3.7 Clear floating zero (G12)

This function will zero out the floating zero's. See G50 section for set floating zero's.

3.8 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.

3.9 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. In metric the actual machine position may not exactly agree with the program position because of the conversion. Feedrate in the metric mode is in millimeters per minute (mmpm), or millimeters per revolution (mmpr).

Notes on Inch/Metric

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.

3.10 Safe zone on/off (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 tool cannot enter. If the tool is programmed into this area when the safe zone check is enabled, a safe zone error will be displayed. The safe zone is defined in the coordinate parameters. (Refer to the section on parameters.) The check is turned on with a G22 and off with a G23. The normal state of this check is off.

3.11 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 or two axes are commanded as intermediate points, only the commanded axis will move to there 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

Machine positions to: X1 Z-2

```
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:

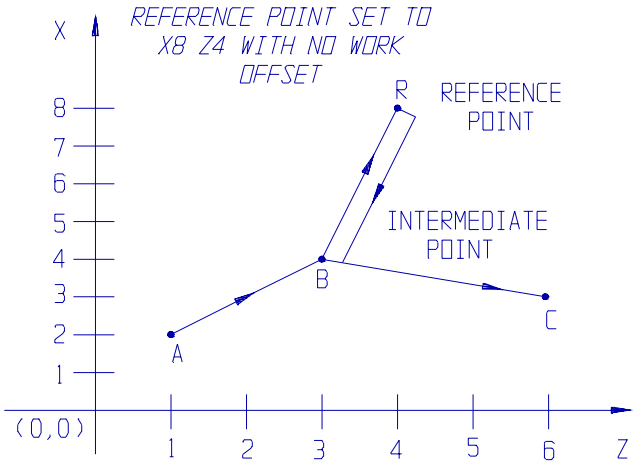
G29 X____ Z____

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  intermediate
   point       point
```

In no P is specified, P2 is assumed. P≤1 and P≥5 are illegal.

3.12 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).

3.13 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 subroutine. 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
```

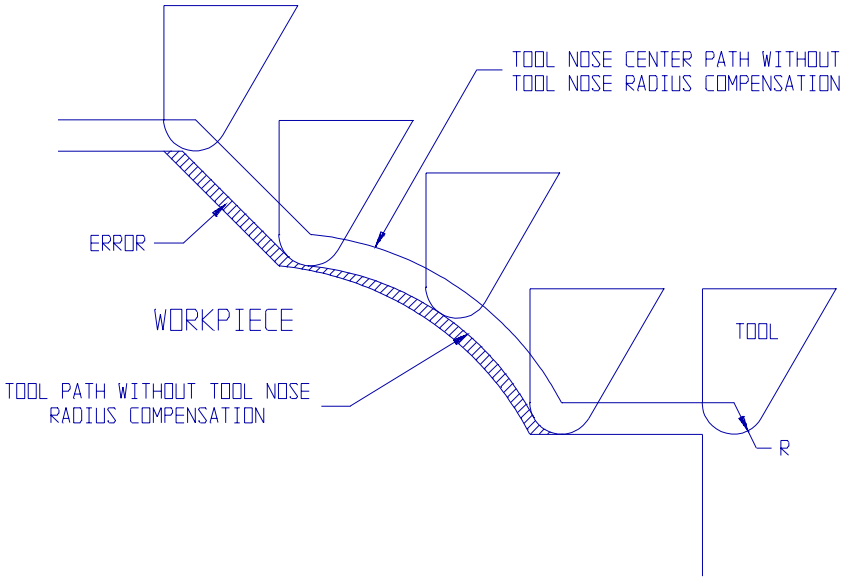
3.14 Cutter compensation (G40, G41, G42)

```
G40  Compensation Off
G41  Left Compensation
G42  Right Compensation
```

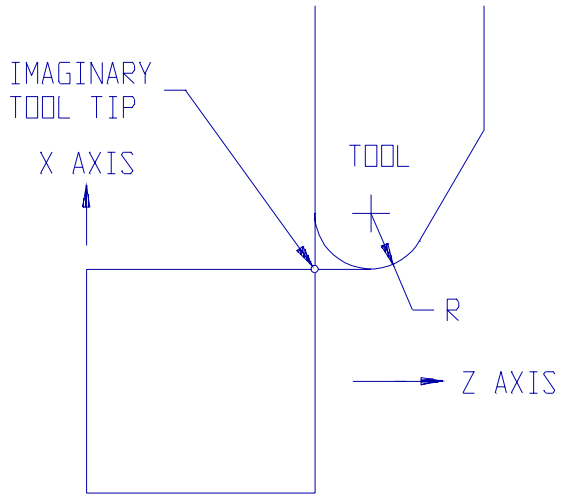
This section will explain how the cutter compensation works and give pointers on how to use it optimally.

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.

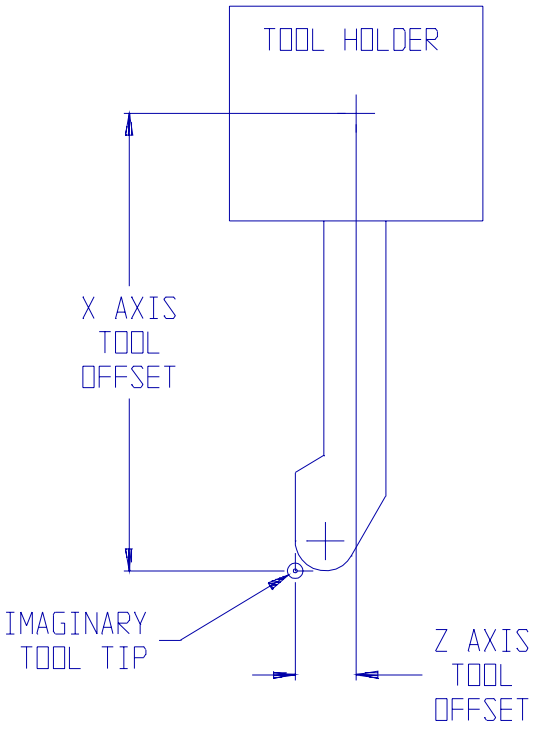


The above illustration shows the path taken by the tool without tool nose radius compensation. The imaginary tool tip follows the programmed path.

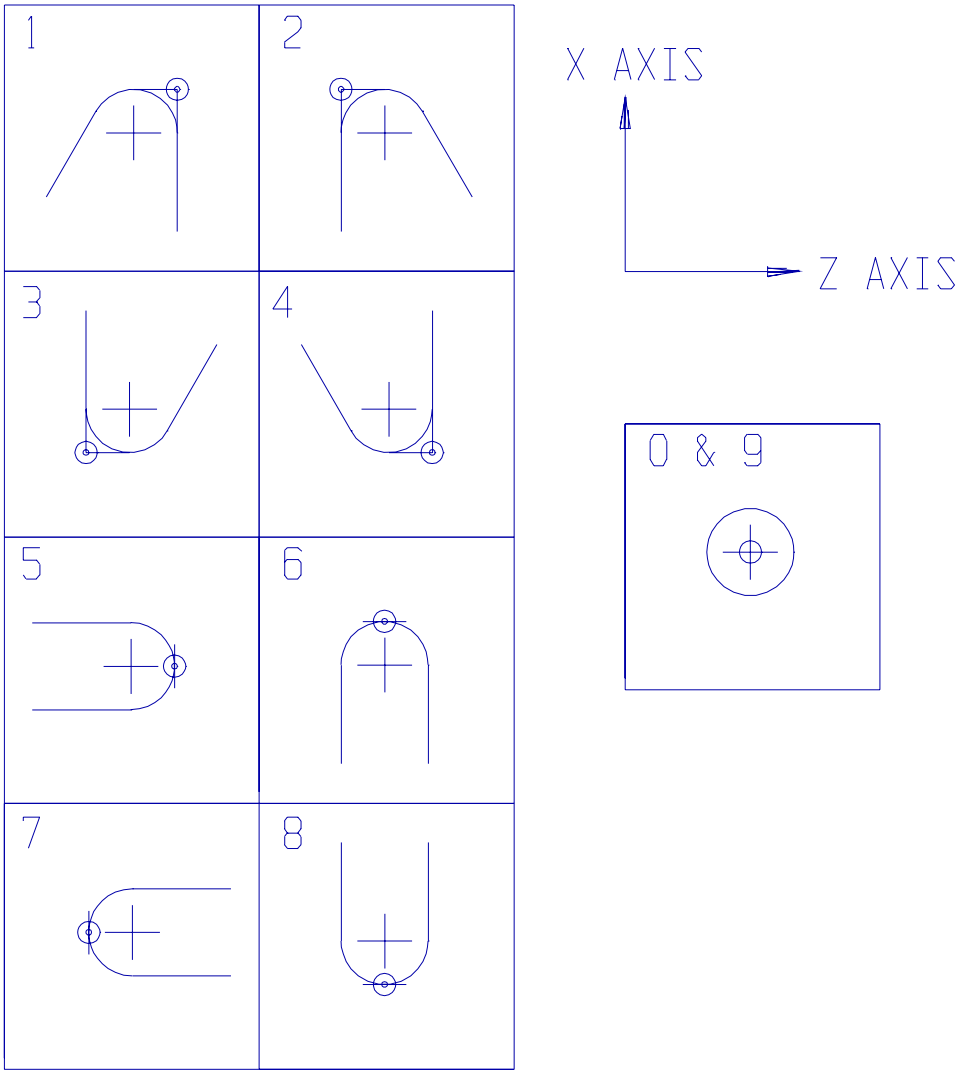


The imaginary tool tip does not actually exist. When position moves are commanded without tool nose compensation, the imaginary tool tip is positioned at the commanded position. The imaginary tool tip is defined by the tool length offsets.

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 ten different tool types are shown below.



Note: *The above tool displays are for systems with the front turret parameter set to number.*

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

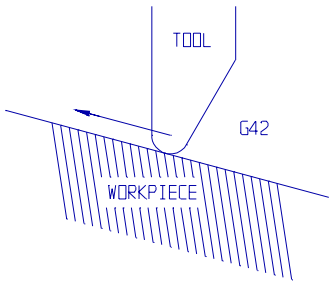
0. Tool Type #0
   *
   *
   *
1. Tool Type #1
   * Inside Turning (chuck toward tail stock)
   *
   *
2. Tool Type #2

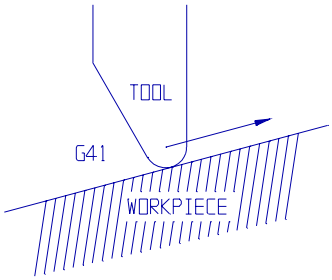
* Inside Turning (tail stock towards chuck)
*
*

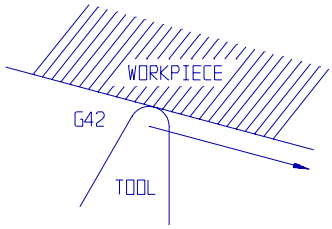
3. Tool Type #3
   * Turning (tail stock towards chuck)
   *
   *
4. Tool Type #4
   * Turning (chuck towards tail stock)
   *
   *
5. Tool Type #5
   *
   *
   *
6. Tool Type #6 and Inside Threading Tools
   * Inside Turning Using a Diamond Insert
   *
   * Boring Tools using Diamond Insert
7. Tool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

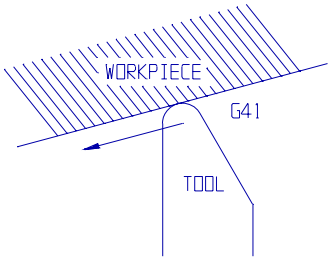
When tools are called using the "T" code, the format used is T####. The first two digits specify the Tool #; the last two digits specify the tool length offsets, tool tip radius and the tool type.

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.

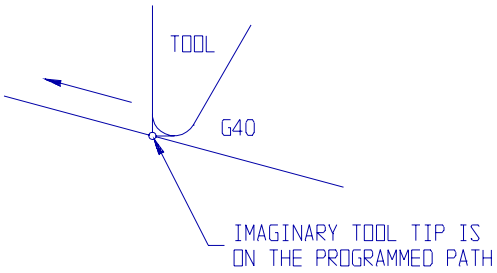




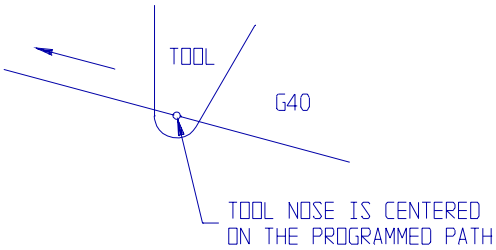




For Tool Types 1-8:



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.

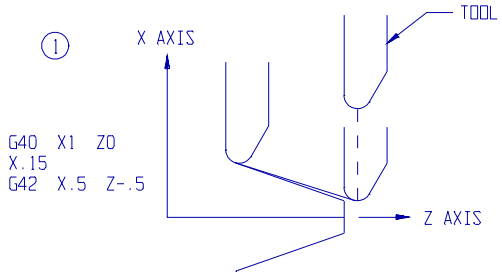


Figure 1 shows the cutter comp is not active until the G42 X.5 Z-.5 block is completed.

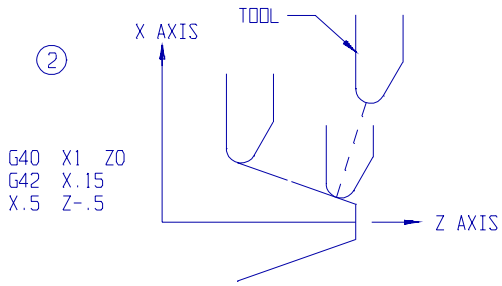


Figure 2 shows a problem on the G42 X.15 block. The desired position at the face of the part is not obtained.

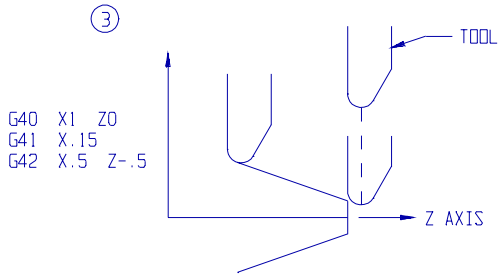
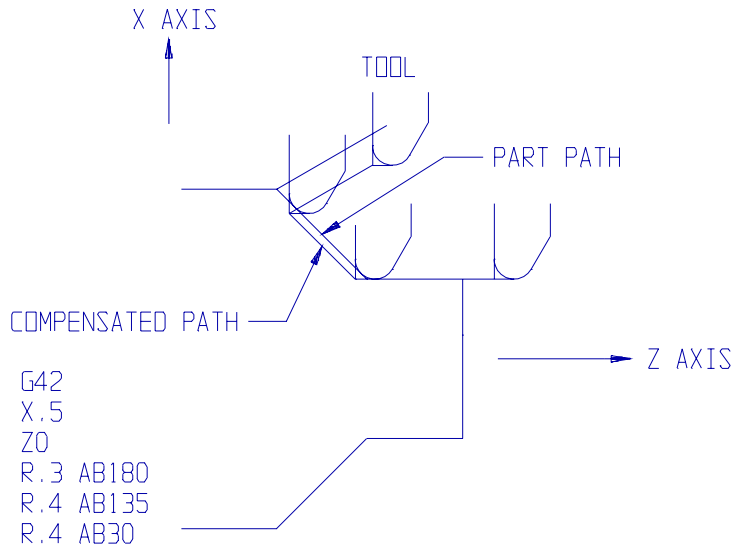
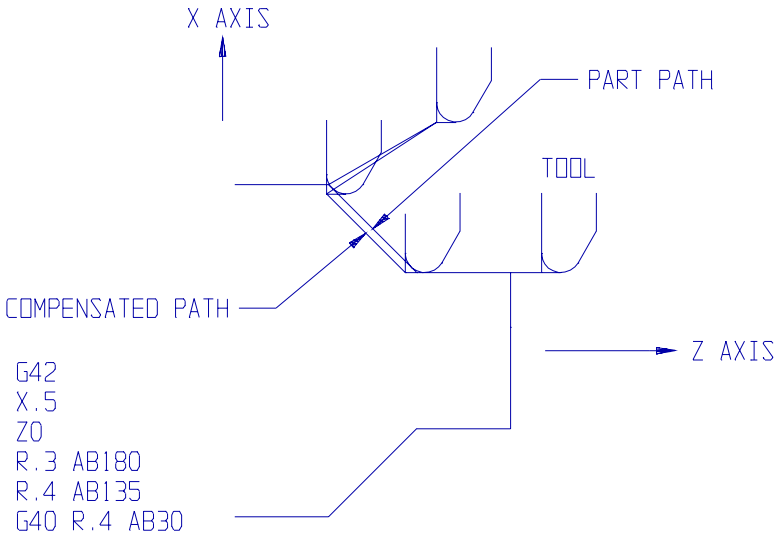


Figure 3 shows an approach that obtains the desired part.

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.

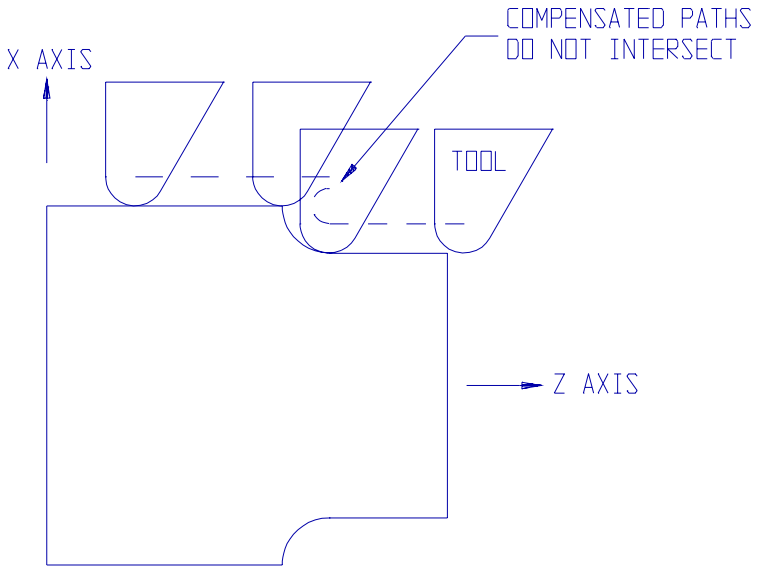
With three points, information on how they are connected, the cutter radius, and whether it is a left or a right compensation, the control can calculate the current compensated point. The control will also employ its Trig Help function discussed earlier to connect lines and arcs during cutter compensation.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

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.

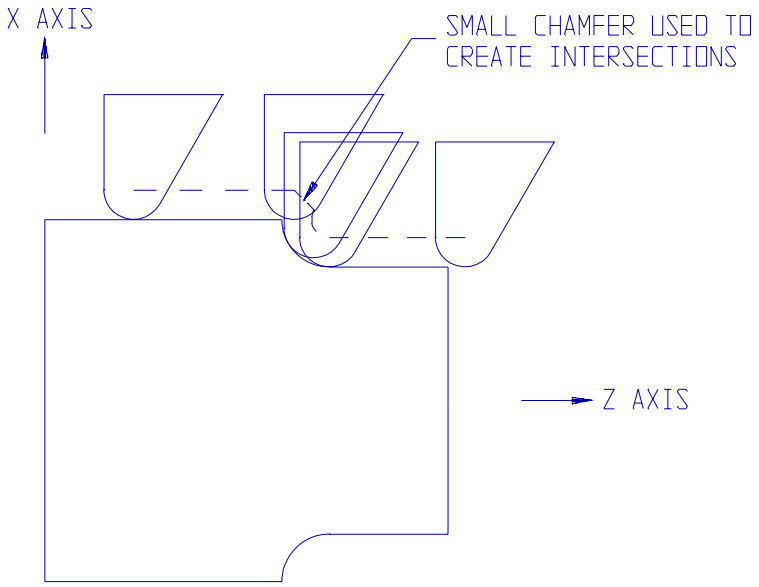
The compensation in this control is truly intersectional. Given these three points, the control calculates the intersection of the compensated path between the previous and the current programmed points and the compensated path between the current and the next programmed points. These paths can be a 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 or round corner at the point between the non-intersecting surfaces.



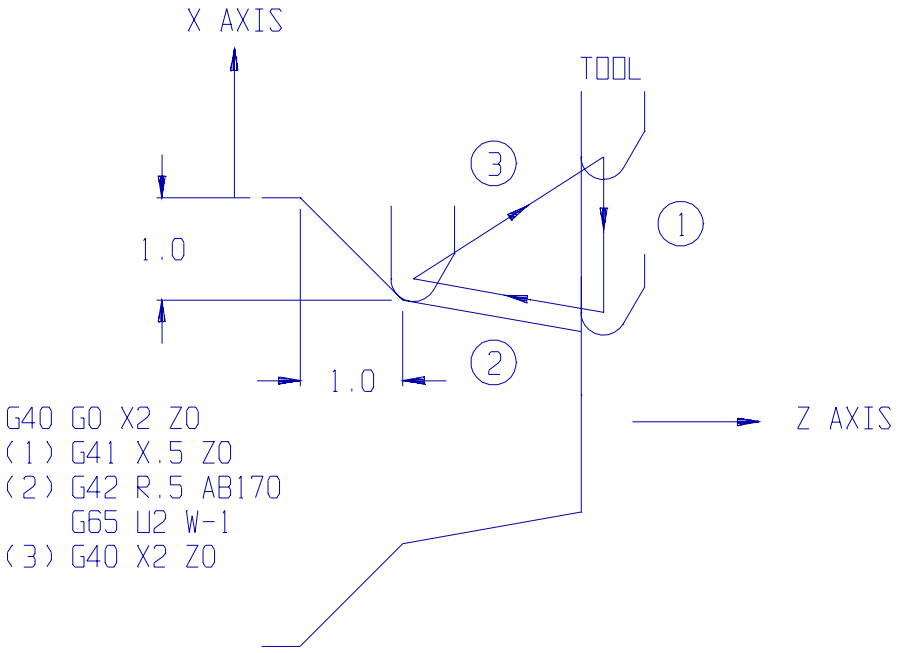
3.15 Non-movement (G65) Starting and Ending Cutter Compensation

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:

* Addresses G, H, M and T do not pass arguments to programs called. Address P does assign its value parameter #16.

* If the program specified by address P does not exist an error will occur.

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

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

Notes on Cutter Compensation

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

** There is no restriction on how many successive blocks can have no axis information.

** There is no restriction on how lines enter and leave arcs. They can have any angle of intersection as long as an intersection exists.

** 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.

** If the programmed point rather than the compensated point is desired, a G40 command should be added to the block containing that point. The G40 could have been a part of the previous block, providing that block had no axis information.

** Similarly, if the point after the desired programmed point is to be compensated, a G41 or G42 should be added to the block containing that next point or to any non-axis command block between the two.

** Roughing and finishing passes can be easily made by first entering a tool radius value larger than the actual measured tool radius by the amount of stock to be left on the part for the finish pass.

When the program is run, the resultant part will be oversize. Now by entering the actual cutter radius into the system and running the program once more, the finished part size will be obtained.

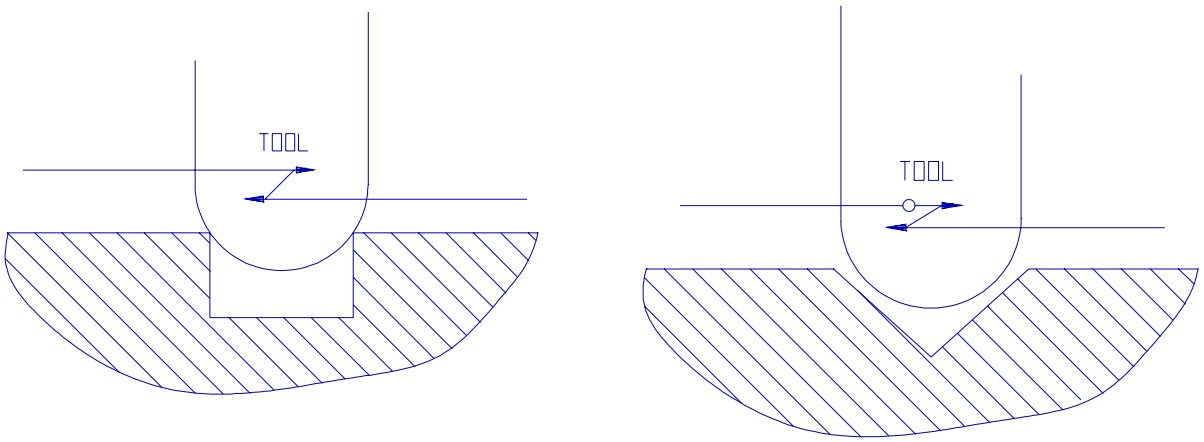
Thus by entering a **larger** or **smaller** tool tip radius, the part can be made **under** or **oversize**.

** 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.

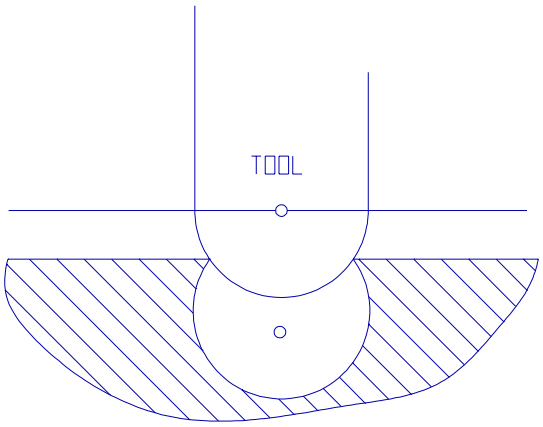
** Cutter compensation can be shut off by setting bit 4 of the special flag parameter.

** A G41 with a negative tool tip radius is the same as a G42 with a positive tool radius.

** 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 vee.



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

3.16 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.

3.17 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.

3.18 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.

3.19 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.

3.20 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 and wear offset table for tool #15.

* *Normally the tool # and the tool offset # are the same.*

* *Offsets 0 are all zero.*

* *Legal tool #'s and offset #'s are 0 thru 24.*

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.

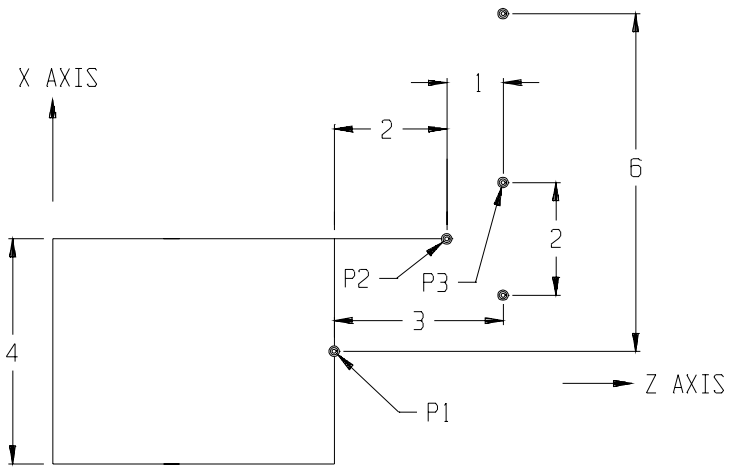
3.21 Floating zero (G50)

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. If this is not desirable, then a new G50 must be set when changing coordinate systems. G50 offsets are zeroed on power up and after homing the machine.

3. On power up and at the start of every program 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.

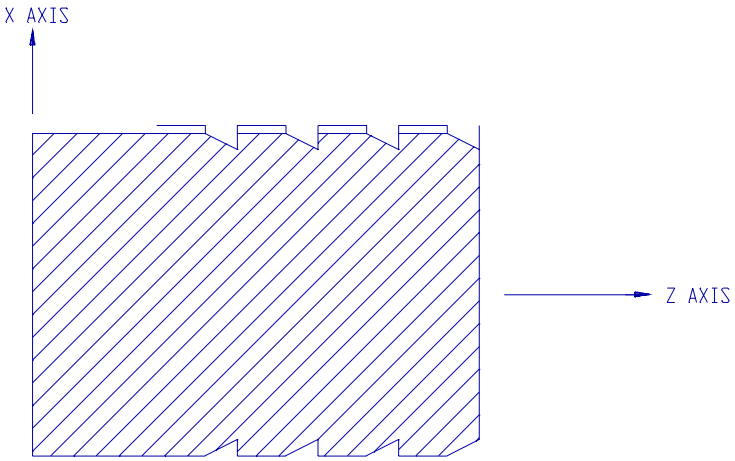
5: The chuck can be assumed Z0, but all examples in the manual assume the face of the part as Z0.

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.

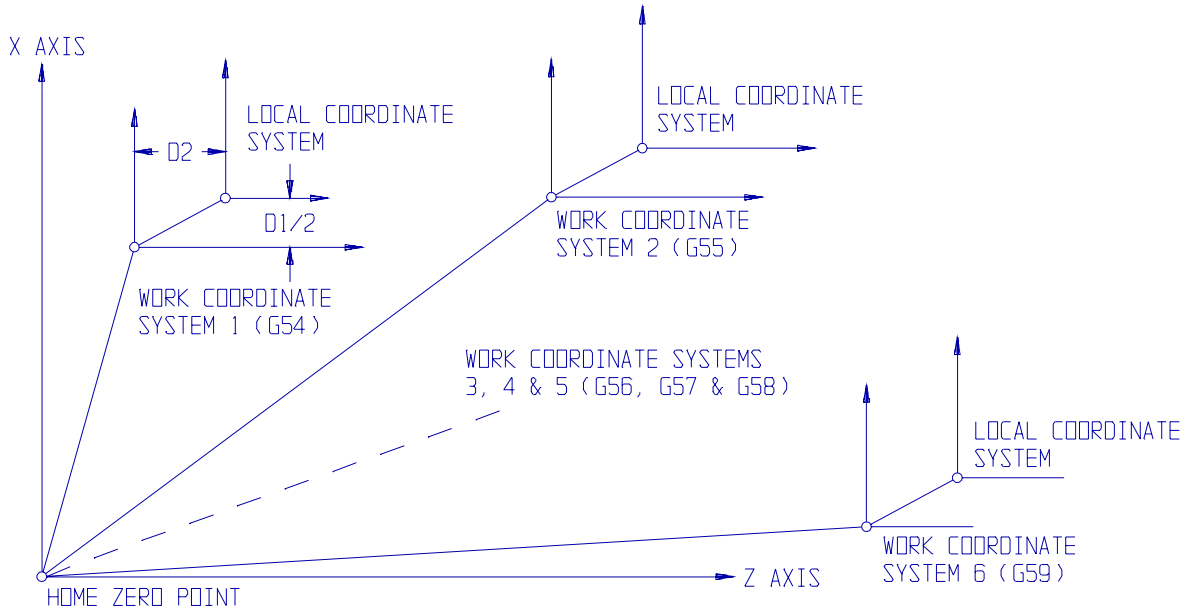
Example:

```
G0 X2.1 Z0
G0SUB 1 L4
M30
N1
G50 X2.1 Z0
X1.8
X2 Z-.2
G0 X2.1
Z-.5
Return
```



3.22 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.*

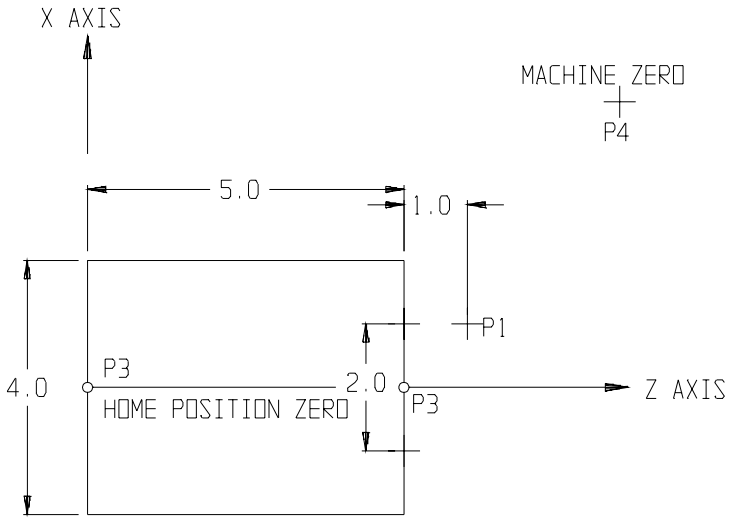
2. *To cancel a G52, enter G52 X0 Z0.*

3. *G52 offsets <u>are not</u> affected by the position of the machine. G50 offsets <u>are</u> affected by the position of the machine.*

4. *G52 offsets are zeroed on power up, after homing, after any G50 command, and at the start of any program.*

3.23 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:

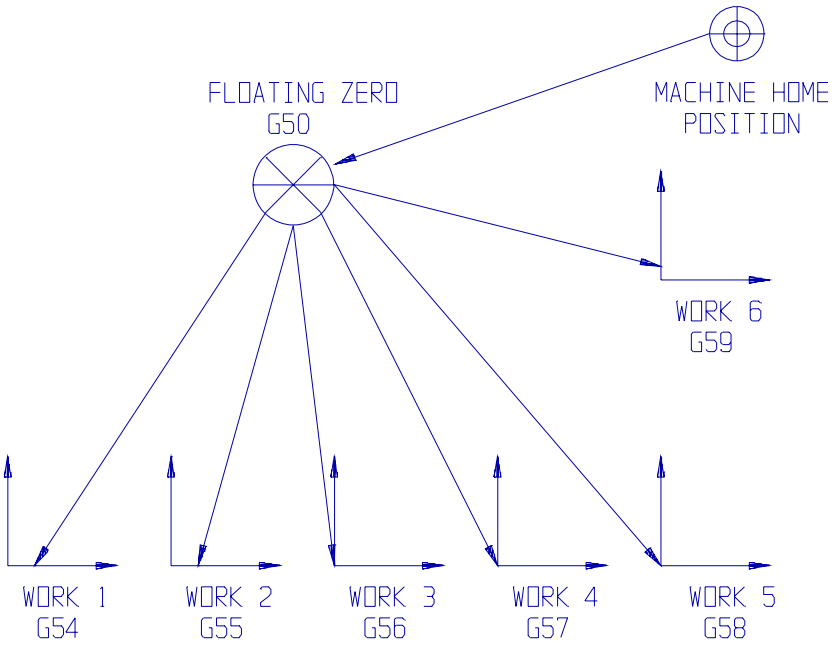
```
X0 Z0 move to P3
X0 Z5 move to P2
G50 X0 Z0
X2 Z1 move to P1
G53 X0 Z0 move to P4
X0 Z-5 move to P3
```

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.

3.24 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 "PARMS" on the main menu, then "COORD". 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
                and is active upon starting a program
```

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

Note #2: See section 5 (Handwheel) for a procedure for setting work and tool length offsets.

3.25 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.

3.26 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.

3.27 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.

3.28 Cutting mode (modal) (G64)

G64 is the default at the beginning of each program. When G64 is commanded, deceleration at the end point of each block thereafter is not performed and cutting goes on to the next block. This command is valid until G61 (exact stop mode) or G63 (tapping mode) is commanded.

3.29 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.

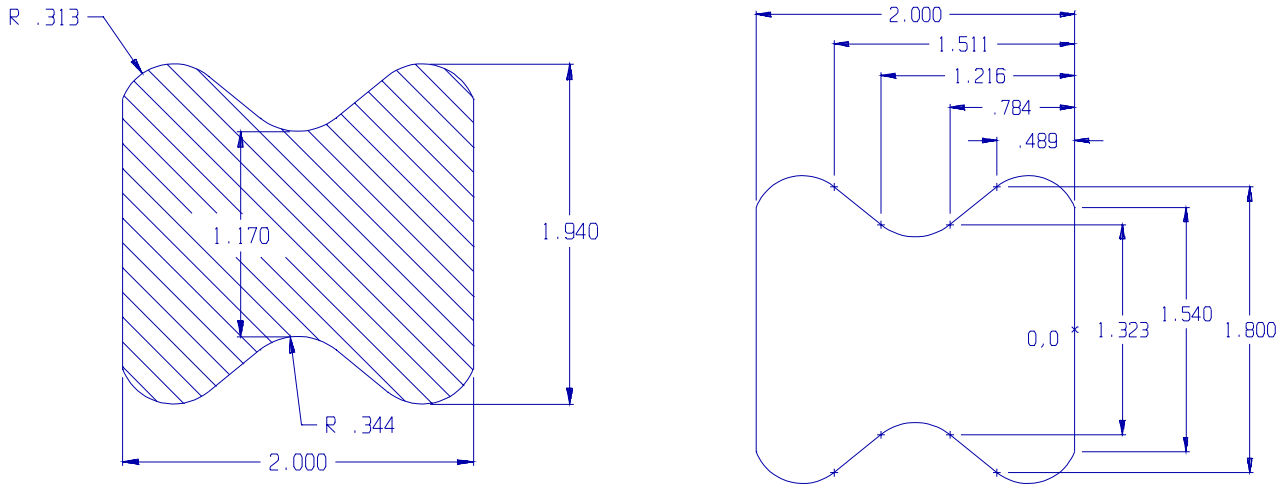
Code used for Finish Cycle:

```
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

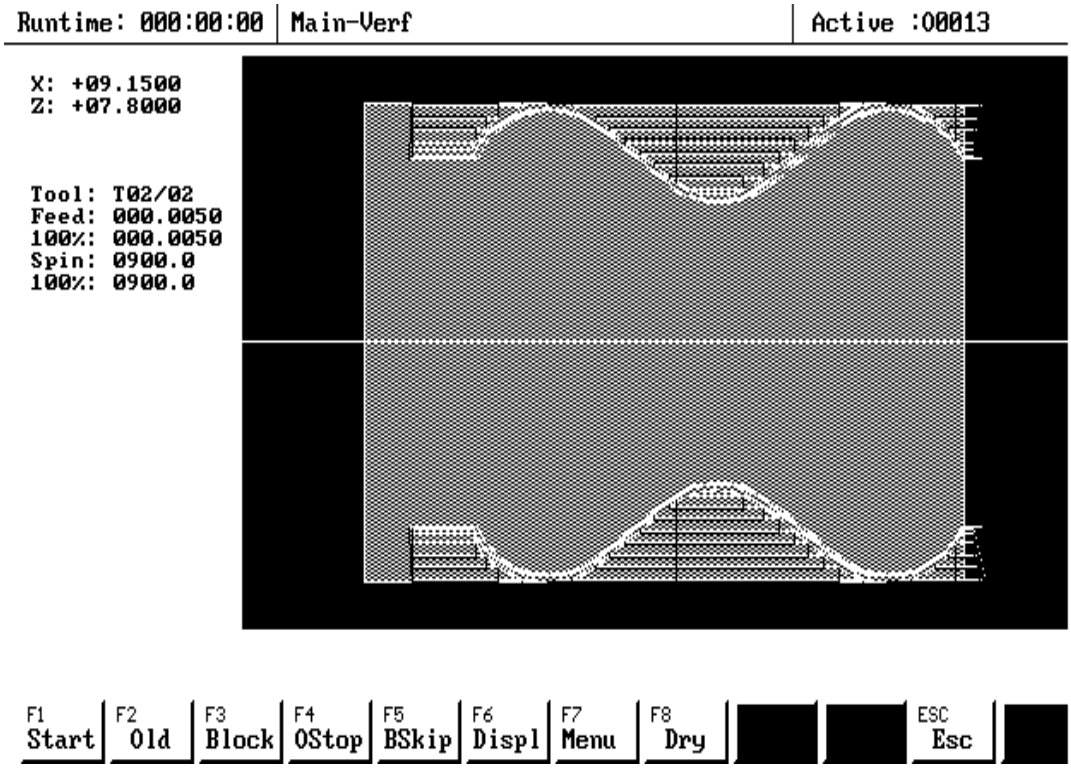


Example program using a rough turning cycle with a finish cycle for the part shown above:

```
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
N2  (End Block)

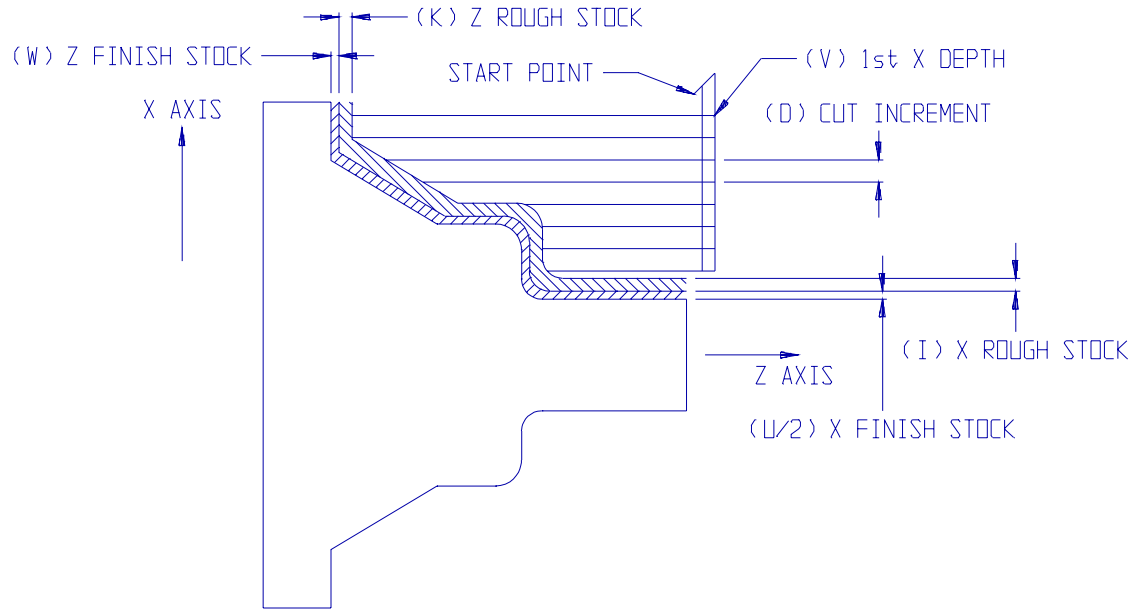
T0202 M6
G99 F.005
G50 S2000
G96 S900
M3
M8
G0 X2.1 Z.1 (Start Point)
G70 P1 N2 (Finish Cycle)
```



3.30 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

G71 U##.#### W##.### I##.### K##.#### V##.#### D##.## R##.### P#### Q####
      3        4       5        6     7          8     9      10     11

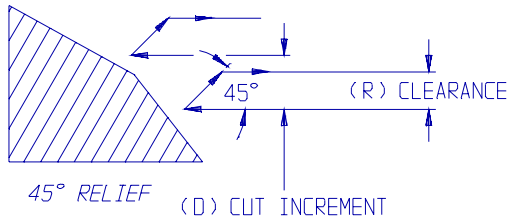
N####
  10
  .          Commands between the Start Block and the End Block
  .          define the shape to rough out.
N####
  11
```

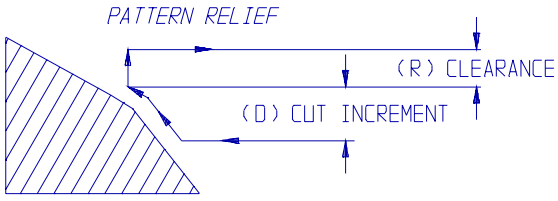
1. (X,Z) Start Point:	X and Z return to the start point at the end of the cycle.
2. Relief (PATTERN or 45 DEGS.):	The type of relief made at the end of the cutting. P117=0 (PATTERN) will cut along the pattern of the part, P117=0 (45 DEGS.) will retract at a 45 degree angle before retracting Z for the next

cut. If the relief type is 45 DEGS., no pockets are allowed in the pattern (radius dimensions).

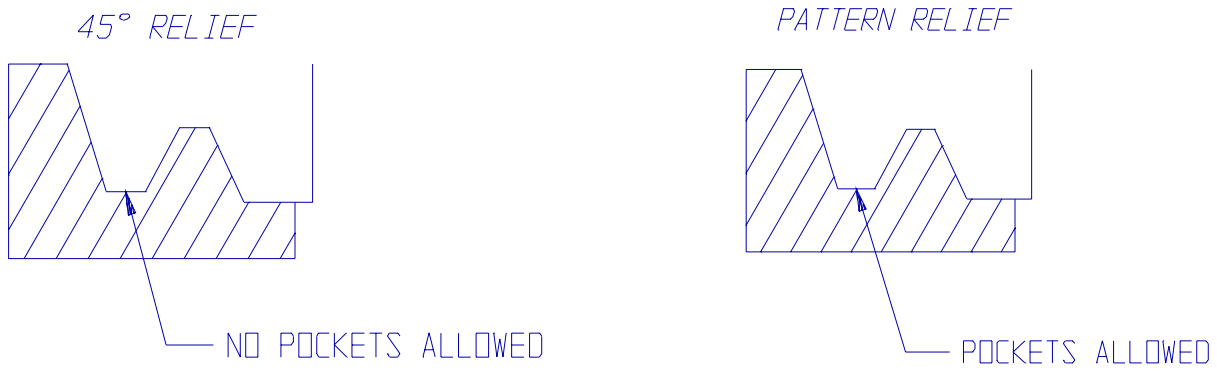
3. (U) X Finish Stock: Distance and direction of the finish diameter dimension allowance in the X axis.
4. (W) Z Finish Stock: Distance and direction of the finish allowance in the Z axis.
5. (I) X Rough Stock: Distance and direction of rough finish allowance in the X axis (radius dimension).
6. (K) 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 0 and 9999.)
11. (Q) End Block: Designates the end of the geometry to rough. (Any number between 0 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.

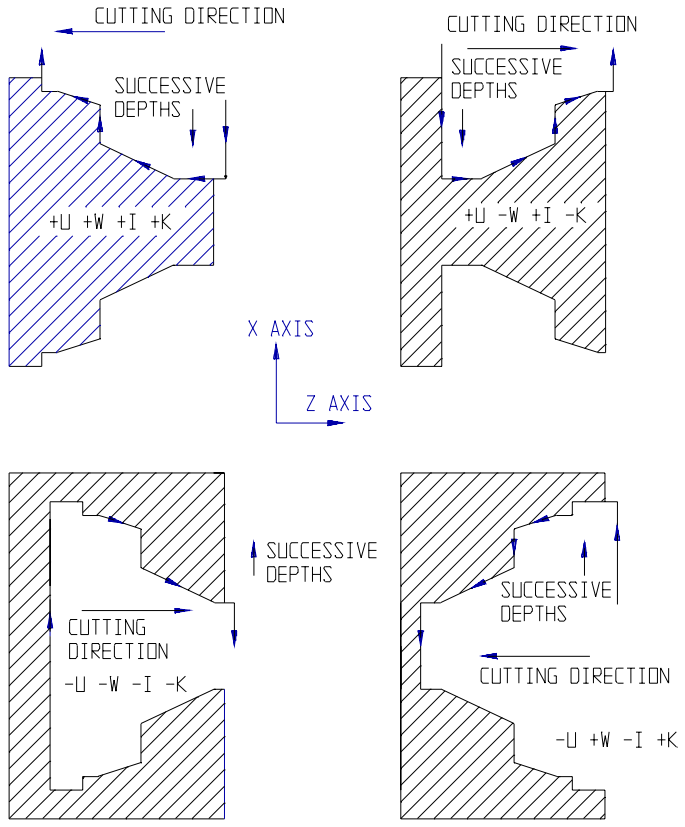




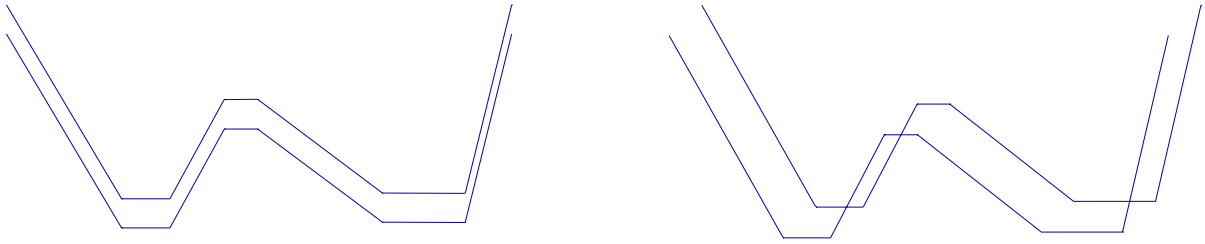
After rough cutting parallel to the Z axis there are two types of relief. 45° relief will rapid away from the pattern at a 45° angle. Pattern relief follows the pattern and then retracts in the X axis. A 0 (zero) clearance is allowed in the pattern relief.



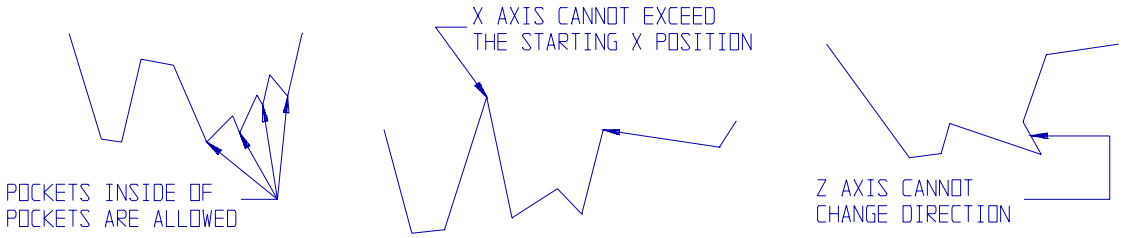
No pockets are allowed when using the 45° relief.



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.

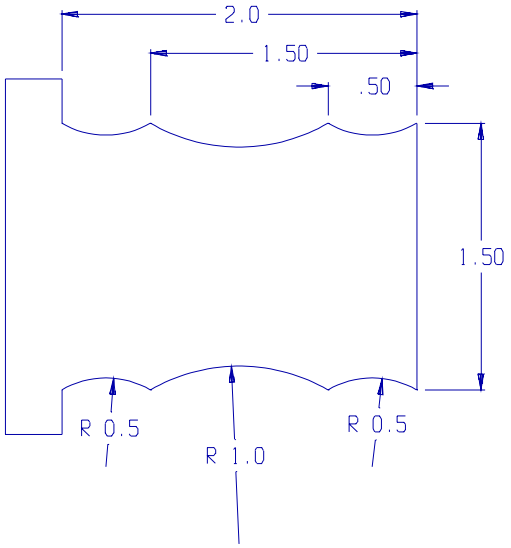


Note 1: *Moves back to the start point from the end of the pattern are rapid moves.*

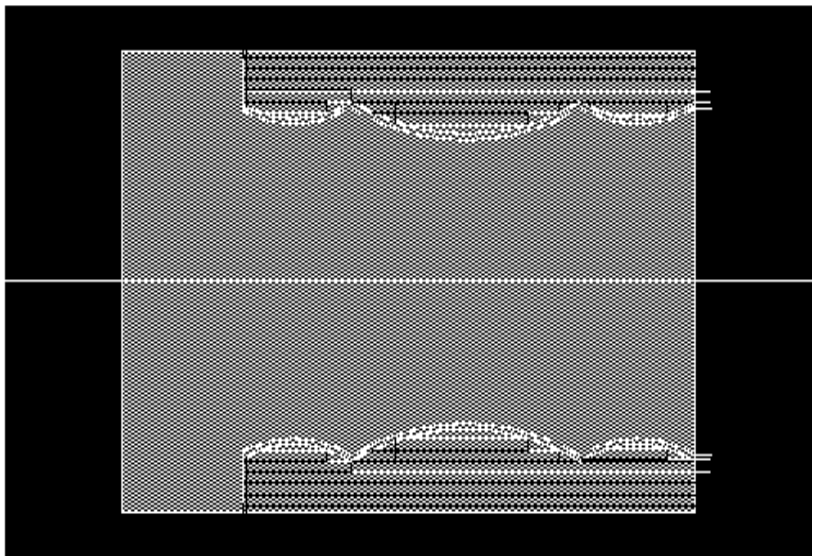
Note 2: *If the X end point in the pattern is not equal to the X start point of the pattern, a shoulder to the X start point will be added.*

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



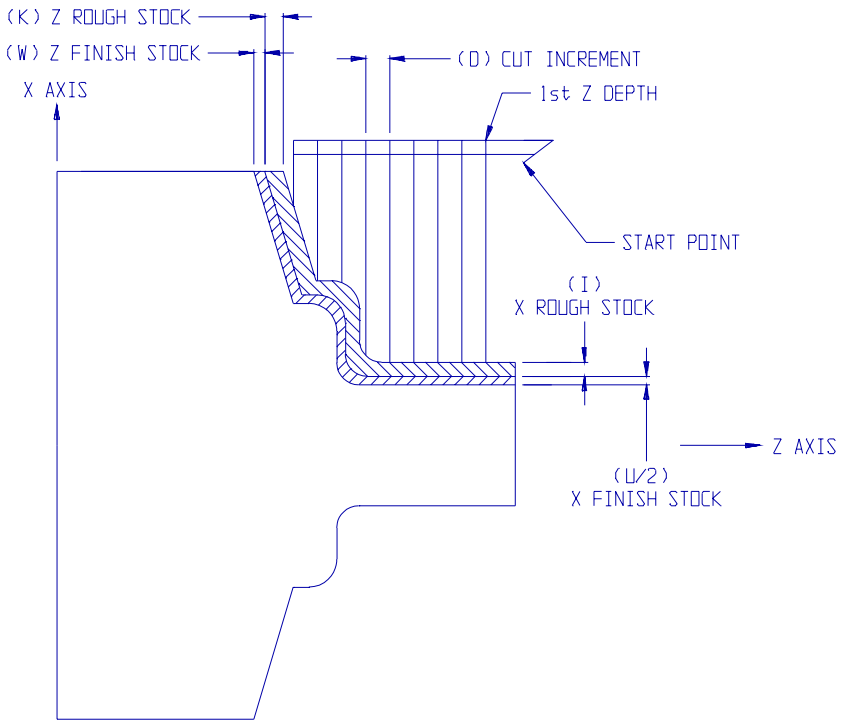

```
G0 X2.1 Z.1  (Start Point)
P117=0 (Use Pattern Relief)
G71 P1 Q2 U0 I.03 K0 R0 V1.95 D.05 (Roughing Information)
N1  (Start Block)
G42
G0 X1.5
G1 Z0
G2 R.5 Z-.5 (Geometry)
G2 R1 Z-1.5
G2 R.5 Z-2
G1 X2
N2 (End Block)
```



Graphics from Example of Rough Turning Program

3.31 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##.####

     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.

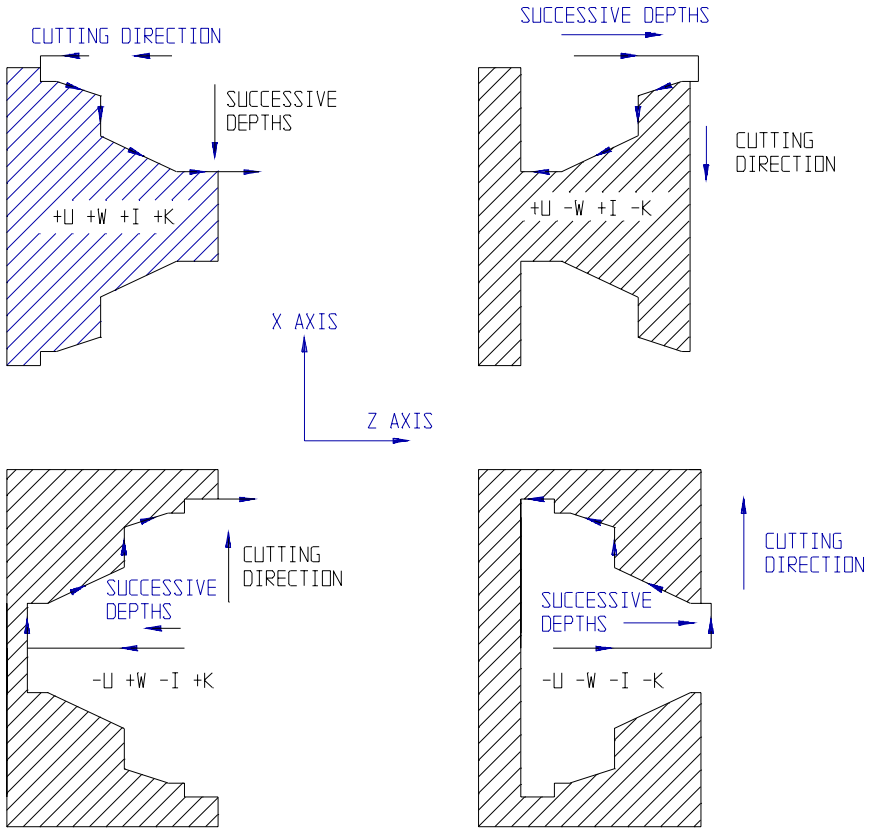
9. (R) Clearance: Distance of the 45 degree relief or an additional distance to pull away after the pattern relief.

10. (P) Start Block: Designates the start of the geometry to face. (Any number between 0 and 9999.)

11. (Q) End Block: Designates the end of the geometry to face. (Any number between 0 and 9999.)

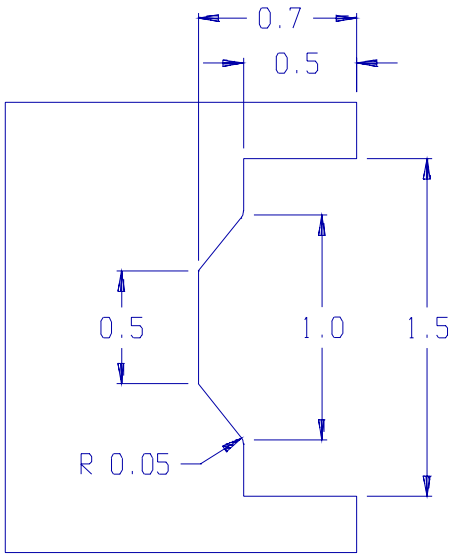
Notes: 1. The start block number should not be the same as the ending block number.

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

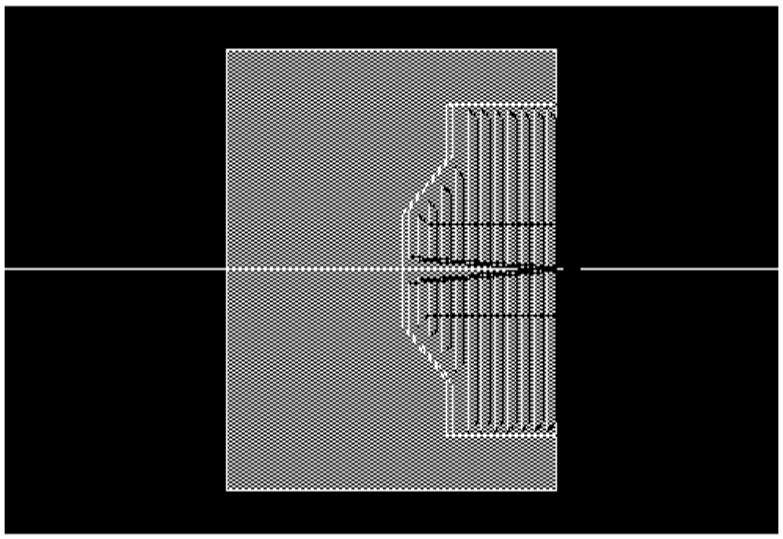


These are four patterns made by the rough facing cycle. Rough cutting for each of these patterns is done parallel to the X axis. The signs of U, W, I and K are shown for each pattern.

Example Rough Facing Cycle Program:



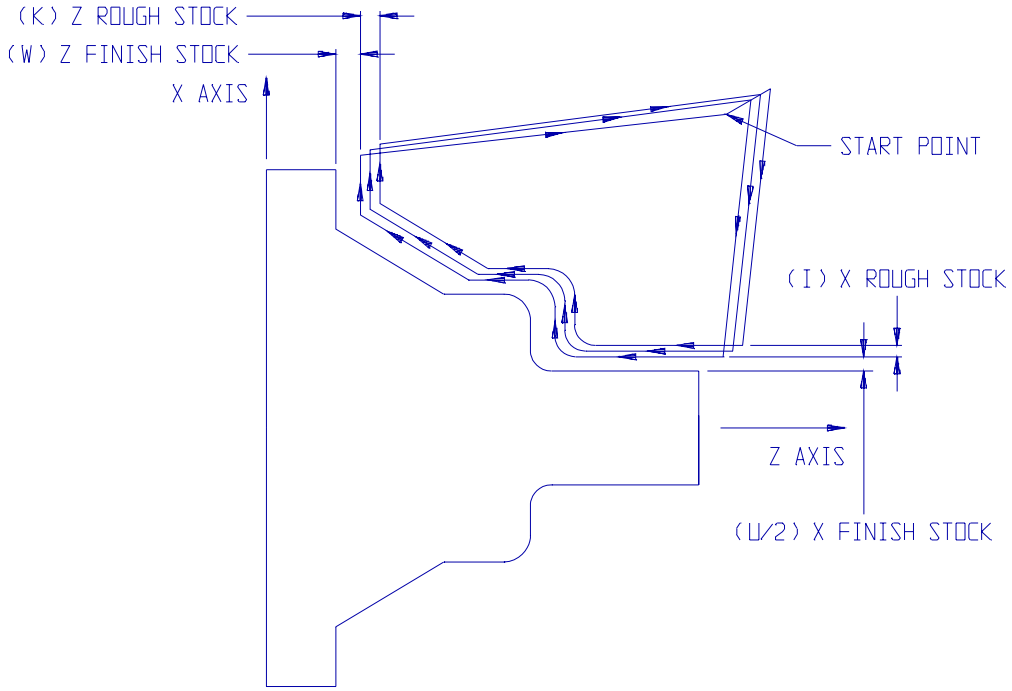

```
G0 X0 Z.1 (Start Point)
P117=1  (45° Relief)
G72 P1 Q2 U0 W0 I-.02 K.04 R.04 V-.04 D.06 (Roughing Information)
N1  (Start Block)
G42
G0 Z-.07          .
G1 X.5            .
X1 Z-.5, R.05   Geometry
X1.5              .
Z0                .
N2 (End Block)
```



Graphics From Example of Rough Facing Program

3.32 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. (P) Start Block: Designates the start of the geometry to cut. (Any number between 0 and 9999.)

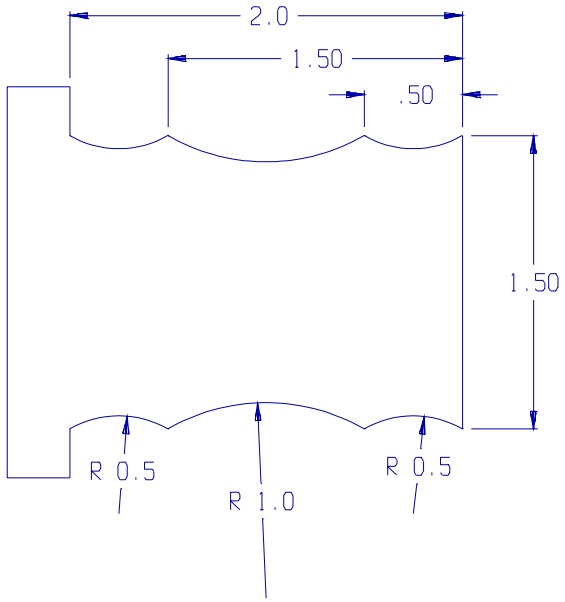
2. (U) X Finish Stock: Distance and direction of the finish allowance in the X axis.
3. (W) Z Finish Stock: Distance and direction of the finish allowance in the Z axis.
4. (I) X Rough Stock: Distance and direction of the rough allowance in the X axis.
5. (K) Z Rough Stock: Distance and direction of the rough allowance in the Z axis.
6. (D) Number of Passes: The number of passes to cut the rough stock.
7. (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.
8. (Q) End Block: Designates the end of the geometry to cut. (Any number between 0 and 9999.)

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

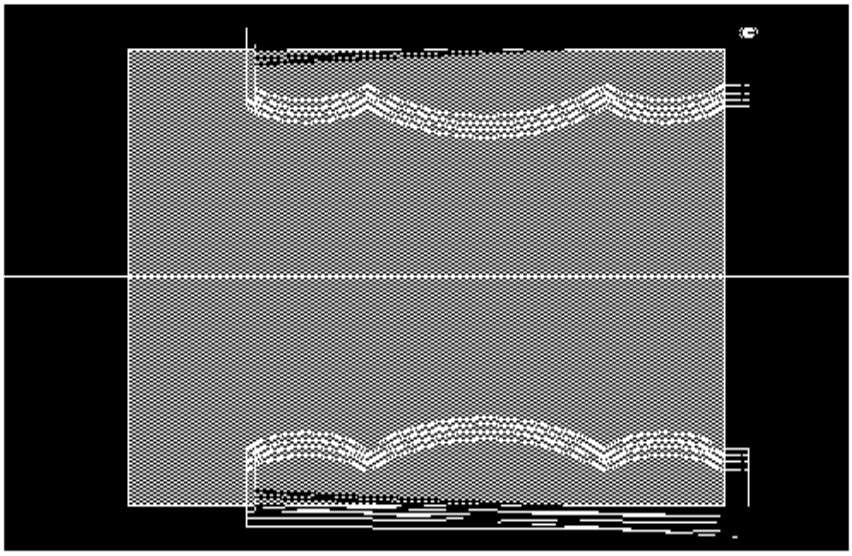
Note 2: Rapid moves within the pattern will be executed as rapid moves during the pattern repeat cycle.

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

Example of Repeat Program:

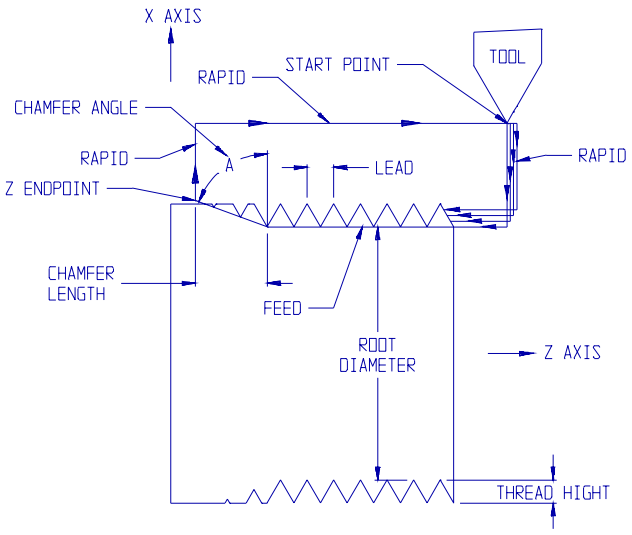


```
G0 X2.1 Z.1                (Start Point)
G73 P1 Q2 I.095 K0 W0 D4   (Repeat Information)
N1                         (Start Block)
G42                  .
G0 X1.5              .
G1 Z0                .
G.2 R.5 Z-.5         .
G2 R1 Z-1.5      (Pattern)
G2 R.5 Z-2           .
G1 X2
N2                 (End Block)
```



Graphics from Example of Repeat Cycle Program

3.33 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.

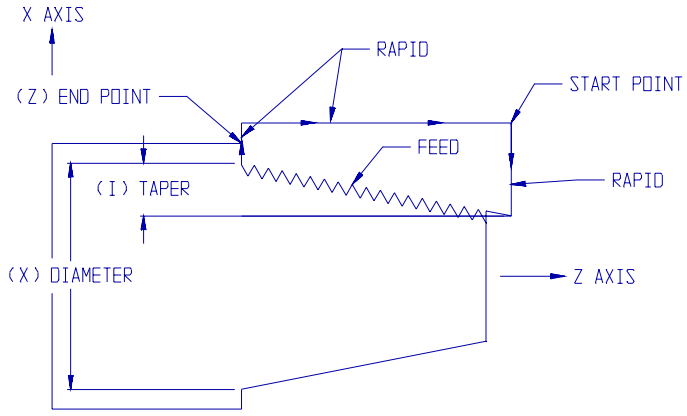
11. (D) First Cut Amount: Cutting depth of the first cut (radius value).

12. Tool Nose Angle: The read angle of the tool nose.

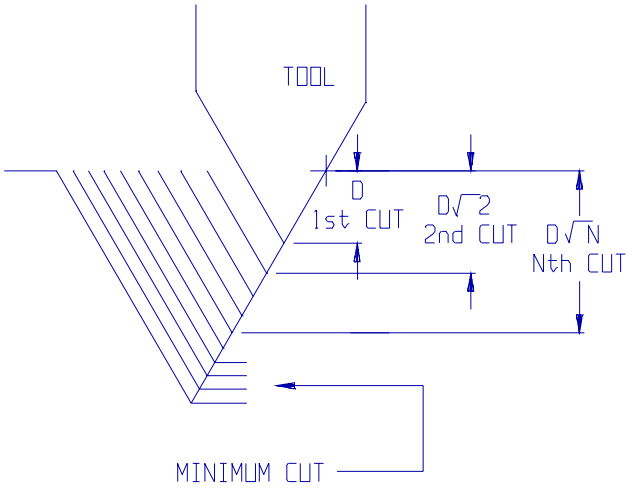
13. Cutting Method: CONSTANT AMOUNT/1 EDGE cutting amount constant, single edge cutting.
CONSTANT AMOUNT/BOTH EDGES cutting amount constant, both edges cutting.
CONSTANT DEPTH/BOTH EDGES cutting depth constant, both edges cutting.

14. (K) Thread Height: Height of the thread (X direction). K is specified as a radius value.

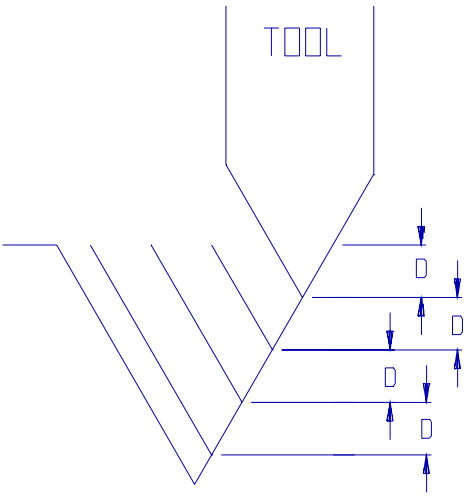
Inside or outside threads are determined by the start point relative to the root diameter.



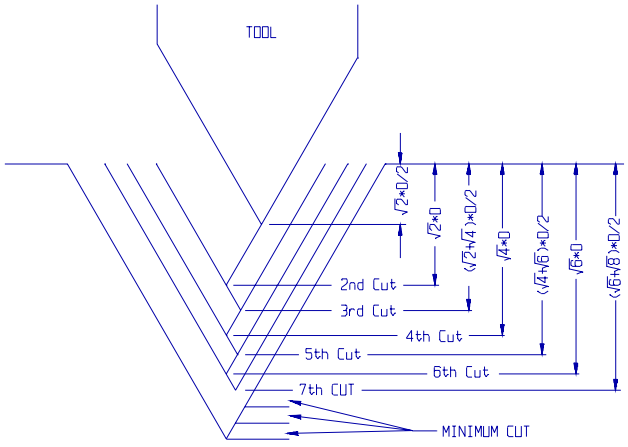
The four cutting methods are shown below:



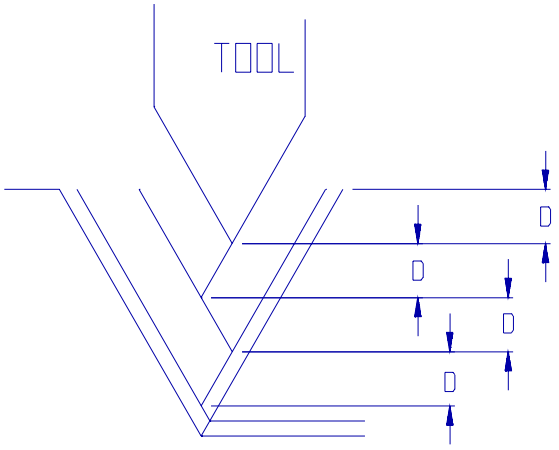
P1: Constant Amount/1 Edge



P3: Constant Depth/1 Edge



P2: Constant Amount/Both Edges

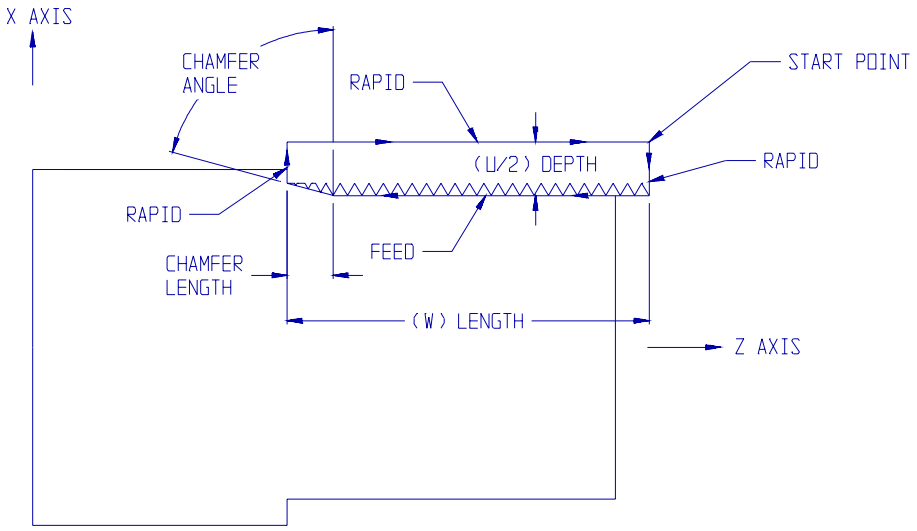


P4: Constant Depth/Both Edges

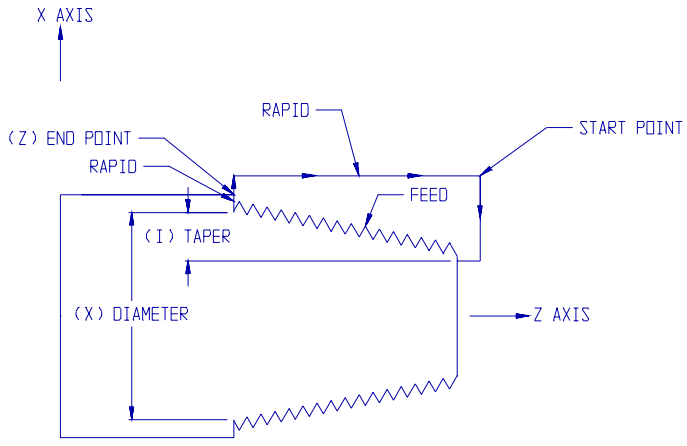
If the cutting depths using constant amount one edge, or constant amount both edges, become less than the minimum cut, the cutting depths are clamped at the minimum cut.

3.34 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:

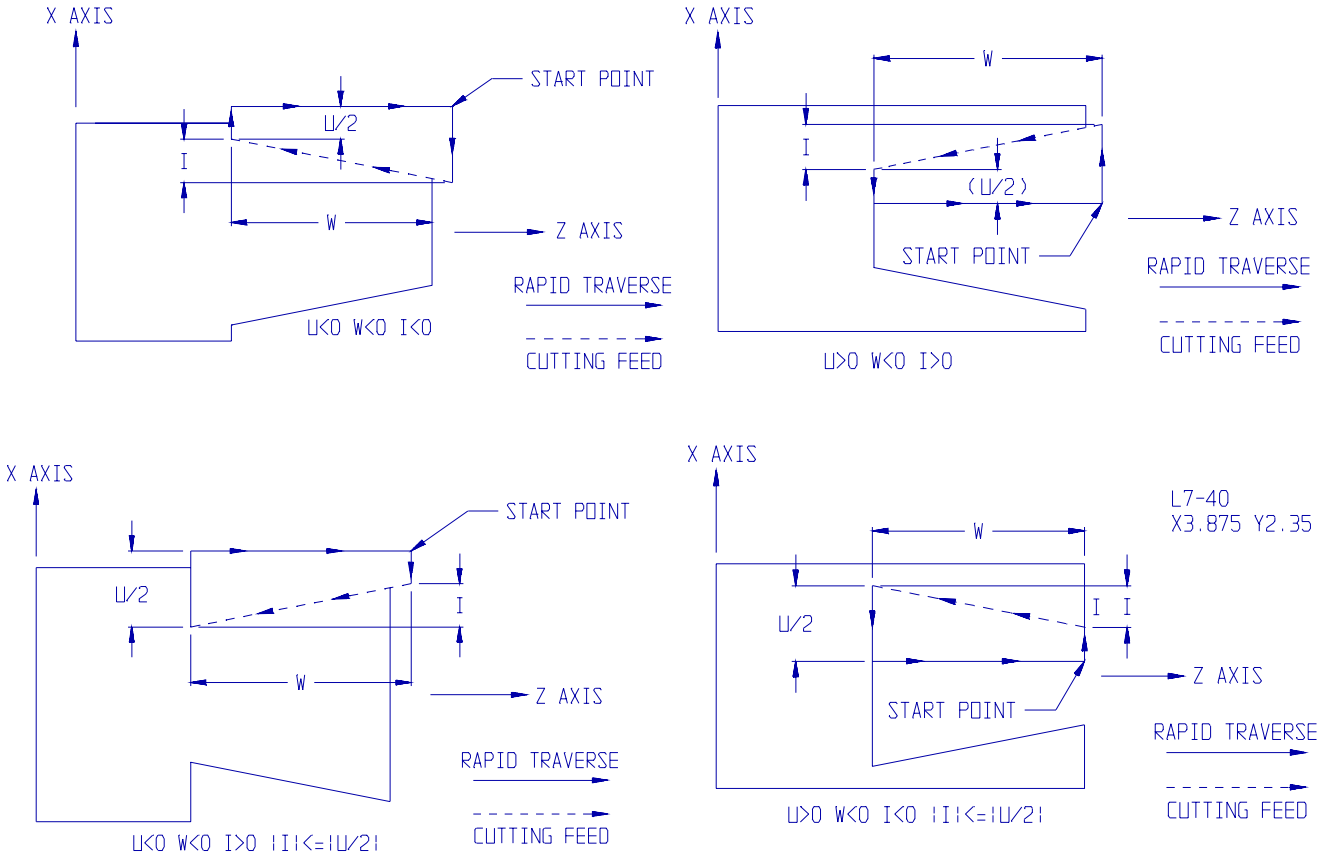
```
G0  X##.####  Z##.####
       1         1

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.



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.*

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	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 3
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
Operation 6: Rapid traverse up to the initial point

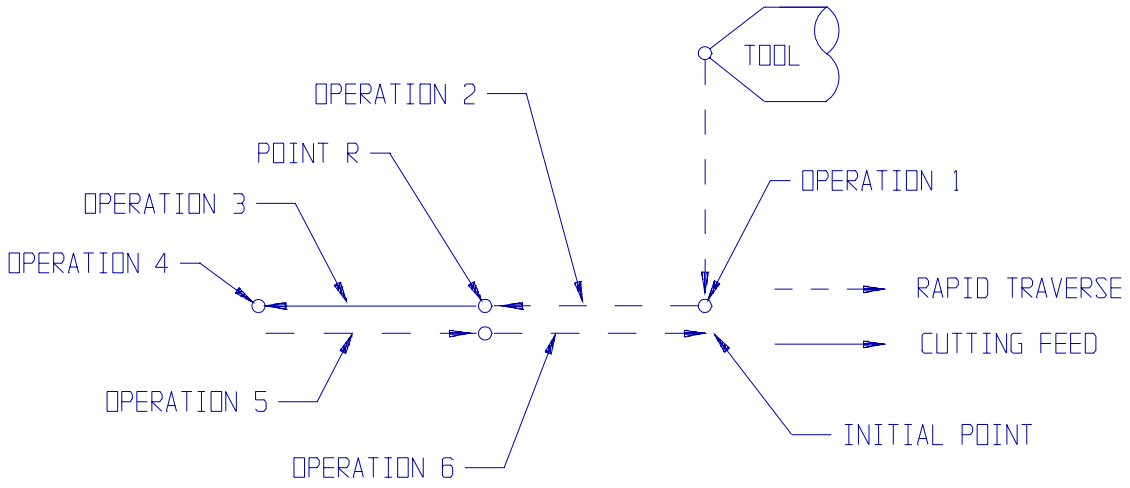


Figure 14.1
Canned Cycle Operation

Canned cycle operations consist of three basic modes which 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
		.	
		.	
		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.

Figure 14.2 shows how to specify data in G93 or G91 mode.

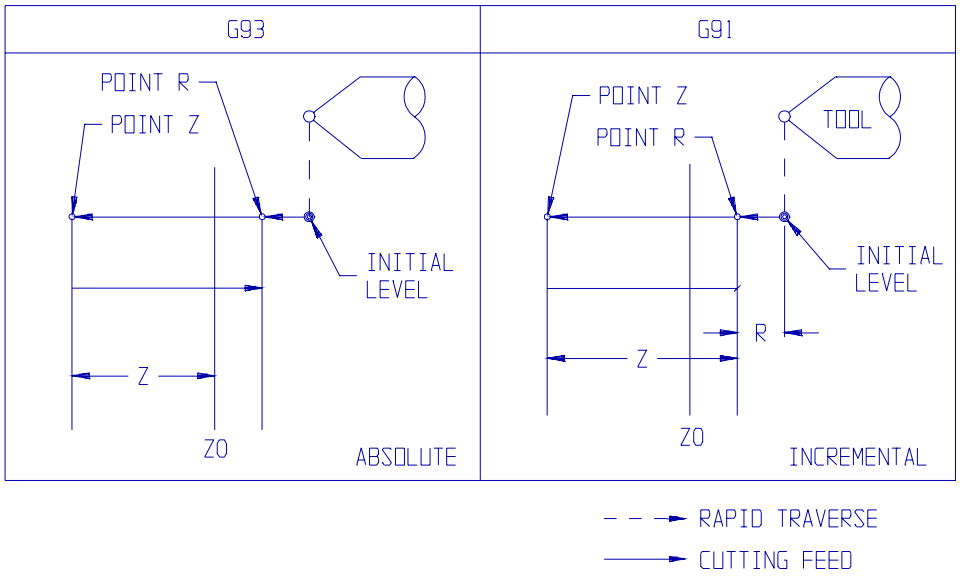


Figure 14.2
Absolute and Incremental Programming

If the tool is to be returned to point R or to the initial level, it is specified by G38 or G39. (See Figure 14.3.)

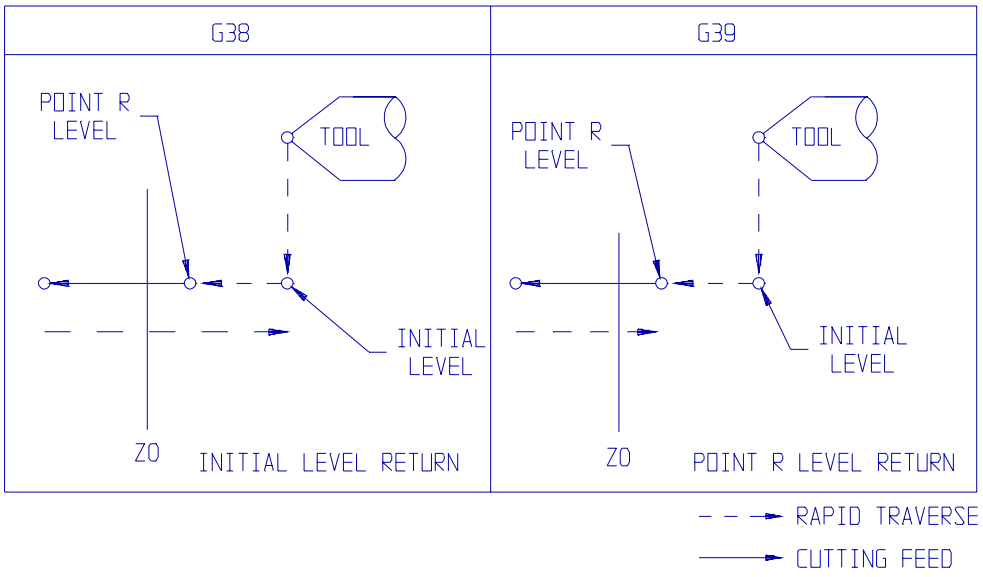
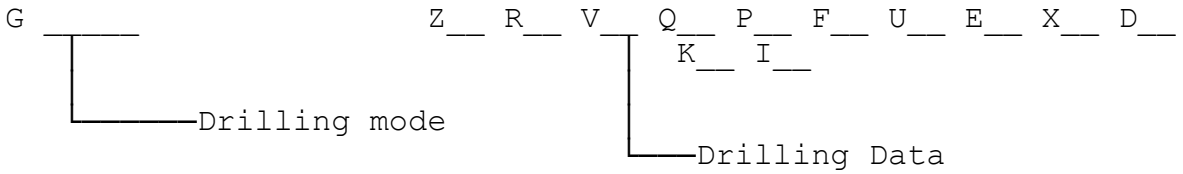


Figure 14.3
Initial Level and Point R Level

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:



Drilling mode . . .	G __ __	See Table 3.
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:

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 at 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.

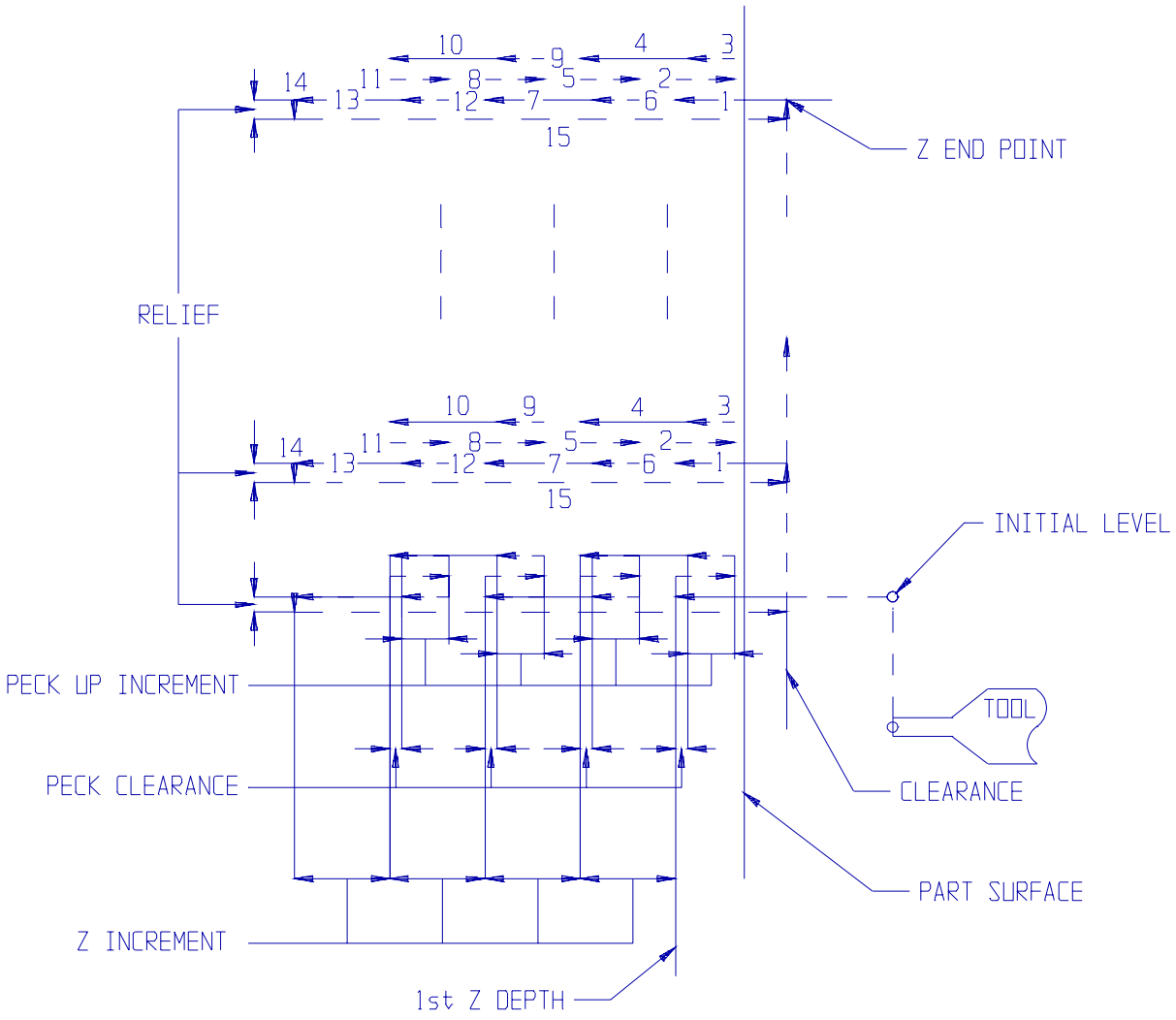
Each cycle is detailed below.

3.35 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.


10
9
4
3
14
11
8
5
2
13
12
7
6
1
15
Z END POINT
RELIEF
INITIAL LEVEL
TOOL
PECK UP INCREMENT
PECK CLEARANCE
CLEARANCE
PART SURFACE
Z INCREMENT
1st Z DEPTH

CODE USED FOR THE FACE GROOVING CYCLE

```
G74 F.#### G39 R##.#### Z##.#### V##.#### Q##.#### E##.#### U##.#### X##.#### I##.#### D##.#### A##.#### C##.####
      1    2    3        4       5       6       7       8       9       10      11      12      13
```

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.

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 final 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 starting diameter to the bottom of the groove.*
6. *Remove the Z stock along the bottom of the groove to the final diameter.*
7. *Retract out of the groove.*

Note 2: No cutter compensation is used when using the Face Grooving Cycle. The dimensions of the groove must account for tool width.

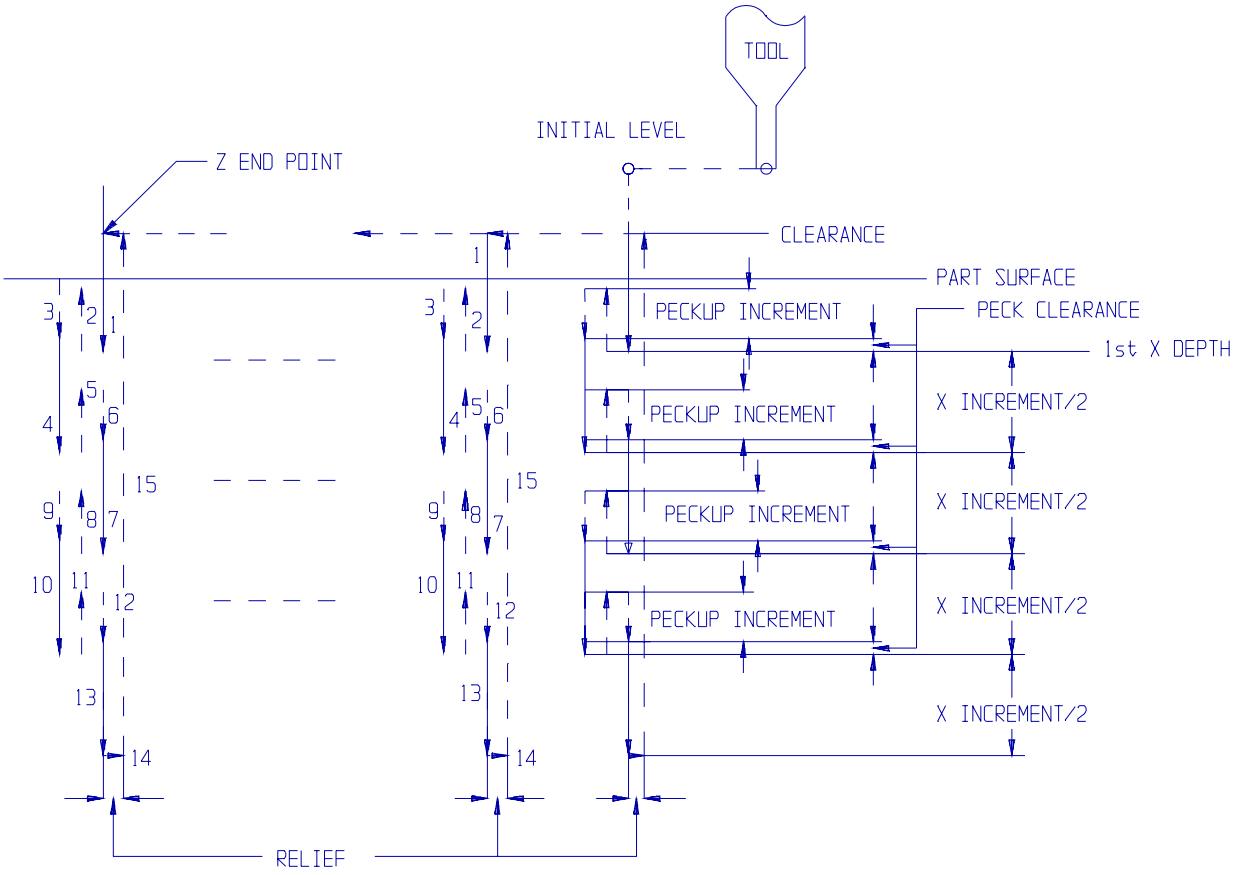
Note 3: A position should be made prior to the Face Grooving Cycle so the X axis is positioned to the start of the groove.

3.36 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 visa versa and outside towards center or visa 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.


TOOL
INITIAL LEVEL
Z END POINT
CLEARANCE
PART SURFACE
PECKUP INCREMENT
PECK CLEARANCE
1st X DEPTH
X INCREMENT/2
PECKUP INCREMENT
X INCREMENT/2
PECKUP INCREMENT
X INCREMENT/2
PECKUP INCREMENT
X INCREMENT/2
RELIEF
1
2
3
4
5
6
7
8
9
10
11
12
13
14
15

CODE USED FOR THE DIAMETER GROOVING CYCLE

```
G75 F.#### G39 R##.#### X##.#### V##.#### Q##.#### E##.#### U##.#### Z##.#### K##.#### D##.#### A##.#### C##.####
      1    2    3         4       5        6        7        8        9        10       11       12       13
```

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.

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:

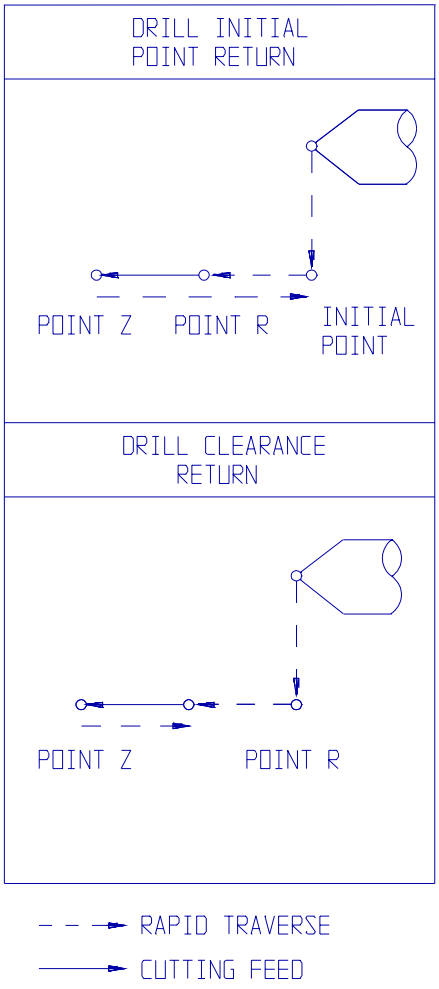
1. *Remove the X finish stock at the final diameter to the bottom of the groove.*
2. *Move to the center of the groove, along the bottom.*
3. *Retract out of the groove.*
4. *Move to the starting diameter.*
5. *Remove the X finish stock at the starting diameter to the bottom of the groove.*
6. *Remove the Z stock along the bottom of the groove to the final diameter.*
7. *Retract out of the groove.*

Note 2: No cutter compensation is 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.

3.37 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.

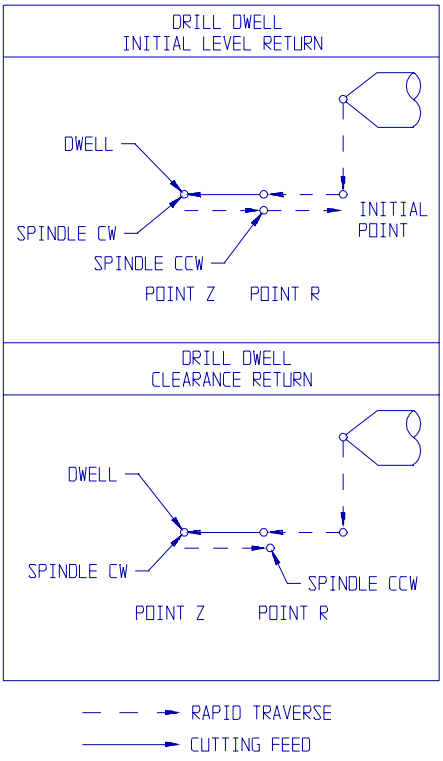
2. Return Point: G39 means the tool will rapid back to the clearance plane at the end of the cycle. G38 means the tool will rapid back to the Z dimension the tool was at before the drill cycle started.

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.

3.38 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

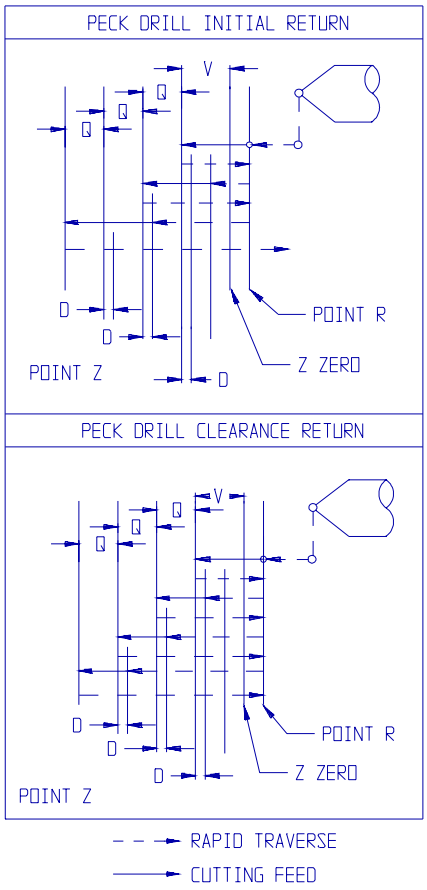
```
G82  F.#### G39 R##.#### Z##.####  P###.#
       1     2     3        4        5
```

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.

3.39 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

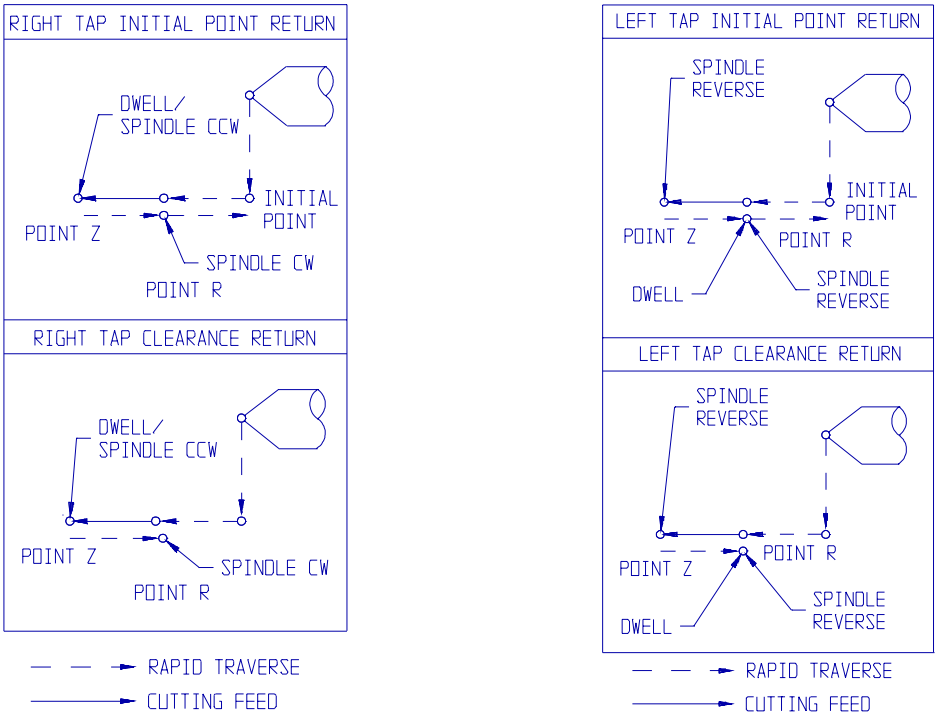
G83 F.#### G39 R##.#### Z##.#### V##.#### Q##.#### E##.####
1 *2* *3* *4* *5* *6* *7*

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.

3.40 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

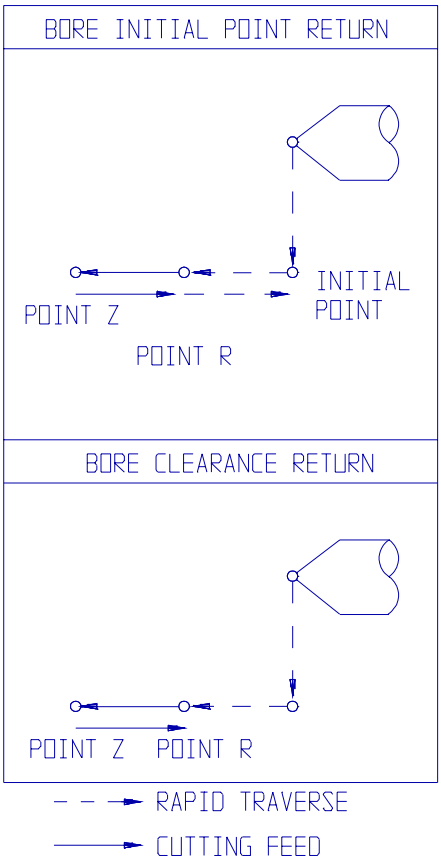
```
G97 G99 M3 S####.#

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 drill cycle started.
3.	(R) Clearance:	Z rapids to the clearance at the beginning of the drill 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.

3.41 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 drill cycle.

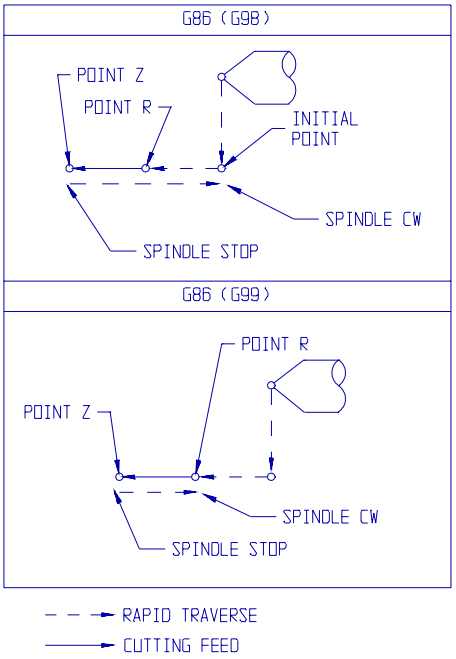
4. (Z) Final Z Depth: The depth to drill the hole to.

3.42 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

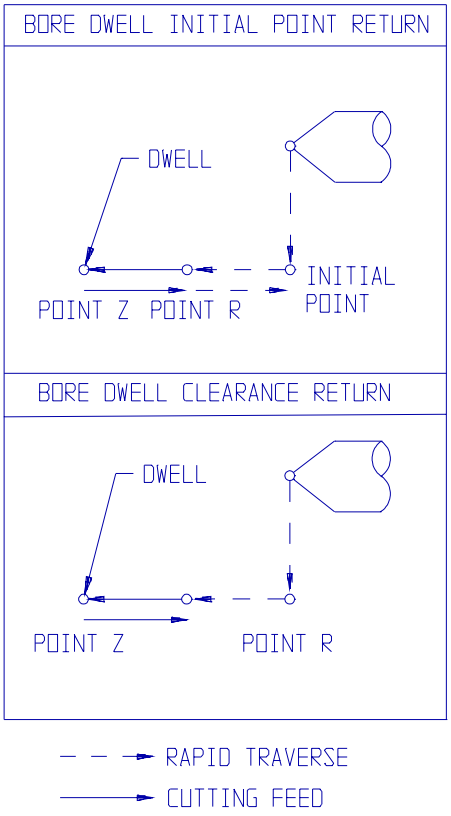
```
G86 F.#### G39 R##.#### Z##.####
      1    2      3       4
```



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 drill cycle.
4. (Z) Final Z Depth: The depth to drill the hole to.

3.43 Bore/Dwell Cycle (G89)

The bore/dwell cycle is the same as the bore cycle, but a dwell is preformed at the bottom of the hole.




```
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 drill 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. (Note: The High Speed Boring Cycle turns the spindle on and off).

```
M3              Spindle CW
 .
 .
 .
G __ __ ..... Correct
 .
 .
 .
M5              Spindle Stop
 .
 .
 .
G __ __ ..... Incorrect (M3 or M4 must be
              specified before this block.)
```

Note 2: Operator Precautions

a) Single block

When a drilling cycle is performed in the single block mode, the control stops at the end of Operations 1, 2 and 6 as shown in Figure 14.1 on page 71. Therefore it must be started at least three times to drill one hole. The tap cycle will not stop at the bottom of the hole in the single block mode.

b) Feedhold

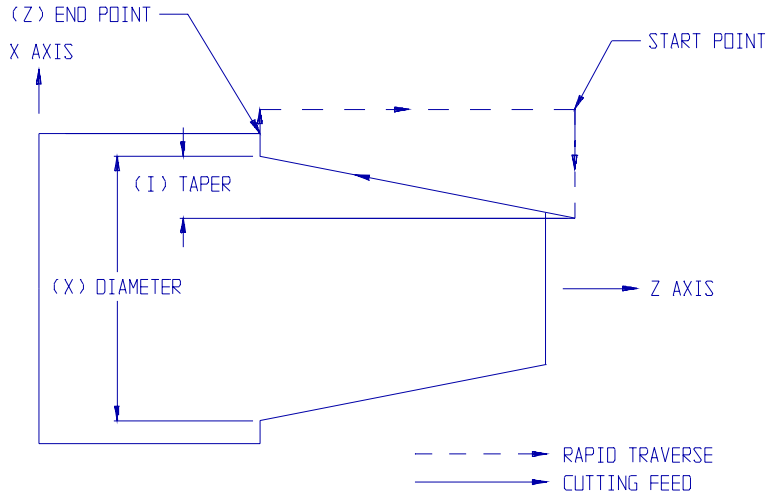
When a feedhold is applied between Operations 3 to 5 in tap cycle G84, the FEEDHOLD lamp immediately lights, but the control continues to operate up to Operation 6 and stops. If a FEEDHOLD is applied during Operation 6, it immediately stops.

c) Override

The feedrate override and spindle override is locked at 100% during the operation of canned cycle G84.

3.44 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.



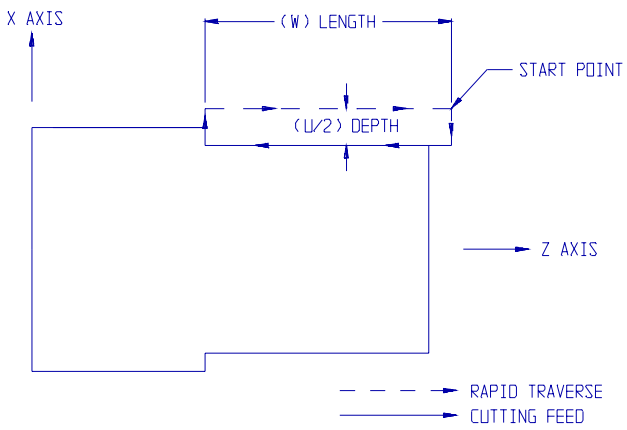
Tapered Cutting Using Cycle A With Absolute Dimension

```
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.

-OR-

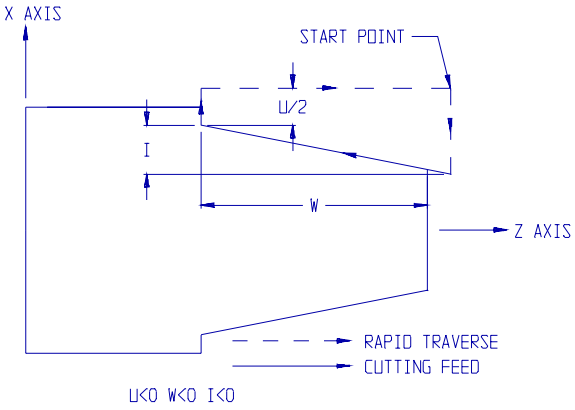


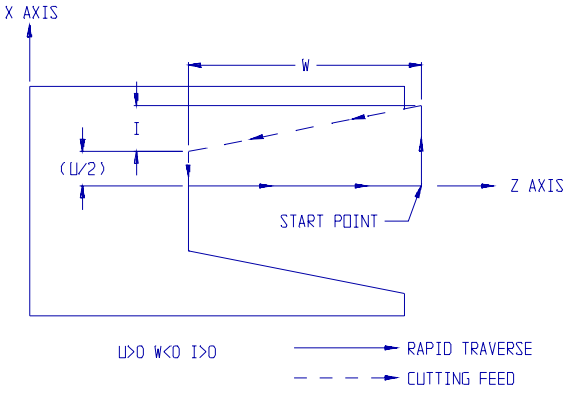
Straight Cutting Using Cycle A With Incremental Dimensions

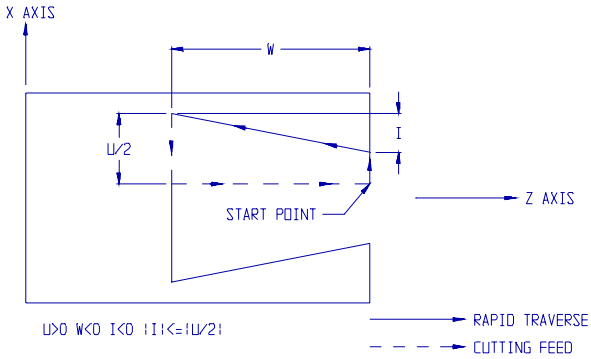
```
G0 X##.#### Z##.####
      1         1

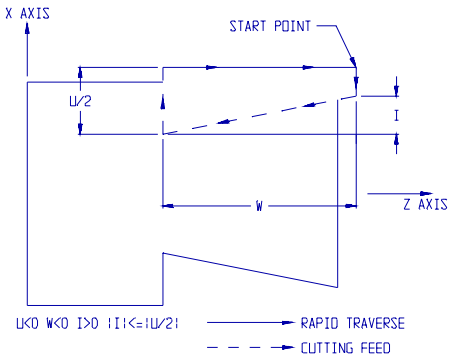
G90  U##.####  W##.####
       2           3
```

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.





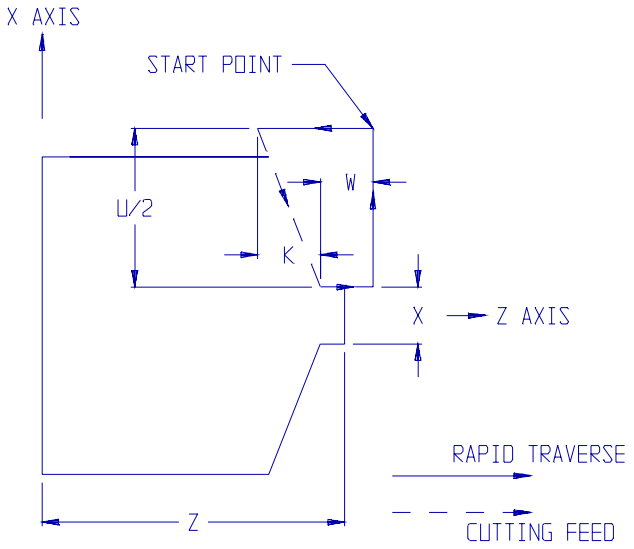




When using Cycle A with incremental dimensions, the signs of the depth, length and taper indicate the directions shown in the above figures.

3.45 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.

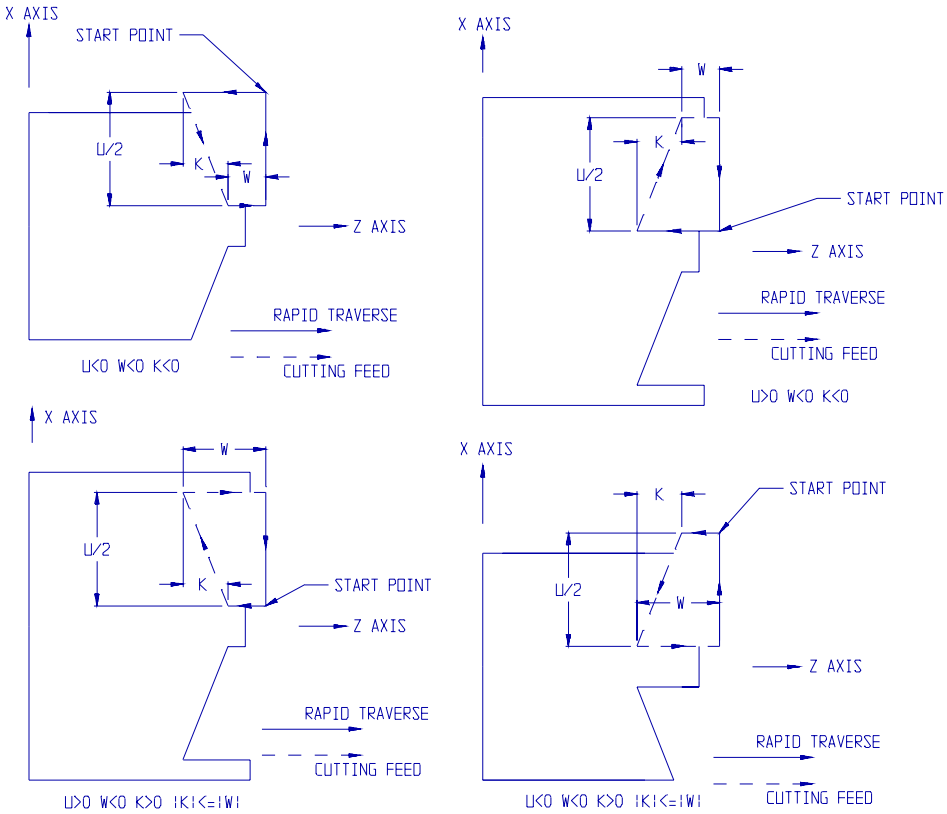
-OR-

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.

3.46 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.

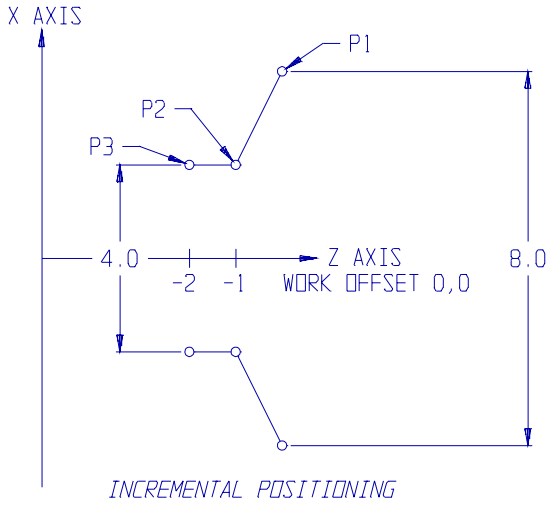
XC, ZC (centers), AA (start angle) and AB (end angle) are always an absolute.

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

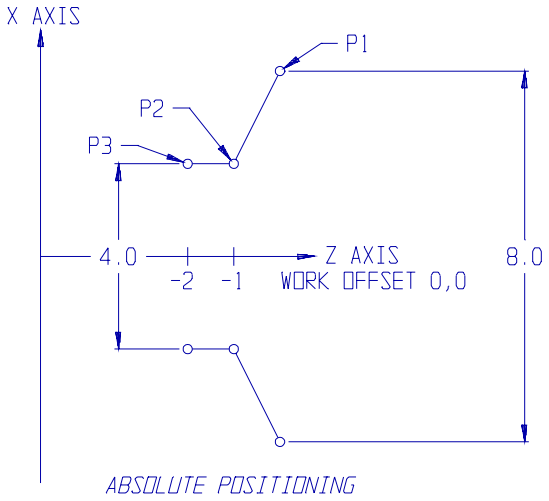
3.47 Absolute mode (G93)

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.



3.48 Constant Surface Speed (CSS) (G96)

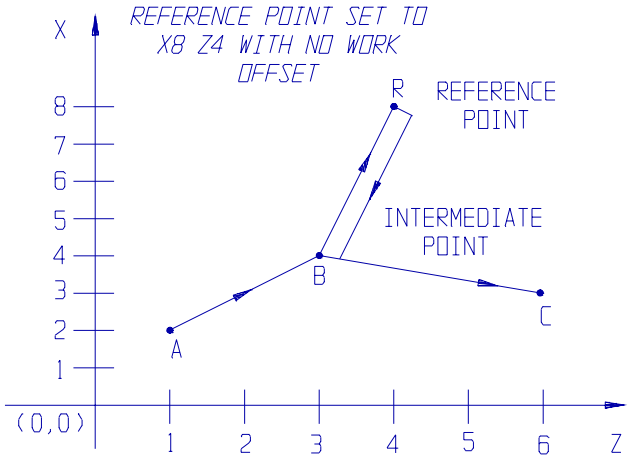
The surface speed (of the tool relative to the work piece) can be controlled by G96 (constant surface speed). Using this code will cause the spindle to rotate faster as the tool moves closer to the center of the work (X=0) and slower as the tool moves further from the center of the work (larger diameters). 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).

3.49 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.

3.50 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.

3.51 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 at percentages of 0%, 10%, 20%, thru 140%.

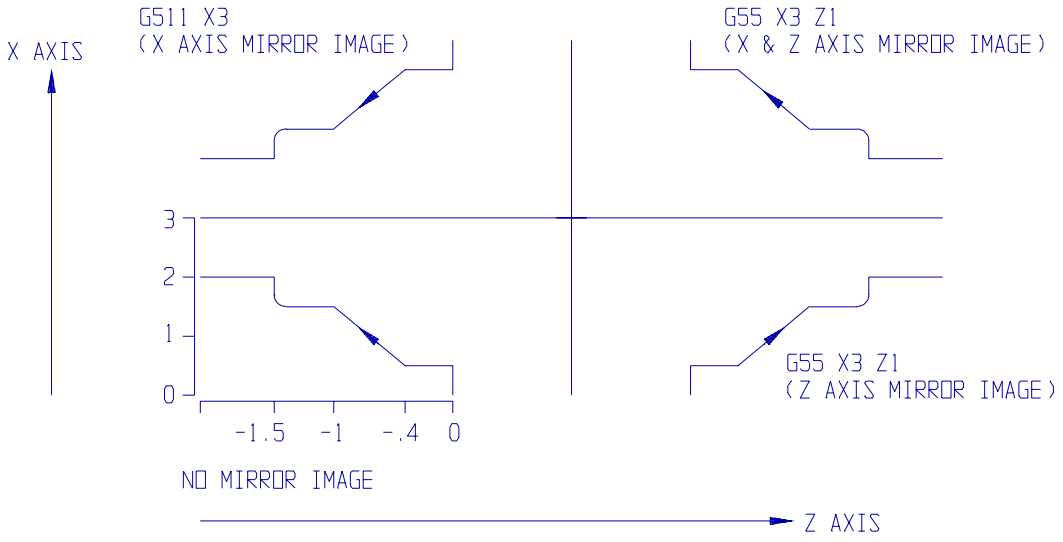
Note 2: When G0 (rapid position) moves are commanded the programmed feedrates are not considered.

3.52 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  X.5
               Z-.4
               X1.5 Z-1 R.1
               X2
```

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.

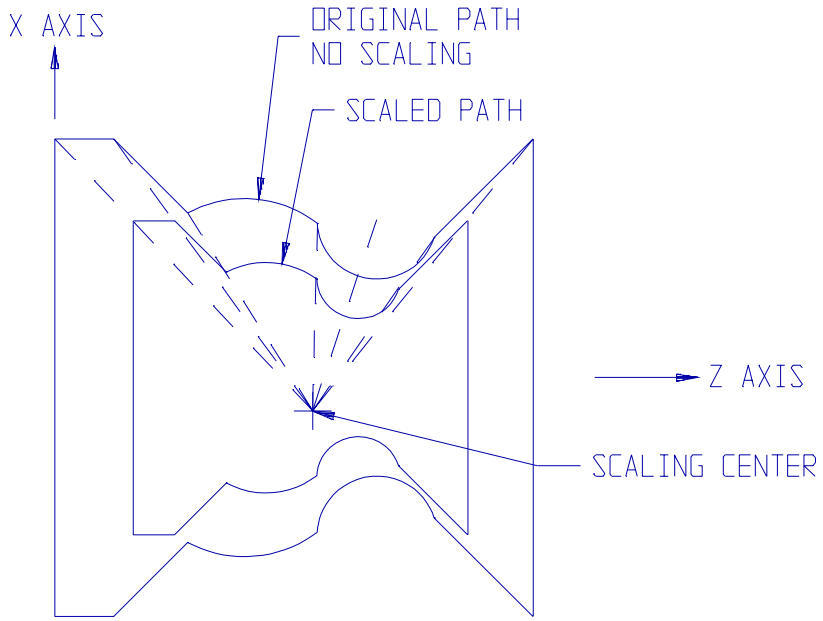
3.53 Cancel scaling (G550)
Set scaling (G551)

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.

3.54 Coordinate system rotation (G568 - G569)

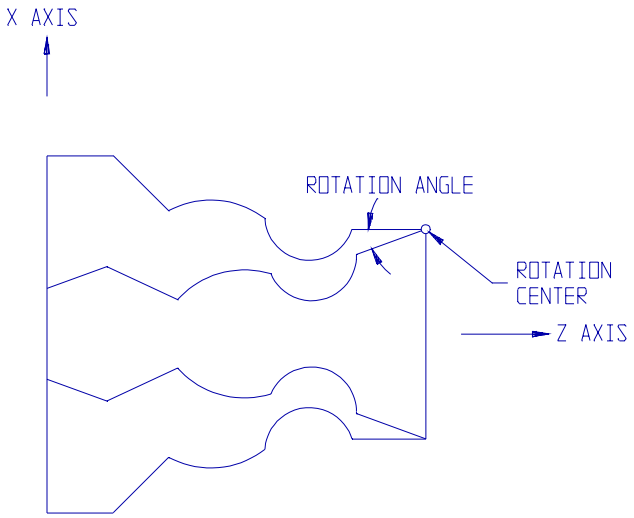
G568 can be used to rotate a programmed shape about a predefined center point. The center of rotation is defined by I and K, and the angle of rotation by AA. The command format is as follows:

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 coordinate system zero point at the beginning of each program. If the IK's are not present in the G568 block the center of rotation will be the last center specified.

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.

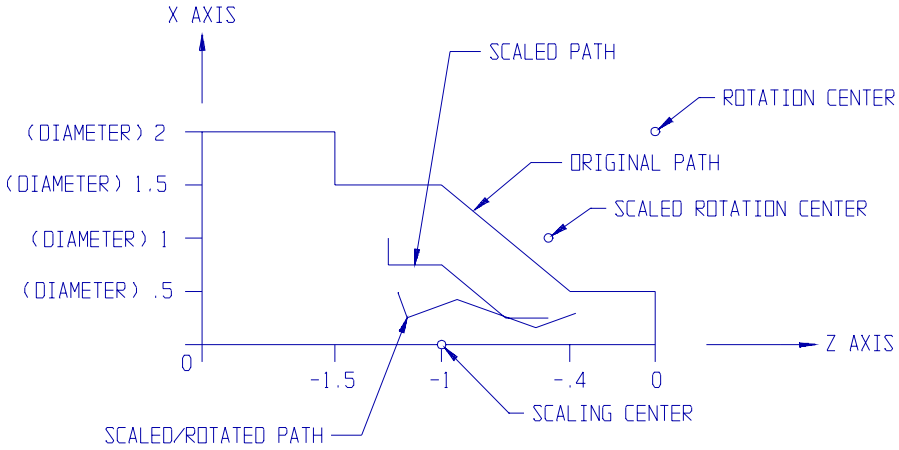
3. The order of on and off is first on last off.

```
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.

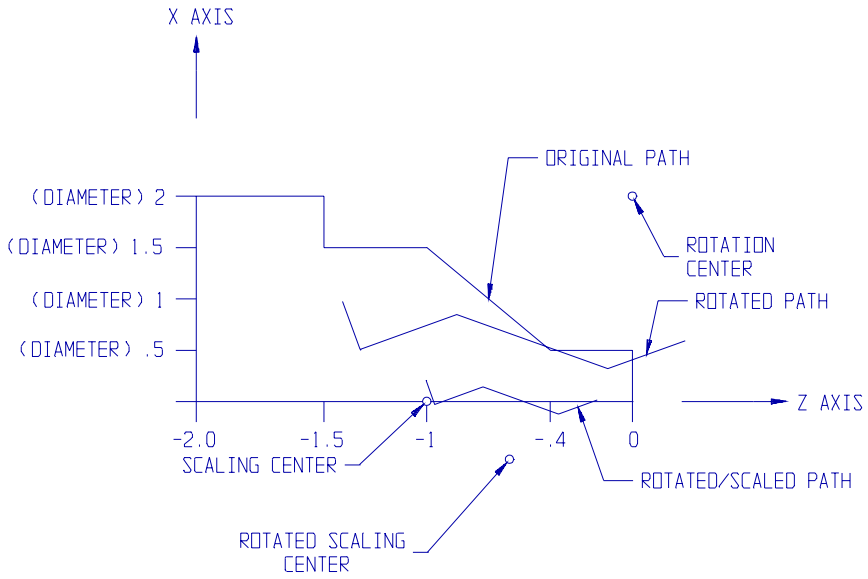
5. X and Z can be used instead of I and K for rotation center.

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
X1.5 Z-1
X-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
```

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 allow parameters to be saved and restored using file names B:\RAM\Q0000-Q9999. Parameters are:

- Pp (base parameter number, default 0),
- L1 (number of parameters, default 10),
- Qq (file identification, default 0).

The G990 or G991 MUST follow the Pp, L1 or Qq.

CAUTIONS: 1. Save and restore only P0-P99.
2. 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)
          M98 (return to caller with original P0-P9)
```

Read byte parameter (G995)

```
P1=b G995 (sets P0 to value of byte b)

     Example:
          P1=79
          G995 (sets P0 to value of byte 79, G18 plane select,
           XZ=0, ZX=1)
          Valid P1 values are 0 to 639.
          Note: P0=PB79 could be used.
```

Write byte parameter (G996)

```
P1=b P0=V sets byte parameter b to value v

     Example:
          P1=82
          P0=1
          G996 (set byte parameter #82, offset round taper walls
           to yes, example 0=no, 1=yes)
          Valid P1 values are 0 to 639
          Valid P0 values are 0 to 255
          Note 1: PB82 = 1 or P3=PB50 can be used instead of
           G995/G996.
          Note 2: Byte parameters are not written in verify mode.
```

Force Error (G997)

```
Forces an error code to be displayed
Error code generated is round (parameter #1)

     Example:
          P1=408
          G997 forces error #408 Y-axis excess error to be
           displayed.  Y-axis does not excess error, it only
           displays the error.
          Notes:  * P1=0 will not produce an error
                  * All valid error codes on the control are
                    between 1 and 999.
```

Beep (G998)

G998 will cause the speaker to beep if a speaker is installed.

Force Fatal Error (G999)

G999 will do division by 0 forcing fatal error #200. This is not recommended for any practical purpose.

4. MISCELLANEOUS FUNCTIONS (M Functions)

These codes are used if the operator is programming the Centurion 5T 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 this table.

M Codes	Function	Executed Before Move	Executed After Move
M00	Program Stop		X
M01	Optional Stop		X
M02	End of Program		X
M30	End of Program		X
M03	Spindle on CW	X *	
M04	Spindle on CCW	X *	
M05	Spindle off		X *
M06	Tool Change		X *
M08	Flood Coolant On	X *	
M09	Coolants Off		X *
M20-M29 M60-M89	These are input/output commands.	X *	
M32	Wait Channel	X *	
M90	Graphics Off	X *	
M91	Graphics On	X *	
M98	Program Call Statement	X *	
M99	End of Program Statement	X *	

** These functions are selectable for either before or after move.*

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 "M00 first M99 last" unless they have been defined to execute after the move statements. (See section on machine setup.)

4.1 Program stop (M00)

The execution of the program is halted on the block containing the M00 and prompts the operator to press cycle start. Program execution will be resumed when CYCLE START is pushed. If M00 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.

4.2 Optional stop (M01)

M01 is the same as M00 except it is only executed if the optional stop switch is enabled.

4.3 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
```

4.4 End of program (M02, M30, M99)

All of these codes can be used to indicate the end of a program. The only difference is that M02 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.

4.5 Spindle on/off (M03, M04, M05)

These codes turn the spindle on CW (M03), CCW (M04) and off (M05). The spindle on commands will be executed before an axis command. M05 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.

4.6 Tool change (M06)

The standard M6 moves X to the tool change position. The command strobes the M function buss and sends out a 150 millisecond (msec) pulse on the M06 I/O output. It then halts program execution and prompts the operator to change the tool. If a comment is on the

M6 block it will be displayed to prompt the operator until it receives a tool change complete signal. The spindle cannot be turned on until the tool change is completed. A CYCLE START is required after the tool change complete signal is received to resume program operation. For safety reasons a manual tool change should never be attempted unless the machine is in an M06 tool change mode.

In many cases the machine tool builder may supply a custom M6 for automatic tool changer or other special applications.

4.7 Coolants on/off (M08, M09)

The Coolant On command (M08) is executed before an axis command. The Coolant Off command (M09) will be executed after an axis command.

4.8 Wait channel (M32)

Waits for the wait input channel (Z input #6).

4.9 (M20 - M29)

These M functions are for general use. They pulse corresponding outputs (X out #1 thru 10) for 150 m/sec. The corresponding input is poled (X in #1 thru 10) program executed is halted until the appropriate input is received. X output #12 is set during the M20-M29 codes.

4.10 (M60 - M69)

For general use, M60-M69 sets X outputs #1 thru 10 respectively. "Set X#" (where # is 1 thru 10) can be also used to control outputs for each axis.

4.11 (M70 - M79)

For general use. M70-M79 clears X output # 1 thru 10 respectively. "CLR X#" (where # is 1 thru 10) can be also used to control outputs for each axis.

4.12 (M80 - M89)

For general use. M80-M89 halts execution of the program and waits for corresponding input (X inputs #1 thru 10) then continues to program.

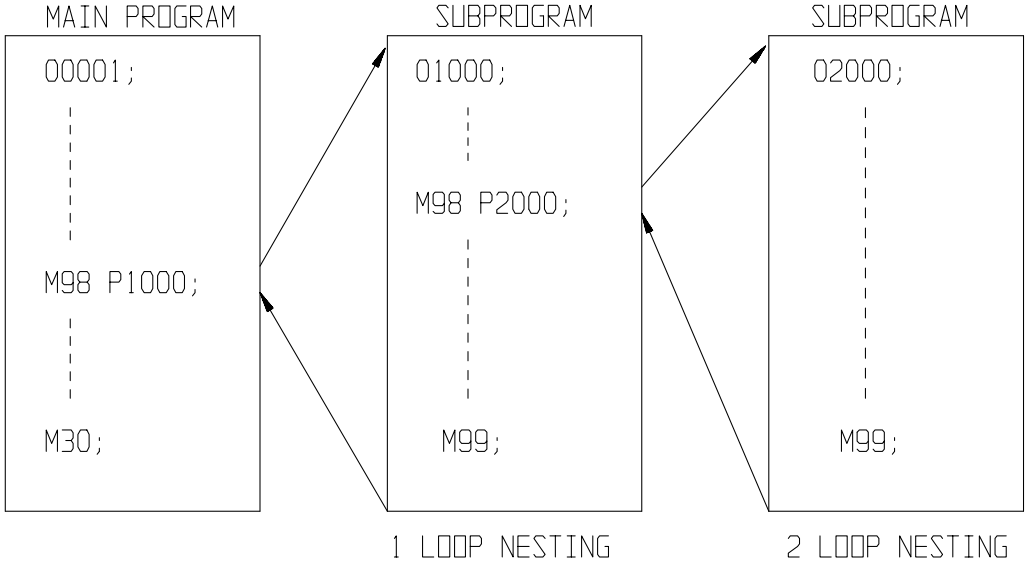
4.13 Graphics off/on (M90, M91)

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.

4.14 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 with up to 50 loops is allowed.

M2 can be used instead of M99. If a subprogram ends without an M2 or M99 it will return to the calling program as if an M2 or M99 was encountered.

Note: "CALL" can be used in place of "M98".

4.14.1 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 M02, M30 or M98 will return. Subprograms are entered into memory the same as normal programs.

Note: M30 also shuts the spindle and coolant off.

4.14.2 Subprogram execution

A subprogram is executed when called by the main program or another subprogram. A subprogram call has the following format:

```
M98 PXXXX LXXX
      |     |
      |     └──── number of times to repeat the subprogram
      └──────────  subprogram number
```

Example: M98 P0002 L5
M98 P2 L5
CALL 2 L5

This command is read like this: call subprogram number 2 five times.

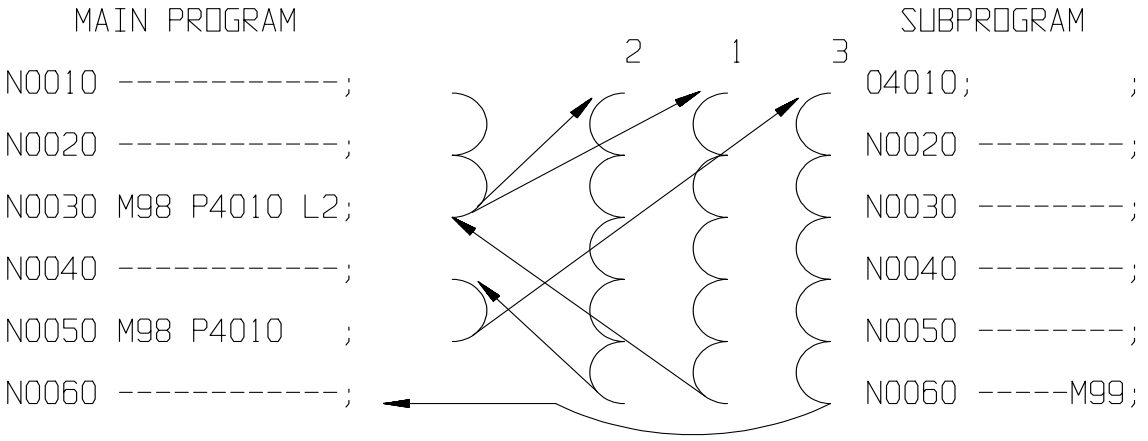
When the loop number is omitted the subprogram is run once.

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, an error message is displayed.

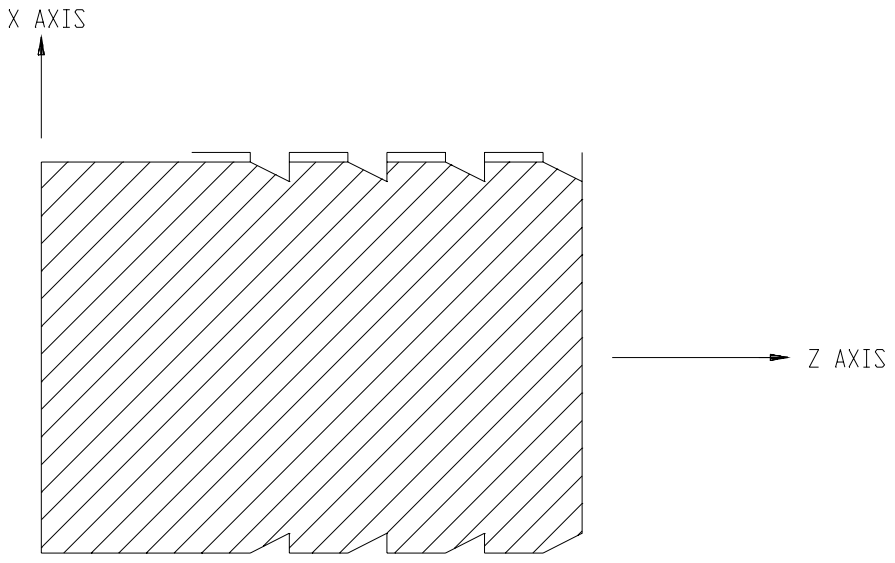
Note 2: A subprogram call "M98___" cannot be executed from MDI. In this case write a short program to call the subprogram.

0XXXX
M98PXXX
M02 *Then execute it in the Run mode.*

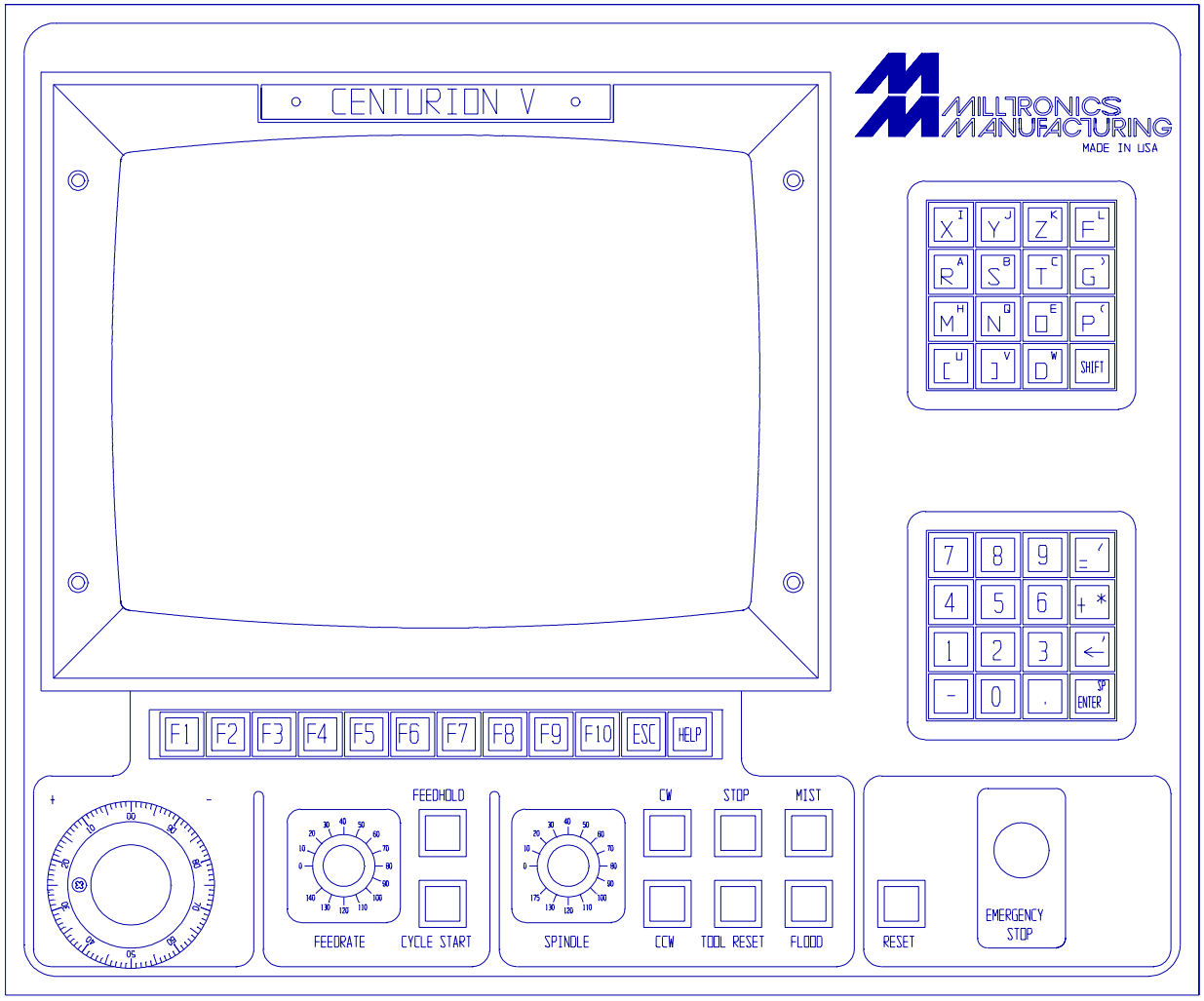
Note 3: If a subprogram modifies the work offset, (normally G50 X0 Z0 on the first block of the subroutine) the work offset will be restored to the same value upon leaving the subprogram as it was when entering the subprogram. This can be useful for cutting the same pattern at different locations.

Example:

```
O####           O0002
X2 Z-1          G50 X0 Z0
M98 P2(cut part at X2 X-1)
X3 Z-2         Geometry
M98 P2 (cuts shape at X3 Z-2)   M99
M98 P2 (cut part at X3 Y2)
M2              M99
```



5. FRONT PANEL OPERATION



Centurion 5T Front Panel

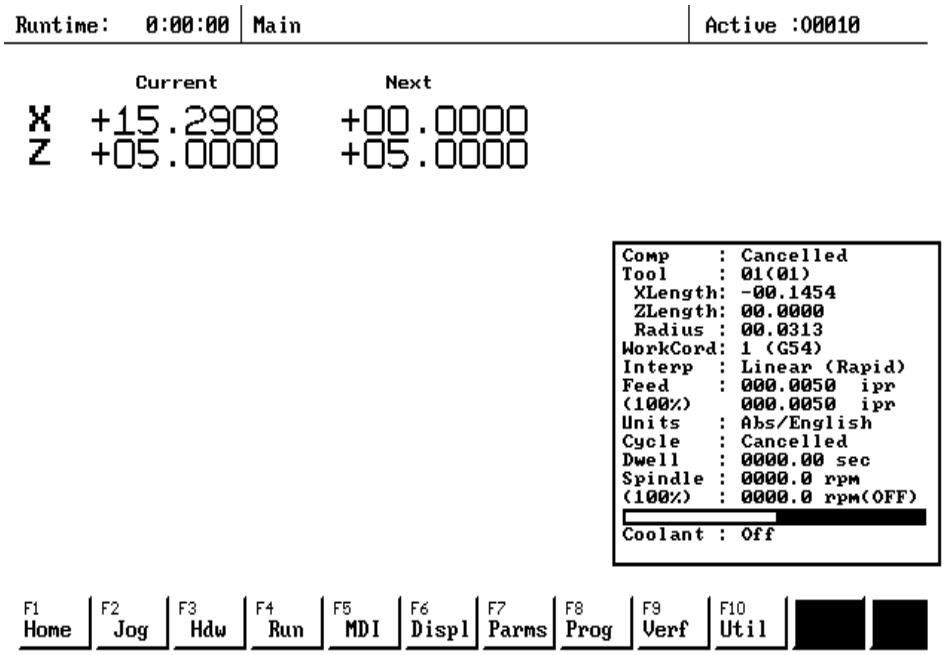
5.1 Front panel description

The Centurion 5T front panel has two 16-key keypads and 12 function keys. The keypads are used to enter the alphanumeric data requested by the CNC. The upper keypad is primarily used to enter alpha characters. To enter one of the shifted characters simply hit SHIFT, then the character. After the character has been entered the control automatically returns to the non-shifted character set. The SHIFT key also works in the same manner on the lower or numeric keypad. When data is entered, spaces between commands are optional, but an ENTER must be pushed to terminate a line of data or to go on to the next function. The operation of the 12 function keys changes

as different menus are displayed on the CRT. The following sections explain in detail each function key meaning.

The lower section of the panel is dedicated to manual machine cycles. 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. Next to the handwheel are the manual feed controls for the machine's axes. The FEEDRATE override switch will modify the current machine feedrate by the indicated percentage. The FEEDHOLD button will cause axis motion to stop whenever it is illuminated. To restart axis motion, press FEEDHOLD again and depress the CYCLE START button. The CYCLE START needs to be depressed anytime a machine command is to be executed. The CYCLE START will blink when it needs to be pushed.

The next section of the panel deals with the spindle and coolant controls. The SPINDLE Override switch will modify the current spindle speed by the selected percentage. The spindle CW, CCW 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 which button is lit. The coolant buttons (MIST and FLOOD) toggle the coolants ON and OFF. When they are illuminated that function is active; however, the coolant will not turn on until the spindle is started. 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 when in a tool change and will need to be pushed after the tool change is completed before program operation can be resumed. The EMERGENCY STOP button, when pushed, will stop all machine actions instantly. Once the EMERGENCY STOP button is pushed, the RESET 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.

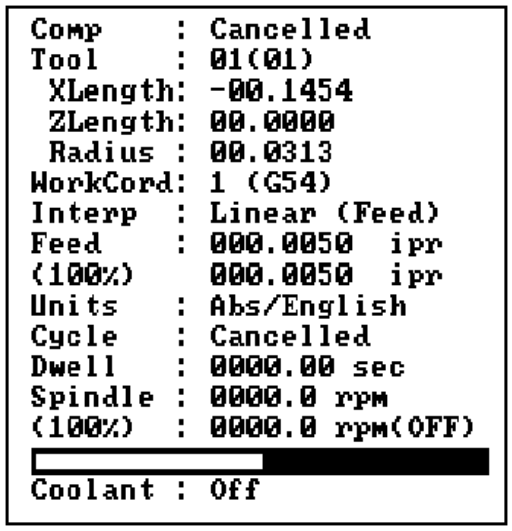


The above figure represents the main menu or the number one screen. All operations will start from this menu. The top of the screen will always display the runtime, screen history, and active program #.

The Runtime at the top of the screen is basically a stop watch which starts when a Run Program command is executed and stops at the end of program or when the program is aborted. The total job time is kept in program parameter #369, job time.

The screen history line will display the various levels or screens gone through to get to the present screen. Each time the ESC key is pushed the CRT will back down one level until the number one or main screen is reached.

The status window shown here displays information related to the current state of the machine.



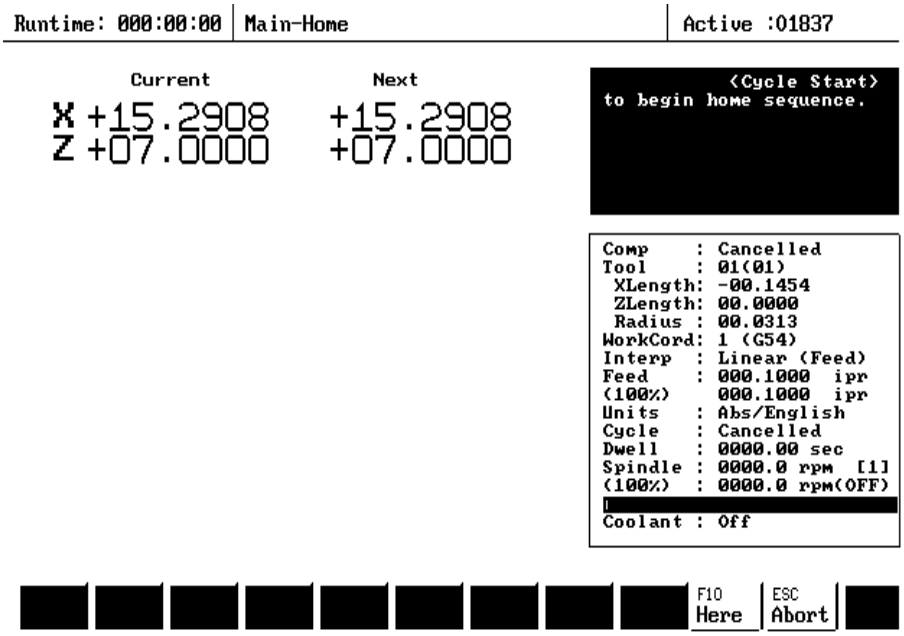
1. Comp: Cancelled for G40 (tool nose radius compensation off), left cut for G41 (cutter compensation left) or right cut for G42 (cutter compensation right)
2. Tool: The first two digits relate to the active tool #. The two digits in parentheses relate to a pending tool (T####) that has been commanded but not active until an M6 has been commanded.
3. X Length: The current X tool length offset
4. Z Length: The current Z tool length offset
5. Radius: The current tool nose radius
6. Work Cord: The current work offset: 1(G54), 2(G55), 3(G56), 4(G57), 5(G58) or 6(G59)
7. Interp: Current interpolation mode: linear (feed), linear (rapid), circular (CW) or circular (CCW)
8. Feed: Current programmed feedrate and its units in ipr (inches per revolution), mmpr (millimeters per revolution), ipm (inches per minute) or mmpm (millimeters per minute)
9. Feedrate Override The current feedrate override position and its percentage of the programmed feedrate. *(Note: If the maximum feedrate of an axis is exceeded, an "*" will be displayed to the right of the override percentage and the clamped feedrate will be displayed in the feedrate field.)*

10. Units: Absolute or incremental and English or metric
11. Cycle: Displays the cycle being executed or "cancelled" if no cycle is active. The cycles that are displayed are drill, drill/dwell, peck drill, tap, bore, bore II, face grooving, diameter grooving, bore/dwell, single turn, single face, single thread, thread cycle, threading.
12. Dwell: The dwell time in seconds, counts down during a dwell
13. Spindle: The programmed spindle speed and its units in feet per minute (fpm), meters per minute (mpm), or revolutions per minute (rpm)
14. Spindle The current spindle override position and its actual spindle speed in revolutions per minute
15. Spindle bar: Indicates the load on the spindle. As the load increases it changes colors from blue to green to yellow to red.
16. Coolant: The state of the coolant pump (off or flood)

The remainder of this section will explain each function that can be executed from the front panel. The history line for each function is the sequence of keys to push to get to that function from the main menu. Highlighted keys on the CRT mean they are the currently selected mode or their functions are available for use on this screen.

5.2 Home sequence F1 HOME (MAIN-HOME)

After a power off sequence the control will always have to be homed. Each axis will seek a home limit switch and a marker pulse on the encoder. After this procedure is finished the machine's reference position will be established and will be remembered until another power off. The sequencing of which axis homes first, the direction of home, and whether a limit switch and marker are used or just the marker pulse, can all be set in the axis parameters sections. To initiate a home sequence push ESC until the Main screen is reached; then push F1 "HOME".



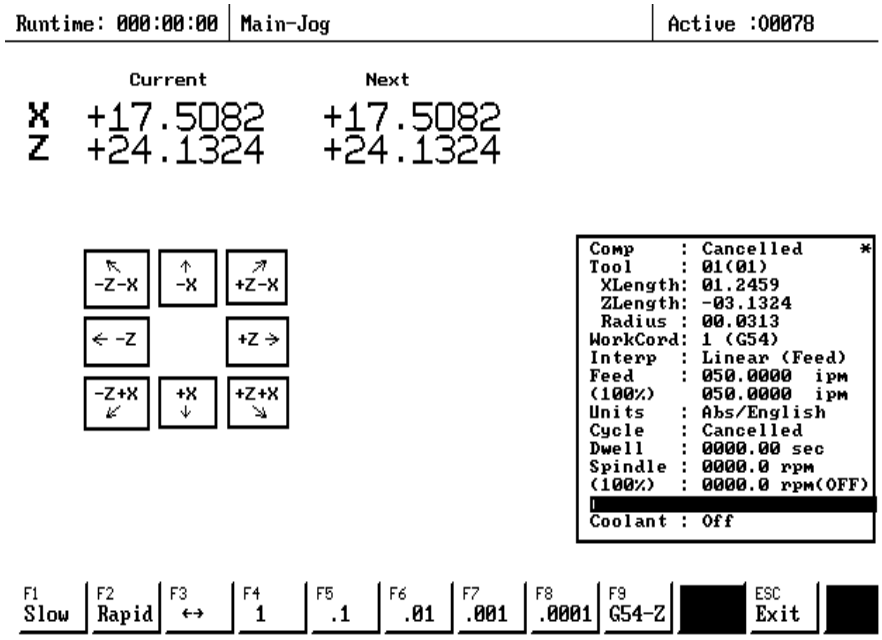
A message requesting that the Cycle Start button be pushed will appear on the screen and Cycle Start button will start flashing. Pushing Cycle Start will start the home sequence and when it is finished the Main screen will return.

5.2.1 F10 HOME - HERE

The machine can be homed without moving the axis by toggling the F10 HERE key. When the key is lit, push the CYCLE START button. The position of the machine will be assumed as home zero. This is useful to allow jogging, handwheeling or MDI-ing without physically homing first. It is not recommended to run programs unless the machine is homed in the normal fashion, as the software limits will not be valid after a home-here.

5.3 Jog F2 JOG (MAIN-JOG)

Jog is used to move the machine around in a manual mode to pick up zeroes and align parts. Upon pushing JOG the following screen appears:



Note: Machine parameters can be set so F9 can be used to store the Z position in the Floating Zero (G50) offset.

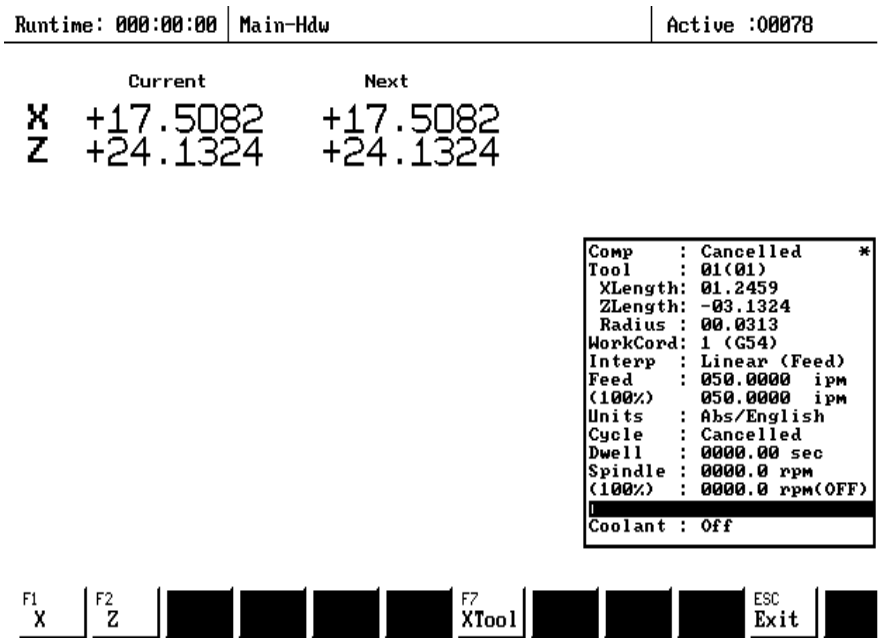
The F keys across the bottom of the screen (soft keys) are used to select the type of Jog desired. F1 selects slow jog. The feed override is active and can be used to speed up or slow down the jog speed. F2 selects rapid jog. F3 selects continuous jog. In continuous jog the selected axis will continue to move until the axis key is released or the software limits are reached. In incremental jog the axis will move the selected increment (F4 thru F8) and then stop each time the axis key is pushed and released. F9 sets a work offset (G54) for a Z axis at the current machine position. ESC (escape) key exits Jog, returning to the main menu. The keyboard diagram shows which direction an axis will move when the corresponding key is pushed on the numeric keyboard.

Note 1: Slow and fast jog feedrates can be modified in the setup parameters.

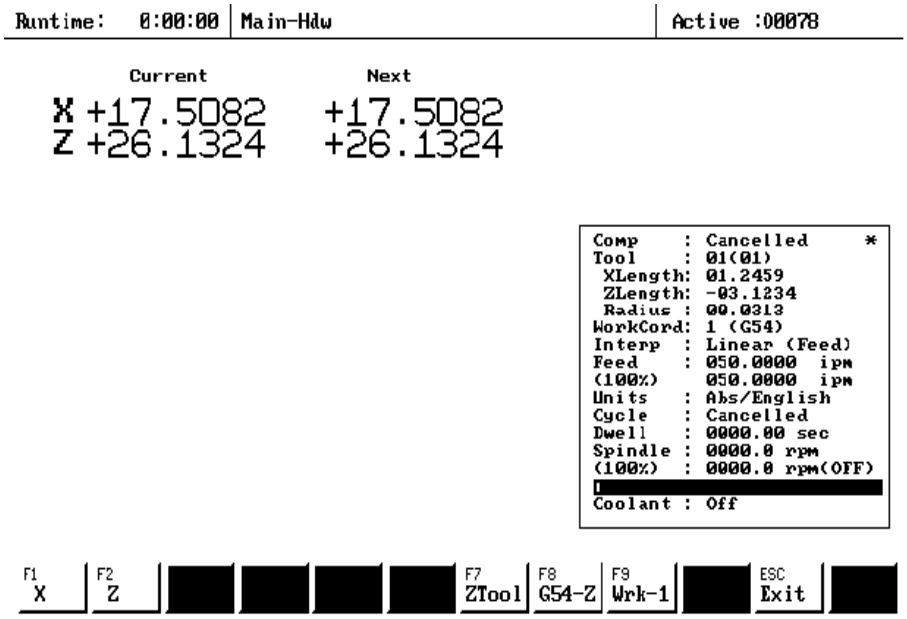
Note 2: Jog key directions can be reversed in the setup parameters.

5.4 Handwheel F3 HDW (MAIN-HDW)

The Handwheel mode is used to move the machine around using the electronic handwheel. It is used mainly to set tool length offsets, floating zeroes and part alignment. Upon pushing HDW the following screen will appear with the X axis enabled:



If the Z axis is enabled the following screen will appear:



The F keys across the bottom of the screen are used to select which axis will move when the handwheel is turned. The feedrate override switch will determine the distance each axis will move for one click of the handwheel. A feed override of 10% will cause the axis to move .0001 inch or .001mm on metric mode for each handwheel position. The active axis key will be highlighted when it is selected to move (F1 for X axis or F2 for Z axis).

Note: Axis distances per handwheel click at feedrate override positions can be modified in the setup parameters.

The F7 key XTOOL or ZTOOL is used to set tool length offsets into the tool table. In the handwheel mode a tool can be moved to a desired location. When F7 is pressed the control will ask for the tool number. It then asks for the position of the tool, i.e. Z position for Z axis or diameter for the X axis.

The F9 key is used to set up a work offset other than work offset #1. Each time F9 is pressed it will advance to the next work offset that is displayed. When (F8) G5#-Z is pressed a Z zero is set for the appropriate work offset. *(Note: F8 and F9 are only active if the Z axis is active.)*

PROCEDURE FOR SETTING TOOL LENGTH OFFSET AND WORK OFFSETS

The following procedure assumes that G54 (Work Offsets #1) offsets are being used instead of G50 (Floating Zero) offsets. It also assumes that the face of the material is considered Z0.

1. Make sure all G50 offsets are zero. They are zeroed on power up or can be zeroed by using F7 (PARMS) - F2 (COORD) and editing the G50 offsets.

2. Make sure G54 X offset is zero. It can be zeroed by using F7 (PARMS) - F2 (COORD) and editing the G54 offset. (This relates to material length.)

3. Make sure tool #1, Z length, is zero. It can be zeroed by using F7 (PARMS) - F3 (TOOLS) and editing the tool table.

4. Make sure all tool wear offsets are zero. They can be zeroed using F7 (PARMS) - F4 (WEAR) and editing the wear offset table.

   Note: When a tool length is set, the wear offset for that tool is zeroed.

5. Make tool #1 active. Do an MDI T0101 M6.

6. Using the handwheel, touch the tip of the tool to the face of the material (or line up the center of the tool tip to the face of the part for diamond shaped turning tools). Then press F8 (G54-Z). This relates to the material length entered in programs for the graphics.

7. Using the handwheel, make a small turning cut on the diameter of the material. Press F7 (XTOOL) and enter "1" for the tool number. Then enter the diameter of your small cut for the position.

   For drills, center the drill at the center of the material. Press F7 (XTOOL) and enter "1" for the tool number. Then enter 0 for the position.

   The X offset for diamond shaped facing tools can be set like a drill or by lining up the center of the tool tip to a diameter on the part. Press F7 (XTOOL), and then enter the tool number and position.

8. Make the next tool active. Do an MDI T#### M6. (T0202 for tool #2, T0303 for tool #3 . . . etc.)

9. Using the handwheel, touch the tip of the tool to face of the material (or line up the center of the tool tip to the face of the part for diamond shaped turning tools). Then press F7 (ZTOOL) and enter the tool number, then enter 0 for the tool position.

10. Using the handwheel, make a small turning cut on the diameter of the material. Press F7 (XTOOL) and enter the tool number. Then enter the diameter of the small cut for the position.

    For drills, center the drill at the middle of the material. Press F7 (XTOOL) and enter the tool number. Then enter 0 for position.

    The X offset for diamond shaped facing tools can be set like a drill or by lining up the center of the tool tip to a diameter on the part. Press F7 (XTOOL), and then enter the tool number and position.

11. Go to step #8 (for the next tool).

12. Enter the tool types and tool nose radii in the tool table by pressing F7 (PARMS) - F3 (TOOLS) and then editing the tool table. F1 (TOOL>) toggles the tool type.

Some sample tool types:

1 - inside turning (chuck towards the tail stock)

2 - inside turning (tail stock towards the chuck)

3 - outside turning (tail stock towards the chuck)

4 - outside turning (chuck towards the tail stock)

5 - ???

6 - inside threading
  - inside turning using a diamond insert
  - boring tools using a diamond insert

7 - drills
  - face grooving
  - facing tools

8 - threading
  - outside turning using a diamond insert
  - cut off tools
  - outside diameter grooving

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.*

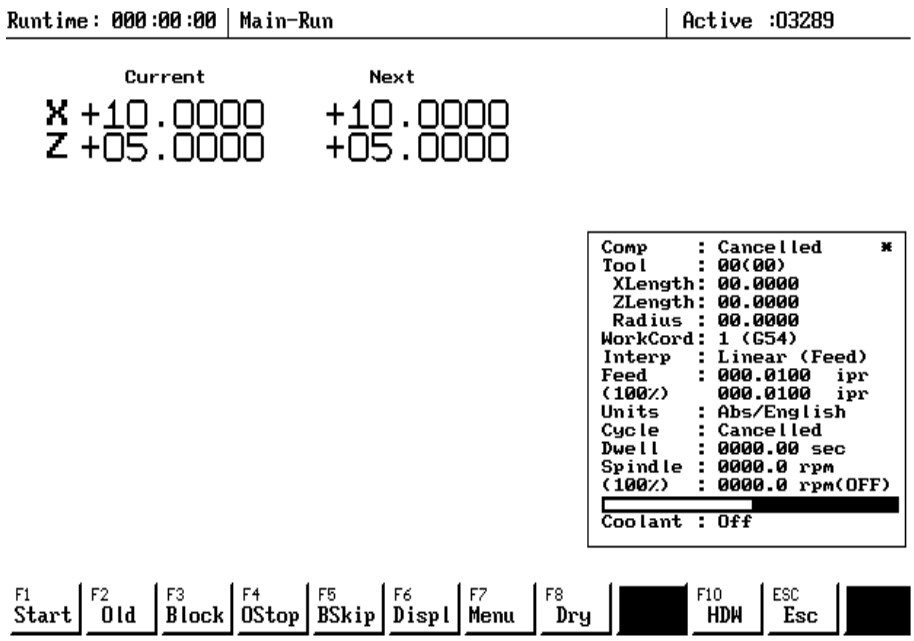
To check the work and tool offset, select [MDI].

For each tool: Type T0101 M6 (T0202 . . etc.) [ENTER]
[CYCLE START] Then type: G0 X0 Z.1 or safe distance

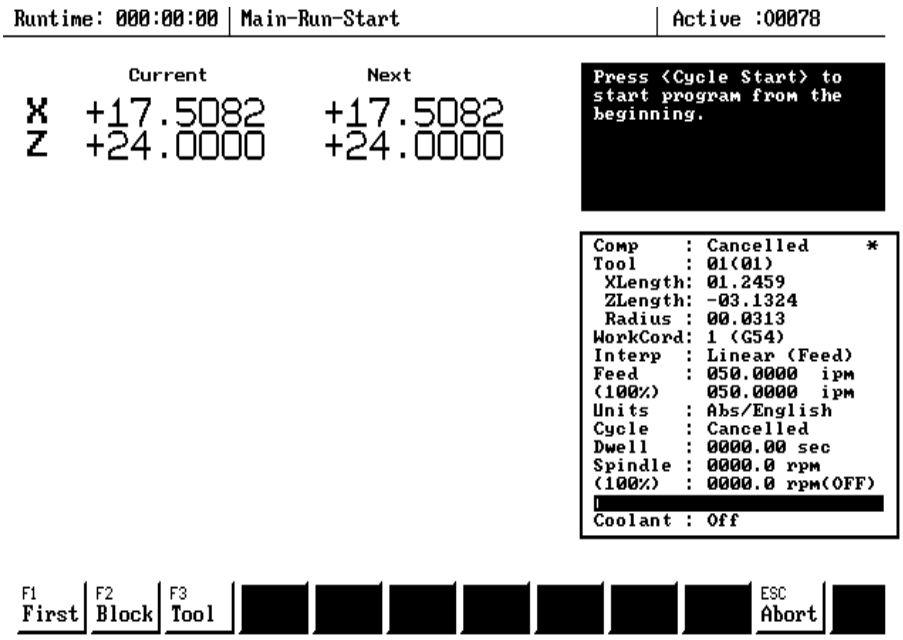
Caution: ***The machine will move at a rapid feedrate. Each tool should position itself to the center of the spindle and 0.1000 or safe distance from the face of the part.***

5.5 Run F4 RUN (MAIN-RUN)

Run is used to execute the active program. Upon pushing the RUN button the following screen appears:

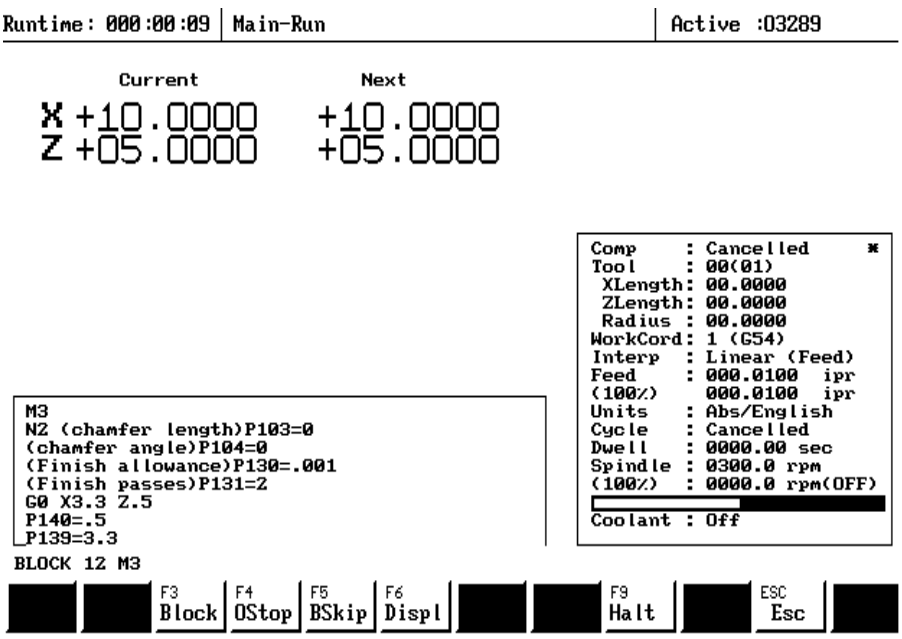


After the above screen appears, F1 START would be pushed to run the active program shown in the upper right-hand box and the following screen would appear:



The F1 FIRST key is automatically selected when entering this screen from the RUN screen. Therefore if it is desired to run the active program from the beginning, all that is necessary is to push the CYCLE START button. If F2 BLOCK is pushed, the control will request that the desired block or sequence number (to start the program from) be typed in, followed by ENTER. If the Cycle Start button is pushed the active program will start running from the selected block number. If F3 TOOL is pushed, the control will request a tool number. After typing the tool number (to start the program from) followed by an ENTER, the CYCLE START button is pushed, and the active program will start running at the desired tool number and the following screen will appear.

Note: If the block number or tool number requested is not found in the active program, a message will say it was not located and Cycle Start will start the program from the beginning.



This screen is the basic run screen after CYCLE START has been pressed. It has two new additions, a block number display and F9 HALT. The block number shows the current line being executed as the program runs. The HALT key is similar to FEEDHOLD in that when pushed the machine will stop. However, unlike FEEDHOLD, the HALT also exits the "Run" mode and allows a new program to be started.

5.5.1 F9 RESUM (MAIN-RUN)

Once a program has been "Halted" the "Resume" feature of the control becomes active. The F9 RESUM 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: Verify, MDI, Home, Emergency Stop or modifying parameters. If none of the above functions are done the axis can be Jogged or Handwheeled away from the work, the spindle turned on or off, or any other function performed, and the "Resume" remains active. As long as the Resume is active the F8 key on the Run screen will show a "Resume" function. If the Resume function is selected, the active program will be resumed at the halted point, when "Resume" "Cycle Start" is executed.

If the Z position is less than the Z clearance:

1. Z rapids to its clearance.
2. X rapids to its halted position.
3. Wait for another cycle start.
4. Z moves to its halted position at its last feed/minute feedrate.

5. Resume program from this position.

If the Z position is greater than the Z clearance:

1. X rapids to its halted position.
2. Z rapids to its clearance.
3. Wait for another cycle start.
4. Z moves to its halted position at its last feed/minute feedrate.
5. Resume program from this position.

Note: Once the "Run" or "Verify" modes have been entered, a "Halt" must be executed if the mode is to be exited prior to program completion.

5.5.2 F2 OLD (MAIN-RUN-OLD)

The OLD key will allow entry, in the message window, of the number of an existing text program. After the number has been entered, the control will check the text programs currently in memory to see if a program by that number is there. If it is, that program will become the active program. If not, an error stating that the program was not found will appear and after pressing the ESC key, another number may be entered.

The F3, F4, F5 and F8 keys on the RUN screen set the mode of operation a program will run in. When these keys are in a highlighted state, the functions will be active in any currently running program or program to be run.

5.5.3 F3 BLOCK (MAIN-RUN)

When the BLOCK switch is activated the program will stop at the end of each block. Each time the CYCLE START button is pushed one more block will be run.

5.5.4 F4 OSTOP (MAIN-RUN)

When the Optional Stop switch (OSTOP) is activated the program will stop at each M01 command. When the CYCLE START button is pushed the program will continue to run.

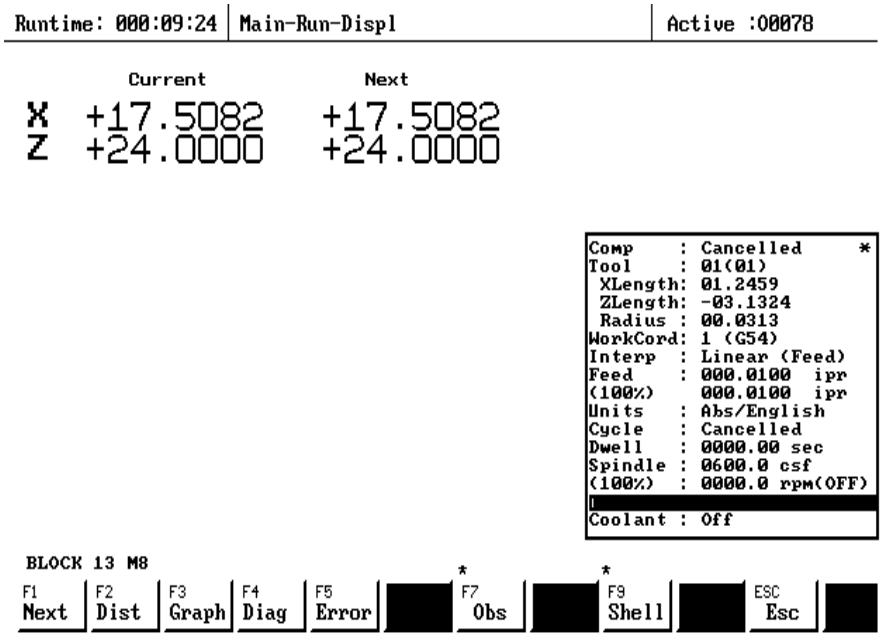
5.5.5 F5 BSKIP (MAIN-RUN)

When the Block Skip switch (BSKIP) is activated the program will skip all blocks started with a "/".

Example: /M5 When the block skip switch is active the spindle off command will not be executed.

5.5.6 F6 DISPL (MAIN-RUN-DISPL)

The F6 DISPLAY key (DISPL) can be accessed from a number of screens. The following screen is shown as though the DISPL was entered from the Main Run 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. When the DISPL key is pushed the following screen will appear:



**Note: F7 and F9 only come up in protected modes and are normally used for service purposes.*

5.5.6.1 F1 NEXT (MAIN-RUN-DISPL)

When the NEXT key is activated the position display on the run screen displays the current position and the next position.

	CURRENT	NEXT
X	+ 00.0000	+ 00.0000
Z	+ 00.0000	+ 00.0000

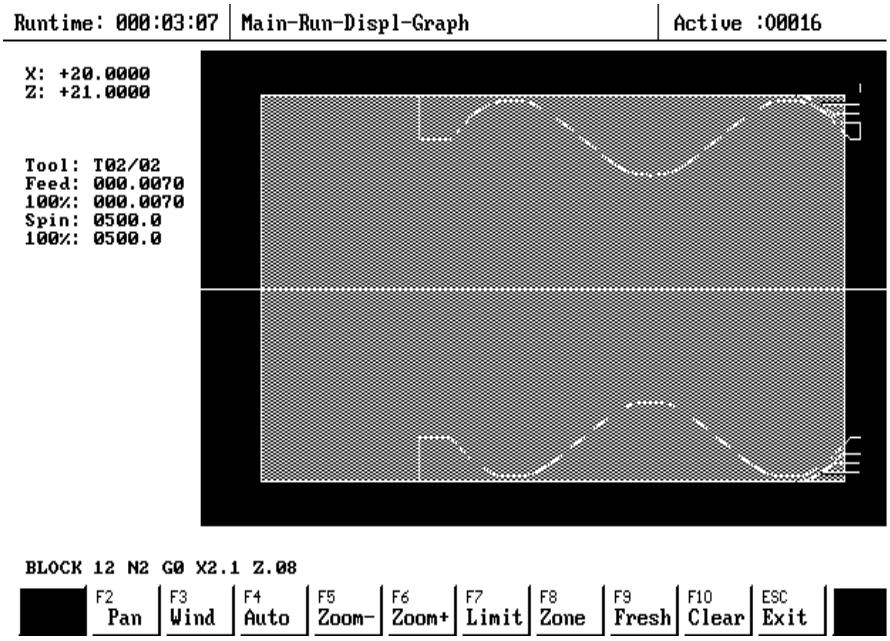
5.5.6.2 F2 DIST (MAIN-RUN-DISPL)

When the DISTANCE key (DIST) is activated the position display on the RUN screen displays the current position and distance to go.

	CURRENT	DIST
X	+ 00.0000	+ 00.0000
Z	+ 00.0000	+ 00.0000

5.5.6.3 F3 GRAPH (MAIN-RUN-DISPL-GRAPH)

If the GRAPH key is pressed the control switches from displaying text to a graphic display of the active part program. The following screen will appear.



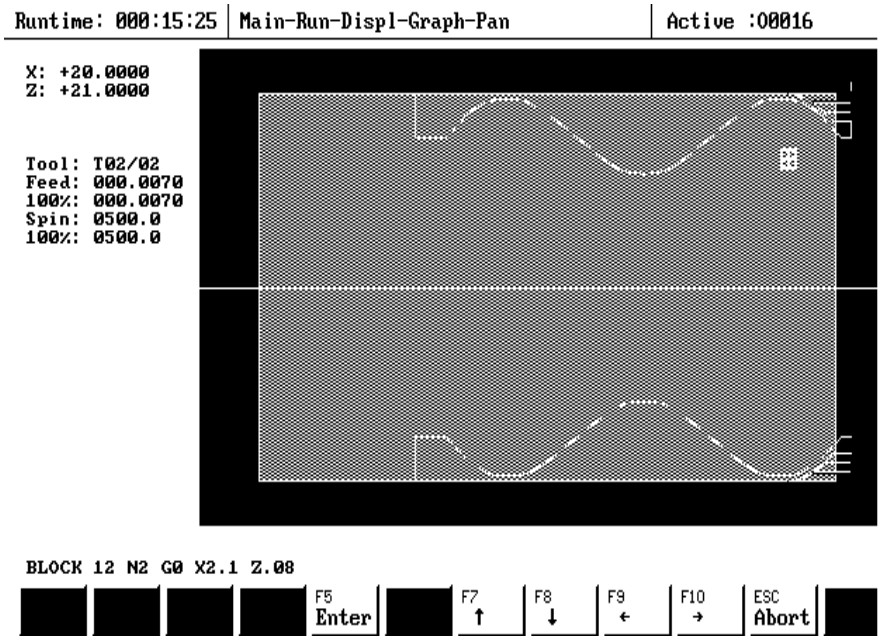
Rapid moves are represented by red lines. Feed moves are represented by green lines. The finish pattern of rough cycles and repeat cycles is presented by yellow lines.

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

Note: The graphics are cleared from the display at the start of run or verify of a program.

5.5.6.3.1 F2 PAN (MAIN-RUN-DISPL-GRAPH-PAN)

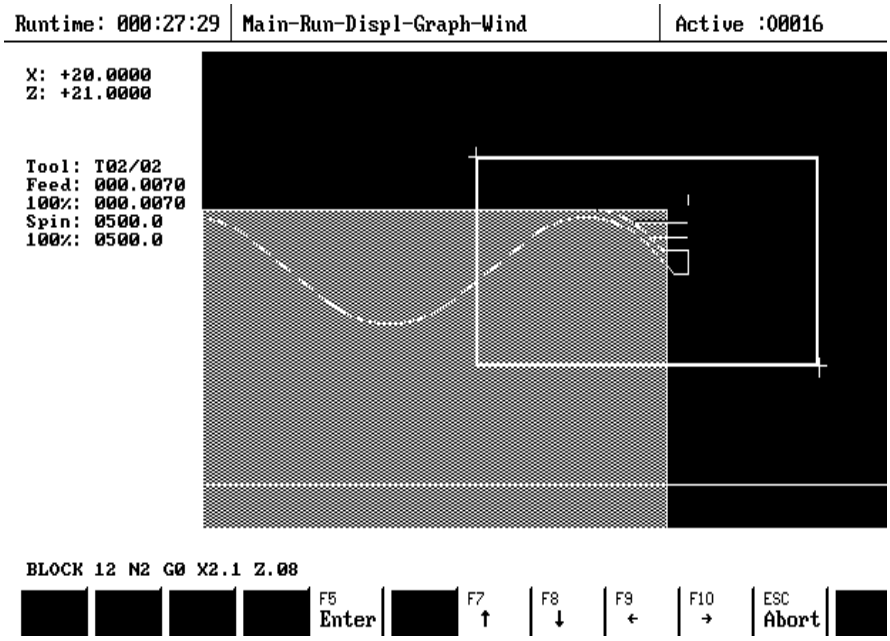
The F2 PAN key selects the Pan function (PAN) which allows the operator to pan around a part. The following display will appear.



Cross hairs appear on the screen and can be moved around using the arrow keys F7 thru F10 or the numeric keypad. To pan, simply move the cursor to the point on the display which is desired to be at the center of the screen and push F5 ENTER or the ENTER key on the keyboard. The display will shift to its new position and panning can be started again. The ESC key will cancel the Pan function and return to the Graph screen.

5.5.6.3.2 F3 WIND (MAIN-RUN-DISPL-GRAPH-WIND)

The F3 WIND key selects the window function which allows the operator to window in on a particular area of the part. The following display will appear when Window is selected.



Cross hairs appear on the screen and can be moved around using the arrow keys F7 thru F10 or the numeric keypad (the same as Pan). To window, move the cursor to the first corner of the window area and hit F5 ENTER or the ENTER key. Then move the cursor around until the desired area of the part to be viewed is enclosed in the rectangular box being drawn on the screen and hit ENTER. The area enclosed in the box will now be displayed on the entire screen. The ESC key will cancel the window function and return to graph screen.

5.5.6.3.3 F4 AUTO (MAIN-RUN-DISPL-GRAPH-AUTO)

The F4 key selects the Auto Zoom function. This function automatically scales and centers the described material size.

5.5.6.3.4 F5 ZOOM- (MAIN-RUN-DISPL-GRAPH)

The F5 key selects Zoom- which decreases by one half the size of the part currently being displayed on the screen. Generally this function is used to view a larger portion of the part.

5.5.6.3.5 F6 ZOOM+ (MAIN-RUN-DISPL-GRAPH)

The F6 key selects the Zoom+ function. This function doubles the size of the part being displayed on the screen. Generally this function is used to enlarge a specific area of a part enabling one to see greater detail.

5.5.6.3.6 F7 LIMIT (MAIN-RUN-DISPL-GRAPH)

The F7 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 overtravel. The machine tool builder sets the machine's software limits.

5.5.6.3.7 F8 ZONE (MAIN-RUN-DISPL-GRAPH)

The F8 key draws a box on the screen which corresponds to an axis safe zone. This is a zone which the tool cannot enter. If the tool is programmed into this box an *"attempted to move into safe zone"* error will be generated.

5.5.6.3.8 F9 FRESH (MAIN-RUN-DISPL-GRAPH)

The F9 key redraws the currently displayed part on the screen.

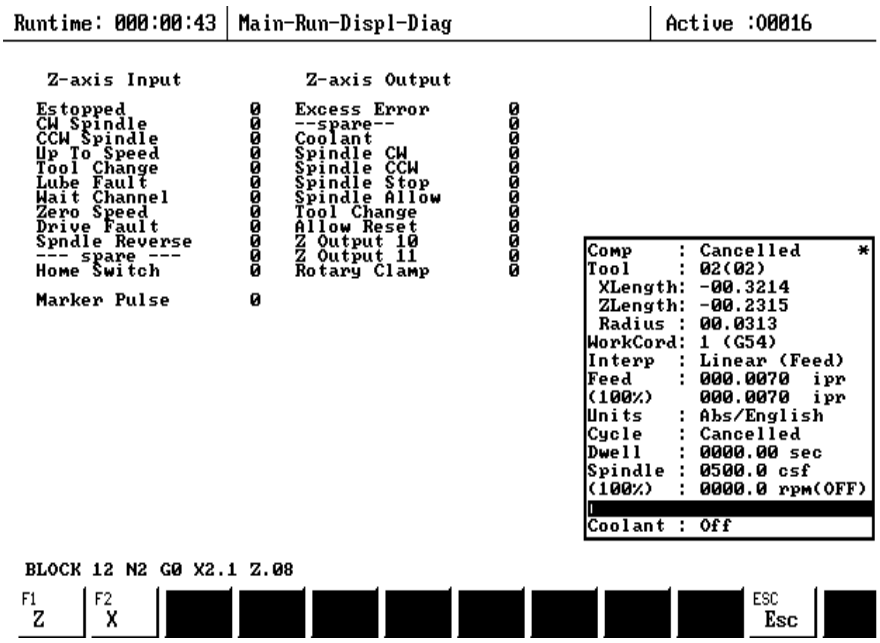
5.5.6.3.9 F10 CLEAR (MAIN-RUN-DISPL-GRAPH)

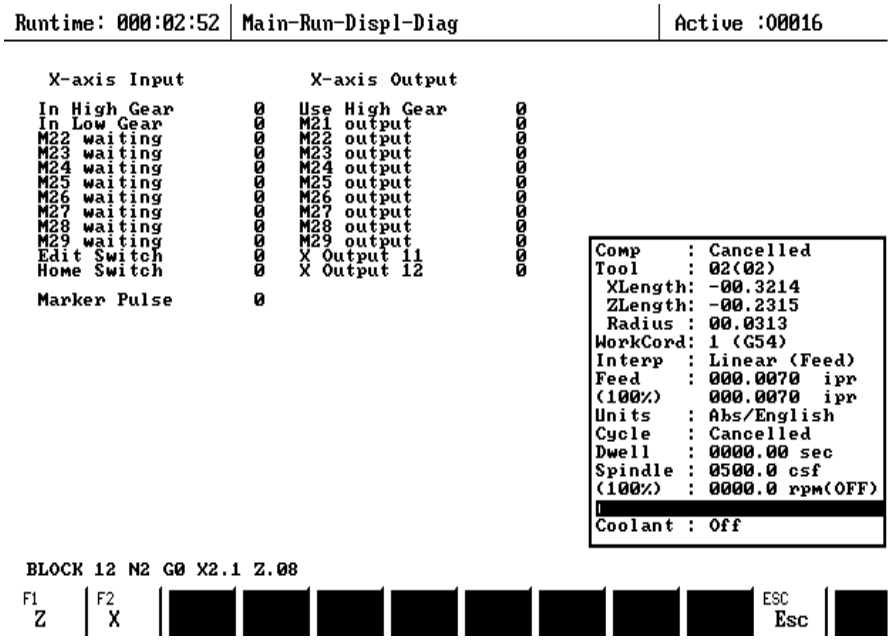
The F10 key clears the current display buffer. After the clear screen command nothing will be displayed until either the program is Verified or Run again. It is generally used to clear MDI moves from the graphic display.

5.5.6.4 (F4 DIAG) MAIN-RUN-DISPL-DIAG

The F4 key is mainly used for machine setup or troubleshooting machine functions. The diagnostic screens shows all the external I/O bits connected to the CNC. The status of each bit is continuously displayed on the screen, and as they change on the machine the screen will be updated. Function keys F1 thru F5 display the various axes enabled on the CNC. Selecting X or Z brings up the appropriate axis I/O channels. A "0" displayed on the screen means the I/O channel is not active. A "1" indicates active.

The following screens represent the displayed information for the various axis selections.





Note: Machines with tool changers, etc. will have different diagnostics screens.

5.5.6.5 F5 ERROR (MAIN-RUN-DISPL-ERROR)

The F5 ERROR key changes the position display to show each axis following error instead of axis position. The following error display is intended to help in machine setup or troubleshooting an axis problem.

5.5.6.6 F7 OBS (MAIN-RUN-DISPL-OBS)

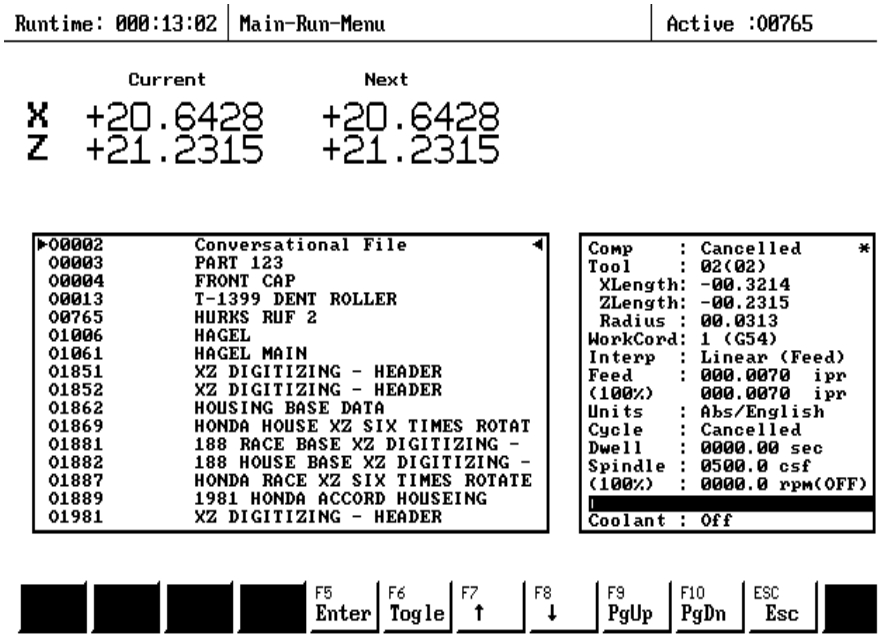
The F7 key will only display in access level 2 or greater. OBS will be highlighted and is only available if the current display is F1 NEXT, F2 DIST or F5 ERROR. It will display information on the motion control cards.

5.5.6.7 F9 SHELL (MAIN-RUN-DISPL-SHELL)

F9 SHELL is available if the control is in access level 2 or greater. It will SHELL out to DOS, to return to the CNC type EXIT.

5.5.7 F7 MENU (MAIN-RUN-MENU)

The F7 MENU key selected from the Run or Verify screen brings up a window containing a listing of all the available programs which can be run. The next screen will be displayed when the program menu is requested.



To activate one of the programs listed in the window, simply use the arrow and page keys to move the cursor to the desired program and push F5 ENTER or the ENTER on the keyboard. The Menu function can be called from other screens but works the same way from all. When called from the Run or the Verify screen, the selected program becomes the active program. When called from an edit screen the selected program becomes the current program being edited. F6 TOGLE will display the file size, last time and date it was edited, and the first 7 characters of the file name.

Note: The file name is the characters enclosed in parenthesis on the first block of the file. If there are no parenthesis on the first block the file name is the first block.

5.5.8 F8 DRY (MAIN-RUN-DRY)

When the DRY RUN switch is active all program feedrates will run at the dry run feedrate.

Note 1: The spindle commands can be ignored during dry run by setting a parameter.

Note 2: Switching to or from DRY RUN can only be done if the program is first halted then switched to or from DRY RUN and then resuming the program. The F8 DRY key will not be visible if the program is running.

5.5.9 F10 HDW (MAIN-RUN-HDW)

When the HDW switch is active, the axes moves in the program will relate to turning the electronic handwheel. The feedrates (feed/rev. or feed/min.) will affect the distance moved per click of the handwheel. There are four miscellaneous parameters that affect the distance moved per click of the handwheel. They are:

1. Handwheel Encoder PPU
   This should be 400 for our current systems.

2. Cranking Minutes/Rev
   This determines how many minutes of normal program execution time should correspond to one turn of the handwheel. Multiplying an IPM feed times this factor results in an IPR feed (inches per turn of the handwheel). This should be about 0.0010 for our current systems.

3. Cranking Max IPM
   This allows for a reasonable handcranked speed for slow feedrate programmed moves, while limiting the speed and avoiding excess errors on high feedrate programmed moves.

4. Cranking Factor
   This is the defined multiplier for each handwheel click. It should be about 100 for our current system.

   For lathe IPR mode:
   Cranking Factor = X*IPR Scale* (Resolution Spindle)/ (Resolution Handwheel); e.g., if X = 10, one turn of the handwheel will have the same effect as would ten turns of the spindle.

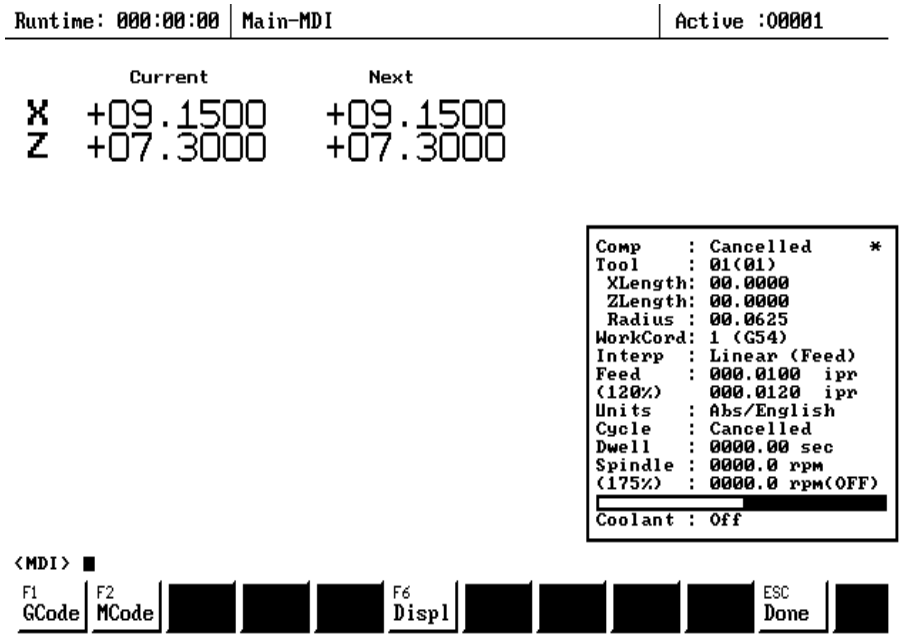
   If X = 10 is desired, then
   Resolution Spindle = 4000
   Resolution Handwheel = 400
   IPR Scale = 1
   Then Cranking Factor = 100

If handwheeling and dry running a program, the distance moved per click of the handwheel relates to the Dry Run Feed parameter.

If handwheeling a program and a tapping cycle is started, the message, *"Tapping cycle will not be in handwheel mode. Press any key to continue."*, will appear.

5.6 Manual data input F5 MDI (MAIN-MDI)

The F5 MDI key from the Main menu selects the MDI (manual data input) function. Through MDI any programmable machine function can be executed one function or one block at a time. When MDI is selected the following screen appears.



As the functions are typed in they will appear on the MDI line at the bottom of the screen. After the data has been typed in, the ENTER key must be pushed to end the block. At this point a Cycle Start will execute the MDI line. When MDI is selected any active program will be halted. During MDI the ESC key will terminate any MDI command. F6 will bring up the normal graphic displays discussed earlier. During MDI the graphic display will graph all the MDI moves as they are executed.

5.6.1 Gcode (MAIN-MDI-MCODE)

GCODE will bring up a list of legal G codes with a short description of each code.

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 Floating Zero	18 ZX Plane
20 English	21 Metric
22 Safe zone check on	23 Safe zone check off
28 Reference Return	29 Return from Reference
30 2nd-4th Ref Return	32 Threading
34 Threading	35 Threading
36 Threading	38 Initial 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 Tool Change	48 X to Tool Change
49 Tool Length Cancel	50 Set FLZ, Max RPM's
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 Move	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	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 Cycle
93 Absolute	94 Facing Cycle B
96 Const Surf Speed	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

5.6.2 Mcode (MAIN-MDI-MCODE)

MCODE will bring up a list of legal M codes with a short description of each code.

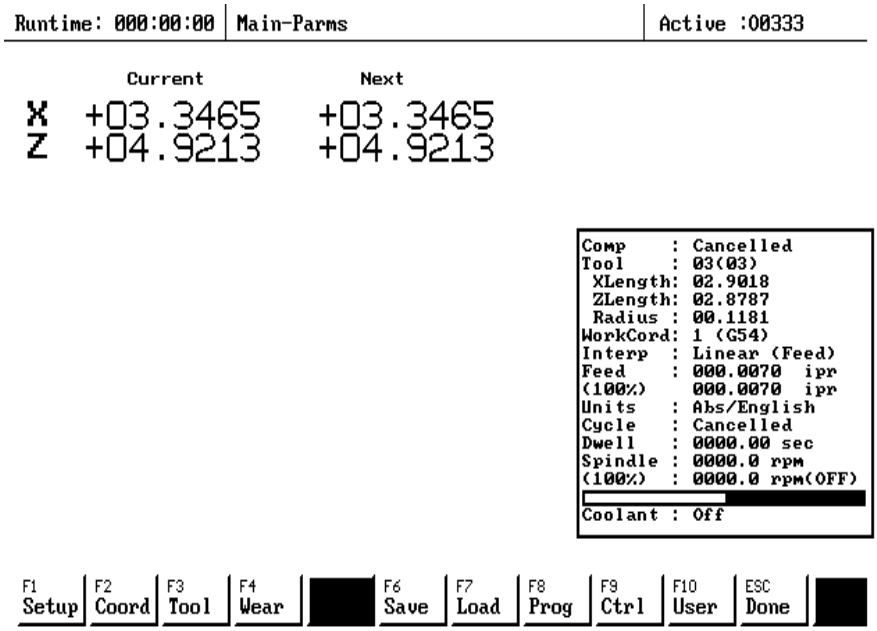
00 Program Stop	01 Optional Stop
02 End of Program	03 Spindle On CW
04 Spindle On CCW	05 Spindle Off
06 Tool Change	08 Coolant on
09 Coolant Off	30 SpindleOff,End of Program
90 Graph Off	91 Graph On
98 Call Program	99 End of Program

5.7 Display F6 DISPL (MAIN-DISPL)

For F6 DISPLAY key, see explanation under MAIN-RUN-DISPL. This function can be entered from several screens.

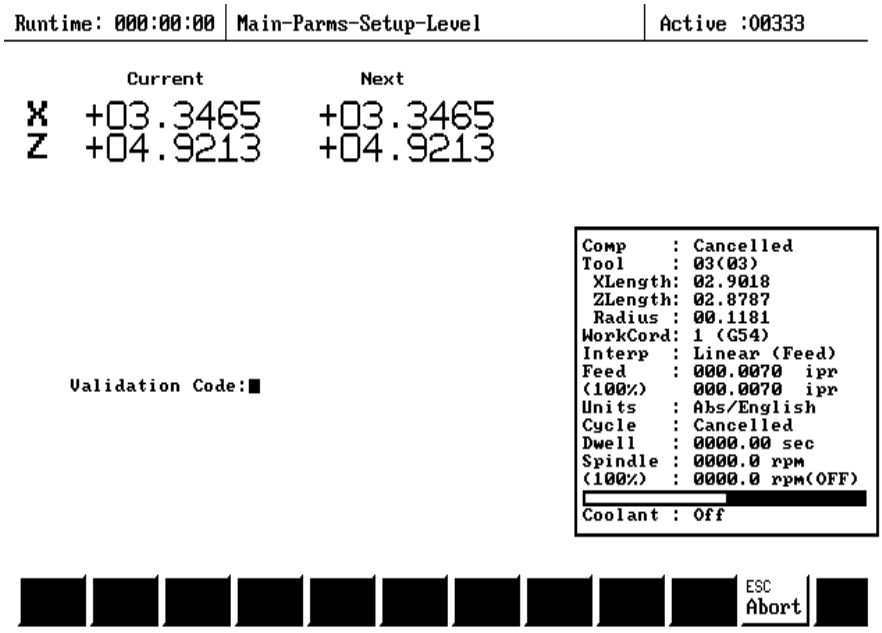
5.8 Parameters F7 PARMS (MAIN-PARMS)

The F7 PARMS key from the main screen brings up this parameter screen.

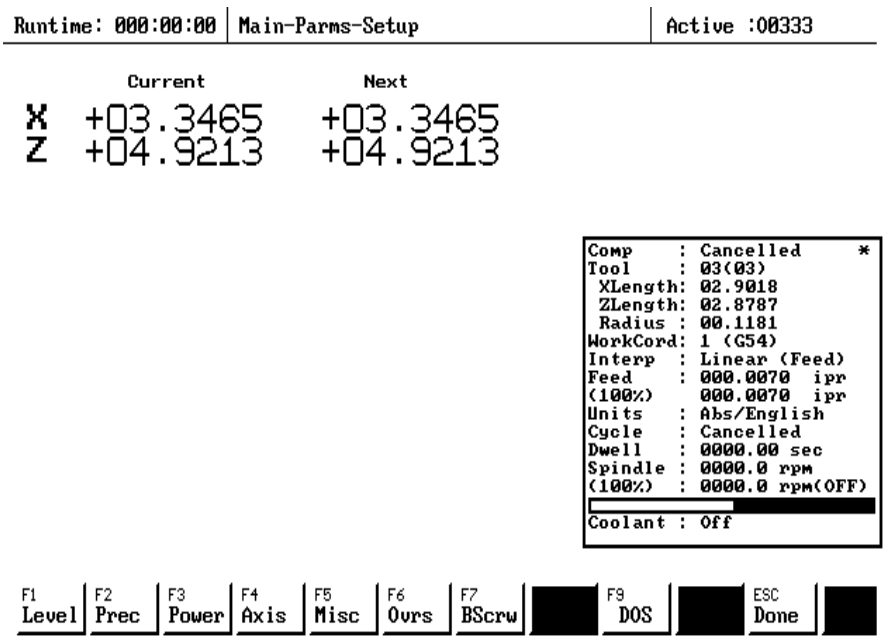


5.8.1 F1 SETUP (MAIN-PARMS-SETUP)

The F1 SETUP selection brings up the parameters which make the control unique to a particular machine or application. When F1 level is selected the following screen appears.



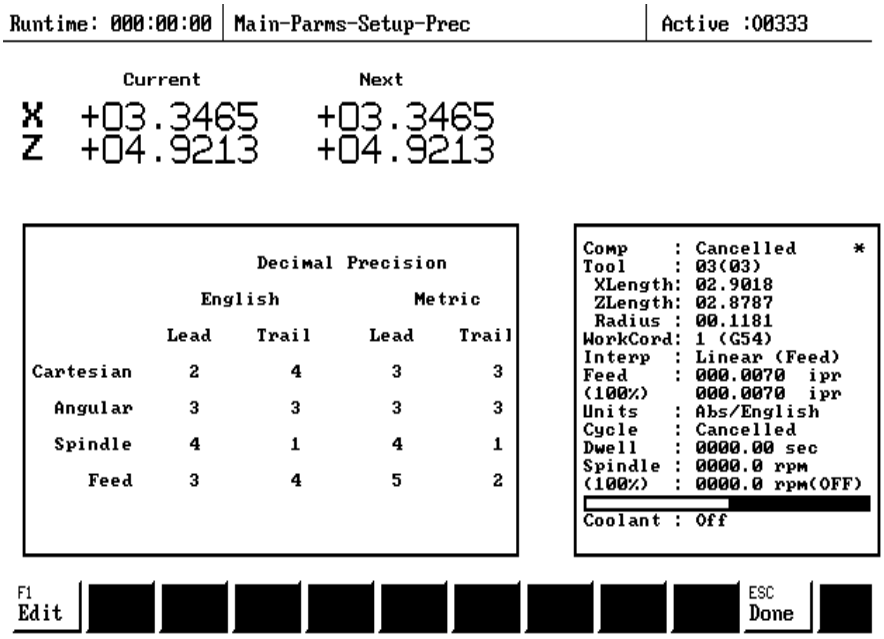
The CNC requires a Validation Code and an Access Level number to allow the machine setup parameters to be displayed or changed. These are supplied by the machine tool builder and should be part of the system parameter setup sheet. Assuming the proper validation code has been entered, the following screen will appear.



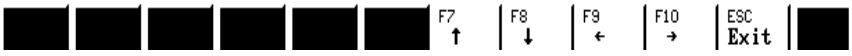
Note: *If changes are made to the setup parameters the new parameters should be saved on a floppy disk for back-up purposes.*

5.8.1.1 F2 PREC (MAIN-PARMS-SETUP-PREC)

If the F2 selection for Machine Precision is made, the following screen will be displayed.



Keys displayed in the Edit Mode:

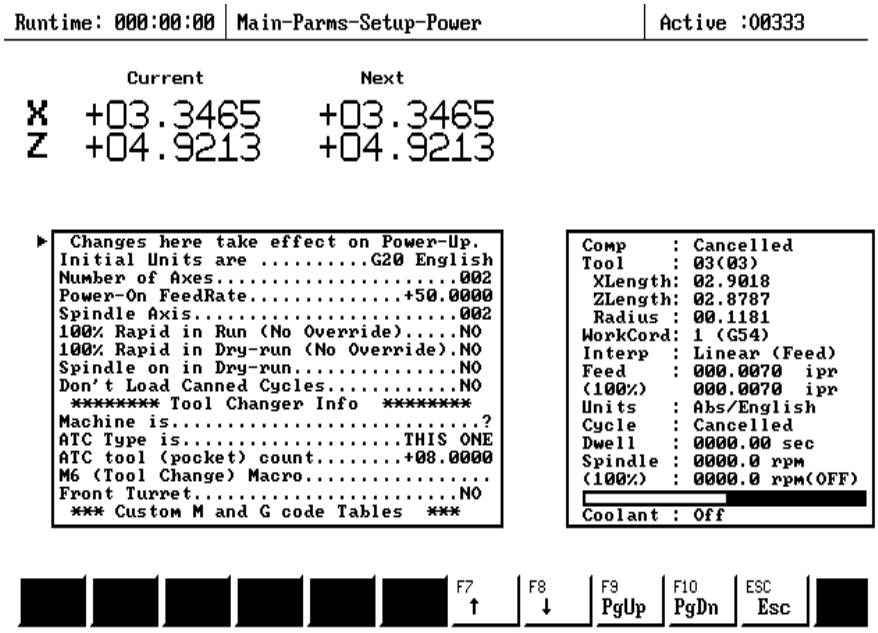


The above screen shows some typical settings for leading and trailing zeroes for the different coordinate systems. The number of leading and trailing zeroes is unlimited, but some practical limits do exist. If the numbers get too large they will not fit on the screen in their allotted space, and if they are smaller than the feedback units they will not cause movement. To change a parameter push F1 EDIT. A series of arrow keys will be displayed; simply use them to move the cursor to the desired parameter and type in the new number. Once all the numbers have been edited, pushing the EXIT key will validate the new numbers and return to the previous screen. These parameters are for all numbers entered into the control except axis. The axis parameters are set separately in the "Axis" parameters.

5.8.1.2 F3 POWER (MAIN-PARMS-SETUP-POWER)

POWER parameters are parameters which directly relate to the configuration of the machine tool and will normally be set by the machine tool builder. Changes to these parameters require the machine to be powered down and back up for the changes to take effect.

The F3 key brings up the power-on defaults as shown in the following screen.



Initial Units are:	G20 English or G21 Metric depending on the feedback or screw type.
Number of Axes:	Can be 1 to 6. (Normally 2).
Power-On Feedrate:	Can be any number up to the maximum feedrate.
Spindle Axis:	1 to 6 1=Z and 2=X (Normally 1)
2nd Handwheel Encoder Axis:	"3" means that it will read the second handwheel from an Acroloop jumpered as #3. "0" means there is only one handwheel.

100% Rapid in Run: (No Override)
NO means the feedrate override will affect rapid in the run mode.
YES means the feedrate override will not affect rapid in the run mode.

100% Rapid in Dry-Run: (No Override)
NO means the feedrate override will affect rapid in the dry run mode.
YES means the feedrate override will not affect rapid in the dry run mode.

Spindle on in Dry-Run:
NO means the spindle will not come on in the dry run mode.
YES means the spindle will come on in the dry run mode.

Don't Load Canned Cycles:
NO means Canned Cycles will be loaded.
YES means Canned Cycles will not be loaded. The canned cycles (drill, tap, groove, etc.) take approximately 14K of RAM.

Block Skip On:
If *YES* the block skip will be active on power up.

Optional Stop On:
If *YES* the optional stop will be active on power up.

* * * Tool Changer Information * * *

ATC tool (pocket count): number of pockets on the tool changer

M6 (Tool Change) Macro: name of the program used for custom M6

Front turret:
YES means the tool will be displayed as coming from the bottom in the graphics. It also relates to the tool types in the tool table as coming from the bottom.

* * * Custom M and G Code Tables * * *

Custom Gcode 09010
Custom Gcode 09011
Custom Gcode 09012
Custom Gcode 09013
Custom Gcode 09014
Custom Gcode 09015

Load the Gcode number into the program # that is the macro for that code.

Custom Gcode 09016	It is not possible to have a
Custom Gcode 09017	custom Gcode for G0.
Custom Gcode 09018	
Custom Gcode 09019	
Custom Mcode 09020	Load the Mcode number into the
Custom Mcode 09021	program # that is the macro for
Custom Mcode 09022	that code.
Custom Mcode 09023	
Custom Mcode 09024	It is not possible to have a
Custom Mcode 09025	custom Mcode for M0.
Custom Mcode 09026	
Custom Mcode 09027	
Custom Mcode 09028	
Custom Mcode 09029	

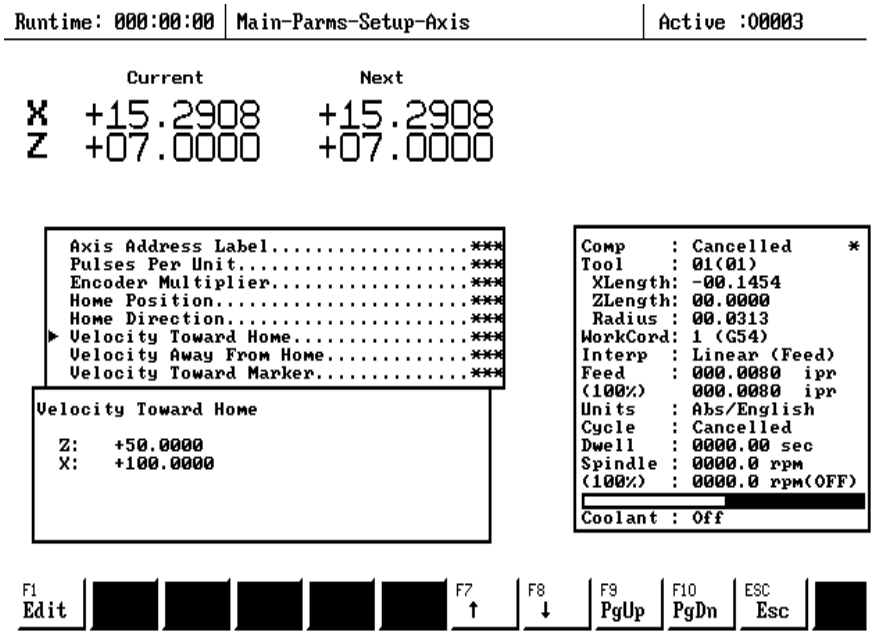
The custom code programs can be in the parts directory or the RAM directory. If they are in both directories the program in the RAM directory takes precedence.

Parameter file version:	This is set by the control and used for version updates.
Use smaller soft keys:	Yes for 14" monitors, no for 12" monitors.

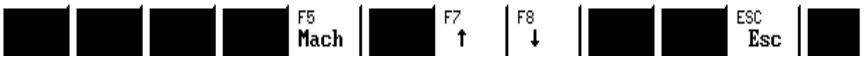
To change a parameter push F1 EDIT. A series of arrow keys will be displayed; simply use them to move the cursor to the desired parameter and type in the new number.

5.8.1.3 F4 AXIS (MAIN-PARMS-SETUP-AXIS)

If the F4 AXIS is selected, the following screen will be displayed.



Keys displayed in the Edit Mode:

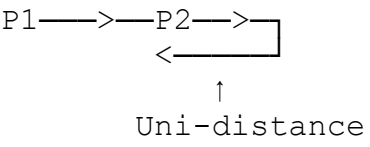


The PGUP, PGDN and arrow keys move through the tables on the upper display. The lower display changes to display the data associated with the cursor position of the upper display. To edit the values in these tables the cursor is positioned via the arrow keys to the desired parameter in the upper screen. When the EDIT key is pushed the cursor will move to the lower screen. At this point again move the cursor using the arrow keys to the axis or parameter desired and type in the new values using the keypad. After the new values have been entered, push the ESC key. The new value will be entered and the cursor will go back to the upper screen and the selection process can be started over. Some parameters can be related to the machine position. To edit or load these parameters, use F5 MACH to load the X and Z positions.

The following is a list of all the selectable parameters displayed in this mode and a description of their functions.

Axis Address Label Z 90.0000 X 88.0000	ASCII code assigned to each axis
Pulses Per Unit Z + 10000.0000 X + 10000.0000	The number of pulses the feedback gives per unit of travel.
Encoder Multiplier Z 02.0000 X 01.0000	Sets an internal multiplier on the number of pulses coming from the encoder. Param set 0 = 4 times multiplication Param set 1 = 2 times multiplication Param set 2 = 1 times multiplication
Home Position Z -10.0000 X -15.0000	The dimension assigned to the machine zero or home position. Normally these are the same as the negative travel of the machine.
Home Direction Z +00.0000 X +00.0000	Defines the direction of rotation of the motor when a home is commanded. CW = 00.0000 CCW = 01.0000
Velocity Toward Home Z 60.0000 X 10.0000	Sets the feedrate at which an axis seeks the home limit switch.
Velocity Away From Home Z 06.0000 X 12.0000	Sets the velocity at which an axis feeds off the home limit switch.
Velocity Toward Marker Z 03.0000 X 06.0000	Sets the velocity at which an axis searches for the encoder pulse.
Home Sequence Z 02.0000 X 01.0000	The Home Sequence numbers determine the order the axes will home in: #1 first and #2 next. Axes with the same number home together. 0 will cause the axis to not home.
Positive Limit Z 00.1000 X 00.1500	Dimension from machine zero where the positive software limit occurs

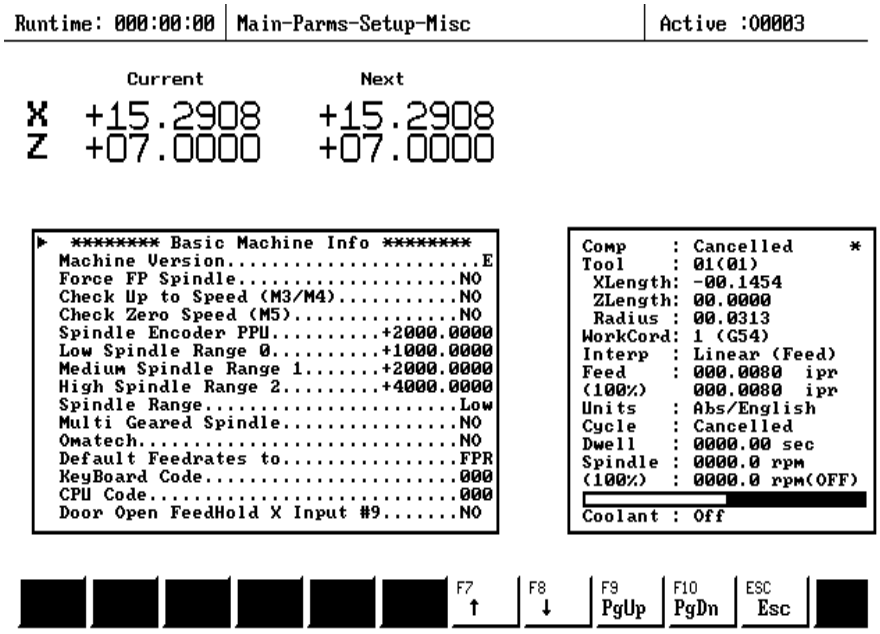
Negative Limit X -31.0000 Z -18.0000	Dimension from machine zero where the negative software limit occurs
Maximum Feed Z 100.0000 X 200.0000	Sets the maximum G01 feedrate in inches per minute or mm per minute
Dry Run Feed Z +75.0000 X 150.0000	Sets the Dry Run feedrate in inches per minute or mm per minute
Rapid Velocity Z 200.0000 X 400.0000	Set the maximum G00 feedrate in inches per minute or mm per minute
Rapid Acc/Dec	40.000 The Acc/Dec constant is a number between 1 and 200 that determines the rate at which the axis velocity is stepped up. The smaller the number the longer the Acc/Dec times will be. Acceleration and deceleration in this control are linear ramps from one speed to another.
Slow Jog Velocity Z 00.0100 X 00.0200	Velocity in IPM or mm per minute
Slow Jog Acc/Dec Z 20.0000 X 20.0000	Parameter set the same as Rapid Acc/Dec
Rapid Jog Velocity Z 100.0000 X 200.0000	Velocity in IPM or mm per minute
Rapid Jog Acc/Dec Z 20.0000 X 20.0000	Parameter set the same as Rapid Acc/Dec
Jog Key Direction Z +01.0000 X +00.0000	Reverse the direction of the Jog keys

In Position Z 00.0100 X 00.0200	Sets the distance in feedback units from the destination point where other axes will start their movement. In inches or metric.
G00 Unidirectional X 00.0000 Z 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 Z 00.0000 X 00.0000	Same as G00 unidirectional except only active in a G60 block
Backlash Z 00.0006 X 00.0009	Sets the distance in inches or mm which the control will compensate for lost motion whenever an axis reversal takes place. Active in all modes.
Excess Error Z 00.2000 X 00.4000	Sets the distance in inches or mm the machine can lag behind the CNC before the CNC will shut the system down due to an excess following error condition. 00.0000 = will never force an excess error
Rotary=0 Linear=1 Z 01.0000 X 01.0000	Sets whether an axis should be treated as circular or linear. In circular the feedrate is interpreted as degrees per minute rather than IPM or mm per minute.
English Leading Z 02.0000 X 02.0000	Sets the number of characters to the left of the decimal point for the inch system, for the specified axis only.

English Trailing Z 04.0000 X 04.0000	Sets the number of characters to the right of the decimal point for the inch system, for the specified axis only.
Metric Leading Z 03.0000 X 03.0000	Same as English Leading except for the metric case.
Metric Trailing X 03.0000 Z 03.0000	Same as English Trailing except for the metric case.
Home Switch=0 Marker=1 Z 00.0000 X 00.0000	Sets whether an axis will seek a home limit switch and then the marker pulse, or just seek the nearest marker pulse.
Max Handwheel Error Z 00.2000 X 00.4000	When the excess error reaches this value, pulses from the handwheel are ignored. Error is specified in inches or mm.
Tool Change Z 00.0000 X 00.0000	X moves to this location on a G48 (X to toolchange) or M6 (toolchange) command. Z moves to this location on a G47 Z to tool change. This position is specified in inches or mm and is relative to the home zero.

5.8.1.4 F5 MISC (MAIN-PARMS-SETUP-MISC)

The F5 MISC key brings up some miscellaneous setup parameters dealing with the spindle, software options and M codes. When MISC is selected the following screen appears:



Keys displayed in the Edit Mode:



Miscellaneous parameters are edited similarly to the POWER parameters. The following is a list and short description of the MISC parameters.

**********Basic Machine Info**********

Machine Version	E (-E is the current model)
Force FP Spindle	YES for Front Panel, NO for CTRL2.XX DAC
Check Up to Speed (M3/M4)	Control will wait for up to speed signal from the spindle controller before continuing on M3 (spindle CW) or M4 (Spindle CCW)

Check Zero Speed (M5)	Control will wait for zero speed signal from the spindle controller before continuing on M5 (spindle off)
Spindle Encoder PPU	Pulses per rev of spindle, used for tapping, threading and feed per rev.
Handwheel Encoder PPU	Pulses per rev of the electronic handwheel should be 400 for current systems.
Low Spindle Range 0	Max spindle speed for low range
Medium Spindle Range 1	Max spindle speed for medium range
High Spindle Range 2	Max spindle speed for high range
Spindle Range	The range the spindle is in: Low, Medium, or High. Scaled assumes the range based on the programmed spindle speed and puts out the appropriate voltage.
Tap Spindle Ramp Time (Sec)	The tapping deceleration point is adjusted based on spindle RPM, pitch and tap spindle ramp time so not to overshoot past the programmed depth. It relates to the time in seconds to decelerate the spindle to 0 RPM's from full RPM's when in the tapping gear (normally low gear).
CPU code	Not used
Door Open Feedhold X Input 9	The X input 9 (door open) shuts the spindle 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 move, will not jog, will not do a move in run, but it will handwheel. The machine will do I/O related functions other than turning on the spindle, i.e., arm in, draw bar, etc. If the machine is tapping and the doors open, the machine will finish the tap and then feedhold and stop the spindle.

	If the doors open while the machine is running: 1. The spindle will shut off. 2. The axes will stop moving. 3. The Feedhold light will come on. 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). If the machine is not running and the spindle is not running, opening of the doors should have no effect on the machine.
Digital Readout in E-Stop	If YES the axis position display will be updated even when E-Stopped.
Probe Axis and Probe Input	This designates the axis and the input number for the Probe1 or Probe2 commands to look at.
Handwheel Encoder PPU	The number is the encoder counts per rev of the electronic handwheel. This should be 400 for the current systems.
Cranking Minutes/Rev	This indicates how many minutes of normal program execution time should correspond to one turn of the handwheel. Multiplying an IPM feed times this factor results in an IPR feed (inches per turn of the handwheel). It should be about 0.0010 for the current systems.
Cranking Max IPM	This allows for a reasonable handcranked speed for slow feedrate programmed moves, while limiting the speed and avoiding excess errors on high feedrate programmed moves.

Cranking Factor	This is the defined multiplier for each handwheel click. It should be about 100 for our current system. For lathe IPR mode: Cranking Factor = X*IPR Scale* (Resolution Spindle)/ (Resolution Handwheel); e.g., if X = 10, one turn of the handwheel will have the same effect as would ten turns of the spindle. If X = 10 is desired, then Resolution Spindle = 4000 Resolution Handwheel = 400 IPR Scale = 1 Then Cranking Factor = 100 For IPM mode, this parameter has less intuitive meaning; however, Cranking Factor/Cranking Mins Per Rev is proportional to the max feedrate allowed while handcranking.
IPR Scale	This allows a higher max IPR than normal, e.g., a factor of two doubles the range at the expense of halving the normal max spindle rate. This should normally be 0 (which is equivalent to 1). Some applications may require a 2.
Handwheel Timing	Handwheel Timing is the minimum time in units of 500 μs (microseconds) it takes to calculate a two handwheel move. This is in addition to the time needed to communicate with the acroloops which is already taken into account. Currently setting this parameter requires trial and error. Numbers around 80 seem to work.
**********Software Options**********	
Max graph file size	8000 for standard systems, this limits the amount of graphics on the screen.
Minimum Parts Space	The system will go to the erase screen if the parts space is less in KB than the parameter value

Special Flags	Bit 1 (#1) puts a circle the size of the tool nose radius on the graphics screen when plotting the tool. Bit 2 (#2) will shut off trig help. Bit 3 (#4) will shut off cutter compensation.
Screen Blank Time (min)	Time in minutes to blank the screen if no key is pressed
Use FLZ instead of G54	Used for setting work offset in jog and handwheel mode. YES means use FLZ (G50 offsets) NO means use G54 offsets.
Extract Input Programs	If set to YES the control will extract programs based on 0###'s in file loaded from the floppy drive or received in RS-232
P899	Secret code ????
Resolve DNC programs	If YES, goto's can be used in DNC
Full DOS File Names	If YES, eleven character DOS filenames (FILENAMES.EXT) may be used.
Disable 417 Errors	If YES parameters may be modified while a program is running or verifying.
Multiple Block Display	If YES the current block being executed plus the next seven blocks will be displayed when programs are running.

**********Post M Codes Table**********

Post Mcode #0	
Post Mcode #1	
Post Mcode #2	
Post Mcode #3	
Post Mcode #4	Mcodes listed here will be executed after all other operations within the block.
Post Mcode #5	
Post Mcode #6	
Post Mcode #7	
Post Mcode #8	
Post Mcode #9	

**********Additional Options**********

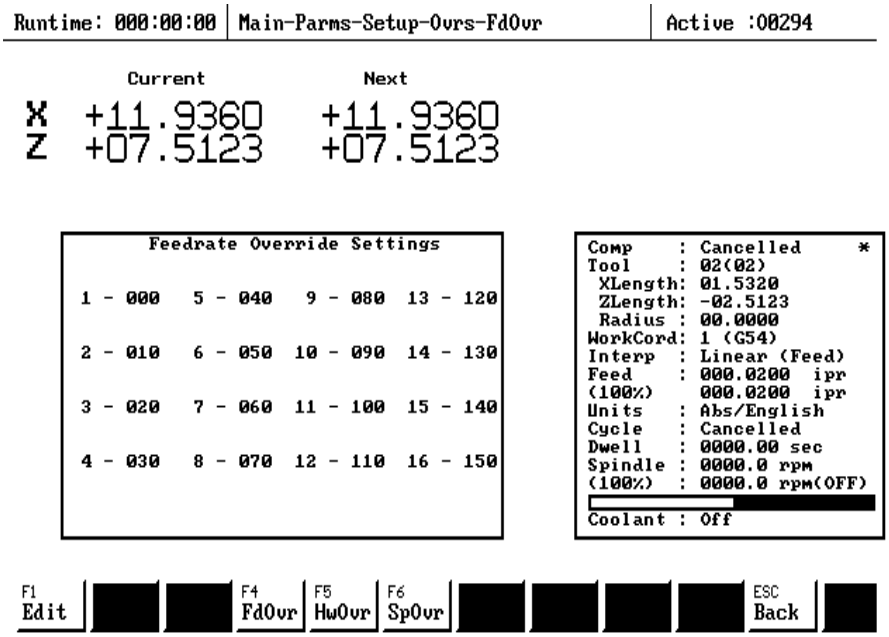
Report File	DOS file names to write DPRNT text to when using POPEN PO
PULSEX pulse delay (MS)	Msec pulse time used with PULSE0 and PULSE1 commands

**********Spindle Power Monitoring**********

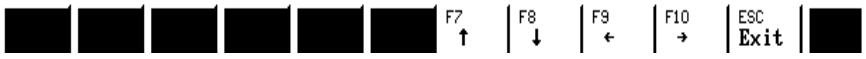
Spindle Power Raw	0..255 value read from ADC
Spindle Power Scale	Multiplier to obtain
Spindle Power Value (AMPS)	Calculated spindle power value
Actual Spindle Factor	Used from displaying actual spindle RPM

5.8.1.5 F6 FDOVR (MAIN-PARMS-SETUP-OVRS-FDOVR)

The F6 FDOVR key brings up the feedrate override parameter settings. These settings determine which percentage will be used for each of the 16 feedrate override switch positions. (i.e. entry 5-040 means 40% at 5th position, 1 is 0%, 2 is 10%, 3 is 20% . . . 16 is 150%) The following screen is displayed when F4 FDOVR is selected. To edit the parameters, select EDIT and move the cursor to the value to be changed. Type in the desired change and hit ESC. The maximum feedrate override setting is 199.

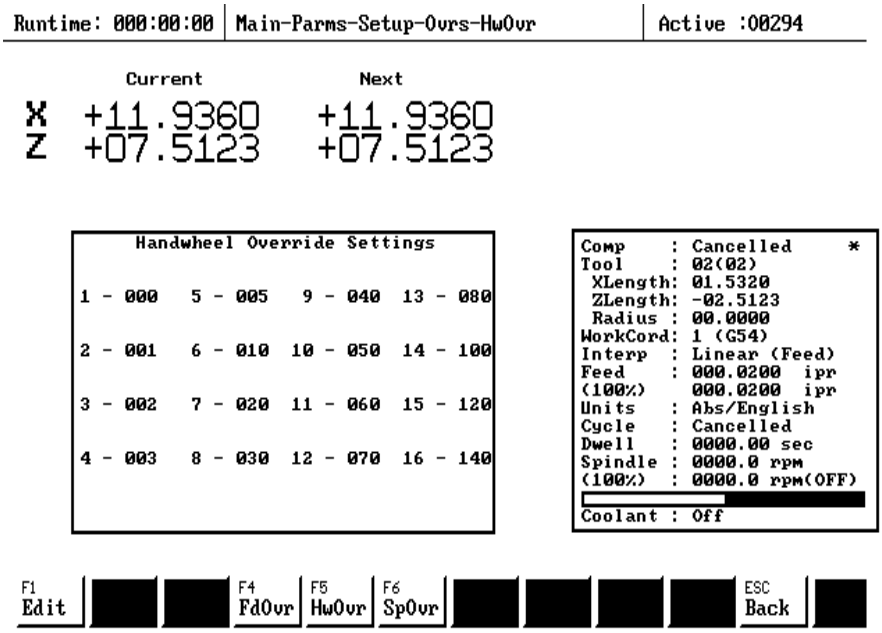


Keys displayed in the Edit Mode:



5.8.1.6 F5 HWOVR (MAIN-PARMS-SETUP-OVRS-HWOVR)

The F5 HWOVR key brings up the handwheel switch settings for the feedrate override switch. These settings determine how far an axis will move for one increment of the handwheel (001=1 pulse). Editing is performed the same way as the feedrate override parameters. The following screen displays the Handwheel Override Settings. The maximum handwheel override setting is 255.

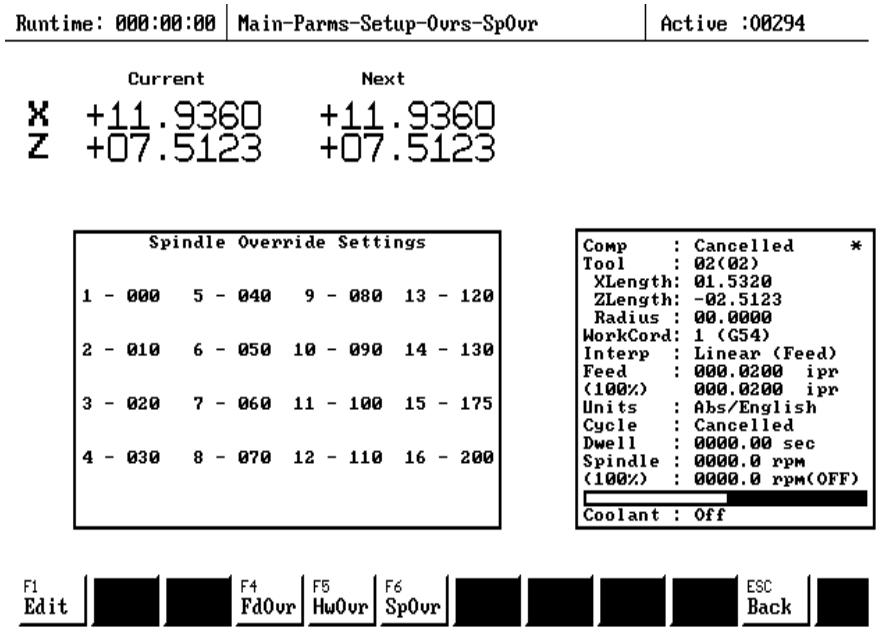


Keys displayed in the Edit Mode:

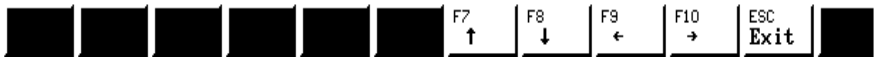


5.8.1.7 F6 SPOVR (MAIN-PARMS-SETUP-OVRS)

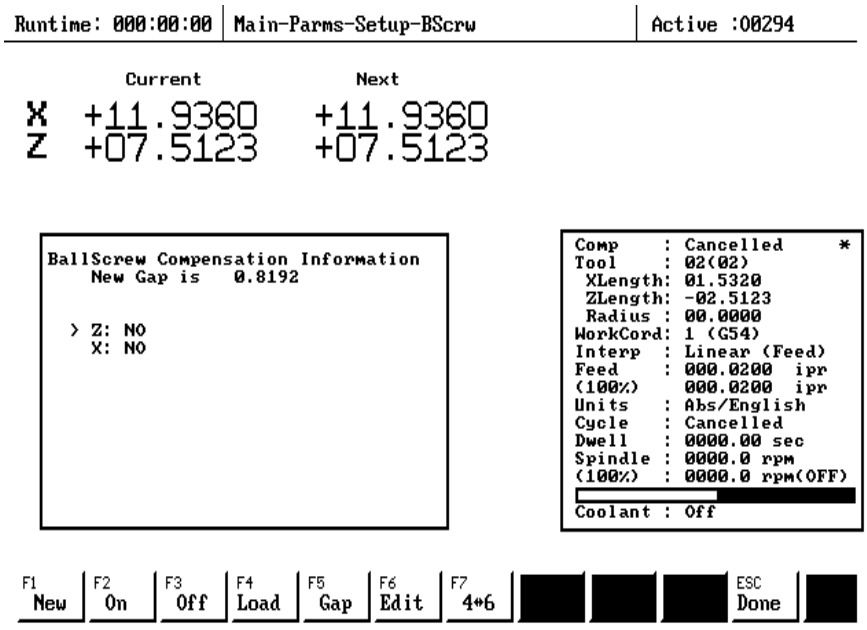
The F6 SPOVR key brings up the 16 spindle override switch settings. These settings are the percentages a spindle command will be overridden at each switch position. The spindle override parameters are changed the same as the feedrate override parameters. The Spindle Override screen is displayed below. The maximum spindle override setting is 255.



Keys displayed in the Edit Mode:



5.8.1.8 F7 BSCRW (MAIN-PARMS-SETUP-BSCRW)



Ball Screw Compensation Table Creation Help

Type X or Z (A,B,C,U,V,W) to select the axis.

F1 (NEW)	creates a new, zero ball screw table
F2 (ON)	turns ball screw comp on for given axis
F3 (OFF)	turns ball screw comp off for given axis
F4 (LOAD)	loads the ball screw table into the axis
F5 (GAP)	change gap
F6 (EDIT)	jumps into editor with ball screw table
F7 (4<>6)	chooses 4 or 6 decimal places.
ESC	DONE

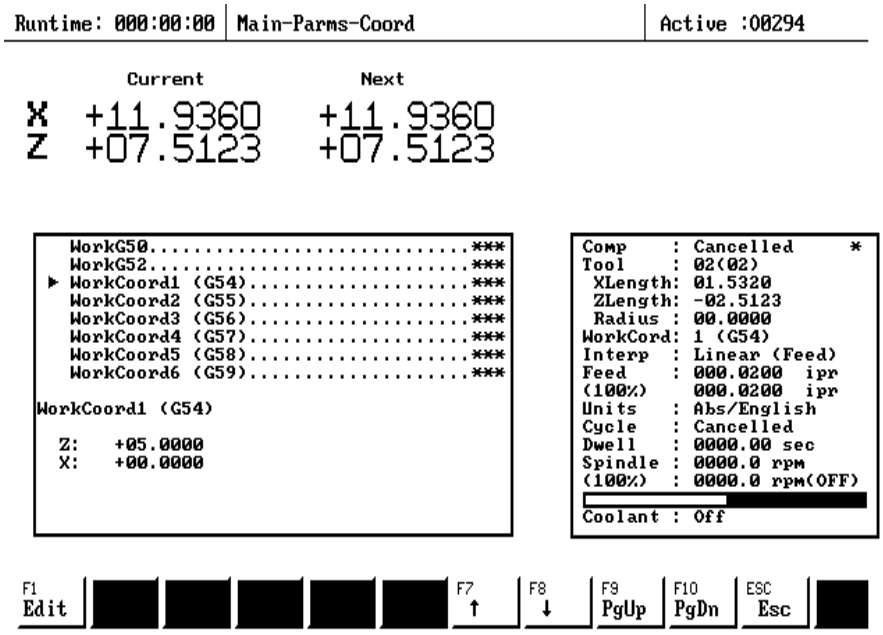
Note: Ball screw compensation is turned on after homing the machine if a ball screw compensation table is present.

5.8.1.9 F9 DOS (MAIN-PARMS-SETUP-DOS)

F9 DOS will EXIT the CNC software and return to the DOS prompt.

5.8.2 F2 COORD (MAIN-PARMS-COORD)

The F2 COORD key off the parameter screen brings up the parameters dealing with the various coordinate systems in the control.



To edit the work coordinate parameters use the PGUP, PGDN and arrow keys to position the cursor to the correct parameter, and then push the EDIT key and arrow to the desired axis. Type in the new values and push ESC.

Keys displayed in the Edit Mode:



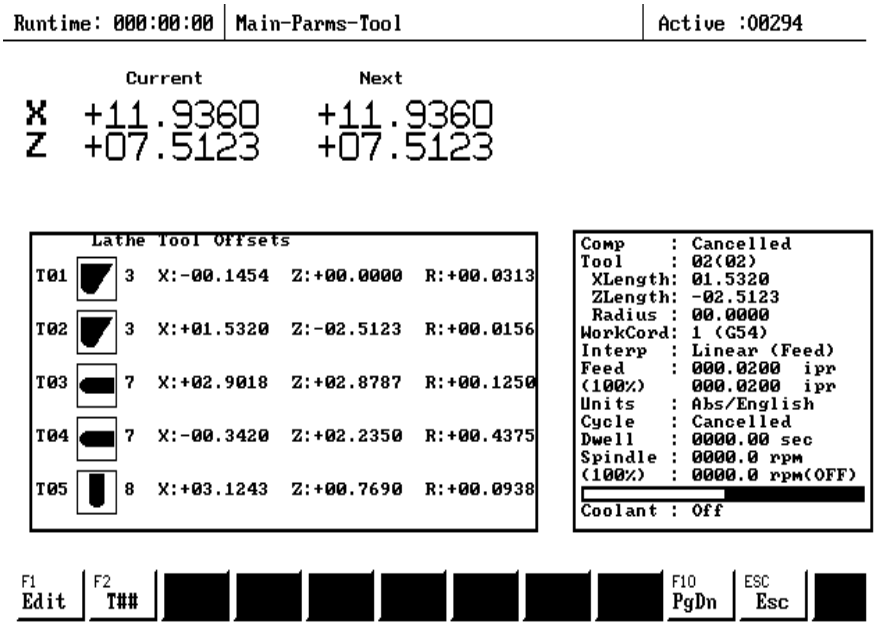
Operation of the Work Coordinate Systems, G50 and G52, was discussed in the Gcode section. These parameters are positions relative to the machine zero and will become the new zero point when they are used. The F5 MACH key in the edit mode enters the current machine position as the work coordinate zero point for X, Y and Z axes.

The next page of this display contains parameters for:

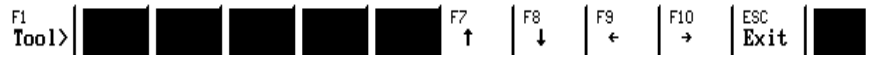
Positive Safe Zone Z 00.0000 X 00.0000	A position relative to machine zero which, along with the negative safe zone position, describes a box which the tool cannot enter.
Negative Safe Zone Z 00.0000 X 00.0000	If the tool is programmed into this box an error will be displayed.
G28 Reference Point 1 G30 Reference Point 2 G30 Reference Point 3 G30 Reference Point 4	Are described in the Gcode section.

5.8.3 F3 TOOL (MAIN-PARMS-TOOL)

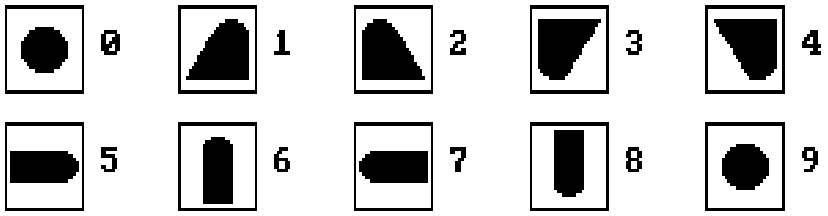
The F3 TOOL key brings up the following screen.



Keys displayed in the Edit Mode:



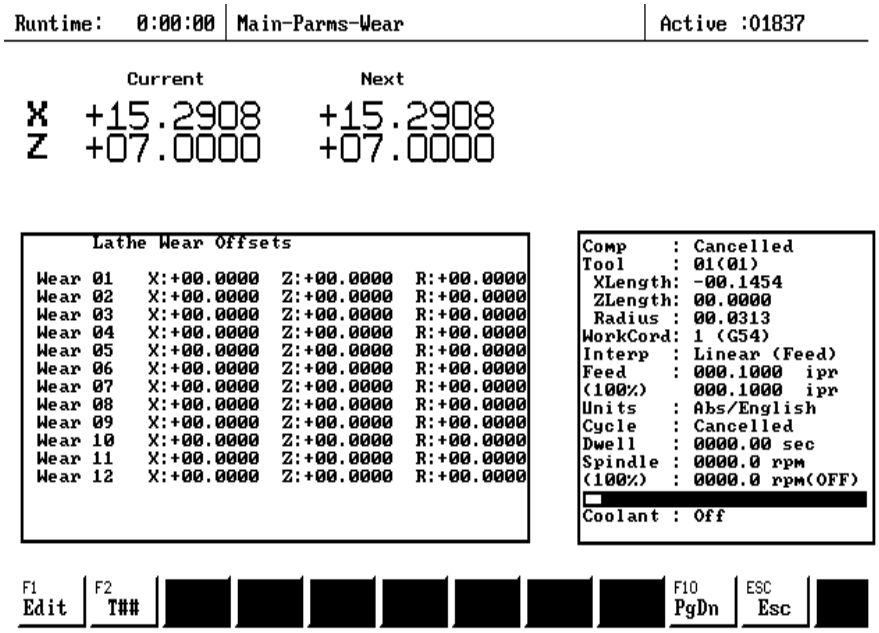
The editing on this screen is similar to all the other PARMS screens. The tool length offsets normally set in the handwheel, can be modified in the tool table. The tool radius and types can only be set in the tool table. Tool types are changed by pushing F1 (TOOL>). There are 24 sets of tool numbers and offsets which can be accessed. The tool types are:



Note: The above tool displays are for a system with the front turret parameter set to NO.

5.8.4 F4 WEAR (MAIN-PARMS-WEAR)

The F4 WEAR key brings up the following screen.



Keys displayed in the edit mode:



Editing the wear offsets is similar to all other PARMS screens. When a tool length offset is set via the handwheel screen, the corresponding wear offset is zeroed. There are no provisions for zeroing the wear offset radii. The operator should zero these when he enters the new values in the tool table. The actual offset is the sum of the values in the tool table and the wear offset table. There are 24 sets of tool wear offsets that can be accessed.

5.8.5 F6 SAVE (MAIN-PARMS-SAVE)

The F6 SAVE key saves the RAM files to a floppy disk (includes parameters, ball screw files, etc.).

5.8.6 F7 LOAD (MAIN-PARMS-LOAD)

The F7 LOAD key loads RAM files from a floppy disk (includes parameters, ball screw files, etc.).

5.8.7 F8 PROG (MAIN-PARMS-PROG)

This set of parameters gives the machine programmer access to all the internal parameters the CNC is using to execute a program. Normally these parameters would be used for display purposes only or 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. These parameters are displayed and edited in the same manner as the coordinate parameters. The following is a list and description of these parameters.

P200 thru P207	Contains the previous programmed position relative to the current work offsets of each axis. P200=Z P201=X . . . etc.
P208 thru P215	Contains the current programmed position relative to the current work offsets of each axis. P208=Z P209=X . . . etc.

P216 thru P223	Contains the previous machine position relative to the machine zero of each axis. P216=Z P217=X . . . etc.
P224 thru P231	Contains the current machine position relative to the machine zero of each axis. P224=Z P225=X . . . etc.
P232 thru P239	Contains the work coordinate offset relative to the machine zero of each axis. P232=Z P233=X . . . etc.
P240 thru P247	Contains the active tool length (H) parameter for each axis. P240=Z P241=X . . . etc.
P248	Contains the current arc radius
P249	Contains the current arc K value (Z center)
P250	Contains the current arc I value (X center)
P251	Current feedrate in feed per minute or feed per revolution
P252	Current dwell time
P253	Current programmed spindle speed in revolutions per minute or constant surface speed
P254	Last Z, X . . . etc. valves executed
P260	Contains active tool number
P261	Contains active tool nose radius
P262	Contains active tool shape
P266	Contains safe zone status
P267	Contains ratio of feedback pulses to program units (English or Metric)

P268	Pending tool number
P270, P271	Last KI values executed (Arc center)
P298	Max spindle speed clamped for constant surface speed
P304	Status if control is in data mode or normal programming
P306	Status of 0=G0, 1=G1, 2=G2, 3=G3 mode
P309	Cutter comp. status 0=G40, 1=G41, 2=G42
P310	Current active canned cycle number: 0=Cancelled, 1=Drill, 2=Drill/Dwl, 3=Peck I, 4=Tap, 5=Bore, 6=Bore II, 7=Face Grooving, 8=Diameter Grooving, 9=Bore/Dwl, 11=Single Turn, 12=Single Face, 13=Single Thread, 14=Multiple Thread, 15=Threading
P311	Absolute or Incremental mode
P312	Feed Unit 0=Feed per minute 1=Feed per revolution
P313	Spindle Unit 0=revolutions per minute 1-Constant surface speed
P315	Inches or Metric dimensions
P316	Scaling on/off
P317	Rotation on/off
P318	Mirror image on/off
P319	Current work coordinate number
P323	Return to R-plane (G39) or Initial Level (G38)
P324	Tapping mode on/off
P365	Last feed per minute feedrate executed
P366	Last feed per revolution feedrate executed

P367	Last RPM spindle speed executed
P368	Last CSS spindle speed executed
P370	True tool number used for tool changers
Block Rate	Average time in seconds per block of the last program run or verified.

5.8.8 F9 CTRL (MAIN-PARMS-CTRL)

This set of parameters is an extension of the Prog parameters. This group deals primarily with parameters used to create the autoroutine and canned cycles. These parameters can be accessed and changed the same as all other parameters in the control. The following is a list and description of these parameters. All coordinates in these parameters are relative to the current work coordinate system.

P100	Thread Variable Lead
P101	Thread Lead
P102	Thread Start Shift Angle
P103	Thread Chamfer Length
P104	Thread Chamfer Angle
P105	Thread Type
P106	Thread Height
P107	1st Thread Cut Depth
P108	Tool Nose Angle
P110	Start Block of Rough, Repeat and Finish Cycles
P111	End Block of Rough, Repeat and Finish Cycles
P112	X Finish Stock of Rough, and Repeat Cycles
P113	Z Finish Stock of Rough, and Repeat Cycles
P114	X Rough Stock of Rough and Repeat Cycles
P115	Z Rough Stock of Rough and Repeat Cycles
P116	Cut Increment
P117	Roughing Type (Pattern or 45 degree retract)
P118	Relief
P119	First Depth
P121	Cut Length
P122	Cut Depth
P123	End Point
P124	Diameter
P125	Turning Taper
P126	Facing Taper
P130	Minimum Thread Cut Amount
P131	Threading Finish Passes
P136	Probe Radius
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
P156	Material Length
P157	Material Diameter
P172	Z Mirror Position
P173	X Mirror Position
P180	Z Scale Position
P181	X Scale Position
P188	Z Scale Factor
P189	Z Scale Factor
P196	Z Rotate Position
P197	X Rotate Position
P198	Angle of Rotation
P199	Incremental or absolute dimensions on G76 . . G79

Spindle Range	Which gear range the spindle is in: low, medium or high. Scaled assumes the range based on the programmed spindle speed and puts out the appropriate voltage.

Comm port, baud rate, parity, data bits and stop bits are communications parameters. See the RS-232 section for more information.

**********Serial Port Data**********

Primary Serial Port	NONE, COM1 or COM2
COM1 Baud Rate	110, 150, 300, 600, 1200, 2400, 4800, 9600, or 19200
COM1 Parity, Data and Stop Bits	N/7/1, N/7/2, N/8/1, N/8/2, E/7/1, E/7/2, E/8/1, E/8/2, O/7/1, O/7/2, O/8/1, O/8/2
COM2 Baud Rate	110, 150, 300, 600, 1200, 2400, 4800, 9600, 19200

COM2 Parity, Data and Stop Bits	N/7/1, N/7/2, N/8/1, N/8/2, E/7/1, E/7/2, E/8/1, E/8/2, O/7/1, O/7/2, O/8/1, O/8/2
Secondary Serial Port	NONE, COM1, or COM2
Tape Start Character	ASCII Value used sent at start of reading paper tape
Tape Stop Character	ASCII Value used sent at end of reading tape
RS232 Buffer Size	For users doing long RS232 communications, up to 256 to improve RS232 in DNC.
LF in change to CR	YES means change all carriage returns received on RS-232 to line feeds
End of Block send CR/LF	YES means send a carriage return/line feed at the end of blocks sent out on the RS-232
RS-232 EOF character	ASCII value of the character to send at the end of the file sent out on the RS-232

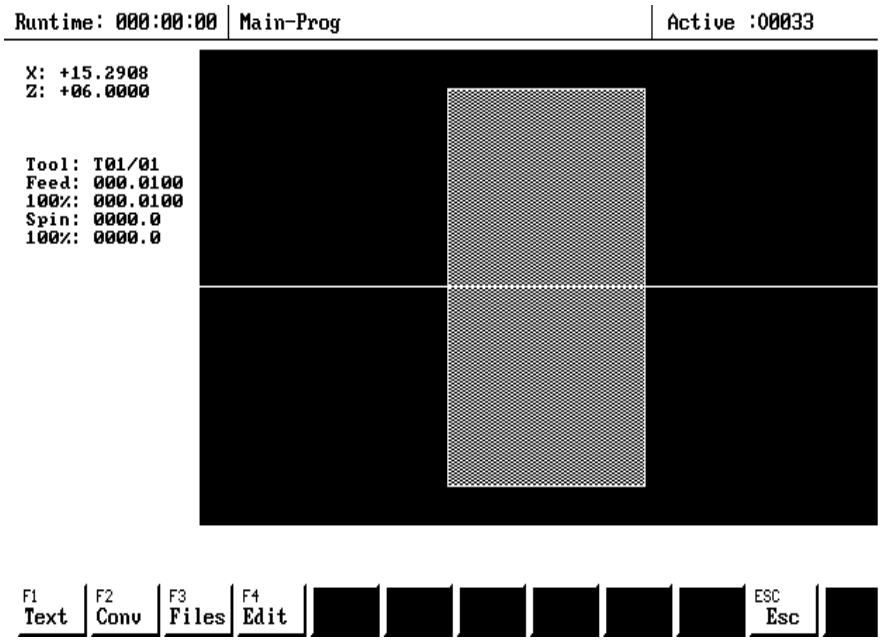
5.8.9 F10 USER (MAIN-PARMS-USER)

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. The editing and displaying formats are identical to the other parameters discussed in this section.

5.9 Programming F8 PROG (MAIN-PROG)

There are two modes of program file creation/editing available on the Centurion 5 control: text and conversational. Access to these features is through the PROG soft key.

Pressing the PROG key will change the soft keypad to allow selection of the type of programming wished, or to transfer programs to or from the floppy disk drive.

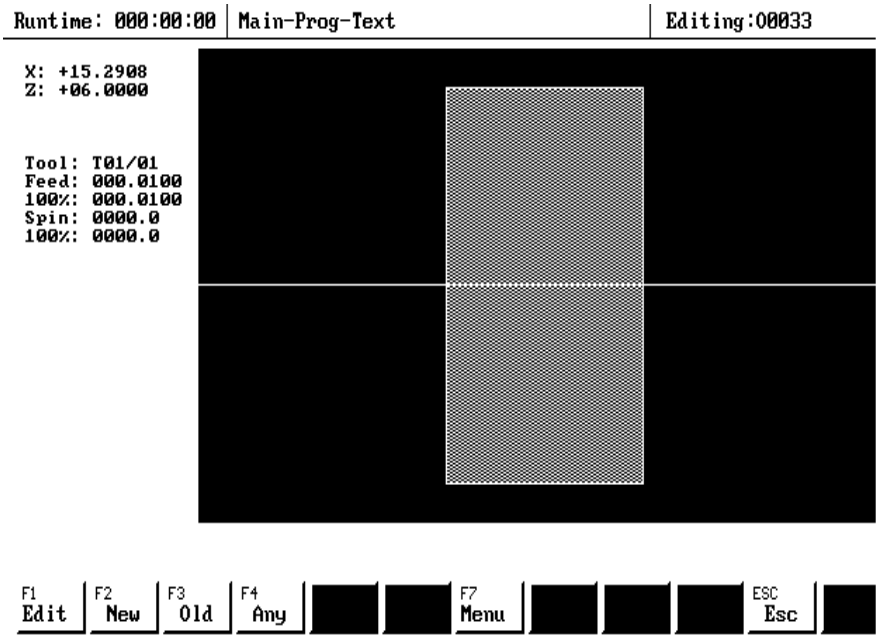


Text and conversational programs are stored in the control in different file formats and have different prefixes to distinguish them. Text programs are prefixed with the letter "O" and are in ASCII format. Conversational programs are in two formats: ASCII, prefixed with an "O"; and conversational, prefixed with a "P".

Only programs prefixed with an "O" may be run or verified.

5.9.1 F1 TEXT (MAIN-PROG-TEXT)

Upon entering the text programming mode the upper right-hand box (containing the active program number) will switch to show the last text program edited.



5.9.1.1 F1 EDIT (MAIN-PROG-TEXT-EDIT)

The F1 EDIT key will select the program shown in the upper right corner as the active text edit program and enter the text editing system.

Text Editing Terms

Editor	A program that allows entry and modification of information, then stores it in a file for later retrieval and/or further modification
Text	Text refers to a sequence of characters and/or lines being edited. The individual characters are manipulated using the widely accepted American Standard Code for Information Interchange (ASCII).
Cursor	The cursor is a small box on the screen that marks where changes are being made to the text

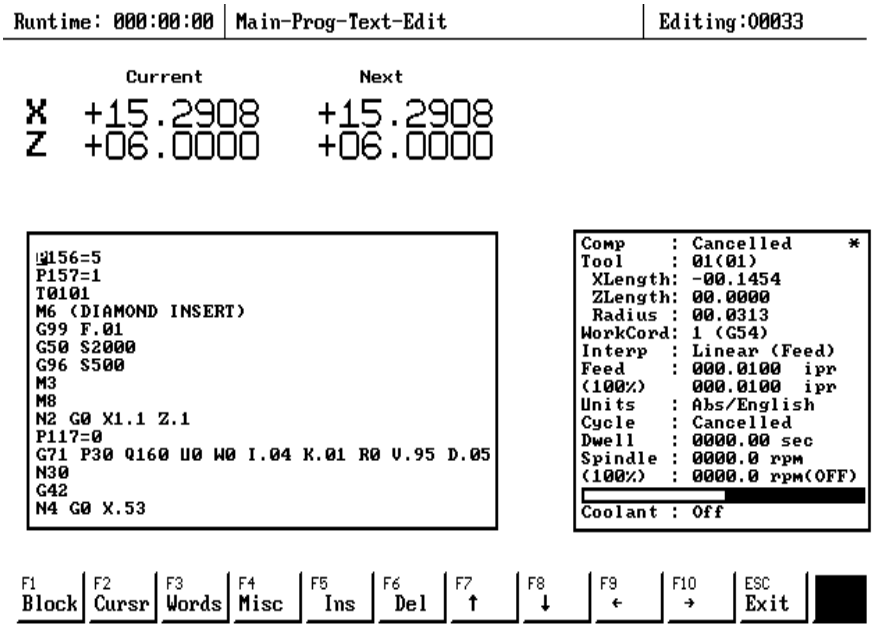
Entering and Editing Text

You enter text in much the same way as you enter text on a typewriter, and most of the keys on the keypads behave in the same fashion (pressing ENTER terminates a block, for example). But there are many important differences.

The cursor always indicates where the new text will be entered, and you can move the cursor in a number of ways. You can copy and move text with block commands. You can locate a particular string of text with the FIND command, and optionally replace it with another string using the CHNG command. And, in most cases, you can even undo your last few changes with the REST (restore line) or UNDO commands. These commands, and many more, are described briefly in the following sections.

(MAIN-PROG-TEXT-EDIT)

The first screen you will see when entering the text editor is the edit screen with the first 16 lines of the program displayed. At the bottom of the screen will be the main edit soft keys.

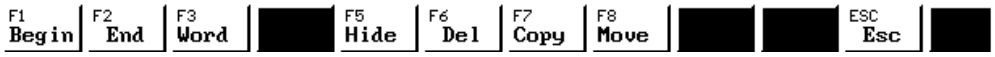


Text may now be entered at the current cursor position.

5.9.1.1.1 F1 BLOCK (MAIN-PROG-TEXT-EDIT-BLOCK)

A block is any arbitrarily defined contiguous unit of text. A block can be as small as a single character or as large as an entire program. Mark a block by placing a begin-block marker at the first character in the desired block, and an end-block marker just beyond the last character. Once marked, the block can be copied, moved, or deleted.

Although marked blocks are normally highlighted so you can see what you've marked, the block may be hidden (or made invisible) with the Hide block command.



F1 BEGIN Marks the beginning of a block. The marker itself is not visible on the screen, and the block becomes visible only when the end-block marker is set.

F2 END marks the end of a block. Like the begin-block marker, the end-block marker is invisible, and the block itself will not be displayed unless both markers are set.

F3 WORD Marks a single word as a block, combining the functions of the Begin-block and End-block commands. If the cursor is positioned within a word, that word will be marked. If it is not within a word, then the word to the right of the cursor will be marked. And if there is no word to the right of the cursor, then the word to the left will be marked.

F5 HIDE Toggles off and on the visual marking of a block.

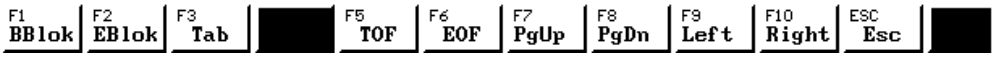
F6 DEL Deletes a marked and displayed block. Although the UNDO last deletion command can usually restore portions of an accidentally deleted block, **there is no command** to restore a deleted block in its entirety, so use this command with care.

F7 COPY Creates a copy of a marked and displayed block at the current cursor position. The original block is left unchanged, and the markers are placed around the new copy of the block.

F8 MOVE Moves a marked and displayed block from its current position to the cursor's position. The markers remain around the block at its new position.

5.9.1.1.2 F2 CURSR (MAIN-PROG-TEXT-EDIT-CURSR)

The F2 CURSR menu contains extended cursor movement commands:



F1 BBLOK Moves the cursor to the position of the block-begin marker.

F2 EBLOK Moves the cursor to the position of the block-end marker.

F3 TAB Moves the cursor to the beginning of the next word.

F5 TOF Moves the cursor to the first character of the program. (Top of file).

F6 EOF Moves the cursor to the last line of the program. (End of file).

F7 PGUP Moves cursor up 15 lines.

F8 PGDN Moves cursor down 15 lines.

F9 LEFT Moves cursor to beginning of line.

F10 RIGHT Moves cursor to end of line.

5.9.1.1.3 F3 WORDS (MAIN-PROG-TEXT-EDIT-WORDS)

The F3 WORDS soft keys represents some of the reserved words that may be used for programming the control.



Pressing a key will cause that word to be printed on the screen. See the section of the manual on programming for the proper usage of these words.

Note: F6 RETRN will print RETURN.

5.9.1.1.4 F4 MISC (MAIN-PROG-TEXT-EDIT-MISC)

This section discusses a number of commands that do not readily fit into any of the other categories.



F1 UNDO Restores whole lines deleted with the Delete line command or the Delete block command. It does not restore single characters or words. To undo your most recent changes to the current line, use the REST command.

F2 REST Will undo any changes made to a line of text as long as you have not left the line. The line is restored to its previous contents regardless of the changes made.

F3 HDW Allows the use of the handwheel to move to a position and have that position inserted in the program by pressing the ENTER key.



F1 - F2 (X/Z) Choice of axis position to insert

F7 Does an End of Block (line feed carriage return) at the cursor position.

F6 LNDEL Deletes the line containing the cursor and moves any lines below it up one line. The cursor moves to column 1 of the next line.

F7 CHNG This operation works the same as the Find command except that you can replace the found string with any other string of up to 67 characters. After entering the search string, you are asked to enter the replacement string. The last replacement string entered, if any, will be displayed; you may accept it, edit it, or enter a new string. Finally you are prompted for options. The options you used last are displayed at first. You may enter new options (cancelling the old ones), edit the current options, or select them by pressing ENTER. The options available are the same as those for the Find command.

F8 FIND Lets you search for a string of up to 67 characters. When you enter this command you will be asked for a search string. The last search string entered (if any) will be displayed. You can select it again by pressing ENTER, edit it, or enter a new search string. After the search string is entered you must specify your search options. The options you used last, if any, are displayed. You can enter new options (cancelling the old ones), edit the current options, or select them again by pressing ENTER. The following options are available:

B Searches backwards from the current cursor position toward the beginning of the program.

G Searches globally. The entire program is scanned for the search string regardless of the current position of the cursor. The search starts at the beginning of the program if searching forwards, at the end if searching backwards.

L Limits searches to the currently marked block.

U Ignores case; treats all alphabetic characters as if they were uppercase.

W Searches for whole words only; skips matching patterns embedded in other words.

If the text contains a target matching the search string, the cursor is positioned just beyond it.

Note: An additional option is available for the change operation.

N: Without the "N" option, before changing each string, a prompt "Replace Y/N/A/Q" will allow the operator to

Y: YES - change it

N: NO - don't change it

A: REPLACE ALL without prompting (used with "G" global option)

Q: QUIT - stop (used with "G" global option)

ESC: Does the same thing as quit

With the "N" option, NO prompt is displayed before it changes the string or strings.

F9 FNEXT Repeats the last search operation. If the last search command called for a Find operation, the same search string and options will be repeated; for a Find-and-replace operation, the replacement string will be reused as well.

5.9.1.1.5 F5 INS (MAIN-PROG-TEXT-EDIT-INS)

If the F5 INS key is on, the editor is in Insert mode and characters will be inserted at the cursor position. If the INS key is off, the editor is in Overwrite mode and characters will overwrite any previous character at the cursor position.

5.9.1.1.6 F6 DEL (MAIN-PROG-TEXT-EDIT-DEL)

The F6 DEL key, deletes the character under the cursor and moves any characters to the right of the cursor one position to the left. This command does not work across line breaks.

Note: At any time while in the editor, pressing the ← (backspace) key on the numeric keypad will move the cursor one character to the left and delete the character positioned there. Any characters to the right of the cursor are moved one space to the left.

F7 ↑Moves cursor up one line.

F8 ↓Moves the cursor down one line.

F9 ← Moves the cursor to the left.

F10 → Moves the cursor to the right.

ESC EXIT Upon pressing the 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 "Y" will accept the changes and alter the program file. Pressing "N" will abort the changes and leave the file unchanged. ESC will return to the editor.

5.9.1.2 F2 NEW (MAIN-PROG-TEXT-NEW)

The F2 NEW key will allow entry (in the message box) of a number for a new text program. After the number has been entered, the control will check the text programs to see if a text program by that number is already there. If it is found, a warning will be displayed and the operator will be allowed to OK the erasing of the text program prior to entering the text editor.

5.9.1.3 F3 OLD (MAIN-PROG-TEXT-OLD)

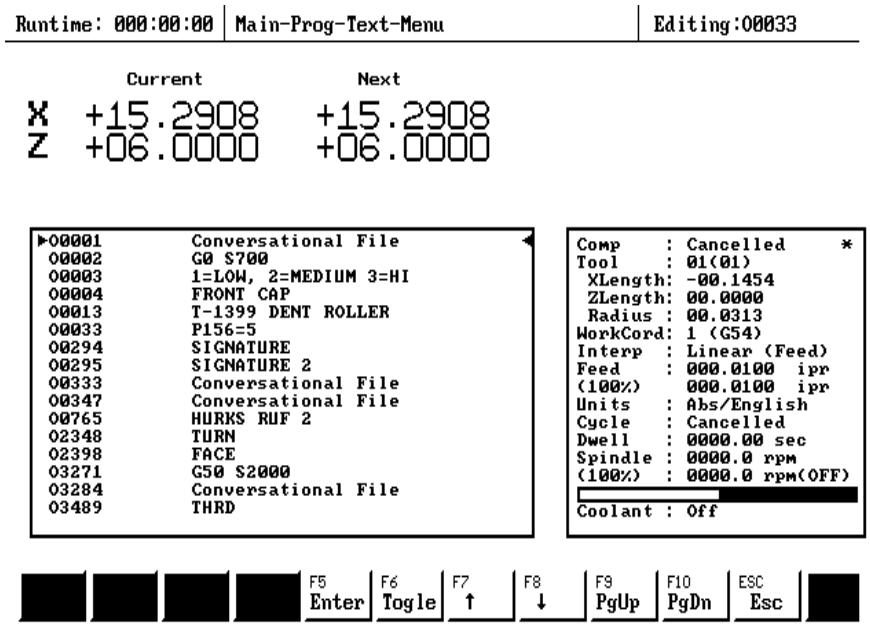
The F3 OLD key will allow entry, in the message window, of the number of an existing text program. After the number has been entered, the control will check the text programs to see if a program by that number is there. If it is, the Edit Window will appear displaying the first 16 lines of the program. If not, an error stating that the program was not found will appear and, after pressing the ESC key, another number may be entered.

5.9.1.4 F4 ANY (MAIN-PROG-TEST-ANY)

The F4 ANY key will allow entry in the message window of any file. Specify the drive, path, file name and extension.

5.9.1.5 F7 MENU (MAIN-PROG-TEXT-MENU)

The F7 MENU key will display a list of all text programs. By using the F7 - F10 keys the file selection arrows are positioned at the program to edit, and the F5 ENTER key is pressed to make a selection.



The edit window will appear displaying the first 16 programs.

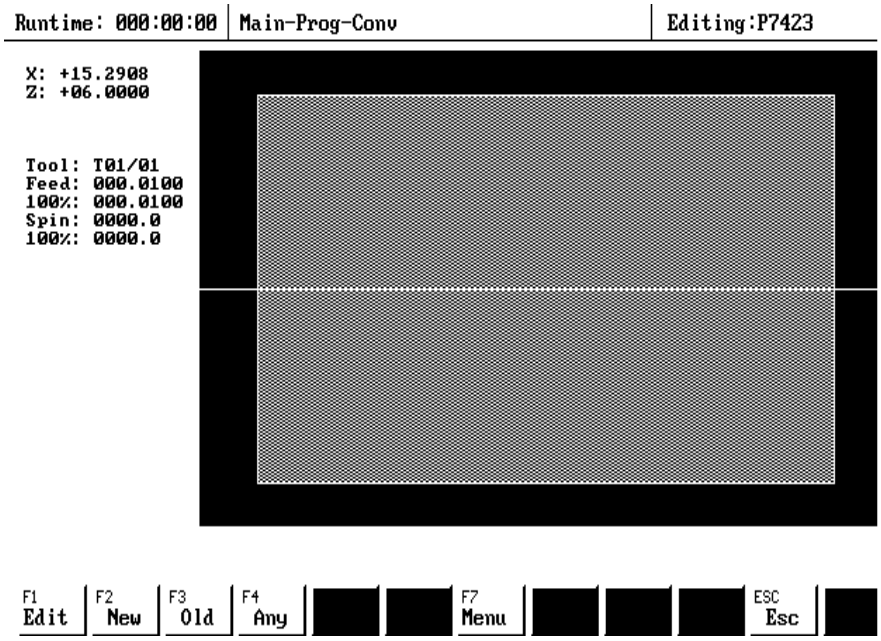
Regardless which selection mode (EDIT, NEW, OLD, ANY or MENU) is used, whenever the edit window is displayed the number of the text program being edited will be shown in the active window.

F6 TOGLE will display the file size, last time and date it was edited, and 7 characters for the file name.

Note: The file name is the characters included in parentheses. If there are no parentheses on the first block, the name is the first block.

5.9.2 F2 CONV (MAIN-PROG-CONV)

This next section will deal with selecting conversational programs. Upon entering the conversational programming mode, the active window in the upper right-hand corner will switch to show the last conversational program edited.

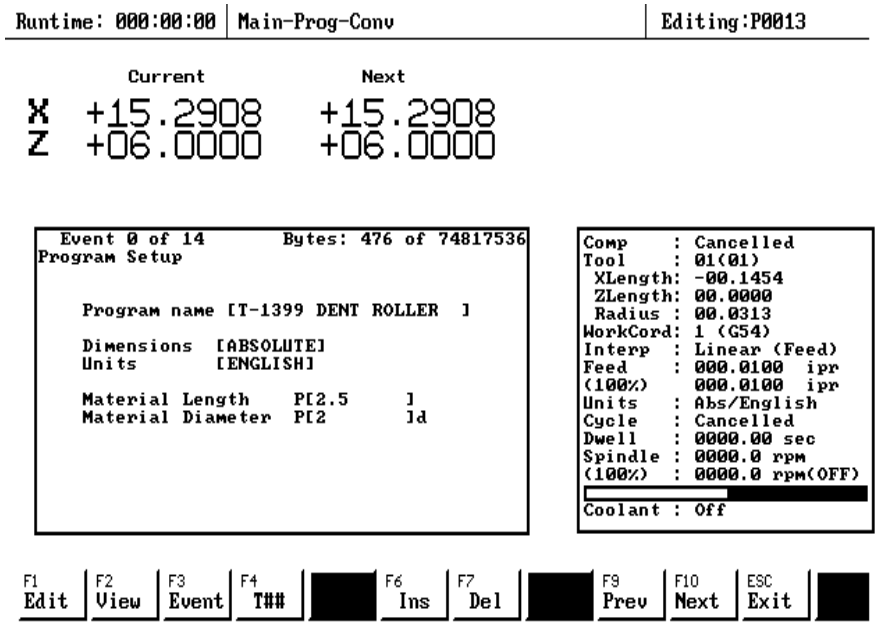


Five options are available with which to choose the conversational program to edit.

5.9.2.1 F1 EDIT (MAIN-PROG-CONV-EDIT)

Pressing the F1 EDIT key will select the active conversational program and enter the conversational programming system.

5.9.2.1.1 EDIT KEYS

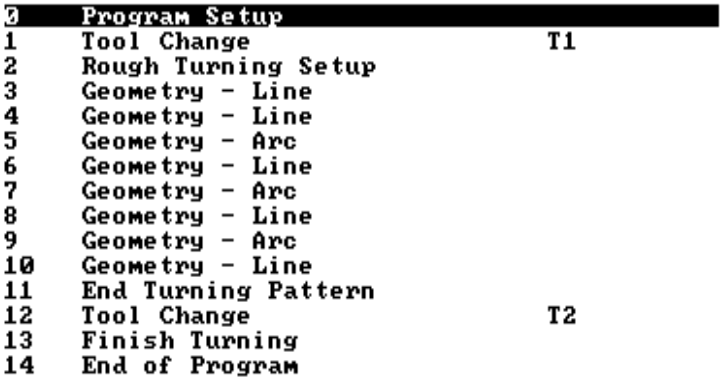


These soft keys will be available whenever the edit menu system has not been entered. At this time it is possible to step through the program, edit events, and insert or delete events.

Definitions of the Edit keys are as follows:

F1 EDIT Pressing the EDIT key will position the cursor at the first field of the current event. The Store/Input keys will appear and the event may be edited.

F2 VIEW This selection allows viewing of the entire program and lets the operator position to any of the events in the program. A window like this will be displayed.



F7 ↑, F8 ↓, F9 PGUP, and F10 PGDN can be used to move to the desired event. Then push ESC or ENTER to display that event.

F3 EVENT This selection allows a search for an event number. If the event number is not found, the -END OF PROGRAM- event will be displayed.

F4 T## This selection allows a search for a tool number. If the tool number is not found the event displayed will not change.

F6 INS Inserts 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 in the menu subsystem. Inserting is not allowed in front of the first event (Program Set Up).

F7 DEL Deletes the event currently being displayed. The first event (Program Set Up) and last event (End of Program) cannot be deleted.

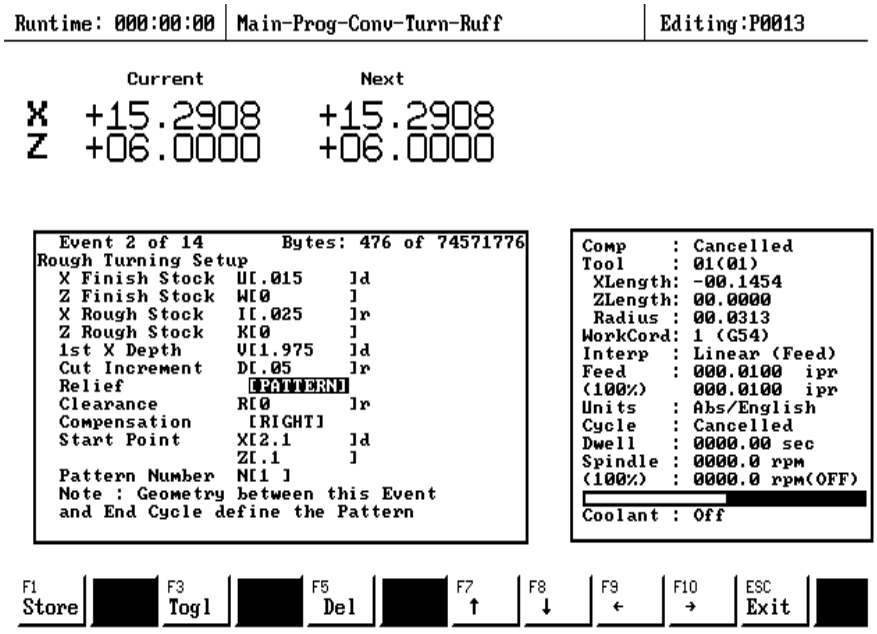
F9 PREV Displays the previous event in the program file.

F10 NEXT Displays the next event in the program file.

ESC EXIT Exits the conversational system and automatically creates the executable text program with an "O" prefix.

While programming or editing in the conversational system, three types of soft key configurations will be encountered. They are:

5.9.2.2 STORE/INPUT KEYS



These soft keys will be available whenever input is expected. At this time, a screen containing any number of fields will be displayed and the cursor will be positioned in one of the fields.

There are two types of fields which may be on an input screen, data and toggle. Data fields are fields in which data is entered using the keyboard and some fields may be left blank. Toggle fields are fields that have a limited number of possible input values and whose value may only be changed by pressing the TOGL key.

Definitions of the Store/Input keys are as follows:

F1 STORE Accepts the entries and adds to the program file. If all required data has not been entered, the STORE key will not work and the cursor will position to the field requiring input. Each screen stored is called an event.

F3 TOGL Pressing this key will result in the next toggle value being displayed in the field. It will have no effect in a data field.

F5 DEL Used to delete an entry from a data field. It will have no effect in a toggle field.

F7 ↑Moves cursor to the previous field.

F8 ↓Moves cursor to the next field.

F9 ← Moves cursor to the left. Has no effect in a toggle field.

F10 → Moves cursor to the right. Has no effect in a toggle field.

ESC EXIT Will abort input and return to the menu keys. Any changes made to the event are ignored.

5.9.2.3 F2 NEW (MAIN-PROG-CONV-NEW)

Pressing the NEW key will allow entry, in the message box, of a number for a new conversational program. After the number has been entered, the control will check the conversational programs to see if a program by that number is already there. If it is found, two warnings will be displayed, and the operator will be allowed to "OK" the erasing of the conversational program and its associated text program prior to entering the conversational system. Remember that conversational programs are stored in both formats.

5.9.2.4 F3 OLD (MAIN-PROG-CONV-OLD)

The OLD key will allow entry, in the message window, of the number of an existing conversational program. After the number has been entered, the control will check the conversational programs to see if a program by that number is there. If it is, the edit window will appear displaying the program setup screen. If not, an error stating that the program was not found will appear and, after pressing the ESC key, another number may be entered.

5.9.2.5 F4 ANY (MAIN-PROG-CONV-ANY)

The ANY key will allow entry, in the message window, of any file. Specify the drive, path, file name and extension. If the name is of the correct format for a conversational program, you will enter the conversation program mode. If not you will enter the text programming mode.

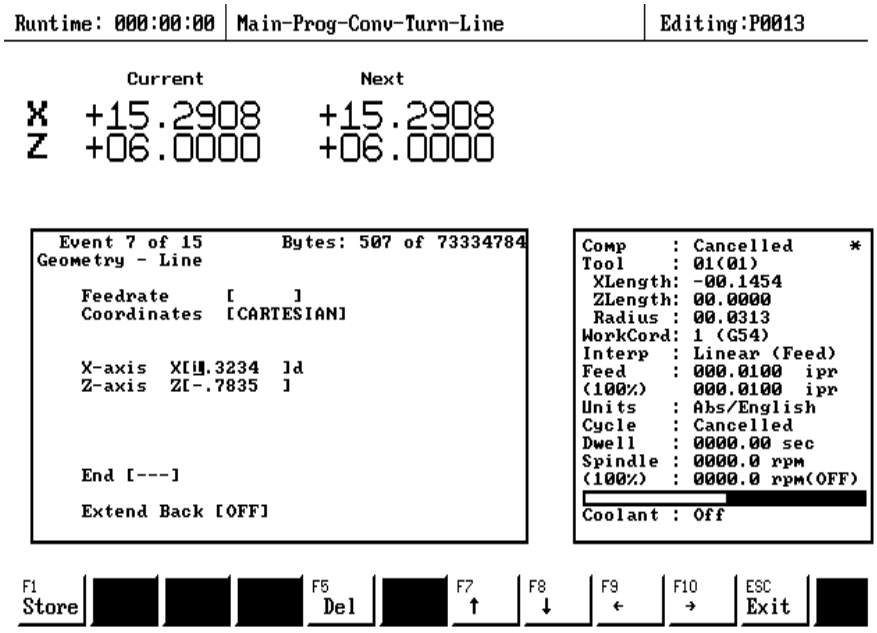
5.9.2.6 MACHINE OPERATION KEYS (MAIN-PROG-CONV)

The menu keys are used to move throughout the conversational system and reach the desired input screen. The menu key sequence is listed on the Conversational System Flow chart shown below.

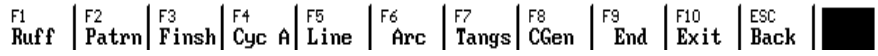
This is what the first level menu keys look like:



F1 will either bring up an input screen (e.g. F1-POS) like this:

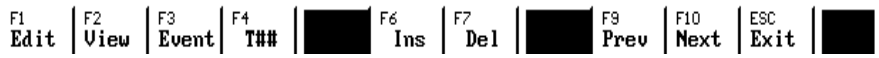


or another menu (e.g. F2 TURN) like this:



Notice that on all levels except level 1 there is an ESC-BACK key. This key will return you to the previous level menu keys.

Pressing the F10 EXIT key will exit the menu subsystem and display the level 1 EDIT keys.



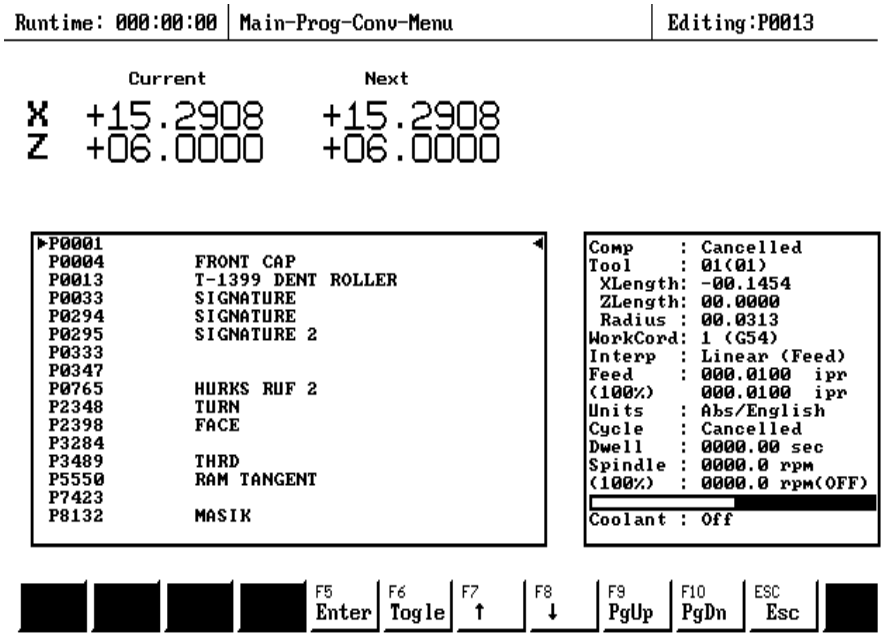
See SECTION 7 for more detailed information on the Conversational Input screens.

5.9.3 F4 EDIT (MAIN-PROG-EDIT)

F4 EDIT from this screen will go to the TEXT EDIT mode or Conversational mode and load the last program that was edited. This is a shortcut for those who normally use one or the other without switching between the two modes.

5.9.3.1 F7 MENU (MAIN-PROG-CONV-MENU)

The MENU key will display a list of all conversational programs.



By using the F7 - F10 keys the file selection arrows are positioned at the program to edit, and the F5 ENTER key is pressed to make the selection. The edit window will appear displaying the program setup screen.

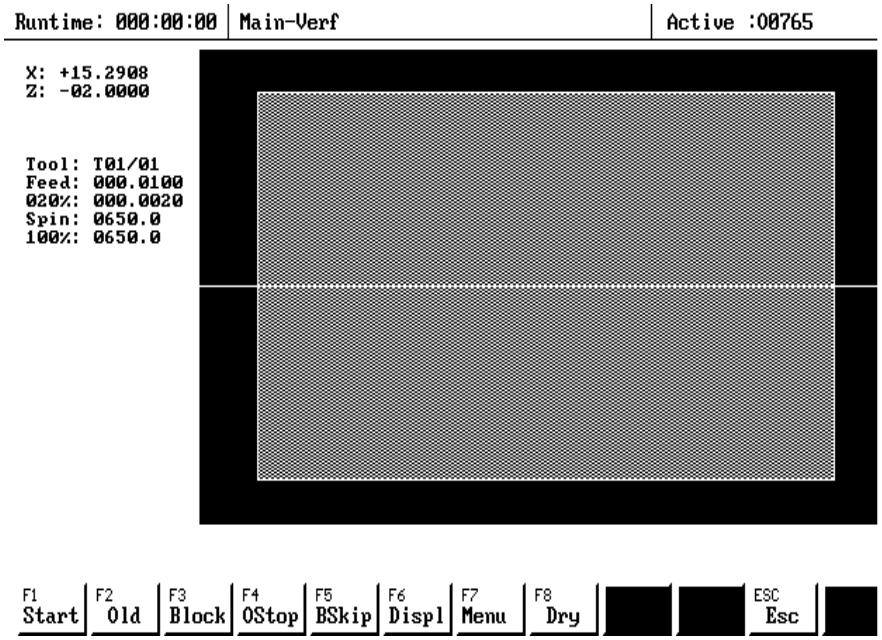
Regardless of which selection mode (EDIT, NEW, OLD, ANY OR MENU) is used, whenever the edit window is displayed the number of the conversation program being worked with will be shown in the editing window.

F6 TOGLE displays the file size, last time and date it was edited, and the first seven characters of the file name.

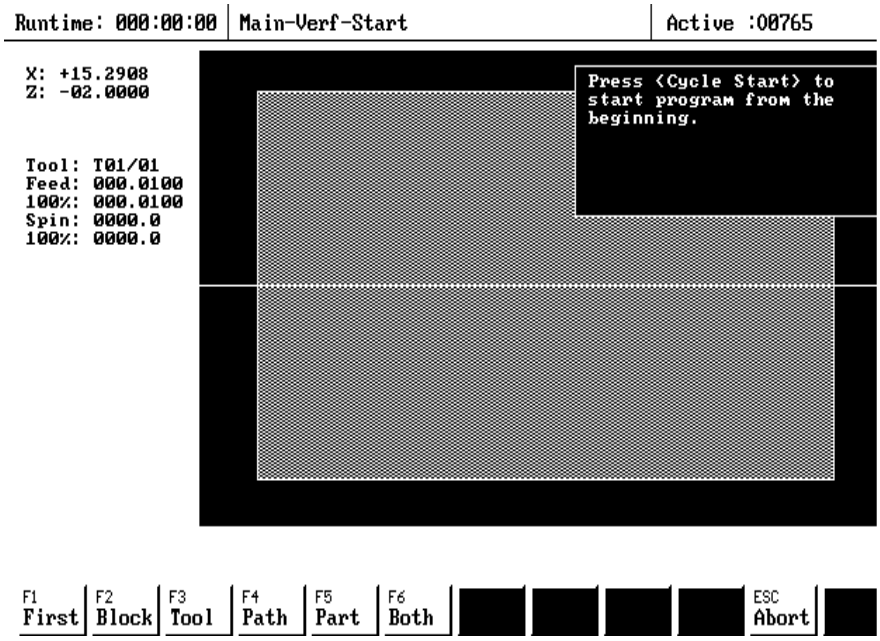
5.10 Verify F9 VERF (MAIN-VERF)

The Verify function is used to verify part programs. Most verification is done in the graphics mode, but it doesn't have to be. The program which 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 in the graphic mode will step around the part and the X and Z display will read out the coordinate values of each point.

Verify is used to verify the active program. Upon pushing the VERF button the following screen appears:



After the above screen appears, F1 START would be pushed and the following screen would appear:



The F1 FIRST key is automatically selected when entering this screen from the VERF screen. Therefore if it is desired to verify the active program from the beginning, all that is necessary is to push the CYCLE START button. If F2 BLOCK is pushed, the control will request that the desired block or sequence number be typed in, followed by ENTER. If the Cycle Start button is pushed the active program will start verifying from the selected block number. If F3 TOOL is pushed, the control will request a tool number. After typing the tool number followed by an ENTER, the CYCLE START button is pushed, and the active program will start verifying at the desired tool number.

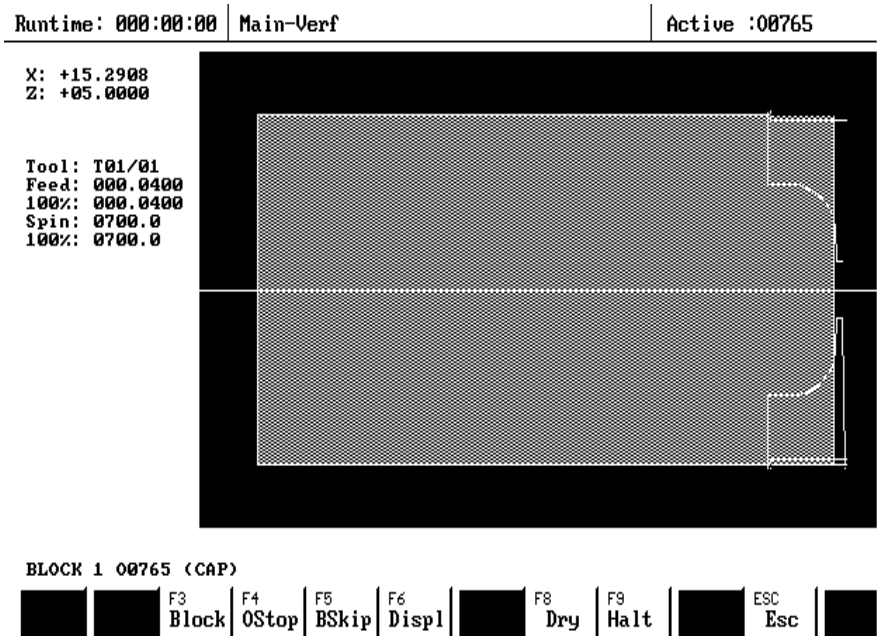
Note: If the block # or tool # typed in is not located within the program, the program will start from the beginning.

The F4 PATH key will show the tool path on the Graphics Screen, which is the default. The F5 PART will show the PART PATH on the graphics screen. The F6 key will show both the PART PATH and then the TOOL PATH on the graphics screen.

Feed moves in the PART PATH are represented by yellow lines. Rapid moves in the PART PATH are not shown. If both paths are displayed, the roughing cuts of rough cycles are not shown and only the finish pattern is shown.

In the TOOL PATH feed moves are shown as green lines, Rapid moves are shown as red lines and Finish Pattern of Roughing Cycles are shown as yellow lines.

After the CYCLE START is pressed the following screen appears:



This screen is the basic verify screen with two new additions, a block number display and F9-HALT. The block number shows the current line being executed as the program verifies. The HALT key is similar to FEEDHOLD in that when pushed the program will stop. However, unlike FEEDHOLD the HALT also allows a new program to be started.

Note: The feedrate overrides and spindle overrides are active when verifying a program.

5.10.1 F3 BLOCK (MAIN-VERF)

When the BLOCK switch is activated the program will stop at the end of each block. Each time the CYCLE START button is pushed one more block will be verified.

5.10.2 F4 OSTOP (MAIN-VERF)

When the Optional Stop Switch (OSTOP) is activated the program will stop at each M01 command. When the CYCLE START button is pushed the program continue to run.

5.10.3 F5 BSKIP (MAIN-VERF)

When the Block Skip Switch (BSKIP) is activated the program will skip all blocks started with a "/".

```
     /F25 When the block skip switch is active the feedrate
          change command will not be executed.
```

5.10.4 F6 DISPL (MAIN-VERF-DISPL)

See explanation under (MAIN-RUN-DISPL). This function can be entered from several screens.

5.10.5 F8 DRY

F8 DRY run in the verify mode will RUN the program as fast as possible.

5.10.6 F9 HALT (MAIN-VERF)

Once a program has been "Halted" the "Resume" feature of the control becomes active. The F9 RESUM key will now be displayed on the Verify screen. If the "Resume" function is selected the active program will be resumed at the halted point. If the Verify screen is exited in the halted mode, then verifying is terminated.

Note: Once the "Run" or "Verify" modes have been entered, a "Halt" must be executed if the mode is to be exited prior to program completion.

The F3, F4, F5 and F8 keys on the VERIFY screen set the mode of operation a program will run in. When these keys are in a highlighted state the functions will be active in any currently running program or program to be run.

5.11 Utilities F10 UTIL (MAIN-UTIL)

When F10 UTIL is pressed the following options are available:



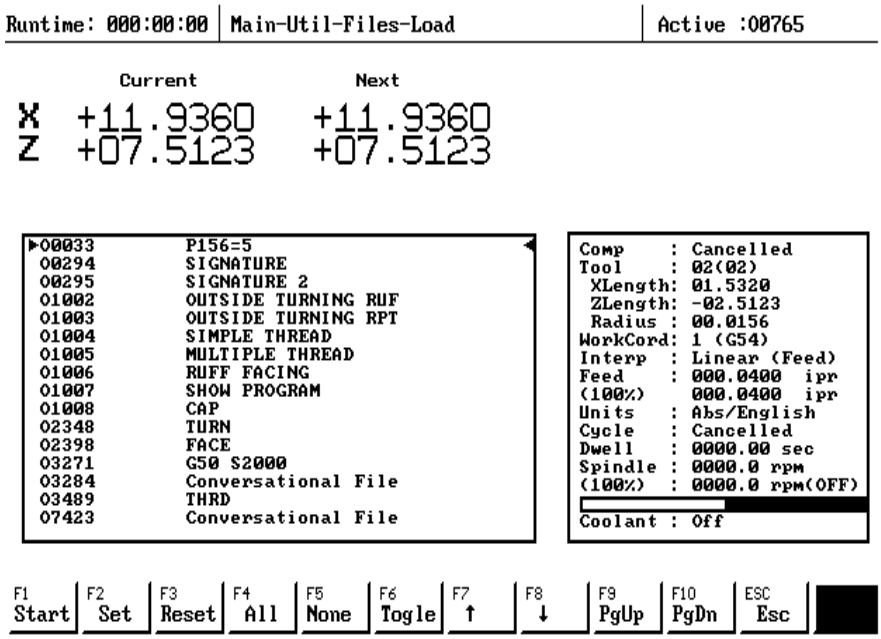
5.11.1 F3 FILES (MAIN-UTIL-FILES)

The Files Utilities contain basic program manipulation functions. They are:



5.11.1.1 F1 LOAD (MAIN-UTIL-FILES-LOAD)

This function is used to load programs from the floppy disk into the control's program memory. When this function is selected the following screen is displayed:



The edit window will display a list of the programs on the floppy drive. The selection cursor (> <) is positioned at the first program.

F1 START Pressing this key will begin the transfer of the selected programs from floppy disk to program memory. The operator will be prompted if the file already exists.

F2 SET Selects the file at the cursor position to be loaded from floppy disk. It will be highlighted.

F3 RESET Unselects the file at the cursor position. Unhighlights highlighted files.

F4 ALL Selects all programs on floppy disk to be loaded. Highlights all files.

F5 NONE Unselects all selected programs. Unhighlights all highlighted files.

F6 TOGLE Shows size and date of the file.

F7 ↑Moves selection cursor up one line.

F8 ↓Moves selection cursor down one line.

F9 PGUP Moves selection cursor up 16 lines.

F10 PGDN Moves selection cursor down 16 lines.

Note: If the parameter to extract files is set, a single file on the floppy, separated by 0####, can be loaded and separated into several files. No checking for files that already exist is done when extracting files.

5.11.1.2 F2 SAVE (MAIN-UTIL-FILES-SAVE)

This function is used to save programs from the control's program memory to the floppy disk.

The SAVE function operates the same as the LOAD function with the exception that the transfer direction is changed.

5.11.1.3 F3 NAME (MAIN-UTIL-FILES-NAME)

This function changes the name of a program. The operator will be prompted if the new file name already exits.

5.11.1.4 F4 COPY (MAIN-UTIL-FILES-COPY)

This function makes a copy of a program under another name. The operator will be prompted if the new file already exists.

5.11.1.5 F5 ▸RAM (MAIN-UTIL-FILES-RAM)
(For controls with extended memory)

This function is used to load programs from a floppy disk to the RAM drive and later DNC'ed from the RAM drive.

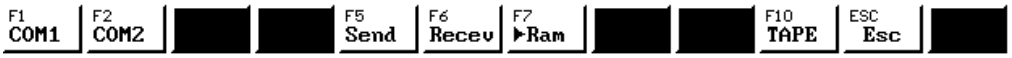
Note: Bigger RAM drives are available as an option.

5.11.1.6 F9 ERASE (MAIN-UTIL-FILES-ERASE)

This function is used to erase programs from the control's program memory. The keys available are the same as those in the SAVE option.

5.11.2 F4 RS232 (MAIN-UTIL-RS232)

The RS-232 utilities contain basic communications functions. They are:



5.11.2.1 F1 COM1 OR COM2

F1 is the primary port. It can be COM1 or COM2 (selectable via RS-232 parameters).

5.11.2.2 F2 COM1 OR COM2

F2 is the secondary port. It can be COM1 or COM2 (selectable via RS-232 parameters). If the secondary port is set to *None* it will not be an option on this screen. If F2 is selected F10-TAPE is also available. This is used for a paper tape reader and start and stop characters will be sent when receiving files from paper tape.

5.11.2.3 F5 SEND (MAIN-UTIL-RS232-SEND)

SEND brings up these keys:



Select what you want to send to the off-line computer.

5.11.2.3.1 F1 TEXT (MAIN-UTIL-RS232-SEND-TEXT)

The SEND option is used to send programs or parameters from the control's program memory to an off-line computer. The soft keys for this function are simply:



F1 BEGIN Starts transmission of the active send program that is shown in the upper right-hand corner.

F7 MENU Allows selection of multiple programs from a menu to send.

Note: If sending parameters, F7 MENU is not an option.

Note: The RS-232 parameters must be set to the same values on both the control and the computer. Always check these parameter settings.

The RS-232 parameters are found in the control parameters (MAIN-PARMS-CTRL). Normal parameter settings are:

Primary Port	*COM1*
Baud rate	*1200 or 9600*
Parity	*Even*
Data bits	*7*
Stop bits	*2*

5.11.2.4 F6 RECEV (MAIN-UTIL-RS232-RECEV)

The F6 RECEV option is used to receive programs or parameters from an off-line computer into the control program memory. Upon pressing the RECEV key (selections are available below), a new program number must be entered in the message window. Once a valid program number has been entered, the edit screen will appear and display the program as it is received. Receiving will continue until the ESC key is pressed.



Select what you want to receive from the off-line computer.

Note: If the program # being received already exists, the operator will be prompted.

If the parameter to extract files is set, several files can be sent from an off-line computer. If they are separated by 0####'s the control will extract them to the correct #### file. No checking is done for files that exist when extracting files being received.

5.11.3 F6 DNC (MAIN-UTIL-DNC)

The DNC mode is used for running large programs. These programs are not loaded into the control's memory, so they cannot have while-wend loops, gosub's or calls.

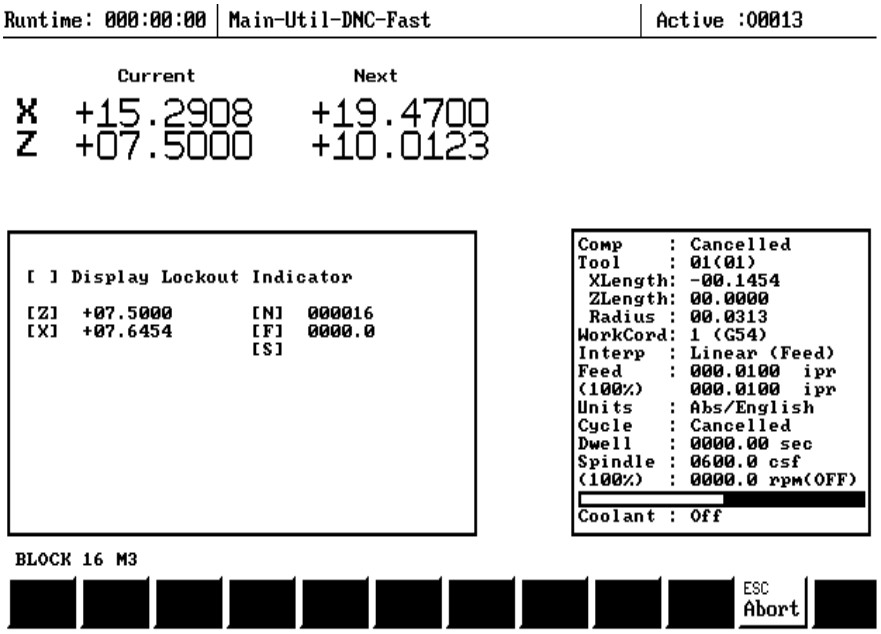
Note: Goto's are allowed if the parameter is set to resolve DNC files. Goto's are never allowed in DNCing from RS-232.

When F6 DNC is pressed, the key shows the DNC modes. They are:



5.11.3.1 F3 FAST (MAIN-UTIL-DNC-FAST)

When DNC FAST is running the display is as follows:



When block time is very short the display lockout indicator will show an (*), and positions, etc. will not be updated. Non-DNC blocks will update the normal display information. When ESC is pressed, the display will show the total DNC time, total distance, average feedrate, average block time, and average blocks/second.

This mode should be used for large programs where a fast block rate is required, e.g. short moves at fast feedrates. No trig help, cutter comp, rotating, scaling, etc. can be done in this mode. Valid data for the fast mode are:

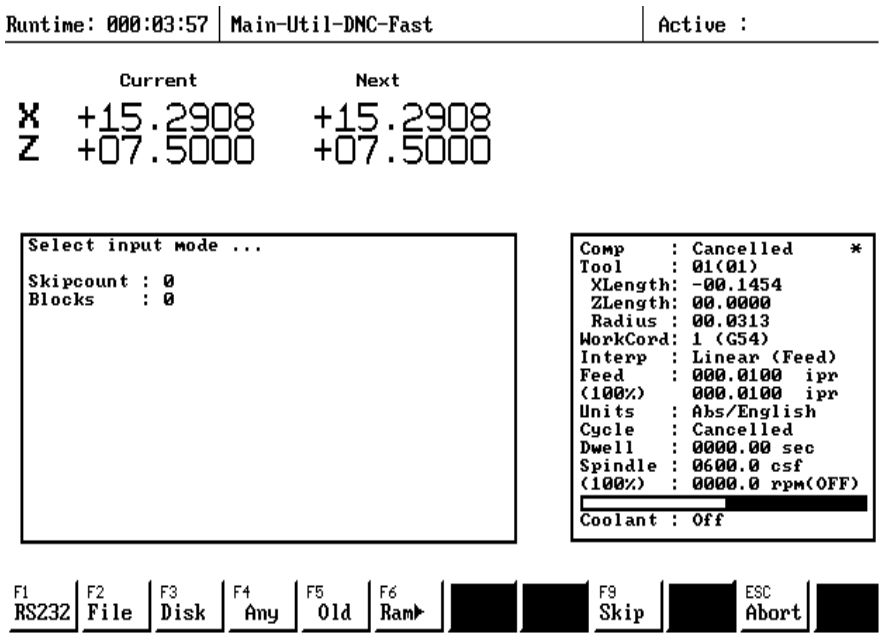
X, Z, I, K, F and N

Valid G codes are:

G0, G1, G2, G3, G90, G93

If a code that is not in these groups is executed, the block time for that block will slow down.

After pushing F3 FAST the control will ask for a source to DNC from. The screen will be:



5.11.3.1.1 F1 RS-232

The control waits for data on the RS-232 then requests a cycle start to begin the program.

5.11.3.1.2 F2 FILE

The control selects a program to DNC from a menu of the programs in the control.

5.11.3.1.3 F3 DISK

The control selects a program to DNC from a menu of the programs on the floppy disk.

5.11.3.1.4 F4 ANY

This key will allows a selected program to DNC from any drive by typing in the name of the drive, path, file and extension. This is useful for machines on a local area network.

5.11.3.1.5 F5 OLD

Type in the number of the file you want to DNC.

5.11.3.1.6 F6 RAM▸

If a program has been loaded to the RAM drive (via RS-232 or floppy disk), this will select that program.

After selecting the INPUT mode the control will request a cycle start to begin the program. All of the above input modes except RS-232 will allow the program to start at locations other than the beginning. The other options are:

F1 FIRST	Start from the beginning
F2 BLOCK	Start from a sequence number
F3 TOOL	Start from a tool number
F4 CONT	This will allow starting from a location where DNC was aborted earlier. When a file is aborted the file position is saved. No checking is done for switching file numbers, edited files, new file names, etc. CONT will continue from the file position that was aborted on any file.

5.11.3.2 F9 SKIP

SKIP will ask for a skip count. If the skip count is non-zero, a Cycle Start will be requested for skipping (or ignoring) that number of blocks. Then a second Cycle Start will run from the next block.

5.11.3.3 F4 RUN (MAIN-UTIL-DNC-RUN)

F4 RUN allows the same selections as F3 FAST for input modes. Skipping works the same. After the input mode has been selected and any blocks have been skipped, the control moves the run screen (MAIN-RUN) and the DNC program can be run similar to any other program.

5.11.3.4 F9 VERF (MAIN-UTIL-DNC-VERF)

F9 VERF works identical to the F4 RUN mode. After the input mode has been selected and blocks have been skipped, the control moves to the verify screen (MAIN-VERF) and the DNC program can be verified similar to any other program except the option for part/path/both is not available.

Note: For F4 RUN and F9 VERF in RS-232 DNC mode, a search for a tool number or block number can be done to start a program from a desired location, but cannot be reset to first after this search.

5.11.4 F7 CHART (MAIN-UTIL-CHART)

The F7 CHART key will display help charts created by the end user specific to their applications. If there is a file called charts.dat in the RAM directory, it will be displayed. The format of this file allows an indexing system to other files and data available to the operator.

Here is a sample listing of charts.dat:

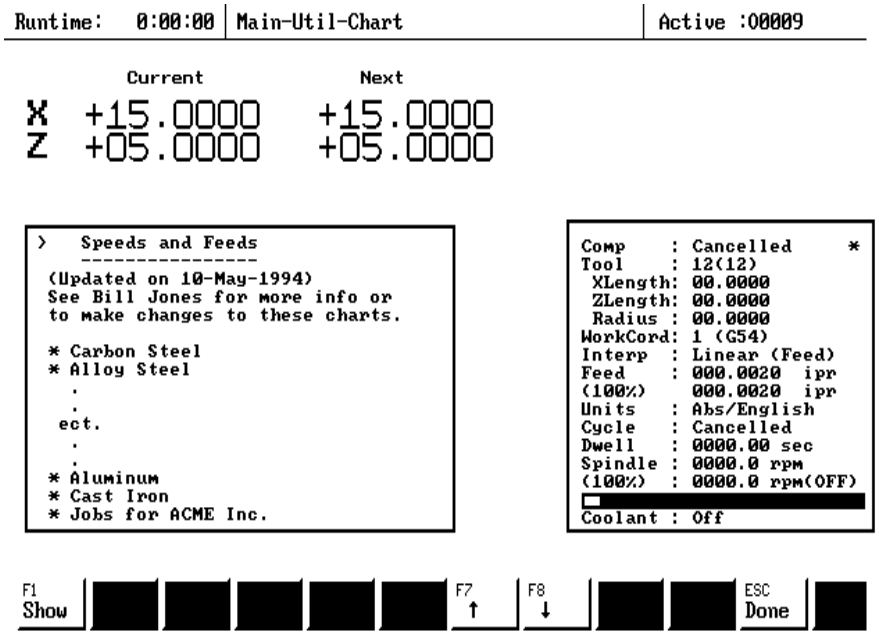
Note: Only the first 16 lines of chart.dat are displayed.

Speeds and Feeds

(Updated on 10-May-94)
See Bill Jones for more information
or to make changes to these charts.

```
CarStl.dat  * Carbon Steel
AllSt.dat   * Alloy Steel
                .
                .
              etc.
                .
                .
Allum.dat    * Aluminum
CstIron.dat  * Cast Iron
Acme.dat     * Jobs for ACME Inc.
```

The first 12 characters from charts.dat are not displayed. They define other file names to index to. After pushing F7 CHART, with the above file in the RAM directory, the screen would look like this:



Move the cursor to the desired field using the arrow keys and press F1 SHOW or ENTER to display the file associated with that field. The operator can then view the file using F9 (Page Up) or F10 (Page Down). Pushing F1 OK or ESCAPE returns to the charts.dat file.

Note: The same charts feature is available for use in the Help Key in conversational programming.

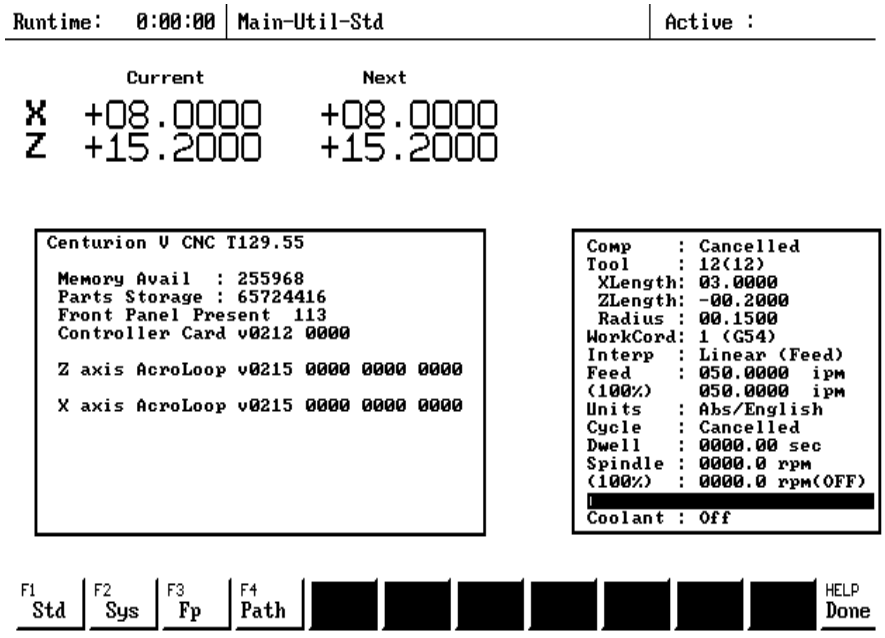
5.11.5 F8 INFO (MAIN-UTIL-INFO)

F8 INFO shows information about the software, hardware, etc. The following function key will be displayed.



5.11.5.1 F1 STD (MAIN-UTIL-INFO-STD)

F1 STD shows standard information.



Memory available gives the system memory available in bytes.

Parts storage gives the amount of available parts storage in bytes.

Front panel present gives the front panel version code.

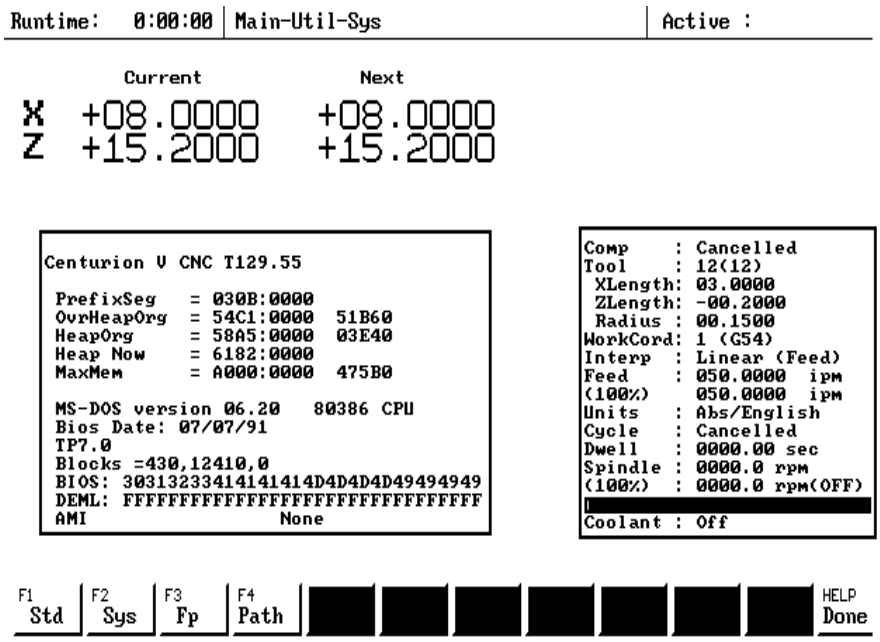
Controller card gives the controller card version (v0206 is 2.06) and an error count (should be zero).

Acroloop information for the X and Z axes is as follows:

First the version, (v0214=2.14),
Then an error count (should be zero),
Then four bytes giving optional Acroloop programs loaded (the .HEX files).
These last four bytes are formatted as:
aamm ssrr, where:
aa = axis (01=Z, 02=X, etc.)
mm = major version
ss = sub version; uniquely identifies .HEX file.
rr = reserved (always zero)

5.11.5.2 F2 SYS (MAIN-UTIL-INFO-SYS)

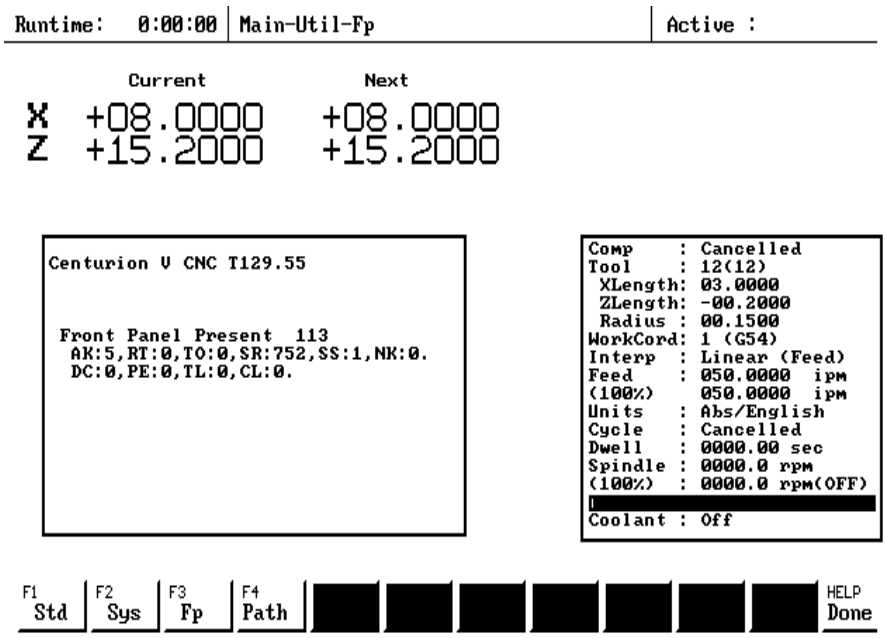
Shows system information.



This screen gives internal information on the system. Lines 1 through 5 show memory allocations to DOS, CNC Overlays, and the heap. Line 6 shows the MS-DOS version and whether the CPU is an 80286, 80386 or 80486. Lines 11 & 12 show the compiler and blocks pre-allocated for canned cycles, text cycles and custom M & G codes. Line 13 is a hex dump of the BIOS ROM Area at F000:0. Line 14 is a hex dump of the Disk Emulator ROM at CA00:0. Line 15 gives a guess at the Bios type and Disk Emulator type.

5.11.5.3 F3 FP (MAIN-UTIL-INFO-FP)

Shows front panel information.



This screen shows some front panel information.

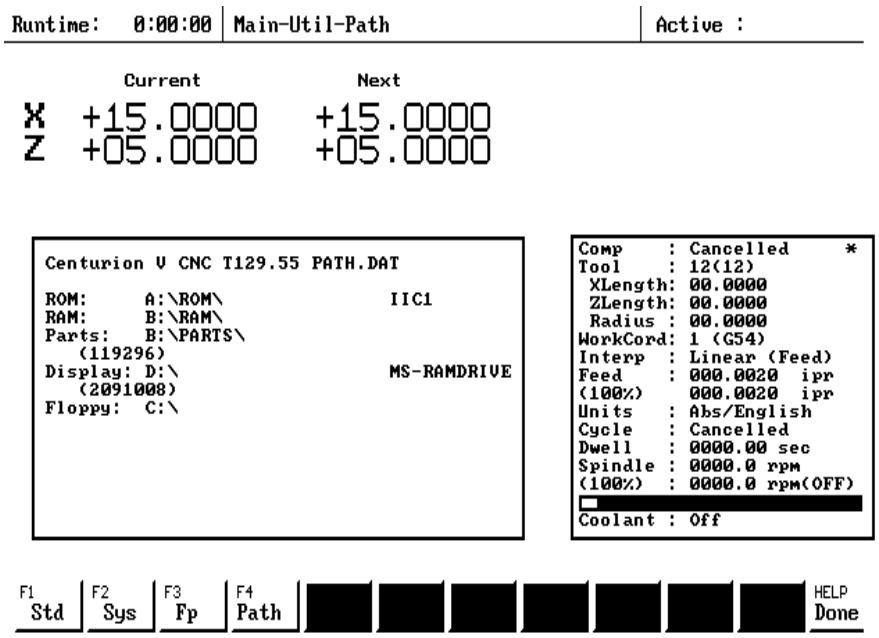
The first line gives the front panel version.

The next two lines show several abbreviations and numbers:

AK=Acks received from keyboard encoder/controller
RT=ReTrans received from keyboard encoder/controller
TO=Timeouts on transmissions to keyboard encoder,
detected by keyboard controller.
SR=Slice resends - timeouts detected by CNC.
SS=Slice sends - standard sends to keyboard encoder.
NK=Nacks detected by keyboard controller.
DC=Spindle DAC value send to keyboard encoder.
PE=Parity Errors detected by Keyboard controller.
TL=Time Limits detected by keyboard controller.
CL=Control pressed received by keyboard controller.
(used to indicate data/command mismatch)

5.11.5.4 F4 PATH (MAIN-UTIL-INFO-PATH)

Displays the paths and the volume label of each.



This screen shows the path file. In the standard order, these are the directories for ROM, RAM, Parts, Display and Floppy. Below the Parts and Display directories is shown the available parts space in bytes. For the ROM, Parts and Display directories, the DOS volume ID for that drive is shown to the right.

NOTE: The path file is reloaded when this information is displayed. Therefore one can edit the path file and reload it by using this command.

5.11.6 F9 BLANK (MAIN-UTIL-BLANK)

F9 BLANK will blank the screen. This will reduce images being burned into the CRT. The screen will remain blanked until any key is pushed.

Block stops, tool changes, request for cycle start, etc. will unblank the screen, prompt the operator then re-blank the screen. There is also a timer to blank the screen if keys are not pushed in a set amount of time (see Misc. Parameter Section). The screen will blank automatically and remain blank until any key is pressed.

6. PARAMETRIC PROGRAMMING

Parametric programming is similar to macro programming in that equations can be used to specify axis position rather than decimal numbers. The Centurion 5 does not restrict the use of parametrics to subroutines or macros. They can be used anywhere throughout a program. Parametric expressions cannot be used to specify M or G codes but can be used to specify axes X Z A etc., and 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. When using parametric expressions in a program the parameters which are used are the 100 "User Parameters" discussed earlier. Values generated by equations can be displayed on "User" parameter screens. Other listed system parameters can be used as input data to parametric equations but under normal circumstances these parameters should not be changed.

6.1 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]
```

6.2 Parametric assignment statement

Assigning is the most basic statement 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 specified by an operand type.

```
Example:    P1 = 1.234
```

In this example the value 1.234 is assigned to parameter 1. Therefore both of the following statements would move to the same coordinate position:

```
X1.234 or X[P1]
```

Notes: 1. If writing to parameters #'s greater than 499, the control needs to be in the data mode (G11).

2. See the Appendix for a list of parameter numbers and their descriptions.

6.3 PB50 Byte Parameter (In this case PB50 is spindle range.)

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. Byte parameter values cannot exceed 255, and even then the value may not make sense. A PB50 value of 4 is meaningless.

Example: PB50=2

Notes: 1. Byte parameters are not written in the verify mode.

2. See the Appendix for a list of parameter numbers and their description.

6.4 Parametric operators

6.4.1 Arithmetic operators

The following table shows the available arithmetic operators:

Operator	Operation
+	addition
-	subtraction
*	multiplication
**	exponent
/	division
DIV	integer division
MOD	remainder

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
```

MOD (modula) remainder

The MOD operator returns the remainder obtained by dividing its two operands.

```
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)
```

6.5 Relational operators

This table 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

6.6 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 that a decimal value is accepted. Each function has one argument which can be specified using any of the four operand types.

The functions supported are:

SIN - returns the sine of the argument
```
SIN [90] = 1
```

COS - returns the cosine of the argument
```
COS [180] = -1
```

TAN - returns the tangent of the argument
```
TAN [135] = -1
```

ATAN - returns the arc tangent of the argument. The argument must be specified in fractional form (e.g. 1/2, 2/1, -5/6, ...).
```
ATAN [1/-1] = 135
```

SQRT - Returns the square root of the argument
```
SQRT [9] = 3
```

ABS - Returns the absolute value of the argument
```
ABS [-15] = 15
```

INT - Returns the integer part of the argument
INT [5.5099] = 5

RND - Returns an integer random number between 0 and its argument -1
RND [25] will return an integer >=0 and ≤24

LN - Returns the natural logarithm of the argument
LN [10] = 2.3026

EXP - Returns the exponential of the argument
EXP [1] = 2.7183

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

6.7 Mathematic expressions

Expressions are made up of arithmetic operators and operands. Any combination of the previously described operands may be used to define an expression. See section on Expressions for detailed information.

Examples: X[SIN[P123]*COS[P124]]
P6=[2.5+[P2/P3]*SQRT[P4]]
Z[[P2DIV3]+[P2MOD3]]

6.8 Conditional statements

The Centurion 5T 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.

6.8.1 The **IF THEN** statement is a way of conditionally executing a block if the results 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 of the program is executed.

Example of General Form

```
IF [any          ] (relational) [any          ] THEN  any action
    [mathematical] (operator  ) [mathematical]         (opt.)
   [expression  ]               [expression  ]
```

```
N20  IF P1 LT P2 THEN GOTO 15
N21
                    or

N22  IF P1 LT P2 GOTO 15
N21
```

The above two statements do the same thing. If the statement is true, N15 is executed; if 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. For example:

`IF P36<5 THEN IF P1<>0 THEN M5`

means if P36 is less than 5 and P1 does not equal 0, shut the spindle off.

Note: The "THEN" portion of the statement is optional.

6.8.2 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 at all.

```
Example:  N20  WHILE [[P2*P3]/COS[P6]] LT P2
          N21  P6 = P6 + 1
          N22  Z[P3]
          N23  X[P6]
          N24  X1 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.

6.9 Transfer statements

These 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.

6.10 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 program 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 1 "N30" in a program, it will transfer to the first N30.

6.11 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       LXXX
        Program   Loop Count
         Number   (Optional)
```

If the L is omitted the called program will be executed once. The call statement is the same as an M98.

Notes: 1. Subprogram call allows nesting 50 levels deep.
2. See G50 in the Gcode section for information on using G50 in subprograms.

6.12 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 would be at the end of the main program.

The GOSUB format is as follows:

```
GOSUB      XXXX      LXXX
          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.

Notes: 1. Subroutine nesting is allowed to 50 levels deep.
2. See G50 in the Gcode section for information on using G50 in subroutines.

6.13 Computational Functions

6.13.1 Tangent Arc [TANA], Tangent Line [TANL] 3 Point Circle Generate [CGEN]

These three functions can be used anywhere throughout a program to solve various intersection problems. These functions receive input data in parameters P90 thru P99, and return the answer in parameters P80 through P89. The answers can then be used in line and circle commands to produce the desired results. The format for these three functions follows.

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
P95 = R2 radius of arc 2
P96 = R3 radius of tangent arc (use in TANA only)

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)

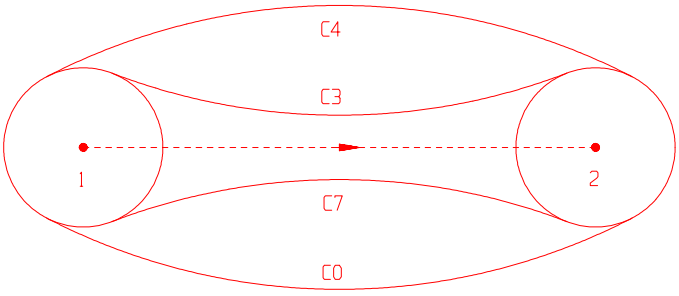
The general form for a tangent arc or line function is

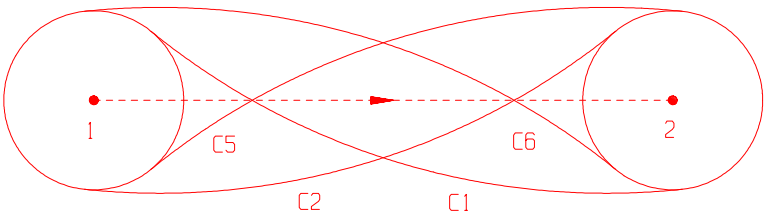
TANA CX or TANL CX

when X 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 CX 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.

TANA CASES

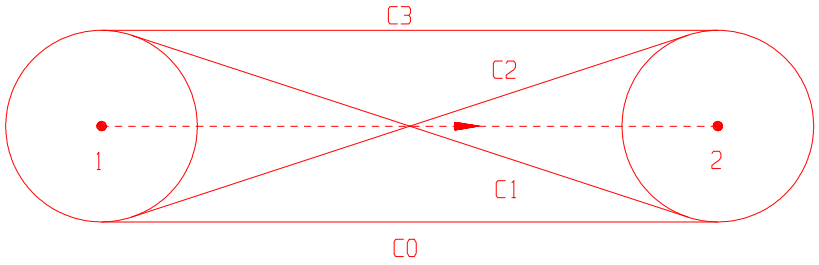
	1st	2nd	Center
C0 =	Right	Right	Left
C1 =	Left	Right	Left
C2 =	Right	Left	Left
C3 =	Left	Left	Left
C4 =	Left	Left	Right
C5 =	Right	Left	Left
C6 =	Left	Right	Right
C7 =	Right	Right	Right





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
N9   G2 R1.5 XC0 ZC0 X[P81] Z[P80]
            (TANA or TANL calculated end points)

N10  G3 R5 XC[P85] ZC[P84] X[P83] 00002]
          (TANA calculated center and end points)
       or

N10  G1 X[P83] Z[P82]
        (TANL calculated end point)

N11  G2 R2 XC4 ZC5 Xend Zend

          depends on next line of program
```

The Circle Generate function will calculate the center and radius of an arc through any three non-colinear 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 Y0       position to start of arc
N6   G2 R[P82] XC[P81] ZC[P80] Xe____Ze____
                                  determined by next line
                                       of program
```

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 in the Centurion 5 control.

Note: The arc direction is returned in P97 so block N6 could be substituted with:

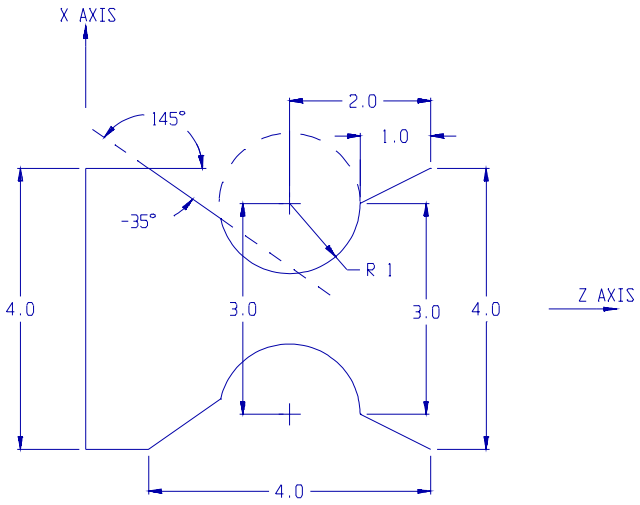
```
G2 IF P97=3 THEN G3
R(P82) XC(P81) ZC(P80) Xe_ Ze_
```

6.13.2 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 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

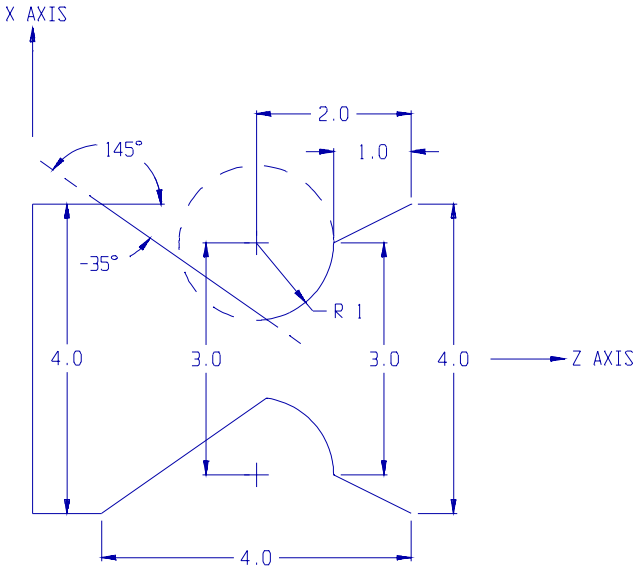


Back extend line backwards from (4,-4)
C0 use the arc intersection closest to (4,-4)
W145 extend the line from (4,-4) at an angle of 145°

Example 2:

(1) X4 Z0
(2) X3 Z1
(3) G2 R1 ZC-2 AB270

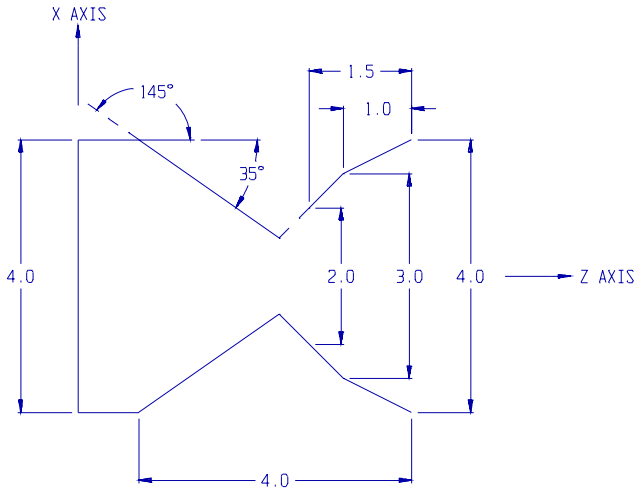
(4) G1 X4 Z-4 BACK C2 W145



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.

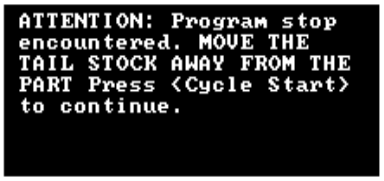
6.13.3 Miscellaneous Commands

Spaces can be used anywhere within the program. For example, *X1.234* can be written as *X 1 . 23 4* if desired.

Blocks without any information are allowed.

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 M6 (tool change), the comment will be displayed in the message window.

Example: M0 (MOVE TAIL STOCK AWAY FROM THE PART)



Example: M6 (3/8" STUB DRILL)



Print is used to print text to the screen. PRINT can be used to display text, parameter values, times, dates, etc.

Example: PRINT [PLEASE CLEAR THE WORK AREA]

#n will show the value of a parameter (n is between 0 and 499).

Example: P87=32.45
PRINT [N=#87] shows "N=32.4500"

#n[LT] will show 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=00.877

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 i)
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=93/12/17"

POPEN command opens a file or RS-232 for output using the DPRNT command.

POPEN P0 command opens a file for output. The file to open is specified in the MISC parameters. If the file name in this parameter is blank, the output will be written to B:\RAM\REPORT.DAT.

POPEN P1 opens serial port 1 for output.

POPEN P2 opens serial port 2 for output.

PCLOS will close a serial port that has been opened using POPEN.

DPRNT outputs text to a file or RS-232 port which is specified by the POPEN command.

```
Example:  DPRNT [PLEASE CLEAR THE WORK AREA]
```

#n writes the value of a parameter (n is between 0 and 499)

```
Example:  PRINT [X#209 Z#208] outputs the current X and
          Z positions to a file or an RS-232 port.
```

#n[LT] will output parameter n with L leading digit 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.

When LT≤0, exceptions apply.

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 )
          PRINT [#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.

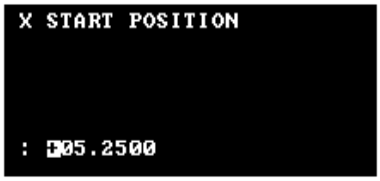
If LT=-2, the parameter is ignored. The date is shown in format yy/mm/dd.

Example: DPRNT [Time=#1[-1] Date=#1[-2]] shows
"Time=012:51:52 Date=93/12/17"

DPRNT can be used to generate report files relating to parts production, parts inspections, digitizing information, etc.

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 pushing the ENTER NEW DATA key. If ESC is pressed during an input statement, the program will be terminated.

The **HDW** command can be used to use the handwheel during a program. A comment may be added to prompt the operator during the HDW command. HDW is ignored when verifying a program or from MDI.

Example: HDW (TOUCH THE TOOL TO PART) prompts the operator with the message.



To manually set and clear input/output pins:

CLR Clear an output pin.
Example: CLR X7 (clear X output #7)

SET Set an output pin.
Example: SET Z3 (sets Z output #3)

PULSE0 Pulses an output pin.
Example: PULSE0 Z10 (clears Z output #10, delays for the number of milliseconds specified by the MISC parameter PULSEx pulse delay (ms) then clears sets output Z10)

PULSE1 Pulses an output pin.
Example: PULSE1 X5 (sets X output #5 delays for the number of milliseconds specified by the MISC parameter PULSEx pulse delay (ms) then clears output X5)

To use pins to make decisions, looping, calculations, etc...

PIN Refers to a pin. Z axis are 1000's, X axis 2000's.

PIN	Z Axis	X Axis
in1	1000	2000
in2	1001	2001
in3	1002	2002
in4	1003	2003
in5	1004	2004
in6	1005	2005
in7	1006	2006
in8	1007	2007
in9	1008	2008
in10	1009	2009
in11	1010	2010
in12	1011	2011
out1	1012	2012
out2	1013	2013
out3	1014	2014
out4	1015	2015
out5	1016	2016
out6	1017	2017
out7	1018	2018
out8	1019	2019
out9	1020	2020
out10	1021	2021
out11	1022	2022
out12	1023	2023

PIN statements can be used in conditional statements such as IF..THEN and WHILE..WEND.

Examples would be: IF PIN[2017] EQ 1 THEN PB50 = 1

```
WHILE PIN[1005] NE 0
M62
M63
WEND
P5=P3 + PIN[2011]
```

6.13.4 Parts Inspection and Digitizing Commands

PROBE1 can be used with any axis move to command an interrupted move. The move will be terminated when Z axis input #11 goes low.

Example: PROBE1 G98 F.01 X1.3 Z-2
The X and Z move toward 1.3 and -2 respectively until the probe contacts the part.

PROBE2 can be used with any axis move to command an interrupted move. The move will be terminated when Z axis input #11 goes high.

Example: PROBE2 G98 F.01 X1.3 Z-2
The X and Z move toward 1.3 and -2 respectively until the probe releases from the part.

SAMPLE PROGRAM USING SOME SPECIAL STATEMENTS

```
(DIGITIZING PROGRAM)
M5
T0101
G00
Z15
M6 (PROBE)

POPEN P0 (open file for output) (SET REPORT.DAT to B:\PARTS\00100)

PB81=7 (set byte parameter special flags to shut off trig help and cutter comp)

INPUT (OUTER DIAMETER)P1 (prompt for data)
INPUT (INNER DIAMETER)P4
INPUT (PROBE INCREMENT)P5

DPRNT [(WHEEL CUTTING)] (write data to file)
DPRNT [M42] (custom code for cutting program to shift to 2nd gear)
DPRNT [PB81=1] (special flags byte parameter to show tool in cutting program)
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
```

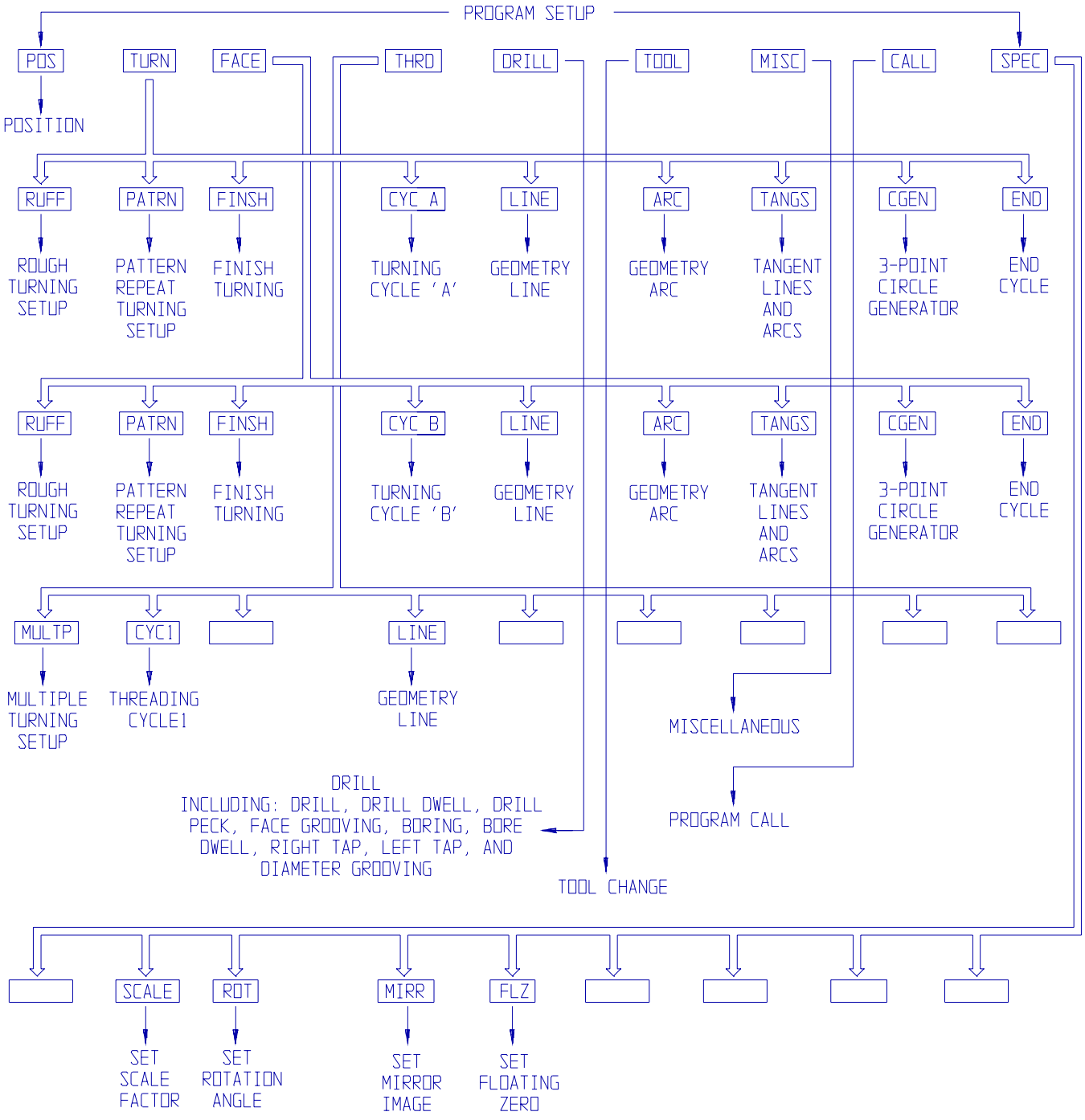
```
DPRNT[G0R.1AB0] (retract for the cutting program)
DPRNT[X6]
DPRNT[Z10.]
DPRNT[M5]
DPRNT[X#1]
```

OUTPUT FROM THE DIGITIZING PROGRAM

```
(WHEEL CUTTING)
M42
PB81=1
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
```

7. CONVERSATIONAL PROGRAMMING SCREENS

FLOW CHART OF CONVERSATIONAL PROGRAMMING SYSTEM



This section contains listings of the conversational programming screens and a small explanation of each screen. Numeric entries can be entered as fractions (i.e; 5/16 instead of .3125). Some entries can be left blank, others are required. A store is not allowed if a required entry is blank.

Note: X dimensions followed by a "d" denote a diameter dimension. X Dimensions followed by an "r" denote a radius dimension.

The program set-up screen always appears at the beginning of every program.

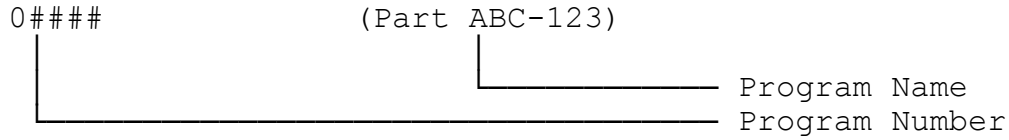
```
PROGRAM SET-UP

1. Program name    [                 ]

2. Dimensions          [ABSOLUTE]
3. Units               [ENGLISH ]

4. Material Length   P[         ]
5. Material Diameter P[         ]d
```

1. Program name: The program name is shown on menus for editing, running etc.
2. Dimension: Absolute or Incremental
3. Units: ENGLISH or METRIC
4. Material Length: This represents the amount of material sticking out of the chuck. It is the offset from Z0 to the face of the chuck. This offset is shown (for work coordinate #1) to the right of the input field (the length is used for graphics only).
5. Material Diameter: The diameter is used for graphics only. For each conversational program a text program (or M and G program) is created. The Program Set-up screen will produce these M and G codes.



```
G20       G93
  |         |
  |         └──────── Absolute or (G91) for Incremental
  └──────────────────── English or (G21) for Metric
```

```
P156 = ##.####
           |
           └──────── Material Length
```

```
P157 = ##.####
           |
           └──────────Material Diameter
```

7.1 POSITION

The Position Screen is normally used for rapid positioning, although feed moves are allowed by toggling the feedrate field.

```
┌──────────────────────────────────────────────────────┐
│    POSITION                                          │
│                                                      │
│    1. Feedrate            [RAPID]                    │
│    2. Coordinates         [CARTESIAN]                │
│                                                      │
│    3. X-axis             X[2        ]d               │
│    4. Z-axis             Z[3        ]                │
│                                                      │
│                                                      │
└──────────────────────────────────────────────────────┘
```

1. Rapid or Feedrate: a feedrate can be entered.
2. Cartesian or Polar: see the G-code section for examples of polar coordinate positioning.
3. X position.
4. Z position.

Text Program Generated for Position Screen

```
G0  X##.####  Z##.####
 |       |        |
 |       |        └──────── Z Position
 |       └──────────────── X Position
 └──────────────────────── Rapid (or G1 F.#### for feed moves)
```

7.2 ROUGH TURNING

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

```
ROUGH TURNING SET-UP

 1. X Finish Stock    U[         ]d
 2. Z Finish Stock    W[         ]
 3. X Rough Stock     I[         ]r
 4. Z Rough Stock     K[         ]
 5. 1st X Depth       V[         ]d
 6. Cut Increment     D[         ]r
 7. Relief             [PATTERN]
 8. Clearance         R[         ]
 9. Compensation       [RIGHT]
10. Start Point       X[         ]d
                      Z[         ]
11. Pattern Number    N[ ]
    Note:  Geometry between this Event
           and End Cycle define the Pattern
```

1. (U) X Finish Stock: Distance and direction of the finish allowance in the X axis.
2. (W) Z Finish Stock: Distance and direction of the finish allowance in the Z axis.
3. (I) X Rough Stock: Distance and direction of the rough finish allowance in the X axis.
4. (K) Z Rough Stock: Distance and direction of the rough-finish allowance in the Z axis.
5. (V) 1st 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).
6. (D) Cut Increment: Depth of cuts. Designated without sign. The cutting direction depends on the direction of the 1st move in the pattern.
7. 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 Z for the next cut. If the

relief type is 45 DEGS. no pockets are allowed in the pattern.

8. (R) Clearance: Distance of the 45 degree relief or an additional distance to pull away after the pattern relief.
9. Compensation OFF, LEFT, or RIGHT: Tool nose radius compensation.
10. (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.
11. (N) Pattern Number: Any number between 0 and 99. The pattern can be called by a finishing cycle later in the program.

CODE GENERATED FROM THE ROUGH TURNING SET-UP

```
G0   Z##.####  X##.####
|        |          |
|        |          └──────X Start Position
|        └─────────────────Z Start Position
└──────────────────────────Rapid
```

```
P140 = ##.####
       |
       └──────────Z Start Position (used for a Z Clearance
                  Plane)
```

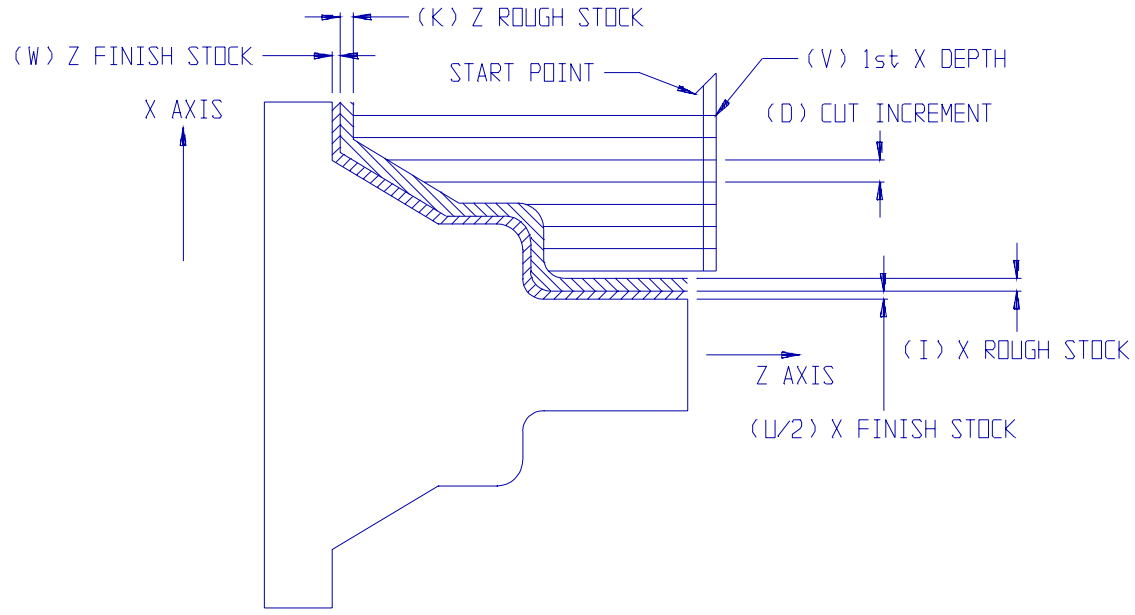
```
P139 = ##.####
       |
       └──────────X Start Position (used for an X Clearance
                  Plane)
```

```
G71 P98## Q99## U##.### W##.### I##.### K##.### R##.### V##.### D##.##
 |      |      |      |      |      |      |      |      |      |
 |      |      |      |      |      |      |      |      |      └──Cut Increment
 |      |      |      |      |      |      |      |      └─────────1st X Depth
 |      |      |      |      |      |      |      └────────────────Clearance
 |      |      |      |      |      |      └───────────────────────Z Rough Stock
 |      |      |      |      |      └──────────────────────────────X Rough Stock
 |      |      |      |      └─────────────────────────────────────Z Finish Stock
 |      |      |      └────────────────────────────────────────────X Finish Stock
 |      |      └───────────────────────────────────────────────────Pattern #
 |      └──────────────────────────────────────────────────────────Pattern #
 └─────────────────────────────────────────────────────────────────Rough Turning
                                                                     Cycle
```

```
N98##
  |
  └──Pattern #

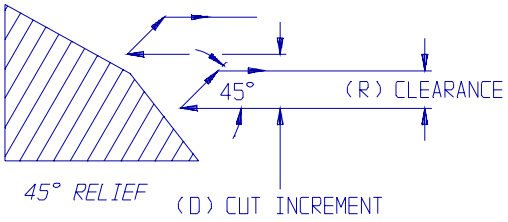
G42
 |
 └──────CutterComp Right or G41 CutterComp Left or G40
                CutterComp  Off
```

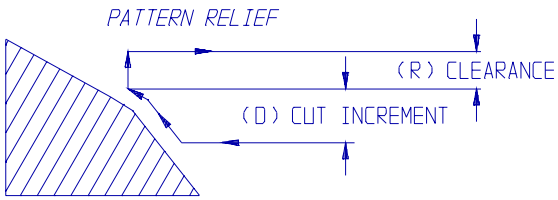
Rough cutting is done by first cutting parallel to the Z axis down to the pattern leaving the rough stock. After cutting parallel to the Z axis is completed, a pass is made along the finish pattern leaving the finish stock.



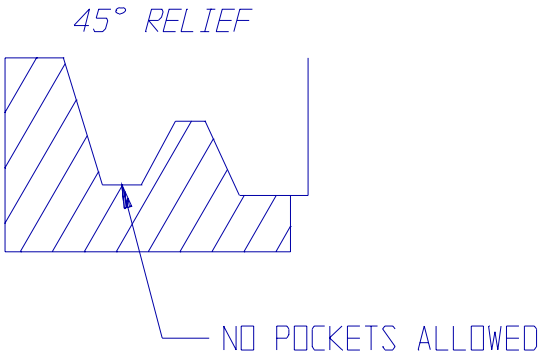
Feed and speeds within the pattern are ignored during the parallel cuts. Feeds and speeds are effective during the pass along the pattern. Rapid moves within the pattern are effective when following the finish pattern.

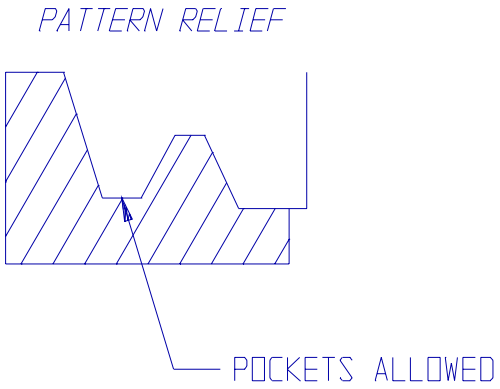
Note: If the X rough stock and the Z rough stock are both zero, the pass made along the finish pattern is skipped.



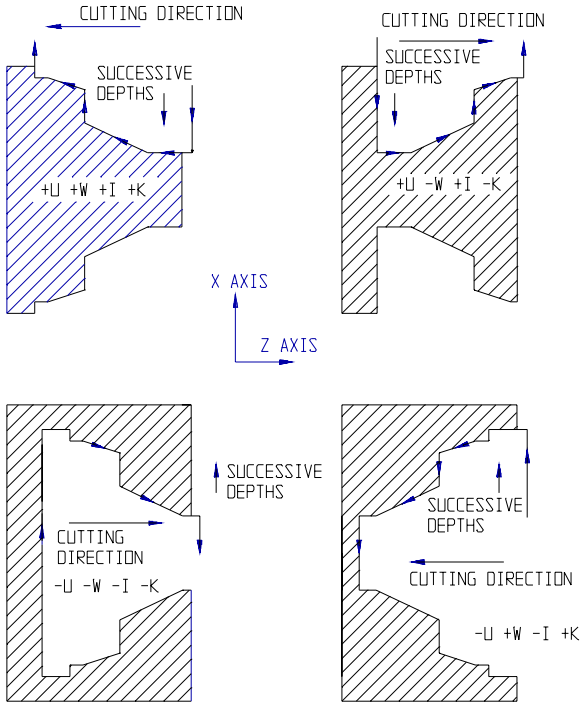


After rough cutting parallel to the Z axis there are 2 types of relief. 45° relief will rapid away from the pattern at a 45° angle. Pattern relief follows the pattern and then retracts in the X axis. A 0 (zero) clearance is allowed in the pattern relief.

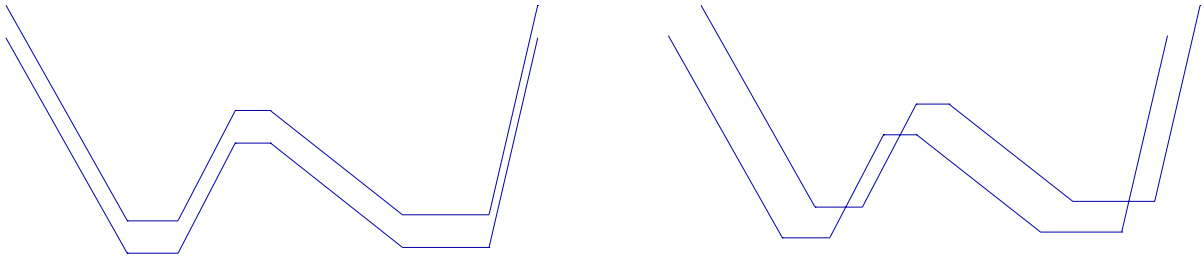




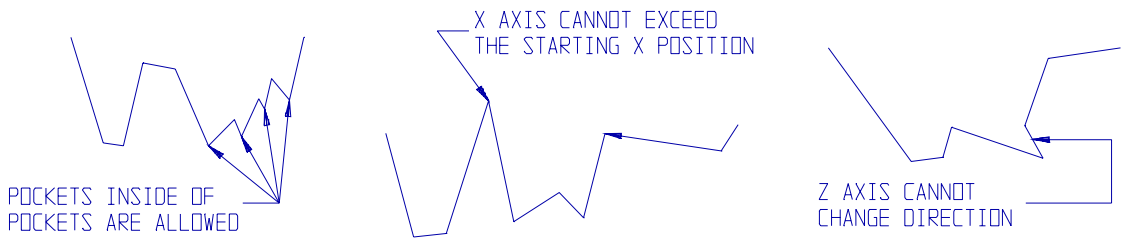
No pockets are allowed when using the 45° relief.



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.*

7.3 PATTERN REPEAT CYCLE

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).

```
PATTERN REPEAT TURNING SET-UP

1. X Finish Stock    U[          ]d
2. Z Finish Stock    W[          ]
3. X Rough Stock     I[          ]r
4. Z Rough Stock     K[          ]

5. Number of Passes  D[  ]

6. Compensation       [RIGHT]
7. Start Point       X[          ]d

8. Pattern Number    N[  ]
   Note:  Geometry between this Event and End
          Cycle define the Pattern
```

1. (U) X Finish Stock: Distance and direction of the finish allowance in the X axis.

2. (W) Z Finish Stock:	Distance and direction of the finish allowance in the Z axis.
3. (I) X Rough Stock:	Distance and direction of the rough allowance in the X axis.
4. (K) Z Rough Stock:	Distance and direction of the rough allowance in the Z axis.
5. (D) Number of Passes:	The number of passes to cut the rough stock.
6. Compensation OFF, LEFT, or RIGHT:	Tool nose radius compensation.
7. (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.
8. (N) Pattern Number:	Any number between 0 and 99. The pattern number can be called by a finishing cycle later in the program.

CODE GENERATED BY PATTERN REPEAT TURNING CYCLE

```
G0   X##.####   Z##.####
          |           |
          |           └──────Z Start Point
          └──────────────────X Start Point
```

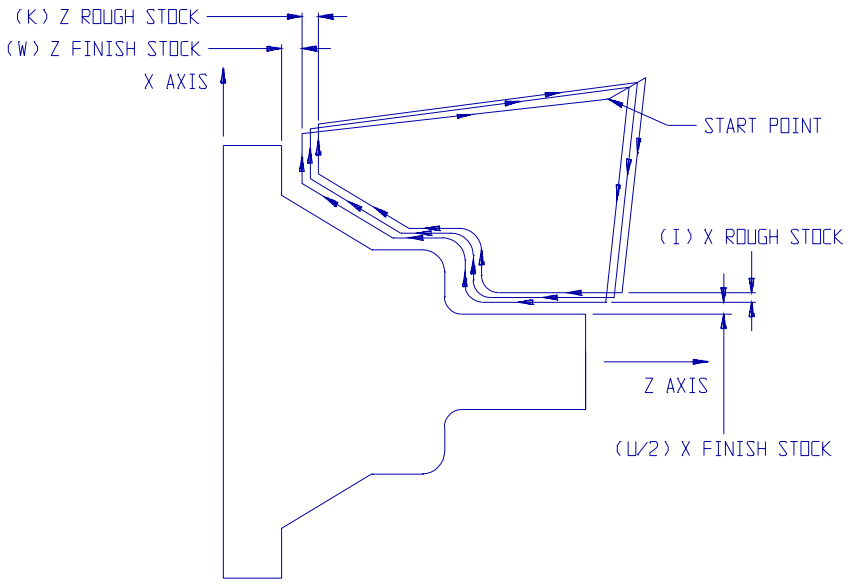
```
P140 = ##.####
           |
           └──────────Z Start Point (used for Z Clearance Plane)
```

```
P139 = ##.####
           |
           └──────────X Start Point (used for X Clearance Plane)
```

```
G73 P8## Q9## U##.### W##.#### I##.#### K##.#### D##
 |    |    |    |    |    |    |    |    |
 |    |    |    |    |    |    |    |    └──Number of Passes
 |    |    |    |    |    |    |    └───────X Rough Stock
 |    |    |    |    |    |    └────────────Z Rough Stock
 |    |    |    |    |    └─────────────────X Finish Stock
 |    |    |    |    └──────────────────────Z Finish Stock
 |    |    |    └───────────────────────────Pattern #
 |    |    └────────────────────────────────Pattern #
 |    └─────────────────────────────────────Pattern Repeat Cycle
 └──────────────────────────────────────────
```

N98##
└──────Pattern #

G42
└──────────CutterComp Right or G41 CutterComp Left



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.

7.4 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.

```
FINISH TURNING

1. Pattern Number     N[  ]

2. Feedrate           F[     ]
3. Spindle Speed      S[     ]

4. Start Point        X[         ]d
                      Z[         ]
```

1. (N) Pattern Number: Any pattern defined by a rough turning or pattern repeat cycle.
2. (F) Feedrate: The feedrate to cut the finish pattern. Any feedrates within the pattern will be effective.
3. (S) Spindle Speed: The spindle speed to cut the finish pattern. Any spindle speed within the pattern will be effective.
4. (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.

CODE GENERATED BY FINISH TURNING CYCLE

```
F.####
   └──────Feedrate

S####
   └──────Spindle Speed

G0  X##.####  Z##.####
 │       │          └──────Z Start Point
 │       └─────────────────X Start Point
 └─────────────────────────Rapid

P140 = ##.####
           └────Z Start Point (used for a Z Clearance Plane)
```

```
P139 = ##.####
           └──────X Start Point (used for a Clearance Plane)

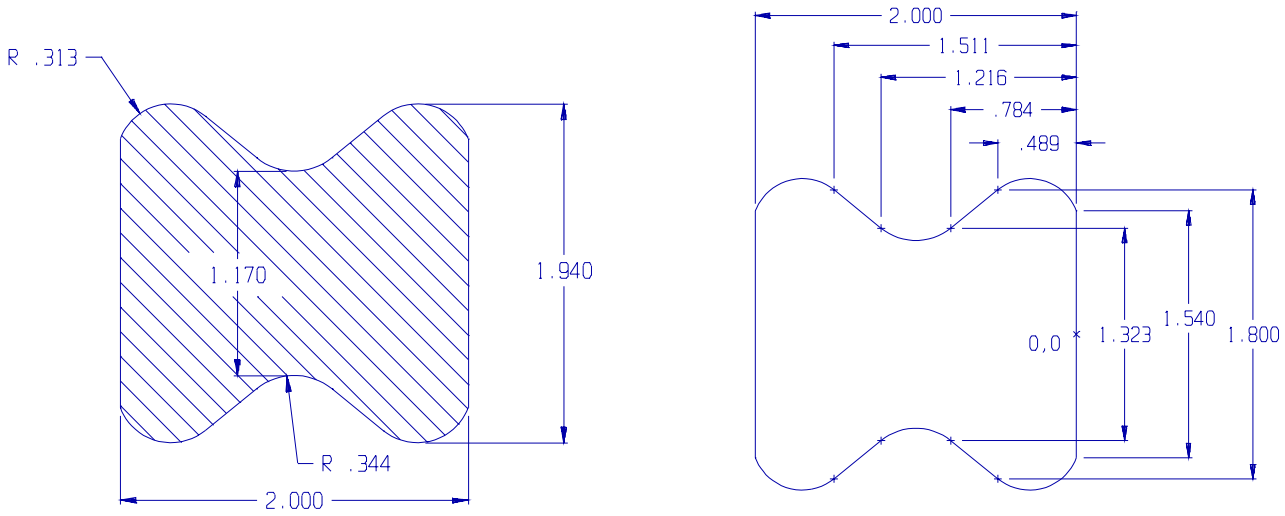
G70  P98##  Q98##
 │     │      └──────Pattern #
 │     └─────────────Pattern #
 └───────────────────Finish Cycle

G0   X##.####
 │      └──────X Start Point
 └─────────────Rapid

Z##.####
   └─────────Z Start Point
```

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



Example program using a rough turning cycle with a finish cycle for the part shown above:

```
Conversational Program C:\LATHE\PARTS\P0099
     Event 0 of 14
Program Set Up

     Program name          [ROLLER              ]

     Dimensions            [ABSOLUTE]
     Units                 [ENGLISH]

     Material Length      P[2.5     ]
     Material Diameter    P[2       ]d
_____________________________________________________

Event 1 of 14
Tool Change
[Tool Change]
Tool Number            T[1 ]
Tool Description [RUFFING TOOL         ]
Feed per [Revolution]  F[.01  ]

Spindle  [Constant Surface Speed]
Max Spindle Speed    RPM[2000   ]
Spindle Speed          S[600    ]

Spindle Restart         [CW]
Coolant                 [ON]
_____________________________________________________

Event 2 of 14
Rough Turning Set-Up
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 14
Geometry - Line
```

```
Feedrate                    [RAPID]
Coordinates                 [CARTESIAN]

X-axis          X[1.54    ]
Z-axis          Z[        ]

End       [---]

Extend Back    [OFF]
```

```
Event 4 of 14
Geometry - Line

Feedrate                    [      ]
Coordinates                 [CARTESIAN]

X-axis                     X[         ]d
Z-axis                     Z[0        ]

End         [----]

Extend Back      [OFF]
```

```
Event 5 of 14
Geometry - Arc

Feedrate              F[     ]
Direction              [CCW]
Center                 [RADIUS ONLY]
Arc Radius            R[.313       ]  [SHORT]

End Point              [ABSOLUTE]
                      X[1.8     ]
                      Z[-.489   ]

End Option       [----]
```

```
Event 6 of 14
Geometry - Line

Feedrate           [      ]
Coordinates        [CARTESIAN]

X-axis            X[1.323   ]d
Z-axis            Z[-.784   ]
```

```
End       [---]

Extend Back    [OFF]
```

```
Event 7 of 14
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 14
Geometry - Line

Feedrate          [      ]
Coordinates       [CARTESIAN]

X-axis           X[1.8     ]
Z-axis           Z[-1.511  ]

End    [---]

Extend Back    [OFF]
```

```
Event 9 of 14
Geometry Arc

  Feedrate         F[     ]
Direction         [CCW]
Center            [RADIUS ONLY]
Arc Radius       R[.313    ]  [SHORT]

End Point         [ABSOLUTE]
                 X[1.54    ]d
                 Z[-2      ]

End Option    [---]
```

```
Event 10 of 14
Geometry - Line

Feedrate          [     ]
Coordinates       [POLAR]

Type          [CURRENT]

Length           R[.3      ]
Angle           AB[180     ]

End   [---]

Extend Back   [OFF]
```

```
Event 11 of 14

End Turning Cycle
```

```
Event 12 of 14
Tool Change
[Tool Change]
Tool Number                T[2 ]
Tool Description [FINISH TOOL            ]
Feed Per [Revolution]   F[.005  ]

Spindle  [Constant Surface Speed]
Max Spindle Speed       RPM[2000 ]
Spindle Speed             S[900  ]

Spindle Restart      [CW]
Coolant              [ON]
```

```
Event 13 of 14
Finish Turning

Pattern Number      N[1 ]

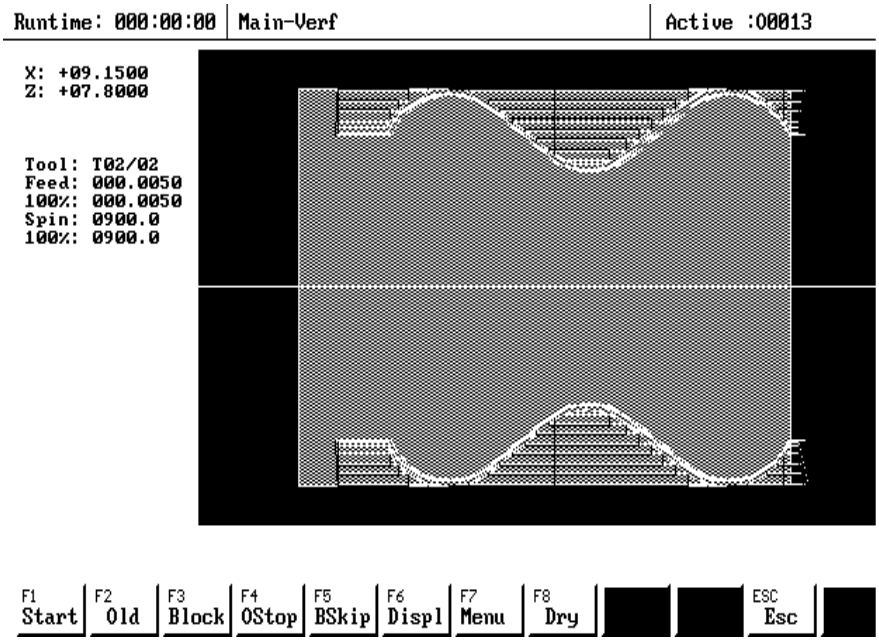
Feedrate            F[     ]
Spindle Speed       S[     ]

Start Point         X[2.1     ]d
                    Z[.1      ]
```

```
Event 14 of 14
End of Program

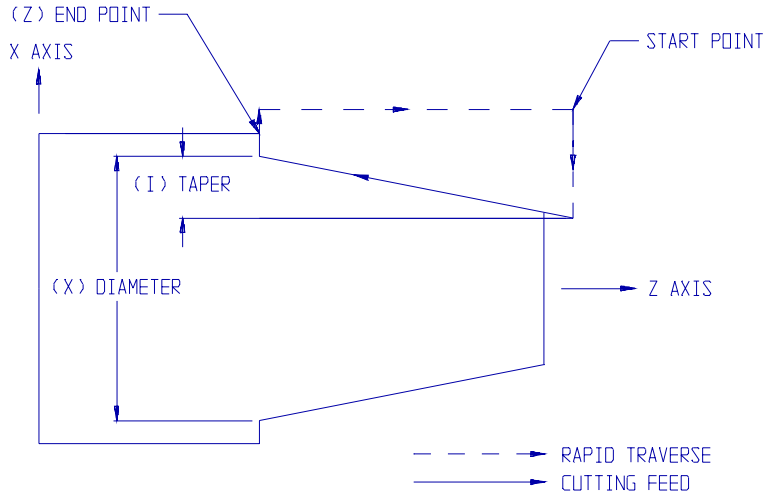
Spindle off          [YES]
Coolant off          [YES]
X Position (home relative)[        ]
Z Position (home relative)[        ]
```

The above program produces the graphics shown below:



7.5 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 Dimension

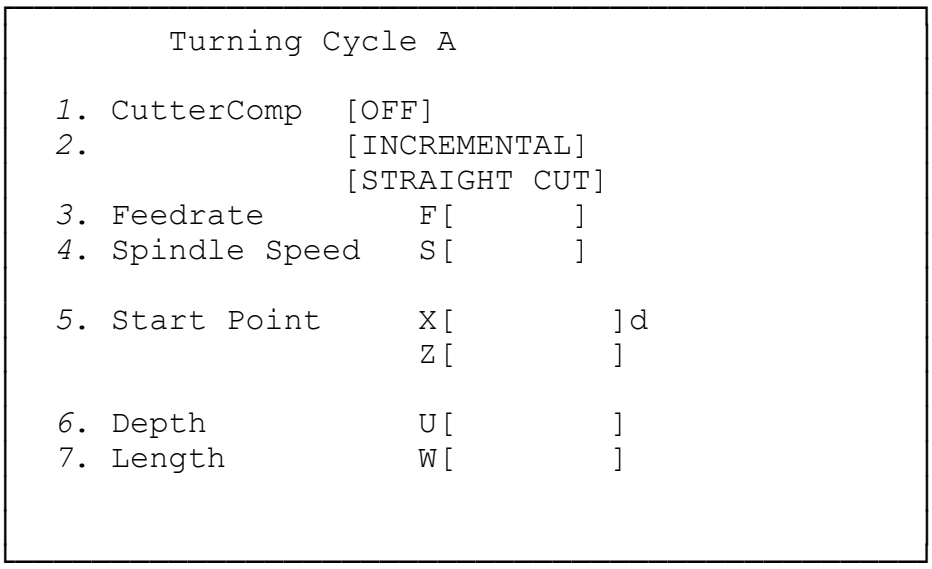
```
        Turning Cycle A

   1. CutterComp [OFF]
   2.            [ABSOLUTE]
                 [STRAIGHT CUT]
   3. Feedrate       F[     ]
   4. Spindle Speed  S[     ]

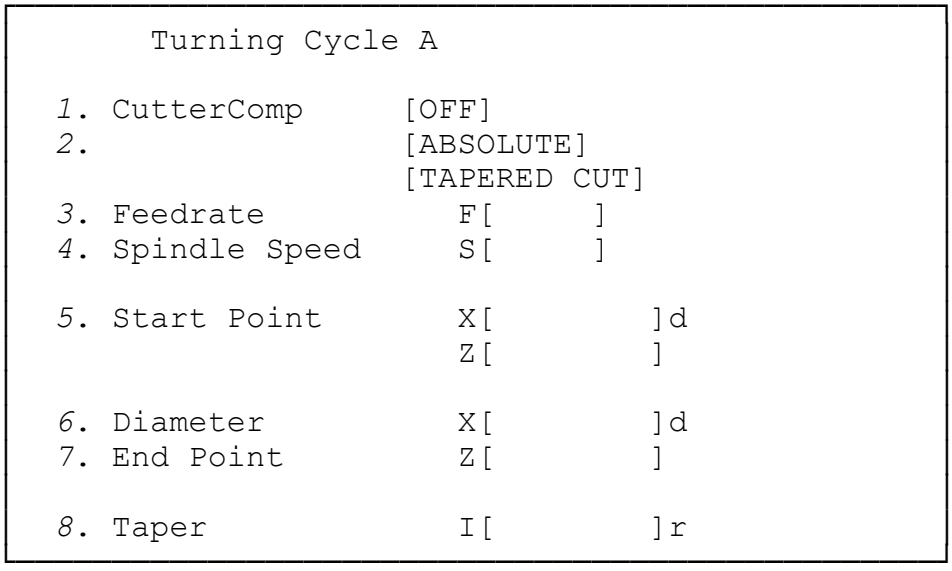
   5. Start Point    X[        ]d
                     Z[        ]

   6. Diameter       X[        ]d
   7. End Point      Z[        ]
```

-OR-



-OR-



1. Compensation OFF, LEFT, or RIGHT:	Tool nose radius compensation.
2. Absolute or Incremental:	Toggles between absolute dimensions (diameter, end point) and incremental dimensions (depth and length).
3. (F) Feedrate:	The feedrate to turn the part.
4. (S) Spindle Speed:	The spindle speed to turn the part.

5.	(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.
6.	(X) Diameter:	The diameter to turn.
7.	(Z) End Point:	The end point of the cut.
6a.	(U) Depth:	The incremental X distance and direction from the start point to the diameter to turn.
7a.	(W) Length:	The length and direction of the cut in Z from the start point.
8.	(I) Taper:	The amount and direction of the taper to turn.

TEXT PROGRAM GENERATED BY CYCLE A
USING ABSOLUTE COORDINATES WITH A TAPER AND CUTTERCOMP RIGHT

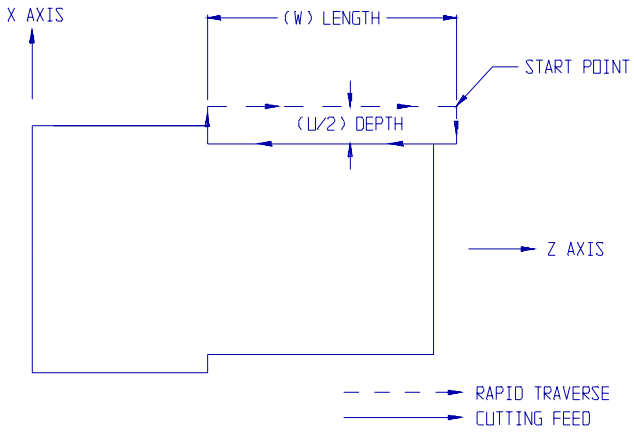
```
G0  X##.####  Z##.####
 |      |          |
 |      |          └────────Z Start Position
 |      └──────────────────X Start Position
 └─────────────────────────Rapid

P140 = ##.####
          |
          └────────Z Start Position (used for a Z Clearance Plane)

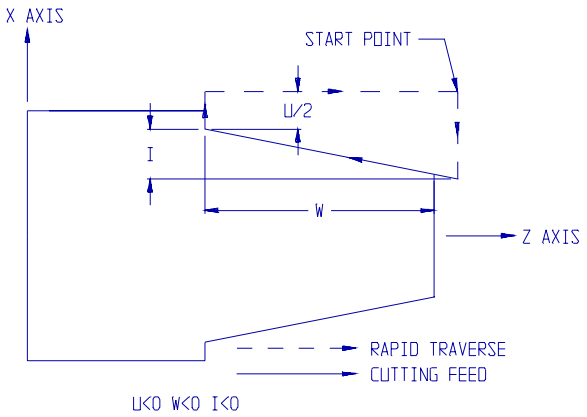
P139 = ##.####
          |
          └────────X Start Position (used for a X Clearance Plane)

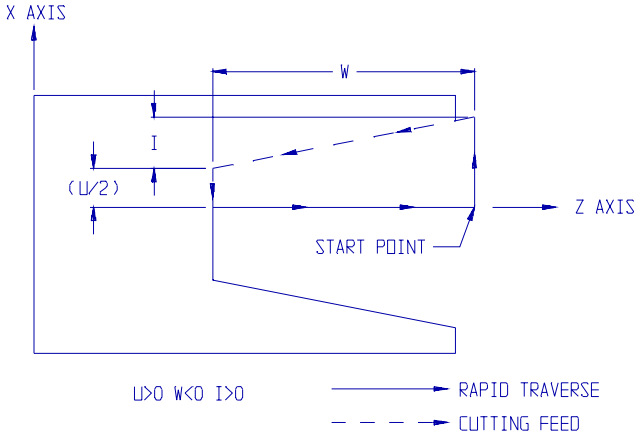
S####
  |
  └─────Spindle Speed

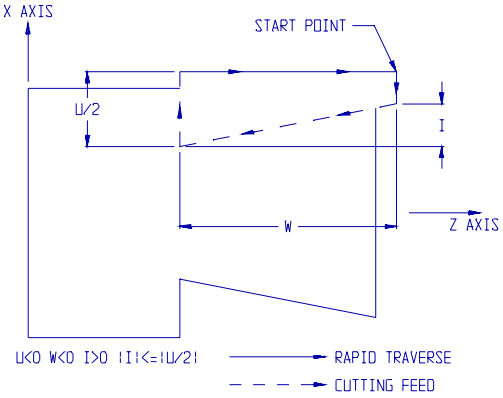
G42 G90 I##.#### X##.#### Z##.#### F.####
 |   |       |         |         |      |
 |   |       |         |         |      └──────Feedrate
 |   |       |         |         └─────────────Z End Point
 |   |       |         └───────────────────────X Diameter
 |   |       └─────────────────────────────────Taper
 |   └───────────────────────────────────────────Turning Cycle A
 └───────────────────────────────────────────────CutterComp RIGHT, or
                                                 G41 CutterComp
                                                 LEFT, or G40
                                                 CutterComp OFF
```

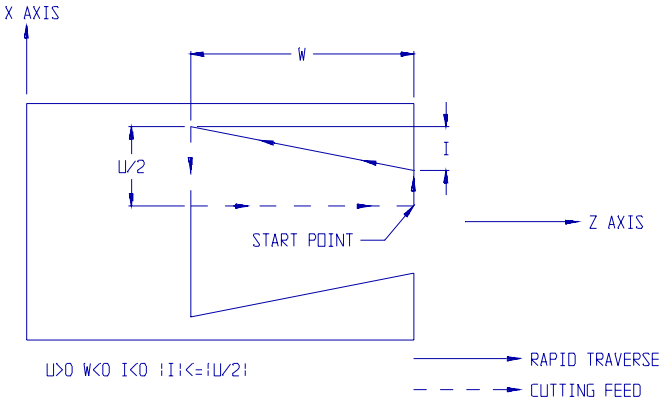


Straight Cutting Using Cycle A With Incremental Dimensions









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

7.6 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.

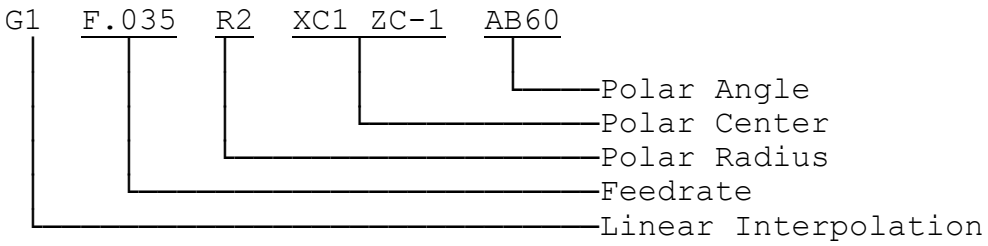
```
┌──────────────────────────────────────┐
│  Geometry - Line                     │
│                                      │
│  Feedrate     F[.02 ]                │   CONVERSATIONAL
│  Coordinates   [CARTESIAN]           │     SCREENS
│                                      │
│  X-axis              X[2        ]d   │   Cartesian Linear
│  Z-axis              Z[2        ]    │     Interpolation
│                                      │
│                                      │
│  End                  [---]          │
│  Extend Back          [Off]          │
└──────────────────────────────────────┘
```

GENERATED CODE FOR LINE

```
G1  F.02  X2  Z2
│    │    │   └──Z Position
│    │    └──────X Position
│    └───────────Feedrate
└────────────────Linear Interpolation
```

```
┌──────────────────────────────────────┐
│  Geometry - Line                     │
│                                      │
│  Feedrate            F[.035 ]        │   CONVERSATIONAL
│  Coordinates          [POLAR]        │     SCREENS
│                                      │
│  Type                 [ABSOLUTE]     │    Polar Linear
│   XC                [1         ]d    │   Interpolation
│   ZC                [-1        ]     │
│  Length            R[2         ]     │
│  Angle            AB[60        ]     │
│                                      │
│  End                  [---]          │
│  Extend Back          [OFF]          │
└──────────────────────────────────────┘
```

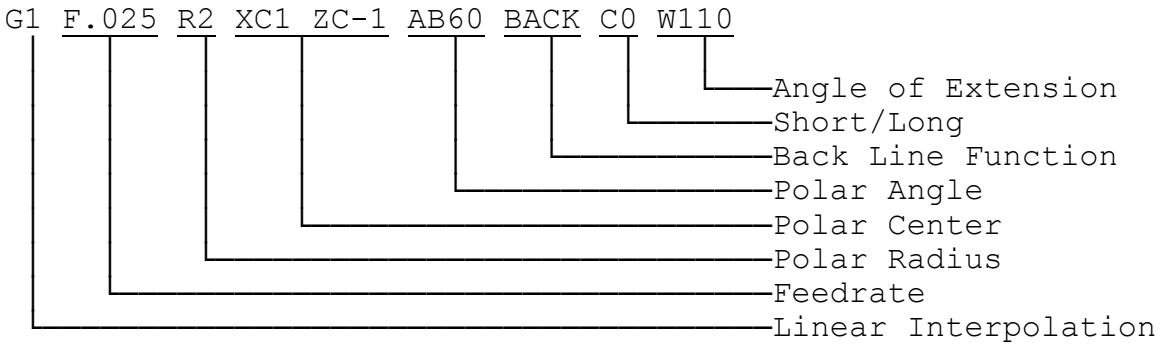
GENERATED CODE FOR LINE



When Extend Back [ON] is selected, these lines will appear:

```
Extended [SHORT]   Angle [110]
```

GENERATED CODE FOR BACK LINE



Line W/Round Corner

```
Geometry - Line

Feedrate  F[.02  ]
Coordinates[CARTESIAN]

X-axis    X[2      ]d
Z-axis    Z[2      ]

End     [Round Corner]
Radius     [.15      ]
```

Line W/Chamfer

```
Mill Geometry - Line

Feedrate   F[.02  ]
Coordinates [CARTESIAN]

X-axis     X[2      ]d
Z-axis     Z[2      ]

End          [Chamfer]
Length      [.15      ]
```

CONVERSATIONAL SCREENS

GENERATED CODE FOR ROUND CORNER

```
G1 F.02 X2 Y2, R .15
                  |
Round Corner Radius
```

GENERATED CODE FOR CHAMFER

```
G1 F.02 X2 Y2, C .15
                  |
Chamfer Length
```

Note 1: Chamfers are allowed for lines only. They are not allowed to or from arcs.

Note 2: For more information on lines see G1 in Section 3.

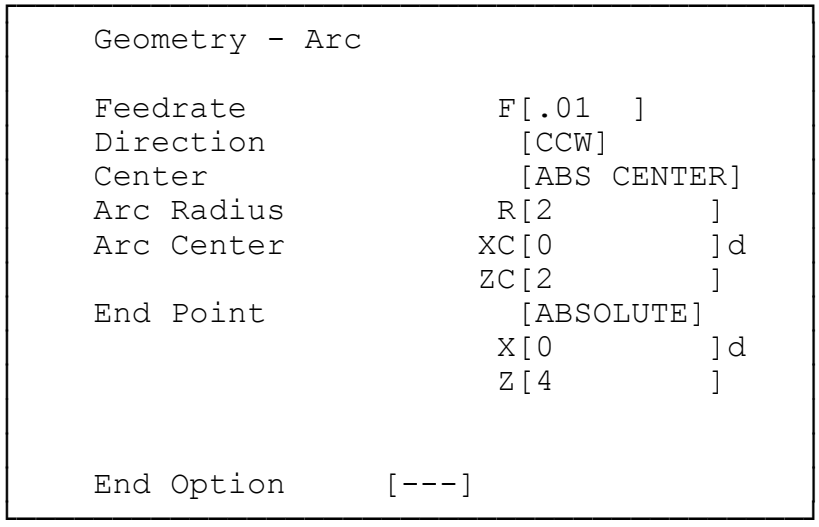
7.7 ARC

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 and G3.

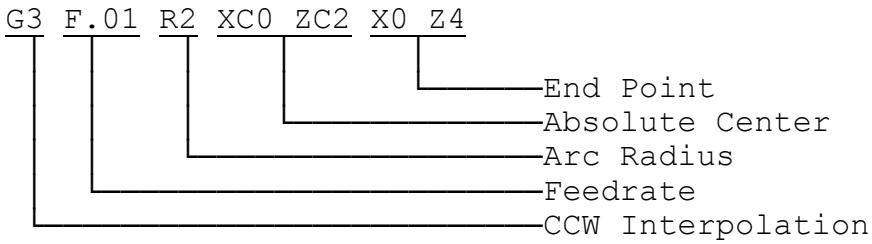
```
┌──────────────────────────────────────────┐
│  Geometry - Arc                          │
│                                          │
│  Feedrate              F[.02  ]          │   CONVERSATIONAL
│  Direction              [CW]             │     SCREENS
│  Center                 [INC CENTER]     │
│                                          │   Incremental Center
│  Arc Center            I[1        ]r     │   CW Circular
│                        K[-1       ]      │   Interpolation
│  End Point              [ABSOLUTE]       │
│                        X[2        ]d     │
│                        Z[0        ]      │
│  End Option     [---]                    │
│                                          │
└──────────────────────────────────────────┘
```

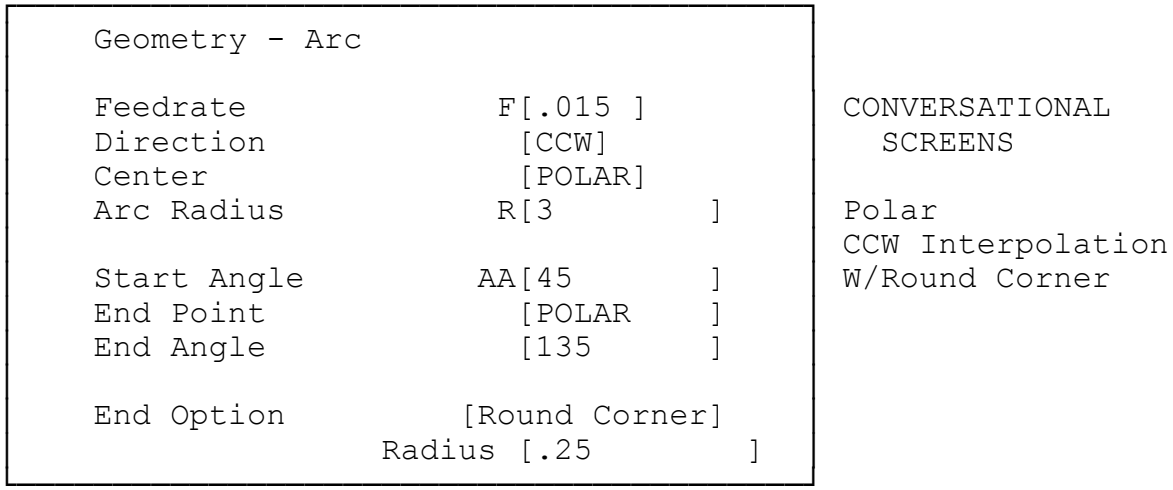
GENERATED CODE FOR ARC

```
G2  F.02  I1 K-1 X2 Z0
|     |       |     |
|     |       |     └──Arc End Point
|     |       └────────Incremental Arc Center Point
|     └────────────────Feedrate
└──────────────────────CW Interpolation
```

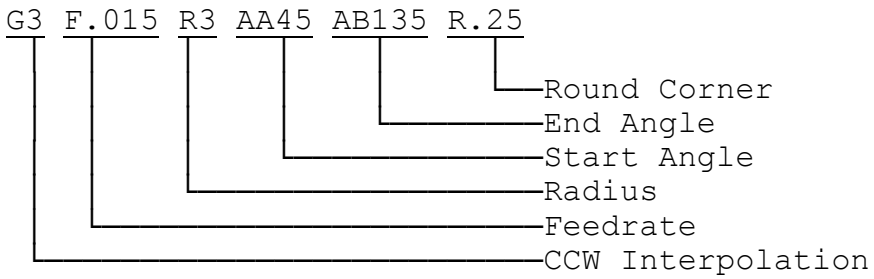


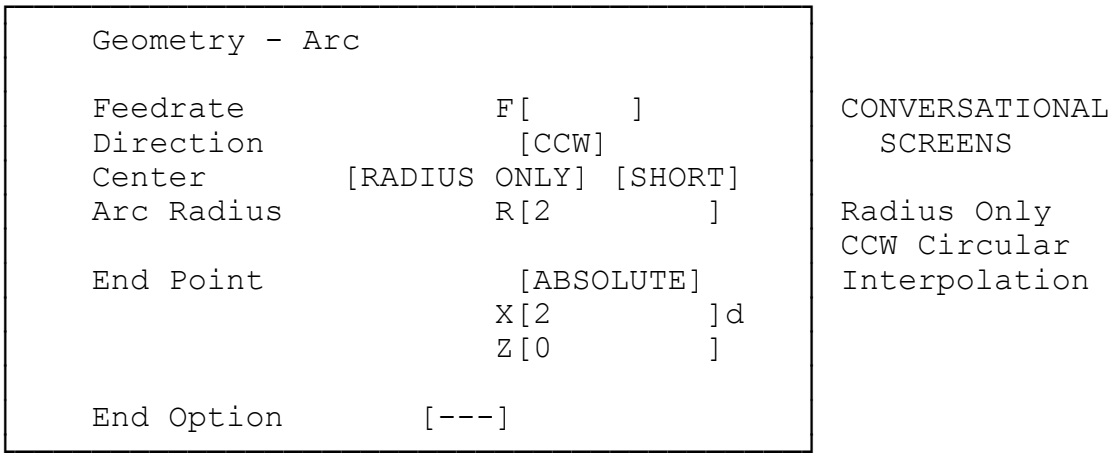
GENERATED CODE FOR ARC



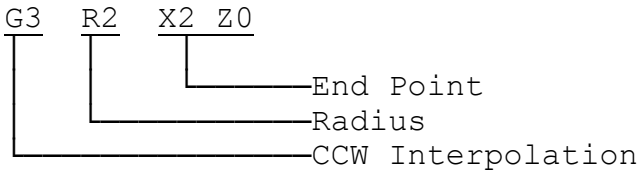


GENERATED CODE FOR ARC





GENERATED CODE FOR ARC

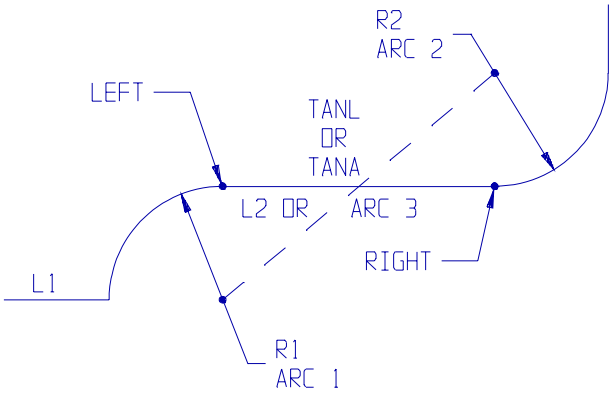


Note: For more information see G2/G3 in Section 3.

7.8 TANGS

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

7.8.1 Tangent Line

```
Connect two arcs with tangent line
    or arc
Cut first arc in direction [CW]
R1    [1.5    ]
XC1   [0      ]d  ZC1[0      ]

Second arc for computation is:
R2    [2      ]
XC2[5      ]d     ZC2[4      ]

Exit 1st arc [LEFT] & enter 2nd arc [LEFT]
  Connect with [a line]
```

CONVERSATIONAL SCREENS

Tangent Line

GENERATED CODE FOR TANGENT LINE

```
P90 = 0                          Z Center of arc 1
P91 = 0                          X Center of arc 1
P92 = 1.5                        radius of arc 1
P93 = 5                          Z Center of arc 2
P94 = 4                          X Center of arc 2
P95 = 2                          radius of arc 2

TANL  C3
  |    |
  |    |_____Tangent line type
  |__________Tangent line function

G2  XC[P91] ZC[P90]  R[P92]  X[P81] Z[P80]
 |      |       |       |_____|_____ End point of arc
 |      |       |                    calculated by
 |      |       |                    TANL function
 |      |       |_____________________Radius
 |      |_____________________________Absolute center
 |                                      of the first
 |                                      Arc
 |_____________________________________CW interpolation

G1  X[P83] Z[P82]
 |      |_____|_____End point of line calculated by TANL
 |_____________________Linear interpolation
```

7.8.2 Tangent Arc

```
Connect two arcs with tangent line
    or arc
Cut first arc in direction [CW]
R1  [1.5   ]
XC1 [0        ]d  ZC1[0    ]

Second arc for computation is:
R2  [2     ]
XC2 [5     ]d  ZC2[4        ]

Exit 1st arc [LEFT] & enter 2nd arc [LEFT]

Connect with [An Arc] Center to the [LEFT]
 Radius [5  ] Arc Direction [CCW]
```

CONVERSATIONAL SCREENS

Tangent Arc

GENERATED CODE FOR TANGENT ARC

```
P90 = 0                          Z Center of arc 1
P91 = 0                          X Center of arc 1
P92 = 1.5                        radius of arc 1
P93 = 5                          Z Center of arc 2
P94 = 4                          X Center of arc 2
P95 = 2                          radius of arc 2

TANA  C3
  |    |
  |    |_____Tangent arc type
  |__________Tangent arc function

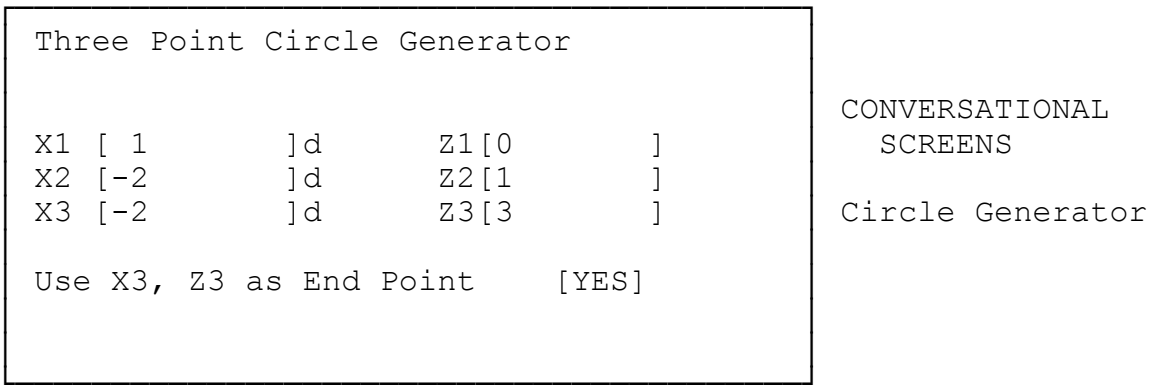
G2  XC[P91] ZC[P90]  R[P92]  X[P81] Z[P80]
 |     |       |       |        |
 |     |       |       |        |_____ End point of arc
 |     |       |       |              calculated by
 |     |       |       |              TANA function
 |     |       |       |_____________Radius
 |     |       |_____________________Absolute center
 |     |                               of the first
 |     |                               arc
 |_____|_____________________________CW interpolation

G3  XC[P85] ZC[P84]  R[P96]  X[P83] ZP92]
 |     |       |       |        |
 |     |       |       |        |_____End Point tangent
 |     |       |       |              arc calculated
 |     |       |       |              by TANA function
 |     |       |       |_____________Radius of Tangent
 |     |       |                       Arc
 |     |       |_____________________Center Tangent
 |     |                               Arc
 |_____|_____________________________CCW Interpolation
```

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.

7.9 CGEN

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.



GENERATED CODE FOR CIRCLE GENERATOR

```
P90 = 1        Z1
P91 = 0        X1
P92 = 2        Z2
P93 = 1        X2
P94 = 2        Z3
P95 = 3        X3

CGEN
   └──────────Circle Generate Function

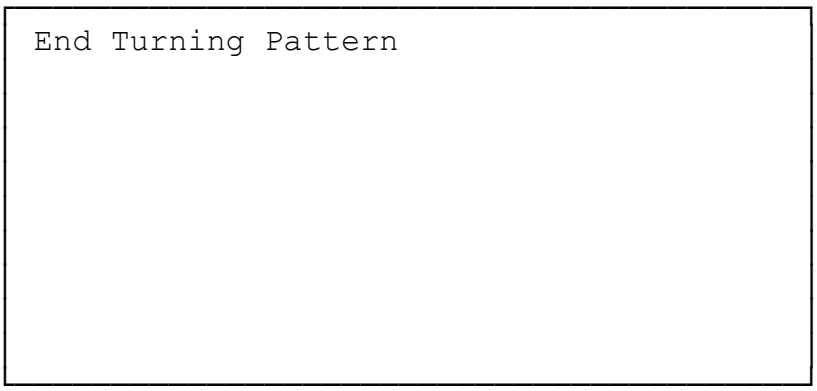
G2 If P97=3 Then G3
          └──────────Selects Arc Direction

XC[P80] YC[P81]  R[P82]  X[P95] Z[P94]
                                └──────End Point (The Last
                                        Point)
                         └──────────────Radius Calculated by
                                        CGEN
          └─────────────────────────────Center Calculated by
                                        CGEN
                    -OR-

XC[P81]  ZC[P80] R[P82] AB90
                          └────End Angle
                   └──────────Radius Calculated by CGEN
          └─────────────────────Center Calculated by CGEN
```

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.

7.10 END



CONVERSATIONAL SCREENS

End Turning Pattern

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

CODE GENERATED FROM THE END TURNING SCREEN

```
N99##
    └──────The Pattern #
```

7.11 Rough Facing Cycle

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.

```
 Rough Facing Set-Up

 1. X Finish Stock     U[          ]d
 2. Z Finish Stock     W[          ]
 3. X Rough Stock      I[          ]r
 4. Z Rough Stock      K[          ]
 5. 1st Z Depth        V[          ]
 6. Cut Increment      D[          ]
 7. Relief              [PATTERN]
 8. Clearance          R[          ]
 9. Compensation        [LEFT]
10. Start Point        X[          ]d
                       Z[          ]
11. Pattern Number     N[ ]

    Note:  Geometry between this Event
    and End Cycle define the Pattern.
```

1. (U) X Finish Stock: Distance and direction of the finish allowance in the X axis.
2. (W) Z Finish Stock: Distance and direction of the finish allowance in the Z axis.
3. (I) X Rough Stock: Distance and direction of the rough allowance in the X axis.
4. (K) Z Rough Stock: Distance and direction of the rough allowance in the Z axis.
5. (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).
6. (D) Cut Increment: Depth of cuts. Designated without sign. The cutting direction depends on the direction of the 1st move in the pattern.
7. 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 Z for the next cut. If the relief type is 45 DEGS. no pockets are allowed in the pattern.
8. (R) Clearance:	Distance of the 45 degree relief or an additional distance to pull away after the pattern relief.
9. Compensation OFF, LEFT, or RIGHT:	Tool nose radius compensation.
10. (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.
11. (N) Pattern Number:	Any number between 0 and 99. The pattern number can be called by a finishing cycle later in the program.

CODE GENERATED FROM THE ROUGH FACING CYCLE

```
G0  X##.####  Z##.####
       |           |
       |           └──────Z Start Point
       └──────────────────X Start Point

P140 = ##.####
          |
          └──────────Z Start Point (used for Z Clearance Plane)

P139 = ##.####
          |
          └──────────X Start Point (used for X Clearance Plane)

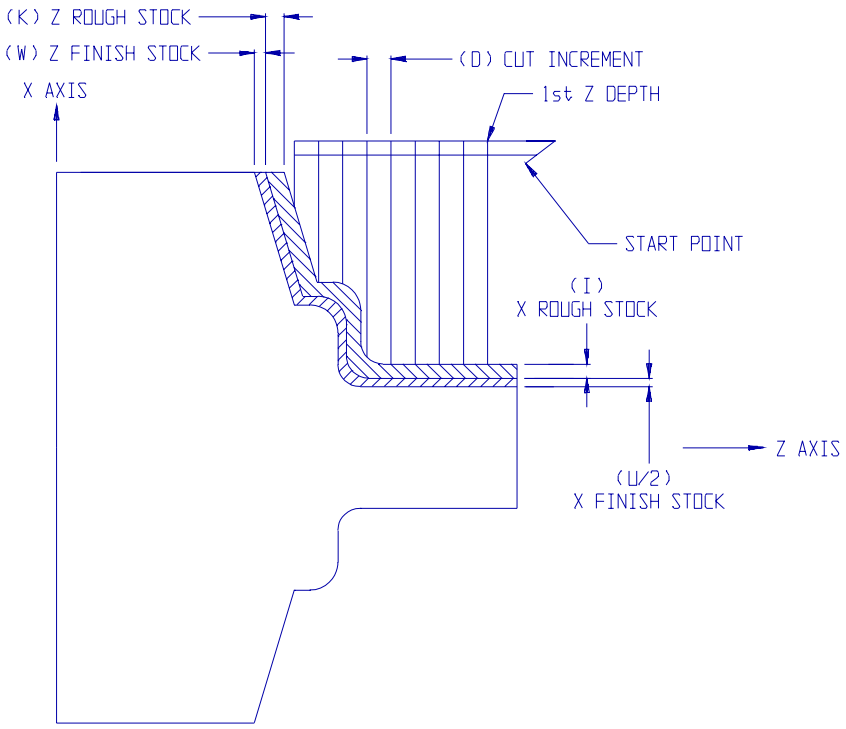
P117 = 0
       |
       └────────0 = Pattern Relief   1 = 45° Retract
```

```
G72 P98## Q99## V##.#### W##.#### I##.#### K##.#### R##.#### V##.#### D##.####
 |    |     |     |         |         |         |         |         |         |
 |    |     |     |         |         |         |         |         |         └──Cut Increment
 |    |     |     |         |         |         |         |         └────────1st Z depth
 |    |     |     |         |         |         |         └──────────────────Clearance
 |    |     |     |         |         |         └────────────────────────────Z Rough Stock
 |    |     |     |         |         └──────────────────────────────────────X Rough Stock
 |    |     |     |         └────────────────────────────────────────────────Z Finish Stock
 |    |     |     └──────────────────────────────────────────────────────────X Finish Stock
 |    |     └────────────────────────────────────────────────────────────────Pattern #
 |    └──────────────────────────────────────────────────────────────────────Pattern #
 └───────────────────────────────────────────────────────────────────────────Rough Facing Cycle
```

```
N98##
   |
   └────Pattern #
```

G41

└──────Left CutterComp or G42 Right CutterComp or G40 CutterComp Off



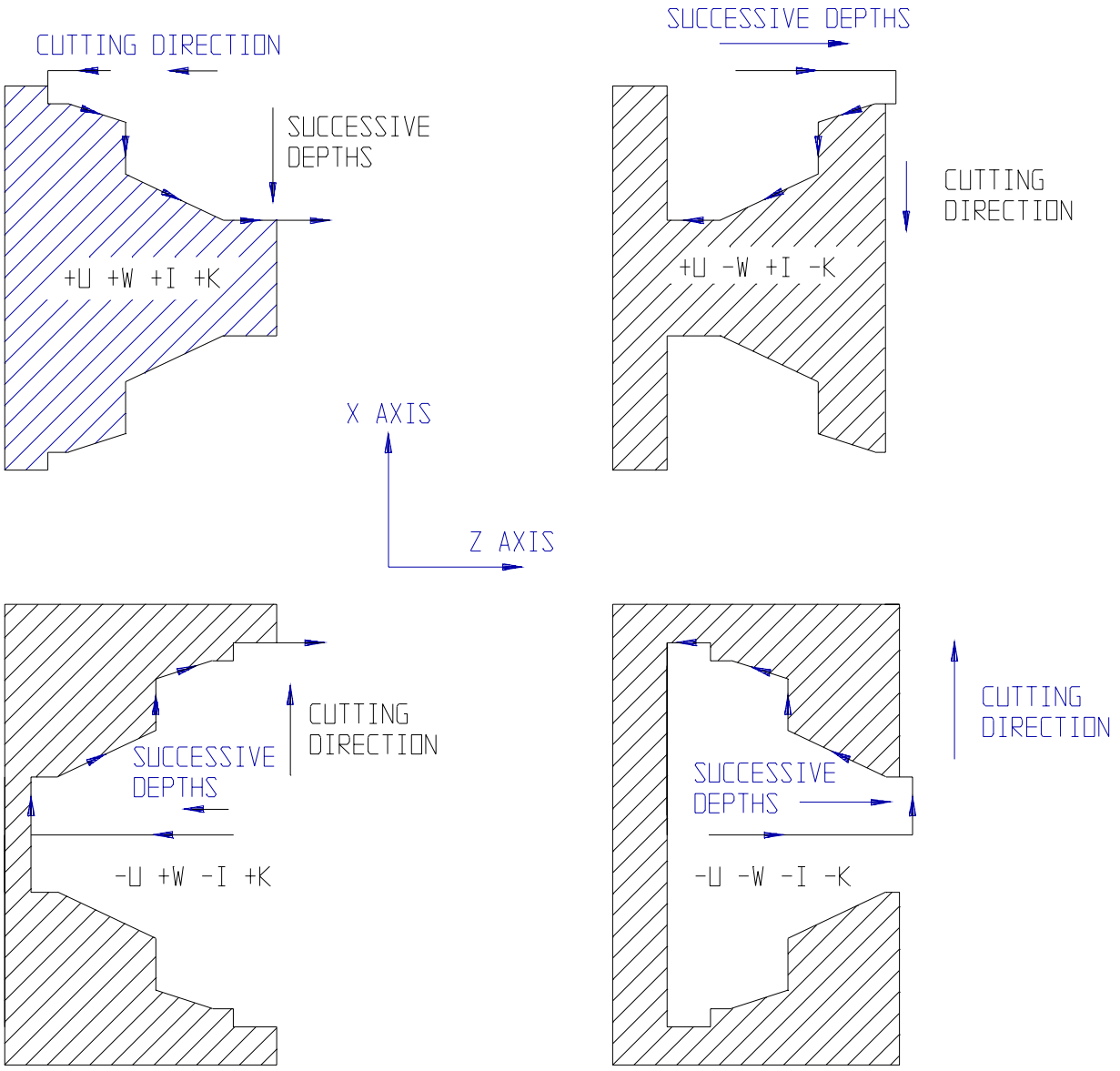
Rough cutting is done by first cutting parallel to the X axis down to the pattern, leaving the rough stock. After the cuts along the X axis are completed, a pass is made along the finish pattern leaving the finish stock.

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.

7.12 Pattern Repeat Cycle

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).

```
Pattern Repeat Facing Set-Up

1.  X Finish Stock    U[        ]d
2.  Z Finish Stock    W[        ]
3.  X Rough Stock     I[        ]r
4.  Z Rough Stock     K[        ]

5.  Number of Passes  D[  ]

6.  Compensation       [OFF]
7.  Start Point       X[        ]d
                      Z[        ]
8.  Pattern Number    N[  ]

Note:  Geometry between this Event
and End Cycle define the Pattern.
```

1. (U) X Finish Stock: Distance and direction of the finish allowance in the X axis.
2. (W) Z Finish Stock: Distance and direction of the finish allowance in the Z axis.
3. (I) X Rough Stock: Distance and direction of the rough allowance in the X axis.
4. (K) Z Rough Stock: Distance and direction of the rough allowance in the Z axis.
5. (D) Number of Passes: The number of passes to cut the rough stock.
6. Compensation OFF, LEFT, or RIGHT: Tool nose radius compensation.
7. (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.
8. (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.

7.13 Facing Finish Cycle

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

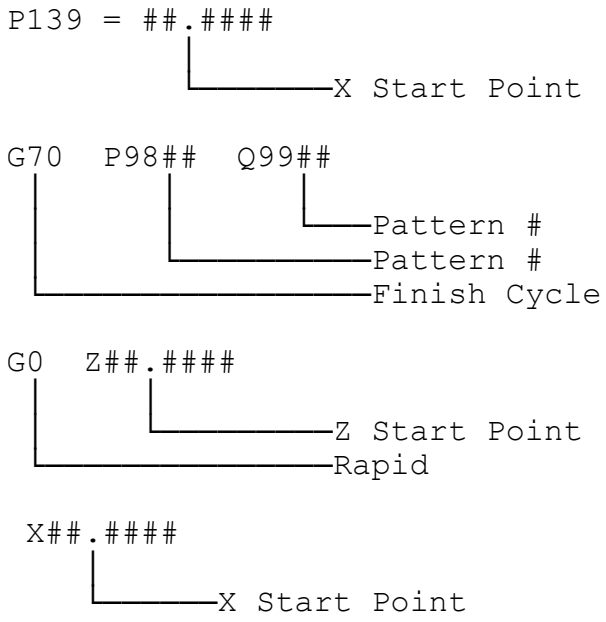
```
┌──────────────────────────────────────────┐
│          Finish Facing                   │
│                                          │
│    1.  Pattern Number    N[  ]           │
│                                          │
│    2.  Feedrate          F[     ]        │
│    3.  Spindle Speed     S[     ]        │
│                                          │
│    4.  Start Point       X[        ]d    │
│                          Z[        ]     │
│                                          │
└──────────────────────────────────────────┘
```

1. (N) Pattern Number: Any pattern defined by a rough facing or pattern repeat cycle.
2. (F) Feedrate: The feedrate to cut the finish pattern. Any feedrates within the pattern will be effective.
3. (S) Spindle Speed: The spindle speed to cut the finish pattern. Any spindle speed within the pattern will be effective.
4. (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.

CODE GENERATED FOR FINISH FACING CYCLE

```
G0  X##.####  Z##.####
 │       │          │
 │       │          └──────Z Start Point
 │       └─────────────────X Start Point
 └─────────────────────────Rapid

P140 = ##.####
          │
          └──────Z Start Point
```



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.

Example program using a Rough Facing Cycle with a Finish Cycle for the part shown above.

```
Conversation Program C:\LATHE\PARTS\P0333

Event 0 of 10

Program Set-Up

     Program name  [EXAMPLE FACING PROG.]

     Dimensions    [ABSOLUTE]
     Units         [ENGLISH]

     Material Length     P[3        ]
     Material Diameter   P[3        ]d
_____________________________________________________

Event 1 of 10
Tool Change
 [Tool Change ]
Tool Number              T[1 ]
Tool Description [RUFF FACING        ]
Feed Per [Revolution]    F[.01  ]
```

```
Spindle  [Constant Surface Speed]
Max Spindle Speed      RPM[2000 ]
Spindle Speed            S[500  ]

Spindle Restart           [CW]
Coolant                   [ON]
```

```
Event 2 of 10
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         [OFF]
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
Geometry - Line

Feedrate          [RAPID]
Coordinates       [CARTESIAN]

X-Axis           X[        ]d
Z-Axis           Z[-1      ]

End     [---]

Extend Back  [OFF]
```

```
Event 4 of 10
Geometry - Line

Feedrate       [      ]
Coordinates    [CARTESIAN]

X-Axis        X[2       ]d
Z-Axis        Z[        ]
```

```
End   [---]
```

```
Event 5 of 10
Geometry - Line

Feedrate        [     ]
Coordinates     [CARTESIAN]

X-Axis         X[1       ]d
Z-Axis         Z[-.5     ]

End     [---]

Extend Back  [OFF]
```

```
Event 6 of 10
Geometry - Line

Feedrate        [     ]
Coordinates     [CARTESIAN]

X-Axis         X[        ]d
Z-Axis         Z[.1      ]


End  [---]

Extend Back  [OFF]
```

```
Event 7 of 10
End Cycle
```

```
Event 8 of 10
Tool Change
 [Tool Change]
Tool Number        T[2 ]
Tool Description [FINISH FACING       ]
Feed Per [Revolution]    F[.007 ]

Spindle  [Constant Surface Speed]
Max Spindle Speed    RPM[2000 ]
Spindle Speed          S[600  ]

Spindle Restart         [CW]
Coolant                 [ON]
```

```
Event 9 of 10
Finish Facing

Pattern Number     N[1 ]

Feedrate           F[      ]
Spindle Speed      S[      ]

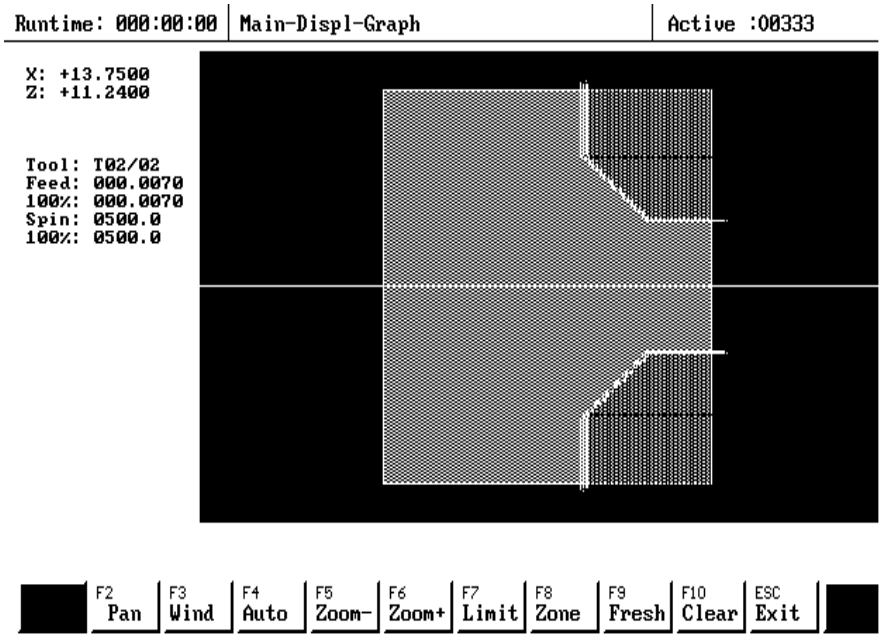
Start Point        X[3.1     ]d
                   Z[.1      ]
```

```
Event 10 of 10
End of Program

Spindle Off      [Yes]
Coolant Off      [Yes]

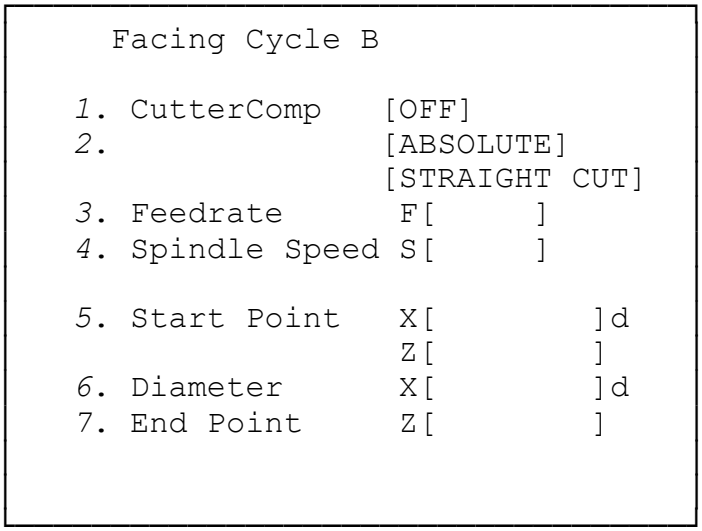
Y Position (home relative) [         ]
X Position (home relative) [         ]
```

The above program provides the graphics shown below.

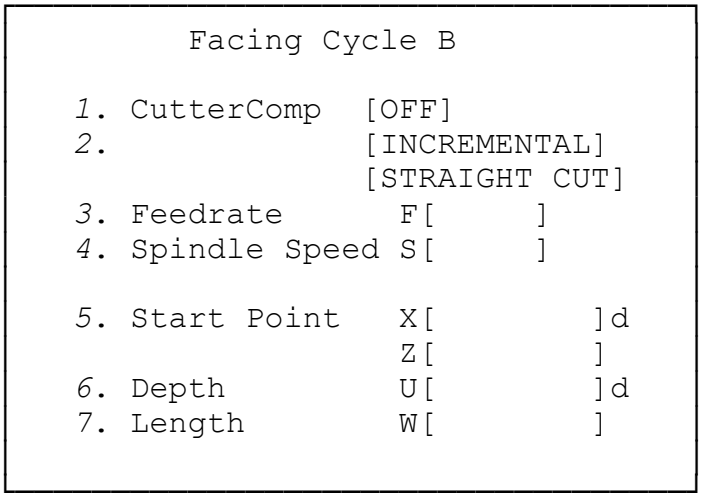


7.14 Cycle B

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-

```
┌──────────────────────────────────────┐
│      Facing Cycle B                  │
│                                      │
│   1. CutterComp  [OFF]               │
│   2.             [ABSOLUTE]          │
│                  [TAPERED CUT]       │
│   3. Feedrate      F[     ]          │
│   4. Spindle Speed S[     ]          │
│                                      │
│   5. Start Point   X[        ]d      │
│                    Z[        ]       │
│   6. Diameter      X[        ]d      │
│   7. End Point     Z[        ]       │
│                                      │
│   9. Taper         K[        ]       │
│                                      │
└──────────────────────────────────────┘
```

1. Compensation OFF, LEFT or RIGHT:	Tool nose radius compensation.
2. ABSOLUTE or INCREMENTAL:	Toggles between absolute dimensions (diameter, end point) and incremental dimensions (depth and length).
3. (F) Feedrate:	The feedrate to face the part.
4. (S) Spindle Speed:	The spindle speed to face the part.
5. (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.
6. (X) Diameter:	The diameter to face to.
7. (X) End Point:	The end point of the cut.
6a. (U) Depth:	The incremental X distance and direction from the start point to the diameter to face to.
7a. (W) Length:	The length and direction of the cut in Z from the start point.
8. (I) Taper:	The amount and direction of the taper to face.

TEXT PROGRAM GENERATED BY CYCLE B USING INCREMENTAL COORDINATES WITH A TAPER AND LEFT CUTTERCOMP

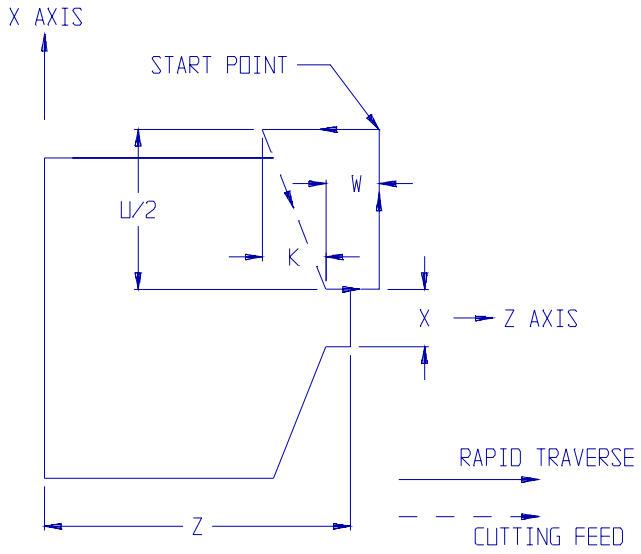
```
G0  X##.####  Z##.####
 │      │         │
 │      │         └──────Z Start Point
 │      └────────────────X Start Point
 └───────────────────────Rapid
```

```
P140 = ##.####
          └──────Z Start Point

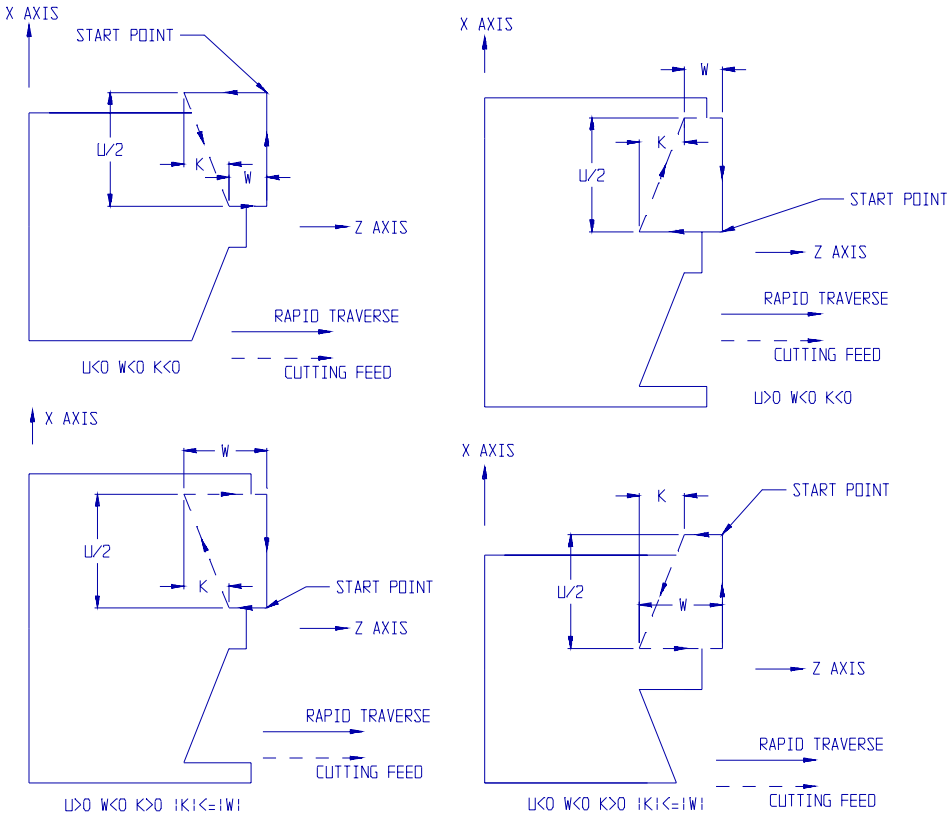
P139 = ##.####
          └──────X Start Point

S####
   └──────────Spindle Speed

G41 G94 K##.#### U##.#### W##.#### F.####
 |   |     |         |         |      └──Feedrate
 |   |     |         |         └─────────Length
 |   |     |         └───────────────────Depth of Cut
 |   |     └─────────────────────────────Taper
 |   └───────────────────────────────────Cycle B (Single
 |                                        Facing)
 └─────────────────────────────────────CutterComp Left or G42
                                        Right or G40 Off
```



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.

7.15 Multiple Thread Cutting Cycle

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

Multiple Threading Set-Up

```
 1. [Crest]                          2.  [Straight]
 3. Start Point       X[       ]d 3a. Z[        ]
 4. Crest Diameter    X[       ]d
 5. End Point         Z[       ]

 7. Lead              F[       ]
 8. Start Angle       Q[       ]
 9. Chamfer Length     [       ]
10. Chamfer Angle      [       ]
11. First Cut Amount  D[       ]r
12. Tool Nose Angle   A[       ]
13. Minimum Cut        [       ]r
14. Finish Passes      [  ]
15. Cutting Method  [Constant Amount/1 Edge]
```

- OR-

Multiple Threading Set-Up

```
 1. [Crest]                          2.  [Tapered]
 3. Start Point       X[       ]d 3a. Z[       ]
 4. Crest Diameter    X[       ]d
 5. End Point         Z[       ]
 6. Taper             I[       ]r
 7. Lead              F[       ]
 8. Start Angle       Q[       ]
 9. Chamfer Length     [       ]
10. Chamfer Angle      [       ]
11. First Cut Amount  D[       ]r
12. Tool Nose Angle   A[       ]
13. Minimum Cut        [       ]r
14. Finish Passes      [  ]
15. Cutting Method  [Constant Amount/Both Edges]
```

Multiple Threading Set-Up

```
 1. [Root and Height]                    2.   [Straight]
 3. Start Point       X[       ]d      3a.  Z[        ]
4a. Root Diameter X[1      ]d   4b. Height  K[        ]
 5. End Point         Z[       ]

 7. Lead              F[       ]
 8. Start Angle       Q[       ]
 9. Chamfer Length     [       ]
10. Chamfer Angle      [       ]
11. First Cut Amount  D[       ]r
12. Tool Nose Angle   A[       ]
13. Minimum Cut        [       ]r
14. Finish Passes      [  ]
15. Cutting Method  [Constant Depth/1 Edge]
```

1. 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.)
2. Straight or Tapered:	If Straight is selected no taper is entered. If Taper is selected then a taper must be entered.
3. (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.
4. (X(Crest Diameter:	The major diameter for external threads or the minor diameter for internal threads.
4a. (X) Root Diameter:	The minor diameter for external threads or the major diameter for internal threads.
4b. (K) Height:	Height of the thread specified as a radius value.
5. (Z) End Point:	The end of the thread.
6. (I) Taper:	The amount and direction of the taper to turn. (This is a radius value.)
7. (F) Lead:	Lead of the thread.

8. (Q) Start Angle:	Shift angle of the thread start angle. Used for multiple threads, i.e. shift 180°, 120° etc. 0° thru 360° are allowed.
9. Chamfer Length:	Length of the chamfer on the end of the thread.
10. 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.
11. (D) First Cut Amount:	Cutting depth of the first cut. (This is a radius value.)
12. Tool Nose Angle:	The read angle of the tool nose.
13. 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).
14. 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.
15. Cutting Method:	Constant Amount/1 Edge Cutting amount constant, single edge cutting. Constant Amount/Both Edges cutting amount constant, both edges cutting. Constant Depth/Both Edges cutting depth constant, both edges cutting.

CODE GENERATED FOR A CREST (SHARP V), STRAIGHT THREAD USING THE MULTIPLE THREADING CYCLE

```
P103 = ##.####
          └──────────Chamfer Length

P104 = ###.###
          └──────────Chamfer Angle

P130 = ##.####
          └──────────Minimum Cut

P131 = ##
        └─────────Finish Passes
P91 = ##.####
          │
          └──────────Crest Diameter
```

```
P101 = ##.####
          │
          └──────────Lead

P108 = ##.####
          │
          └──────────Tool Nose Angle
```

```
P92 = P262
P262 = 0
G65 G42 G0 XC0 ZC0 R2 AB[90-108/2]
G65 X0 Z0
P124 = P209*2 - P261
G65 G40 R2 AB[90 + 108/2]
P262 = P92
P106 = P101 / [Z*TAN[P108/2]] - P124
```

Calculates the thread height and root diameter from the tool nose angle, tool nose radius, crest diameter and the lead.

```
G0 X##.####   Z##.####
     │            │
     │            └──────Z Start Point
     └───────────────────X Start Point

P140 = ##.####
          │
          └──────────Z Clearance

P139 = ##.####
          │
          └──────────X Clearance

P124 = P91 - 2 * P106
If P139 < P91 Then P124 = P91 + 2 * P106
```

More Calculations

```
G76 X[P124] Z##.#### Q###.### F.#### K[P106] A###.### D##.#### P#
 │   │      │        │        │      │       │        │        │
 │   │      │        │        │      │       │        │        └── Cutting Method
 │   │      │        │        │      │       │        │             1=Constant Amount/Single Edge
 │   │      │        │        │      │       │        │             2=Constant Amount/Both Edges
 │   │      │        │        │      │       │        │             3=Constant Depth/Single Edge
 │   │      │        │        │      │       │        │             4=Constant Depth/Both Edges
 │   │      │        │        │      │       │        └──────────── Cutting Depth for the 1st Cut
 │   │      │        │        │      │       └───────────────────── Tool Nose Angle
 │   │      │        │        │      └───────────────────────────── Calculated Thread Height
 │   │      │        │        └──────────────────────────────────── Lead
 │   │      │        └───────────────────────────────────────────── Shift Angle (For Multiple Threads
 │   │      │                                                       i.e. 180°, 120° etc.)
 │   │      └──────────────────────────────────────────────────────End of the thread
 │   └─────────────────────────────────────────────────────────────Calculated Root Diameter for the
 │                                                                 thread
 └────────────────────────────────────────────────────────────────Multiple Threading Cycle
```

CODE GENERATED FROM A ROOT AND HEIGHT, STRAIGHT THREAD USING THE MULTIPLE THREADING CYCLE

P103 = ##.####
- Chamfer Length

P104 = ###.###
- Chamfer Angle

P130 = ##.####
- Finish Allowance

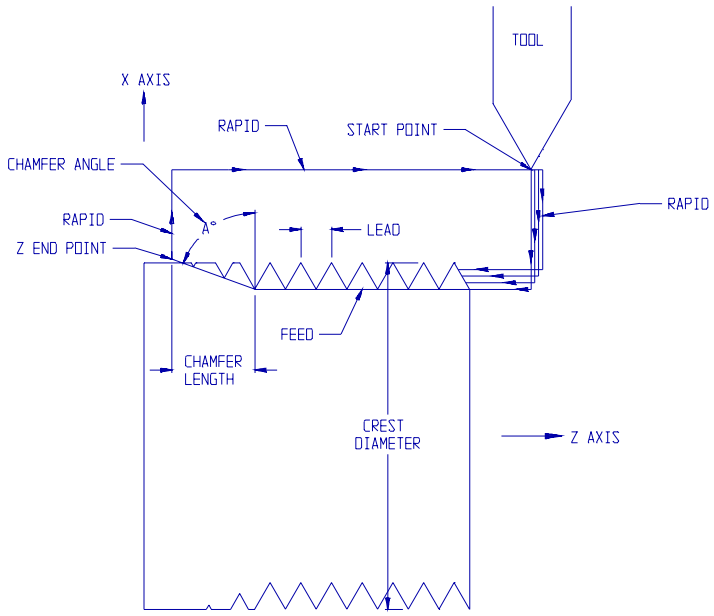
P131 = ##
- Finish Passes

G0 X##.#### Z##.####
- Z Start Point
- X Start Point

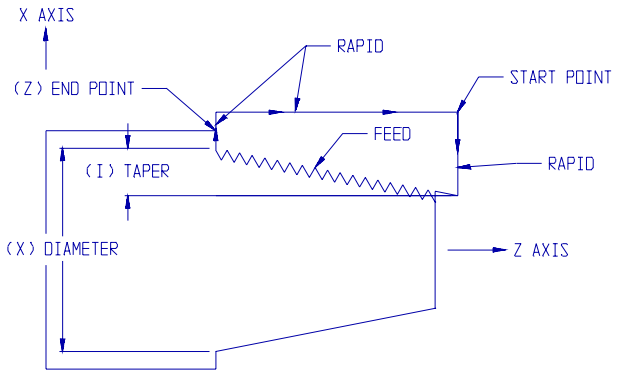
P140 = ##.####
- Z Clearance

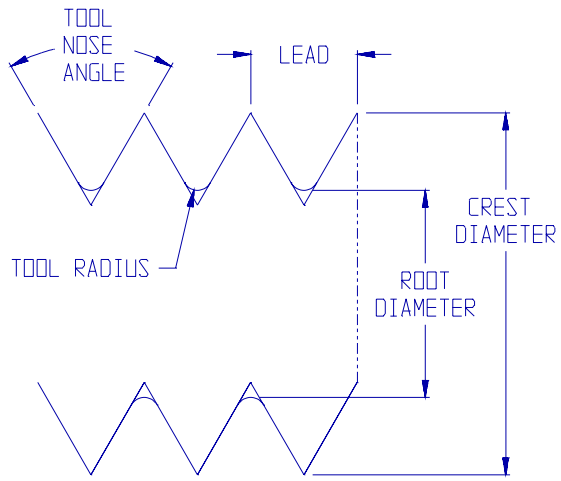
P139 = ##.####
- X Clearance

G76 X##.#### Z##.#### Q###.### F.#### K##.#### A###.### D##.#### P#
- Cutting Method
  1=Constant Amount/Single Edge
  2=Constant Amount/Both Edges
  3=Constant Depth/Single Edge
  4=Constant Depth/Both Edges
- Cutting Depth for the 1st Cut
- Tool Nose Angle
- Calculated Thread Height
- Lead
- Shift Angle (For Multiple Threads i.e. 180°, 120° etc.)
- End of the thread
- Calculated Root Diameter for the thread
- Multiple Threading Cycle



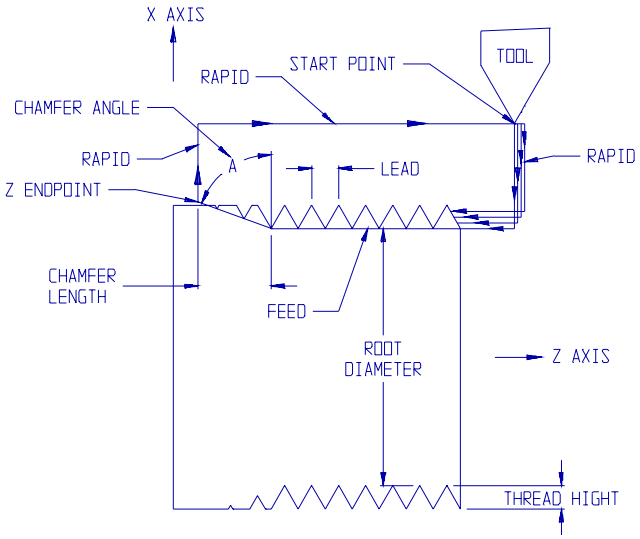
Inside or outside threads are determined by the start point relative to the diameter. The direction of the taper is determined by the sign of I. The tapers are the same as those shown in the cycle A.



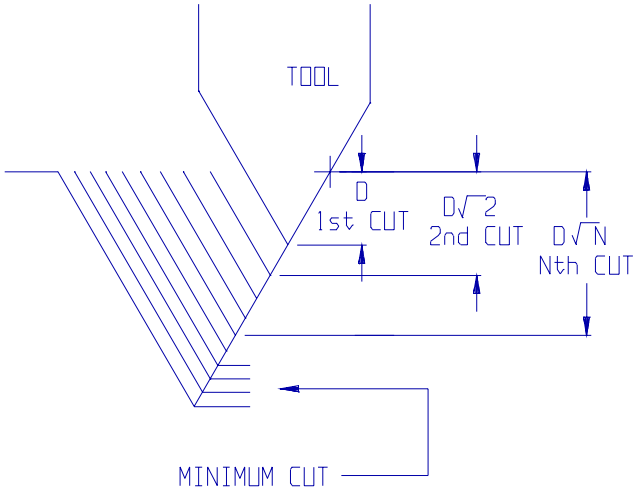


When the crest option is selected, the root diameter is calculated from the crest diameters, the tool nose angle, and the tool nose radius.

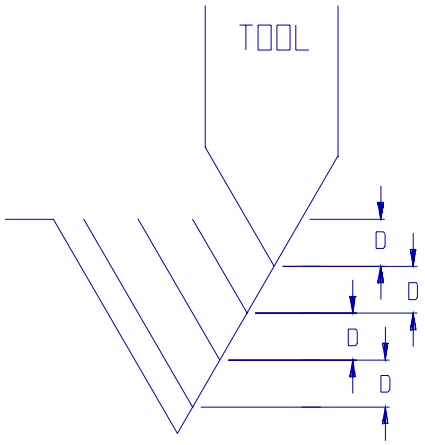
When the root and height option is selected the crest diameter is calculated from the root diameter and the thread height.



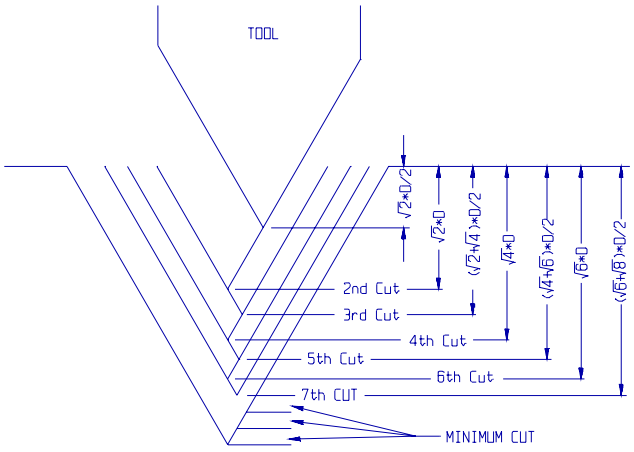
The four cutting methods are shown below:



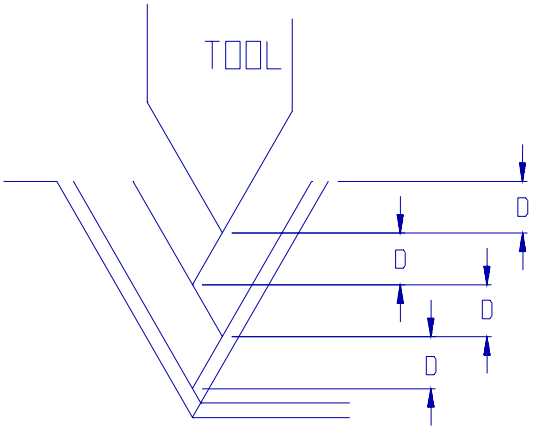
Constant Depth/1 Edge



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.

7.16 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.

```
 Threading Cycle 1

1.    [ABSOLUTE]
      [STRAIGHT CUT]
2. Start Point   X[        ]d
                 Z[        ]

3. Diameter      X[        ]d
4. End Point     Z[        ]

5. Lead          F[        ]
7. Start Angle   Q[        ]
8. Chamfer Length [        ]
9. Chamfer Angle  [        ]
```

-OR-

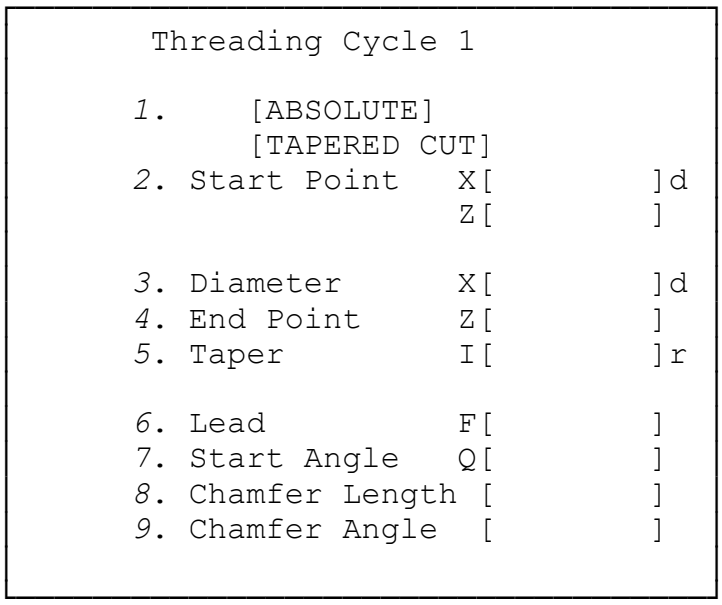
```
     Threading Cycle 1

 1.   [INCREMENTAL ]
      [STRAIGHT CUT]
 2. Start Point    X[        ]d
                   Z[        ]

3a. Depth          U[        ]d
4a. Length         W[        ]

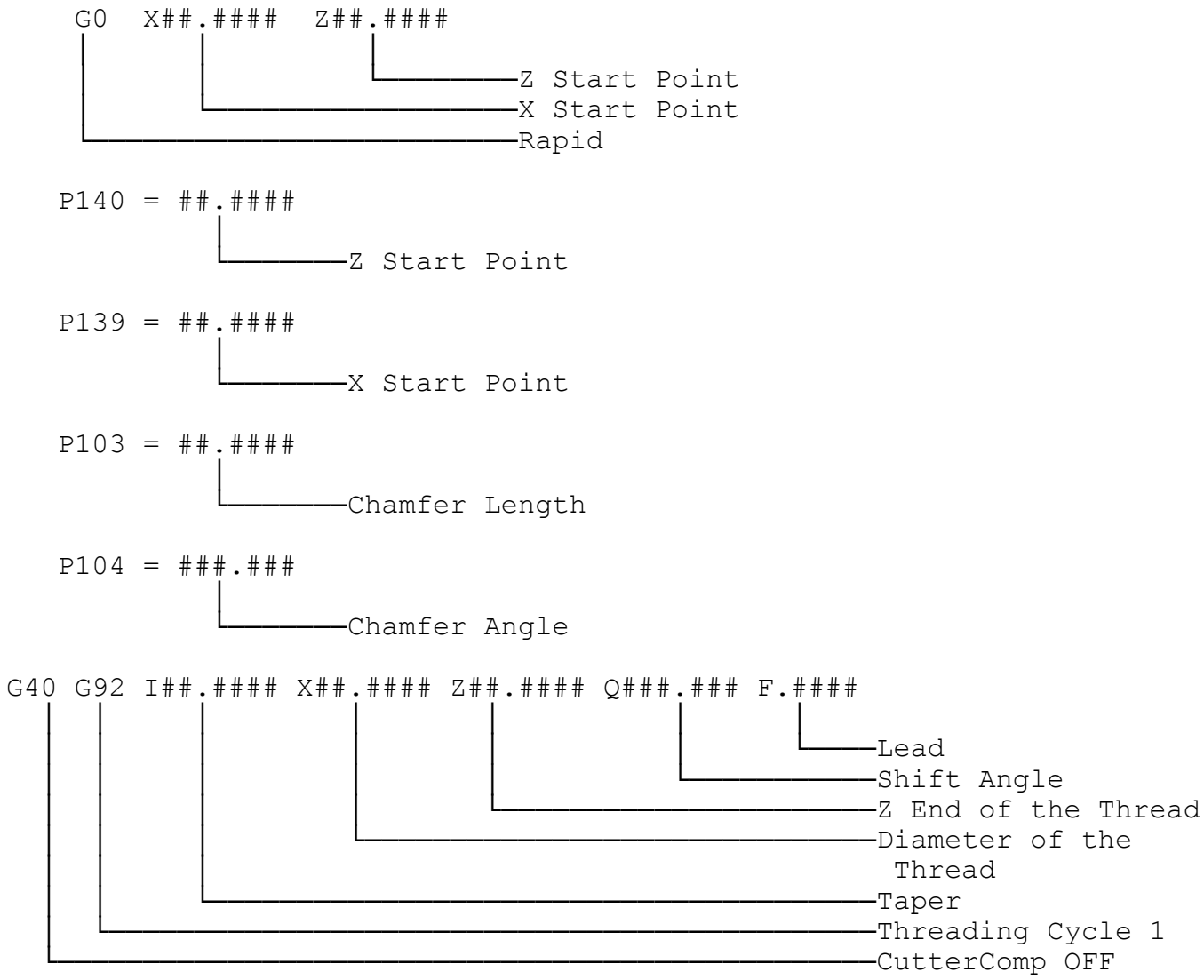
 6. Lead           F[        ]
 7. Start Angle    Q[        ]
 8. Chamfer Length  [        ]
 9. Chamfer Angle   [        ]
```

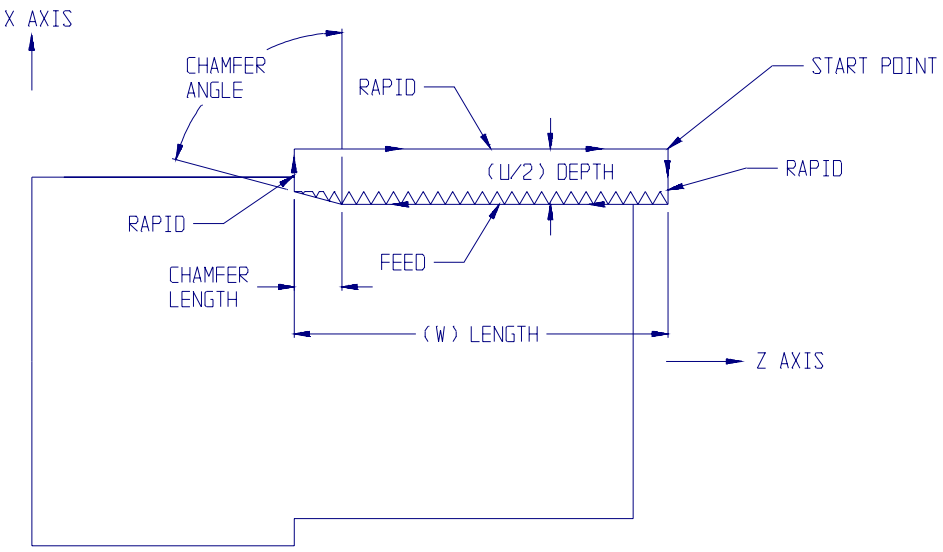
-OR-



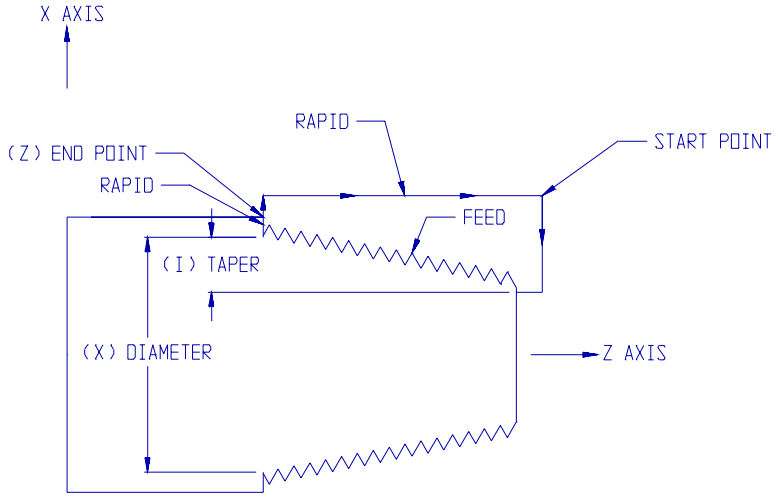
1. ABSOLUTE or INCREMENTAL:	Toggles between absolute dimensions (diameter, end point) and incremental dimensions (depth and length).
2. (X,Z) Start Point:	The start point of Cycle 1. X and Z return to the start point at the end of the cycle at a rapid feedrate.
3. (X) Diameter:	The root diameter to thread.
3a. (U) Depth:	The incremental X distance and direction from the start point to the diameter to thread.
4. (Z) End Point:	The end point of the thread.
4a. (W) Length:	The length and direction of the thread in Z from the start point.
5. (I) Taper:	The amount and direction of the taper to thread.
6. (F) Lead:	Lead of the thread.
7. (Q) Start Angle:	Shift angle of the thread start angle. Used for multiple threads, i.e. shift 180°, 120° etc. 0° thru 360° are allowed.
8. Chamfer Length:	Length of the chamfer on the end of the thread.
9. 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.

CODE GENERATED FROM THE CYCLE 1 SCREEN
USING ABSOLUTE DIMENSIONS WITH A TAPER

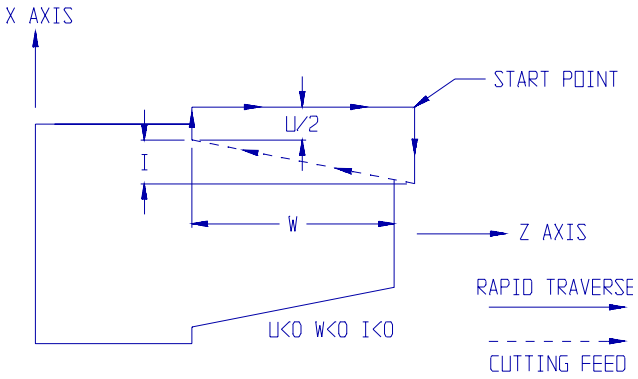


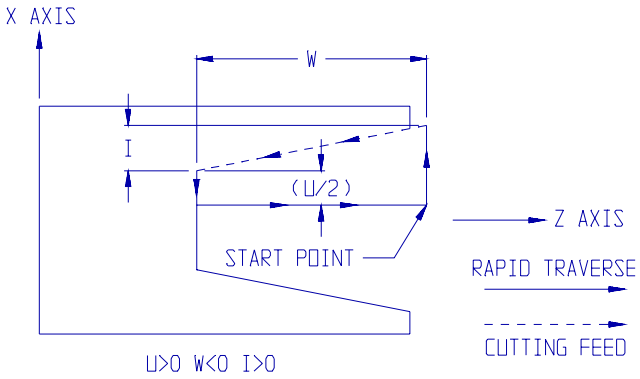


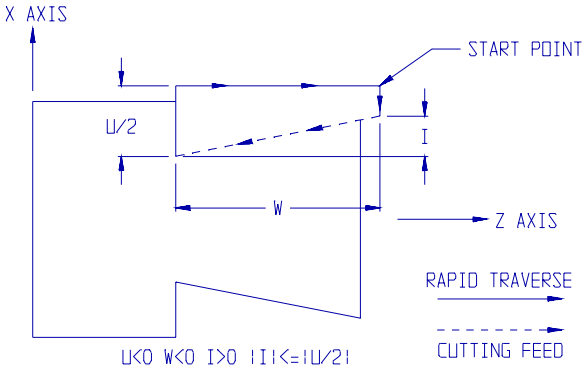
Straight Cutting Using Cycle 1 With Incremental Dimensions



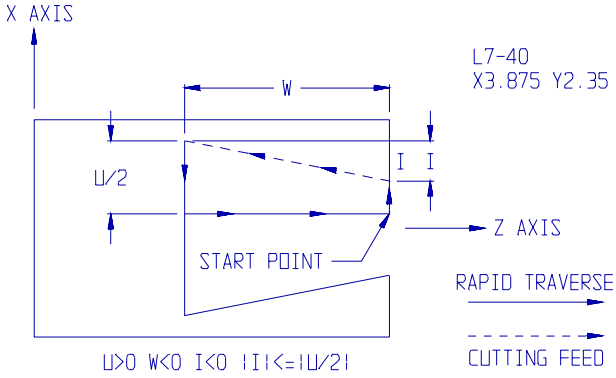
Tapered Cut Using Cycle 1 with Incremental Dimensions







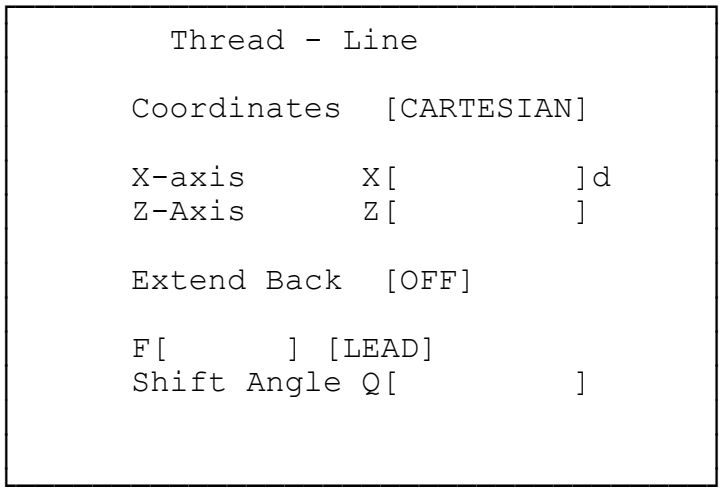
When using Cycle 1 with Incremental Dimensions, the signs of the depth, length and taper indicate the directions shown in the above figures.



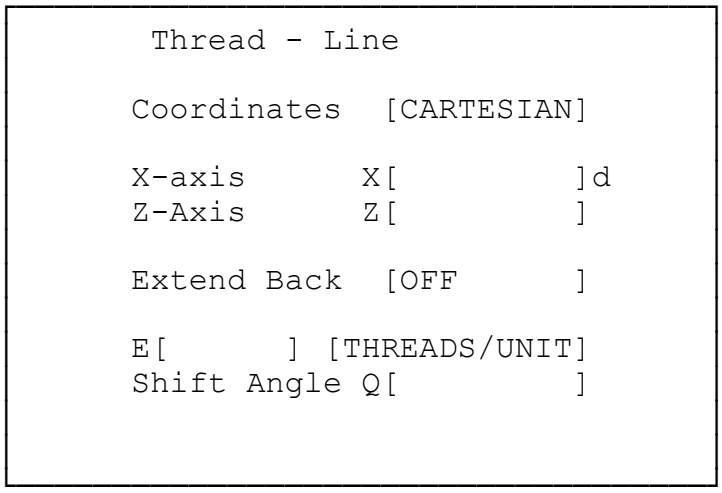
7.17 Thread Lines

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

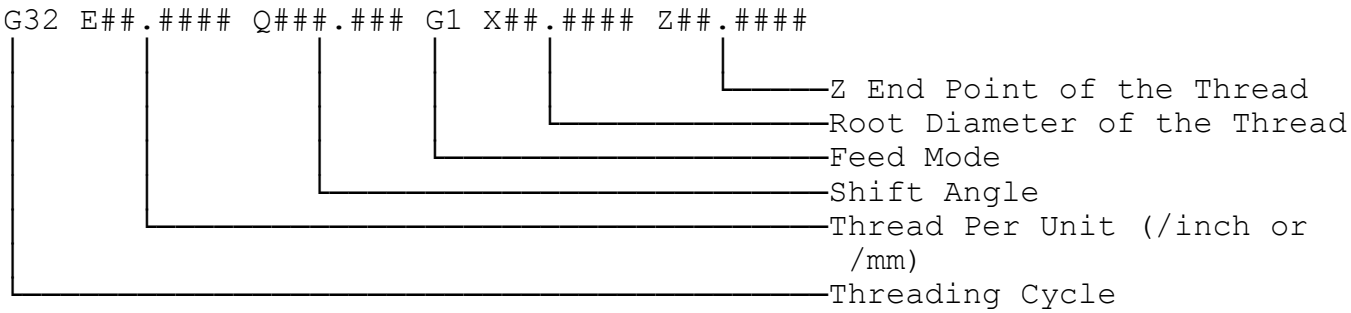
The line defined by the thread line is identical to the lines used in turning or facing with the additional threading information for the line.



-OR-



CODE GENERATED FOR CARTESIAN THREAD-LINE
USING THREADS PER UNIT



7.18 Drilling Cycles

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 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.

7.18.1 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.

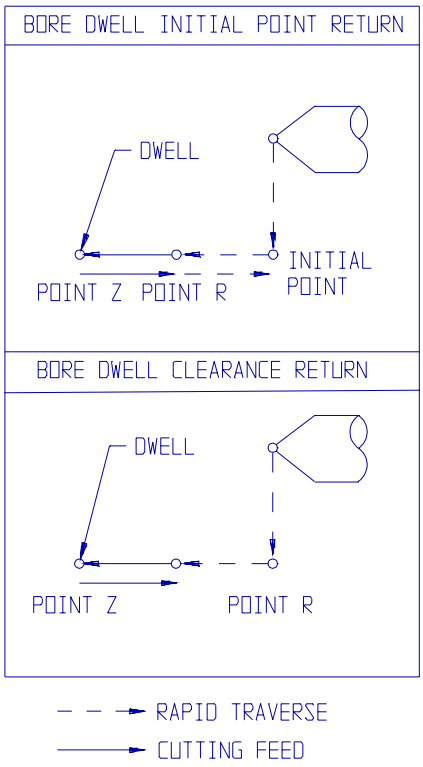
```
Drill
[Drill Cycle]

1. Pierce Feedrate   F[     ]
2. Return Point       [CLEARANCE]
3. Clearance         R[         ]
4. Final Z Depth     Z[         ]
```

1. Pierce Feedrate: Z feedrate for drilling the hole.
2. 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.

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

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

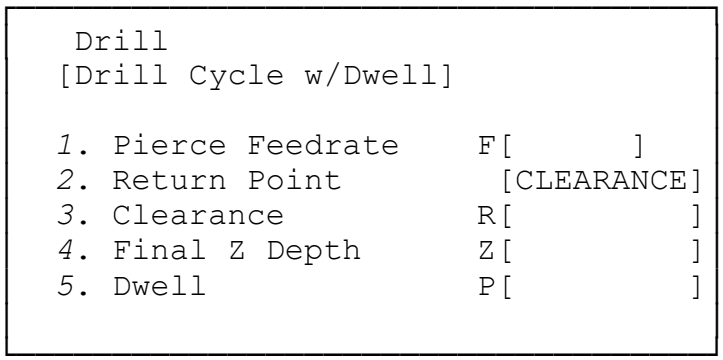


CODE GENERATED BY THE DRILL SCREEN

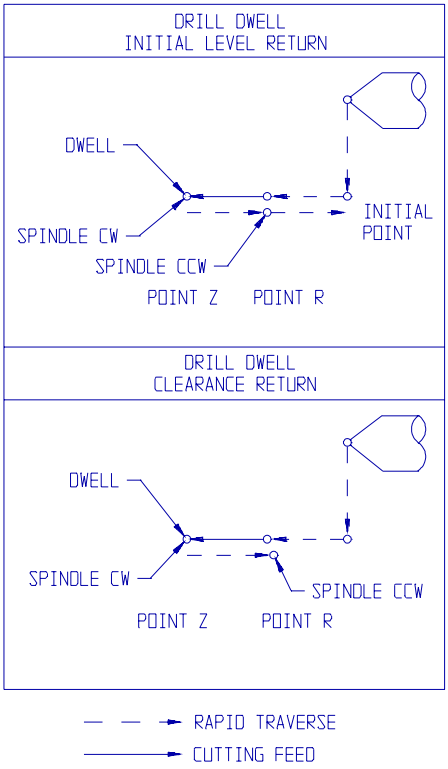
```
G81 R##.#### Z##.#### F.#### G39
 |      |      |      |      |
 |      |      |      |      |____Clearance-Return or G38 for
 |      |      |      |             Initial Point Return
 |      |      |      |___________Feedrate
 |      |      |__________________Z Depth of the Hole
 |      |_________________________Z Clearance Plane
 |________________________________Drill Cycle
```

7.18.2 Drill Dwell Cycle

The Drill Dwell Cycle is identical to the Drill Cycle with the addition of a Dwell at the bottom of the hole.



1. Pierce Feedrate:	Z feedrate for drilling the hole.
2. 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.
3. Clearance:	Z rapids to the clearance at the beginning of the drill cycle.
4. Final Z Depth:	The depth to drill the hole.
5. Dwell:	Time in seconds to dwell at the bottom of the hole.

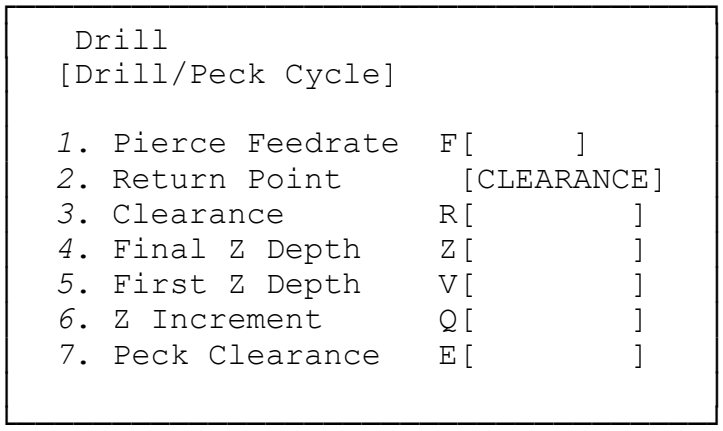


CODE GENERATED BY THE DRILL/DWELL SCREEN

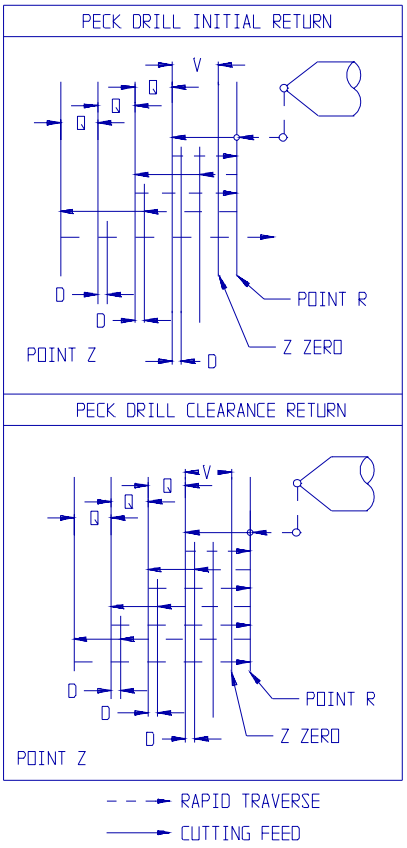
```
G82 P##.#### R##.#### Z##.#### F.#### G39
 |      |         |         |       |    |
 |      |         |         |       |    Return to Clearance Plane or
 |      |         |         |       |     G38 Return to Initial Level
 |      |         |         |       Feedrate
 |      |         |         Z Depth of the Hole
 |      |         Z Clearance
 |      Dwell Time in Seconds
 Drill/Dwell Cycle
```

7.18.3 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.



1. Pierce Feedrate:	Z Feedrate for drilling the hole.
2. 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.
3. Clearance:	Z rapids back to the clearance at the beginning of the drill cycle.
4. Final Z Depth:	The depth to drill the hole to.
5. First Z Depth:	First peck depth.
6. Z Increment:	Unsigned amount to drill in each successive peck.
7. Peck Clearance:	Unsigned distance to rapid to above the previous depth.



CODE GENERATED FROM THE DRILL PECK SCREEN

```
G83 Q##.#### E##.#### V##.#### R##.#### Z##.#### F.#### G39
 |        |         |         |         |         |      |
 |        |         |         |         |         |      Return to Clearance
 |        |         |         |         |         Feedrate
 |        |         |         |         Final Z Depth
 |        |         |         Z Clearance Plane
 |        |         1st Z Depth
 |        Peck Clearance
 Z Increment
Drill Peck/Cycle
```

7.18.4 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.

```
Drill
[Face Grooving]

 1. Pierce Feedrate    F[      ]
 2. Return Point        [CLEARANCE]
 3. Clearance          R[        ]
 4. Final Z Depth      Z[        ]
 5. First Z Depth      V[        ]
 6. Z Increment        Q[        ]
 7. Peck Clearance     E[        ]
 8. PeckUp Increment   U[        ]
 9. Z Finish Stock      [        ]d
10. Finial Diameter    X[        ]r
11. X Increment        I[        ]r
12. Relief             D[        ]r
13. X Finish Stock      [        ]
```

1. Pierce Feedrate:	Z feedrate for grooving the slot.
2. 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.
3. Clearance:	Z rapids to the clearance at the beginning of the grooving cycle.
4. Final Z Depth:	The depth to slot the groove to.
5. First Z Depth:	First peck depth.
6. Z Increment:	Unsigned amount to groove in each successive peck.
7. Peck Clearance:	Unsigned distance to rapid to above the previous depth.
8. Peckup Increment:	Unsigned distance to rapid up each peck.

9.	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.)*
10.	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.
11.	X Increment:	The unsigned amount to step over along the face. (This is a radius value.)
12.	Relief:	Amount to back X away at the bottom of the groove before retracting X to the Z clearance.
13.	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.)

Note 1: If both X and Z finish stocks are 1, then no finish pass is made. If either X or Z stock is 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.

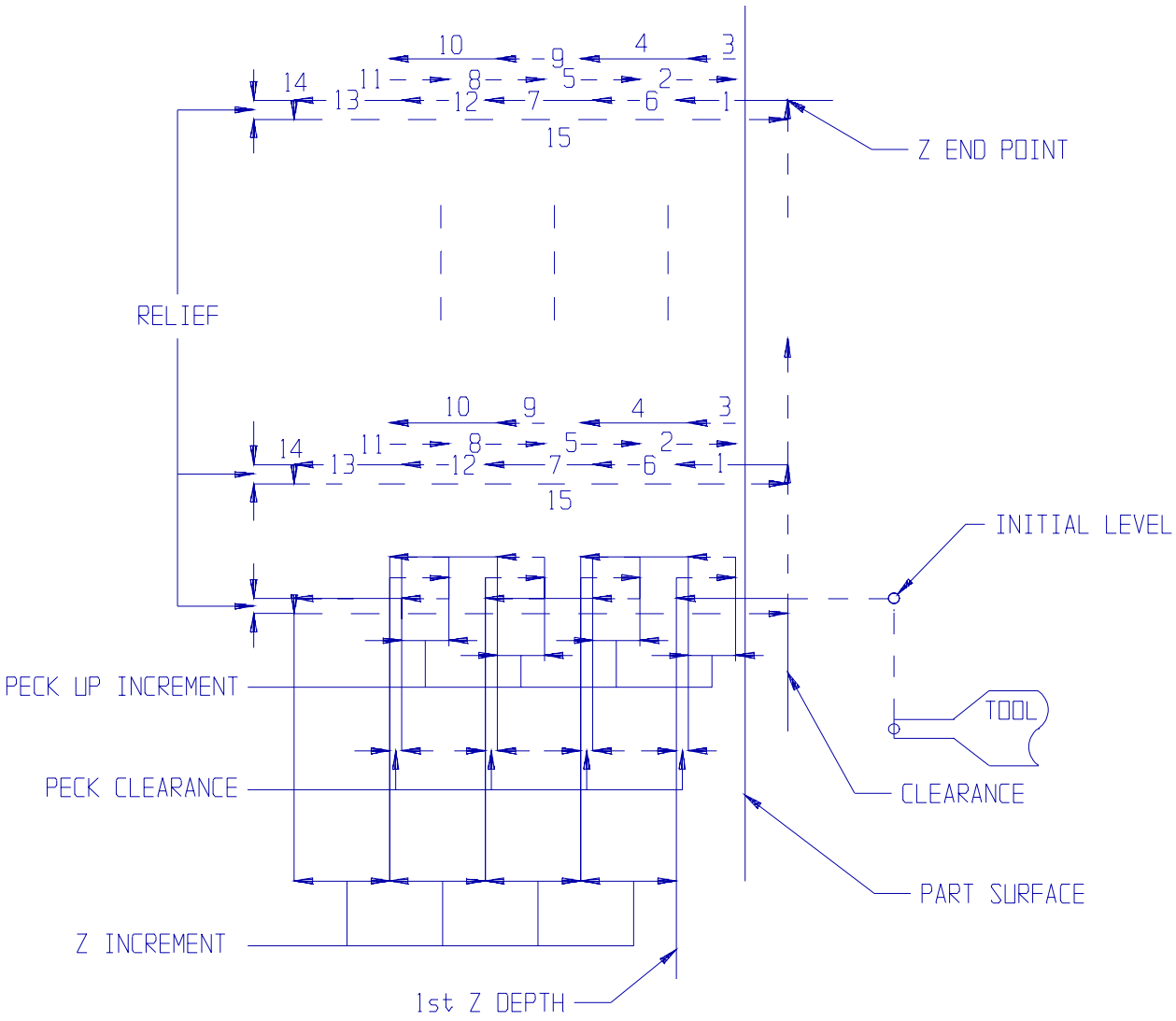
E. Remove the X finish stock at the starting diameter side of the groove to the bottom of the groove.

F. Remove the Z stock along the bottom of the groove to the final diameter side.

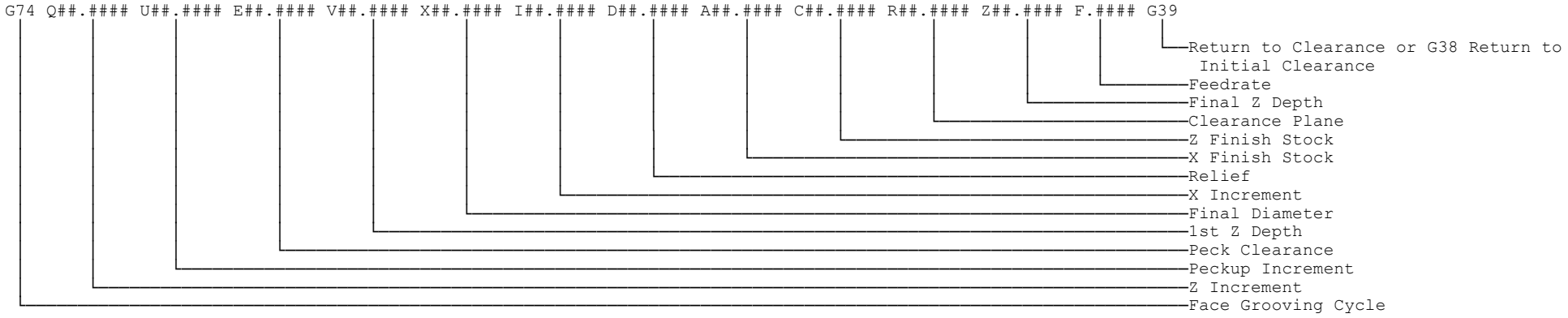
G. Retract out of the groove.

Note 2: No cutter compensation is used when using the Face Grooving Cycle. The dimensions of the groove must account for the tool width.

Note 3: 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.


10
9
4
3
14
11
8
5
2
13
12
7
6
1
15
Z END POINT
RELIEF
INITIAL LEVEL
PECK UP INCREMENT
TOOL
PECK CLEARANCE
CLEARANCE
PART SURFACE
Z INCREMENT
1st Z DEPTH

CODE GENERATED FROM THE FACE GROOVING CYCLE



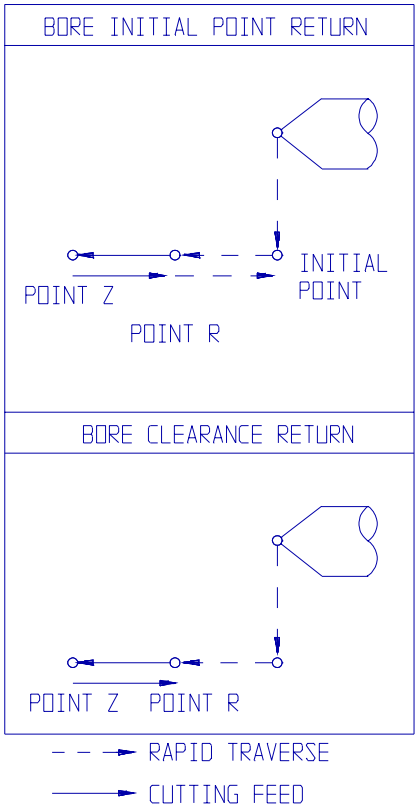
7.18.5 Bore Cycle

The Bore Cycle Feeds to the Z depth and feeds back to the clearance.

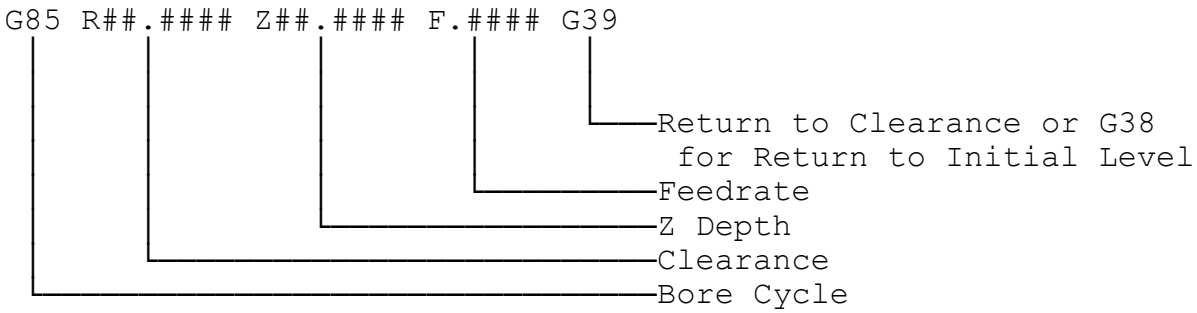
```
Drill
[Bore Cycle]

1. Pierce Feedrate    F[      ]
2. Return Point        [CLEARANCE]
3. Clearance          R[        ]
4. Final Z Depth      Z[        ]
```

1. Pierce Feedrate: Feedrate for drilling the hole.
2. 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.
3. Clearance: Z rapids to the clearance at the beginning of the drill cycle.
4. Final Z Depth: The depth to drill the hole to.



CODE GENERATED FROM THE BORE CYCLE SCREEN



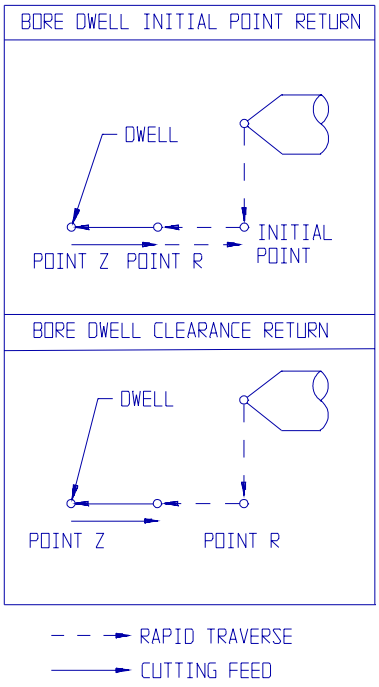
7.18.6 Bore/Dwell Cycle

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

```
Drill
[Bore Cycle W/Dwell]

1. Pierce Feedrate  F[     ]
2. Return Point      [CLEARANCE]
3. Clearance        R[         ]
4. Final Z Depth    Z[         ]
5. Dwell            P[         ]
```

1. Pierce Feedrate: Z feedrate for drilling the hole.
2. 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.
3. Clearance: Z rapids to the clearance at the beginning of the drill cycle.
4. Final Z Depth: The hole will be drilled to this depth.
5. Dwell: Time in seconds to dwell at the bottom of the hole.



CODE GENERATED FROM THE BORE/DWELL CYCLE

```
G89 P##.#### R##.#### Z##.#### F.#### G39
                                      |
                                      Return to Clearance or
                                       G38 for Return to
                                       Initial Level
                                 Feedrate
                            Z Depth
                       Clearance
                  Dwell Time in Seconds
             Bore Dwell Cycle
```

7.18.7 The Right Tap Cycle

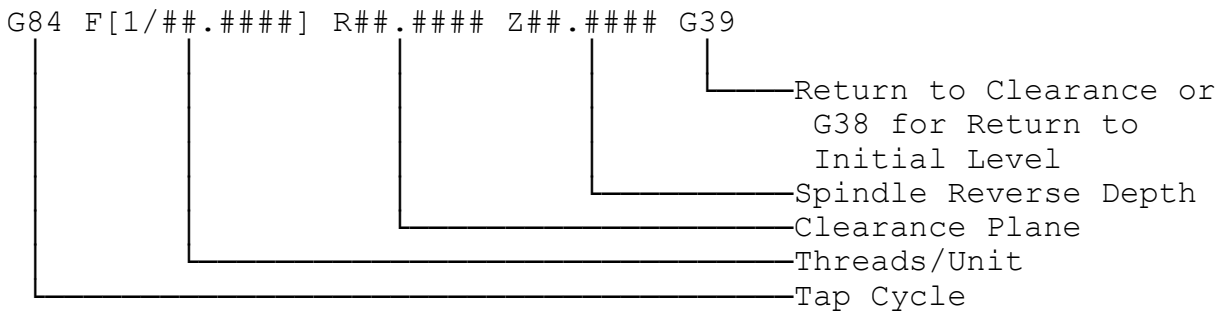
The Right Tap Cycle always runs in FEED/REV and REVS/MINUTE. The spindle is turned on CW at the start of the cycle and reverses direction (CCW) at the spindle reverse depth.

```
┌──────────────────────────────────────────┐
│  Drill                                   │
│  [Right Tap Cycle]                       │
│                                          │
│  1. Spindle on CW RPM      S[        ]   │
│  2. Threads per unit       P[        ]   │
│  3. Return Point            [CLEARANCE]  │
│  4. Clearance              R[        ]   │
│  5. Spindle Reverse Depth  Z[        ]   │
│                                          │
└──────────────────────────────────────────┘
```

1. 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.*
2. Threads per Unit:	The lead of the tap.
3. 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.
4. Clearance:	Z rapids to the clearance at the beginning of the drill cycle.
5. Spindle Reverse Depth:	The spindle reverse direction at this depth. It may coast some and cause the tap to go a little deeper.

CODE GENERATED FROM THE RIGHT TAP CYCLE SCREEN

```
G97 G99 M3 S####.#
 |   |   |    |
 |   |   |    └──────Spindle Speed in RPM's
 |   |   └───────────Spindle Clockwise
 |   └───────────────FEED/REV
 └───────────────────REVS/MINUTE
```



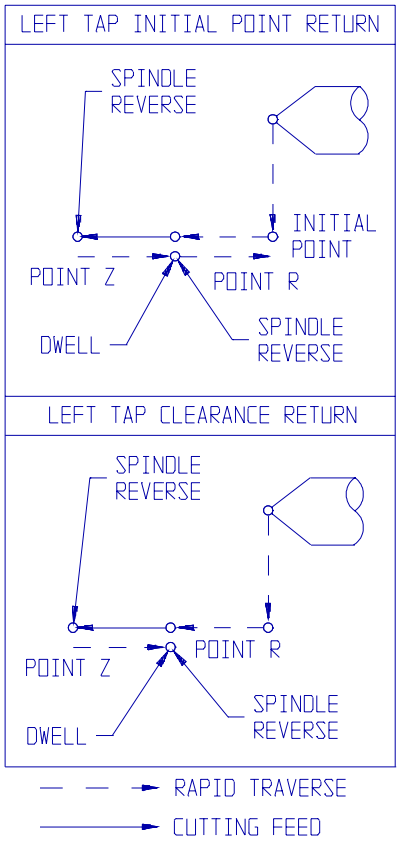
7.18.8 The Left Tap Cycle

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.

```
Drill
[Left Tap Cycle]

1. Spindle on CCW RPM    S[         ]
2. Threads per Unit      P[         ]
3. Return Point           [CLEARANCE]
4. Clearance             R[         ]
5. Spindle Reverse Depth Z[         ]
```

1. 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.*
2. Threads per Unit:	Threads per inch or threads per millimeter.
3. 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.
4. Clearance:	Z rapids to the clearance at the beginning of the drill cycle.
5. Spindle Reverse Depth:	The spindle reverse direction at this depth. It may coast some and cause the tap to go a little deeper.



CODE GENERATED FROM THE LEFT TAP CYCLE SCREEN

```
G97 G99 M4 S####.#
          |
          |_____Spindle Speed
       |_____Spindle on CCW
    |_____FEED/REV
 |_____REVS/MINUTE
```

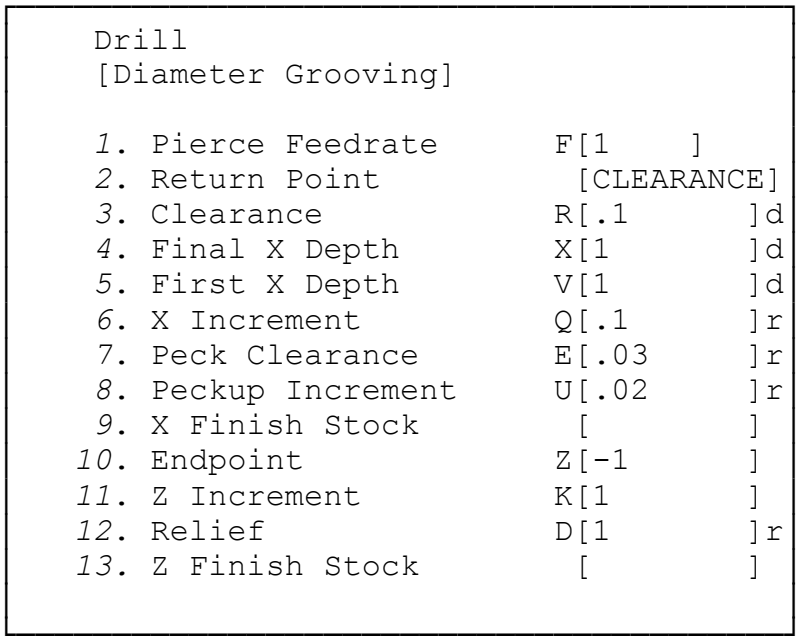
```
G84 F[1/##.####] R##.#### Z##.#### G39
                                  |
                                  |_____Return to Clearance or
                                        Return to Initial
                                        Level
                             |_____Spindle Reverse Depth
                        |_____Clearance Plane
                   |_____Threads/Unit
              |_____Tap Cycle
```

7.18.9 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.



1. Pierce Feedrate:	X Feedrate for grooving the slot.
2. 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.
3. Clearance:	X rapids to the clearance at the beginning of the grooving cycle. (This is a diameter value.)
4. Final X Depth:	The diameter to groove the slot to.
5. First X Depth:	First peck diameter.

6.	X Increment:	Unsigned diameter to groove in each successive peck. (This is a radius value.)
7.	Peck Clearance:	Unsigned distance to rapid up after each peck. (This is a radius value.)
8.	PeckUp Increment:	Unsigned distance to rapid up after each peck. (This is a radius value.)
9.	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.)*
10.	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.
11.	Z Increment:	The unsigned amount to step over.
12.	Relief:	Amount to back Z away at the bottom of the groove before retracting to the X clearance.
13.	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.

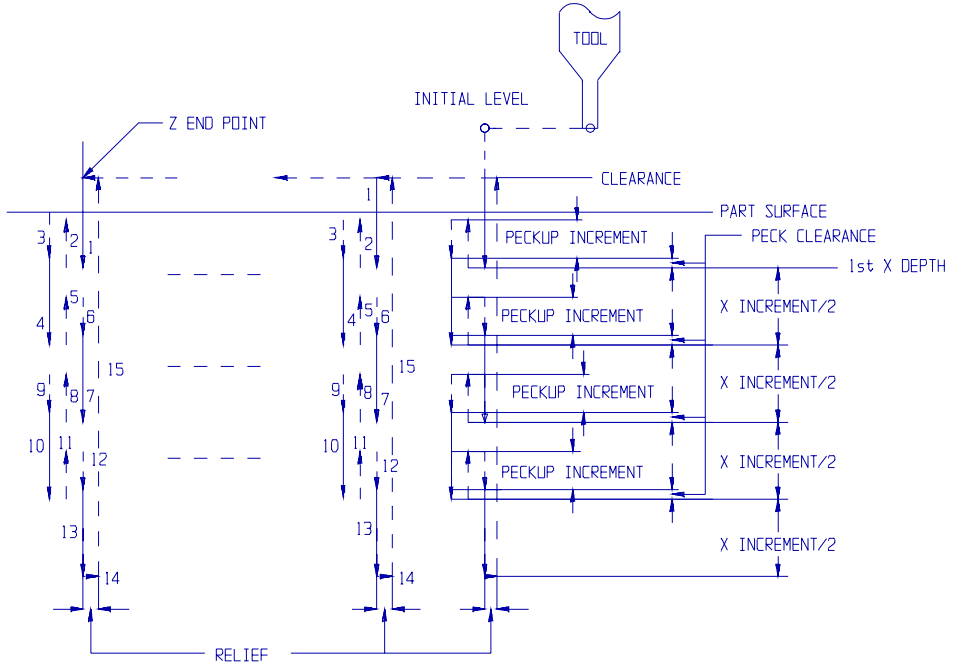
E. Remove the Z finish stock at the starting edge of the groove to the final X depth.

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: No cutter compensation is used in the Diameter Grooving Cycle. The dimensions of the groove must account for the tool width.

Note 3: A position move should be made prior to the Diameter Grooving Cycle so that Z is positioned to the start of the groove.



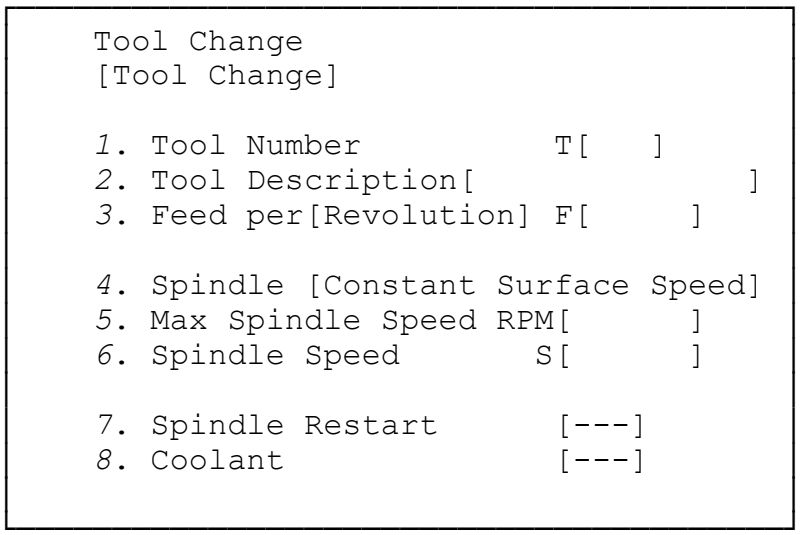
CODE GENERATED BY THE DIAMETER GROOVING CYCLE

```
G75 Q##.#### U##.#### E##.#### V##.#### X##.#### K##.#### D##.#### A##.#### C##.#### R##.#### Z##.#### F.#### G39
```

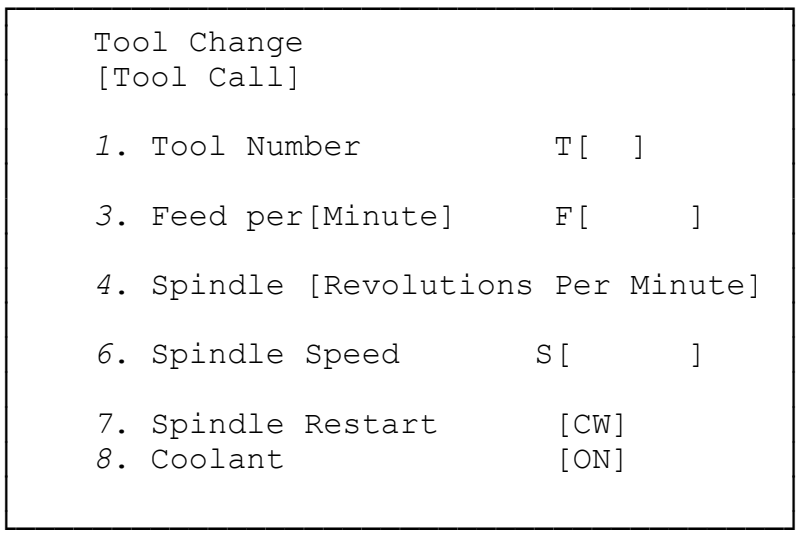
- G39 — Return to Clearance or G38 Return to Initial Level
- F.#### — Feedrate
- Z##.#### — End Point
- R##.#### — Clearance (X)
- C##.#### — Z Finish Stock
- A##.#### — X Finish Stock
- D##.#### — Relief
- K##.#### — Z Increment
- X##.#### — Final X Depth
- V##.#### — First X Depth
- E##.#### — Peck Clearance
- U##.#### — PeckUp Increment
- Q##.#### — X Increment
- G75 — Diameter Grooving Cycle

7.19 Tool Change Screen

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. This tool change sequence is flexible for use with a variety of tool changers.



-OR-



-OR-

```
Tool Change
[Tool Change]

1. Tool Number          T[  ]
2. Tool Description[              ]
3. Feed per[MINUTE]     F[     ]

4. Spindle [Constant Surface Speed]
5. Max Spindle Speed RPM[      ]
6. Spindle Speed       S[      ]

7. Spindle Restart      [CCW]
8. Coolant              [---]
```

1. Tool Number:	The tool call will only load the tool offsets for the specified tool.
2. Tool Description:	The tool description is used to prompt the operator when the control is waiting for the tool change reset signal.
3. Feed Per:	Feed/Revolution or Feed/Minute.
4. Spindle:	Constant surface speed or revolution/minute.
5. Max Spindle Speed:	If constant surface speed is in effect a maximum spindle speed is used to clamp the spindle speed.
6. Spindle Speed:	The programmed spindle speed.
7. Spindle Restart:	Spindle restart CW or CCW.
8. Coolant:	Coolant ON.

CODE GENERATED FROM A TOOL CALL

```
T####
 |  |
 |  └──────Tool Offset
 └─────────Tool #

G99  F.####
 |     |
 |     └─────Feedrate (in inches/rev, mm/rev, Inches/Min or
 |            mm/Min
 └───────────Feed/Rev or G98 for Feed/Minute

G50  S####.#
 |     |
 |     └──────Max Spindle Speed
 └────────────Set Max RPM's
```

```
G96  S####.#
  |       |
  |       └──────Spindle Speed (in Feet/min, mm/Min or RPM's)
  └──────────────Constant Surface Speed or G97 for RPM's

M3
  |
  └────Spindle on CW or M4 for Spindle on CCW

M80  Coolant ON
```

CODE GENERATED FOR TOOL CHANGE

```
G40 G501 G550 G569 G80 M5 M9
 |   |    |    |    |  |  |
 |   |    |    |    |  |  └────Coolant OFF
 |   |    |    |    |  └───────Spindle OFF
 |   |    |    |    └──────────Cancel Canned Cycles
 |   |    |    └───────────────Rotation OFF
 |   |    └────────────────────Scaling OFF
 |   └─────────────────────────Mirror Image OFF
 └─────────────────────────────Tool Nose Compensation OFF

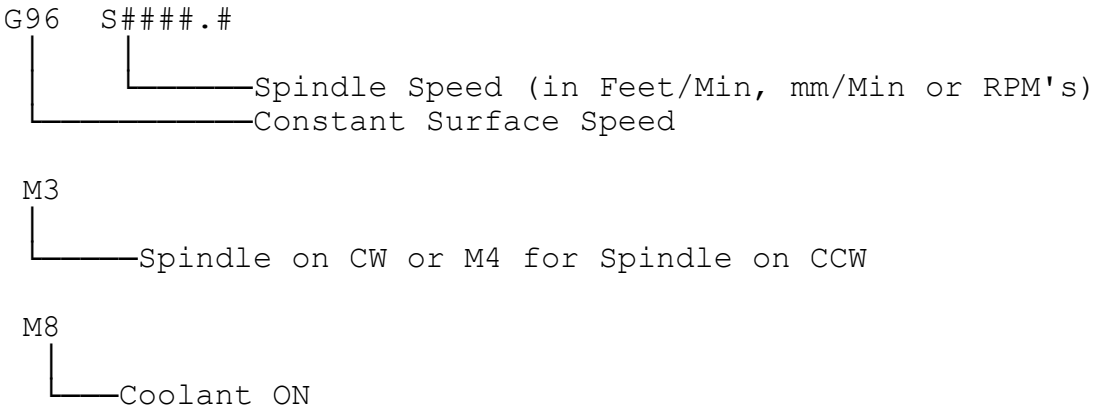
P268 = ##
       |
       └────Tool # (set the pending tool #)

M6  (...Tool Description ...)
 |             |
 |             └──────────────────Used to prompt the operator when
 |                                 waiting for the tool change
 |                                 reset
 └──────────────────────────────Tool Change

 T####
  | ||
  | |└──Tool Offset
  └────Tool#   (i.e. For Tool # 15, T1515

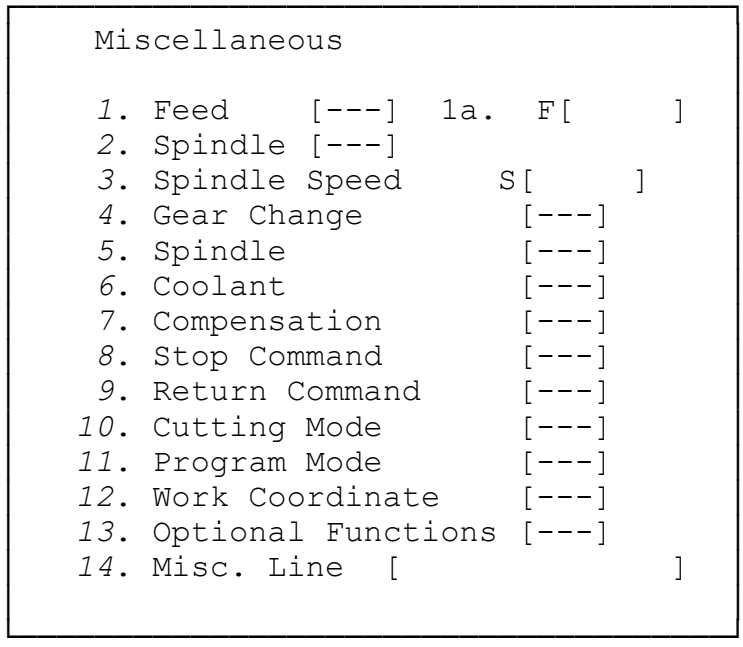
 G99  F.####
  |       |
  |       └──────Feedrate (in Inches/Rev, mm/Rev, Inches/Min or
  |               mm/Min)
  └──────────Feed/Rev or G98 for Feed/Minute

 G50  S####.#
  |       |
  |       └──────Max Spindle Speed
  └──────────Set Max RPM's
```



7.20 Miscellaneous Screen

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.

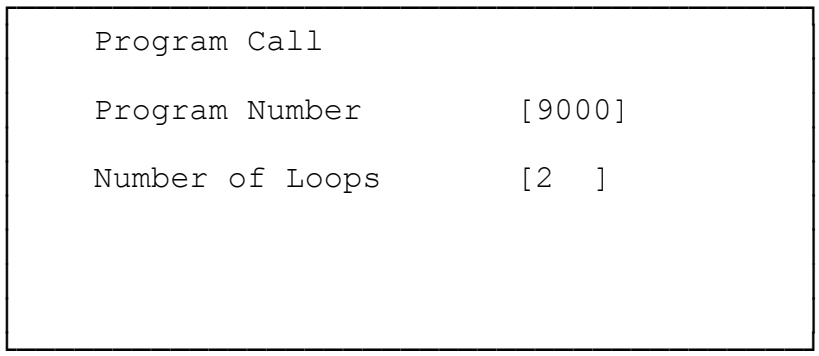


1.	Feed:	Per Rev or Per Minute (posts G99 or G98 respectively).
1a.	Feedrate:	In inches/Rev, mm/Rev, Inches/Min or mm/Min (posts F###.####)
2.	Spindle:	Constant surface speed or revolutions per minute (posts G96 or G99 respectively) (if constant surface speed is selected a maximum spindle speed can be entered)

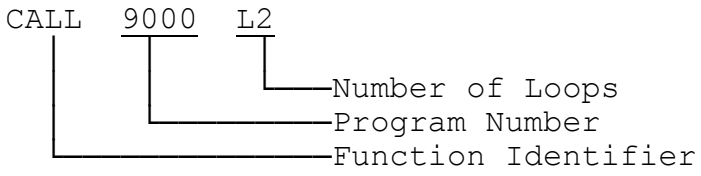
3. Spindle Speed:	In feet/Min, mm/Min or Revs/Min (posts S###.#)
4. Gear Change:	Low, Medium or High (posts M41, M42, or M93 respectively)
5. Spindle:	CW, CCW or OFF (posts M3, M4 or M5 respectively)
6. Coolant:	ON or OFF (posts M8 or M9 respectively)
7. Compensation:	LEFT, RIGHT, or OFF (Posts G41, G42 or G40 respectively)
8. Stop Command:	Program or Optional (posts M0 or M1 respectively)
9. 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
10. Cutting Mode:	Point to Point or Continuous (posts G61 or G64 respectively)
11. Program Mode:	Absolute or Incremental (posts G93 or G91 respectively)
12. Work Coordinates:	1, 2 . . .6 (posts G54, G55, ...G59 respectively)
13. 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 (posts M21, M22, M23... M29, M31, M32,... M25 respectively)
14. Miscellaneous Line:	The Miscellaneous Line is used to type in any M Code, G Code or text that is not on the standard list.

7.21 F7 CALL

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



GENERATED CODE FOR PROGRAM CALL

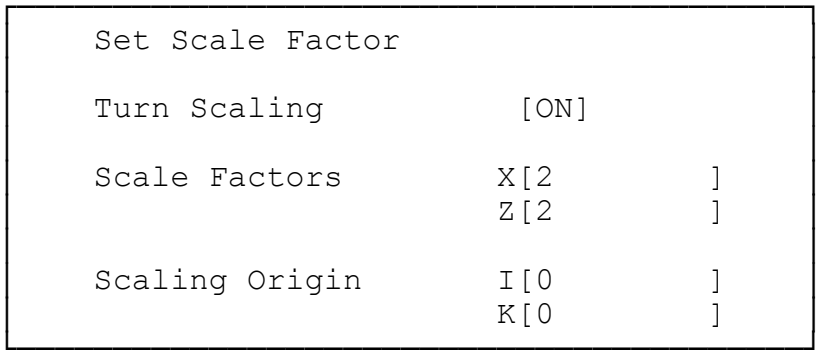


PARAMETER SETTING KEYS

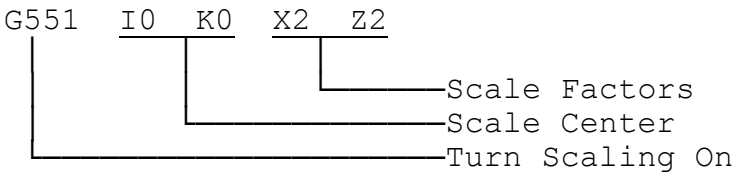
7.22 F8 SPECIAL (SPEC)

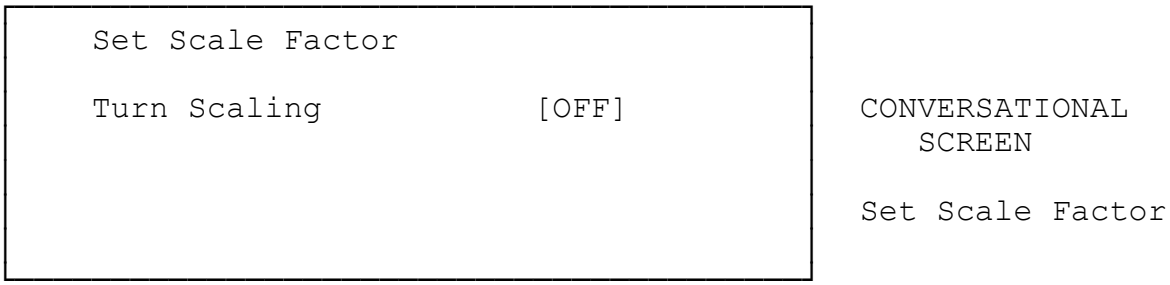
These are screens for setting or adjusting various parameters in the Centurion 5T control. The parameters control various functions such as scale factors, rotation angles, mirror image, and floating zeroes.

7.22.1 F3 SCALE (SPEC-SCALE)



GENERATED CODE FOR SCALING

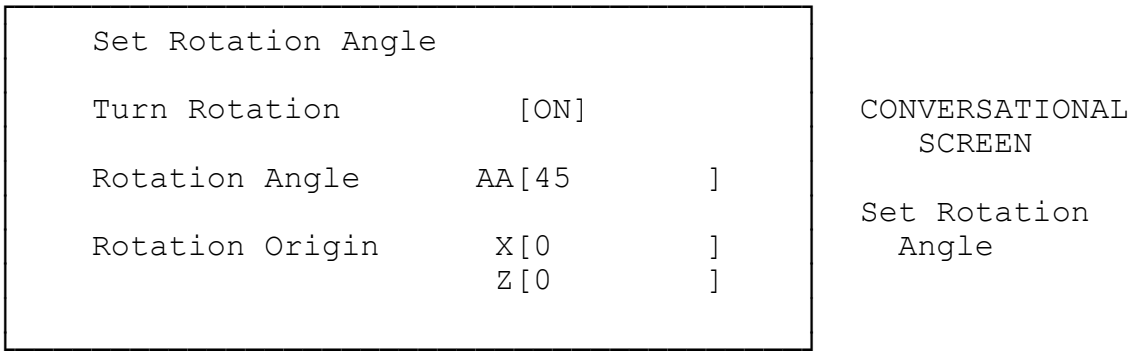




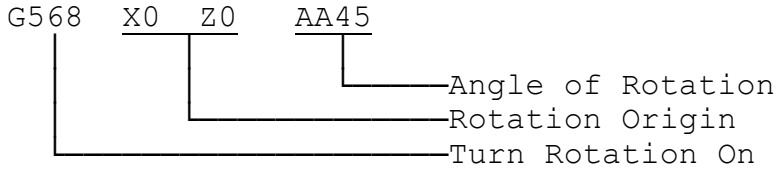
GENERATED CODE FOR SCALING OFF

G550 Turn Scale OFF

7.22.2 F4 ROT (SPEC-ROT)



GENERATED CODE FOR ROTATION



G569 Turn Rotation OFF

7.22.3 F5 MIRR (SPEC-MIRR)

```
Set Mirror Image

Turn Mirror Image     [ON]

Mirror Axis          X[0        ]
                     Z[0        ]
```

CONVERSATIONAL SCREEN

Set Mirror Image

GENERATED CODE FOR SET MIRROR IMAGE

```
G511  X0  Z0
 |     |   |
 |     |   └──Mirror Z axis around Z0
 |     └──────Mirror X axis around X0
 └────────────Turn on mirror image
```

```
Set Scale Factor

Turn Mirror Image      [OFF]
```

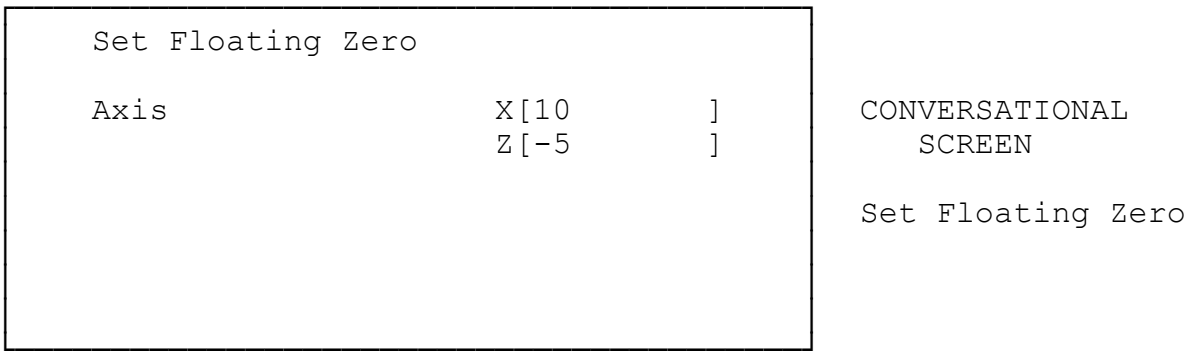
CONVERSATIONAL SCREEN

Set Mirror Image

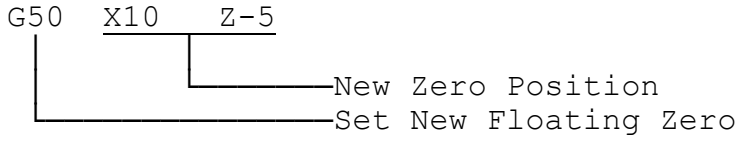
GENERATED CODE FOR MIRROR OFF

```
G511
 |
 └──────Turn Mirror Image OFF
```

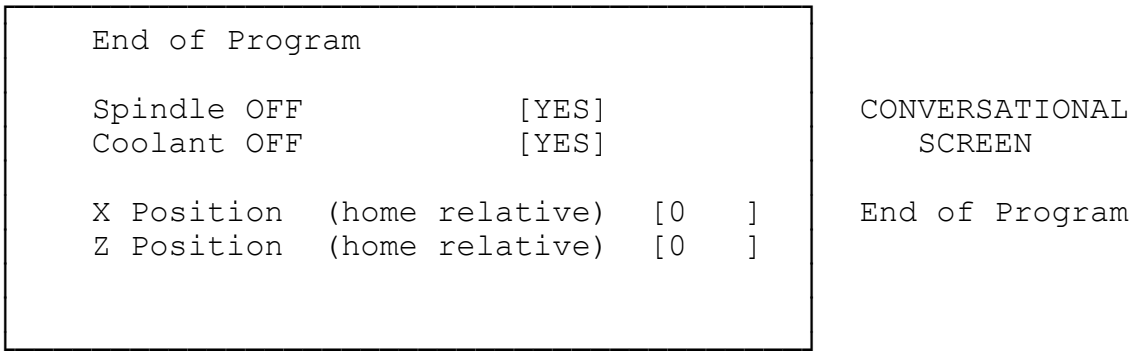
7.22.4 F7 FLZ (SPEC-FLZ)



GENERATED CODE FOR SET FLOATING ZERO



7.22.5 END OF PROGRAM



CODE GENERATED BY END PROGRAM

```
   M5        Spindle OFF
   M9        Coolant OFF
G53 G0 X0 Z0 X and Z to Home Zero
```

APPENDIX

CENTURION 5T
ERROR MESSAGES

001 Invalid function number
002 File not found
003 Path not found
004 Too many open files
005 File access denied
006 Invalid file handle
007 Memory control blocks destroyed
008 Insufficient memory
009 Invalid memory block address
010 Invalid environment
011 Invalid format
012 Invalid file access code
013 Invalid data
015 Invalid drive specified
016 Cannot remove current directory
017 Cannot rename across drives
018 No more files

100 Disk read error
101 Disk write error - Parts Memory is full
102 File not assigned
103 File not open
104 File not open for input
105 File not open for output
106 Invalid numeric format
150 Disk is write-protected
151 Unknown unit
152 Drive not ready
153 Unknown command
154 CRC error in data. Diskette is Bad or it is not formatted
155 Bad drive request structure length
156 Disk seek error
157 Unknown media type
158 Sector not found
159 Printer out of paper
160 Device write fault
161 Device read fault
162 Hardware failure
163 Sharing Violation

200 Division by zero
201 Range check error
202 Stack overflow
203 Heap overflow - Insufficient RAM memory
204 Invalid pointer operation
205 Floating point overflow
206 Floating point underflow

207 Invalid floating point operation
208, 209 TP Overlay Error
210..214 TP Object Error

300 Program already exists
301 Invalid program number
302 No programs to select from
303 Problem saving program(s) to disk
304 Problem loading program(s) from disk
305 Not formatted for conversational, try text editor
306 Conversational system has been corrupted
307 Illegal Event number
308 Invalid tool number
309 Can't copy or rename a file to itself
310 File not formatted for conversational or parameters
311 Parameter file not valid
312 Insufficient parts storage
313 Insufficient storage for compression, unable to post the file
314 Insufficient storage, post has been aborted
315 Out of storage space on the floppy
316 Not enough storage to create a new file
319 Send file aborted

400 Home required
401..
406 Axis-axis software limit overtravel
407..
412 Axis-axis excess error condition
413 Attempted to move into safe zone
415 Can't establish DNC link while program is running
416 Out of position, arc end point is not on the arc
417 Can't edit parameters while program is running
418 Controller Card Error
451 Lube fault, Check Way lube Level
452 Tool not found in auto tool sequence
453 Tool pot not up during turret movement
454 Not at tool change position
455 ATC arm is not out, axis movement not allowed
456 Can't process auto tool change after switching to manual
457 Move Length too great for control. (Split into smaller moves)

500 Last softkey pressed is not supported at this time
501 Illegal address encountered
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

505 Radius too small to span given points
507 After compensation line to arc lacks intersection
509 Arcs lack intersection
517 Parameter out of range
518 Illegal program statement
519 Feedrate out of range
520 Spindle Speed out of range
521 Negative Arc Radius
522 Negative Polar Radius
523 Illegal Tool Number
524 Illegal Offset Number
526 Invalid Access Code
527 Invalid Access Level
529 Duplicate address encountered
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 < 0
536 Can't chamfer and round the same corner
537 Can't chamfer to or from arcs
538 Loop counter out of range
539 Dwell time out of range
540 Illegal dwell time encountered
541 No axis moves are allowed on a G31 or G32 block
542 G30 Illegal return to reference parameter on G30 block
543 Illegal G10 statement
544 Too many digits in number
545 Illegal K value for number of holes
546 Nested calls or gosubs to deep
547 Comment not closed
549 Unrecognized G-Code
550 Bad numeric format
551 Multiple decimal points
552 Missing "]"
553 Missing "["
554 Tangent function overflow
555 Missing "/"
556 Negative SQRT argument
557 Unknown function
560 Illegal relational operator
567 Unresolved Call
568 Unresolved Goto or Gosub
569 The tool is too large to cut inside the arc "Compensated radius is too small"
570 The tool is too large to cut inside the arc "1st compensated radius in arc to arc is < 0"
571 The tool is too large to cut inside the arc "2nd compensated radius in arc to arc is < 0"
577 Input statements must precede axis moves
578 Undefined text cycle { should display offending character }
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 "**" function was zero or negative
582 X must be monotonously increasing or decreasing in type 1 of G71 rough cycle
583 Z must be monotonously increasing or decreasing in G71 rough cycle
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
586 X must be monotonously increasing or decreasing in G72 rough cycle
587 Z value exceeds starting Z in G72 rough cycle
599 syntaxstring;

600 Can't nest -WHILE WEND loops-
601 Missing WHILE statement
602 Missing WEND statement
603 Program does not exist
605 Can't modify dry run status while program is running
606 Program is empty
607 Can't exit DNC run mode while program is running
608 ''P'' expected in M98 block
609 G20/21 not allowed in DNC Fast.

900 RS232 overrun error
901 RS232 parity error (The system sending data may be not have the same parity as the CNC)
902 RS232 framing error (Remote system and CNC may not have the same line settings or a loose cable)
903 RS232 break detected (RS232 cable may be loose)
950 Obsolete controller software for this system.
951 Obsolete acroloop software for this system.
952 Obsolete keyboard software for this system.