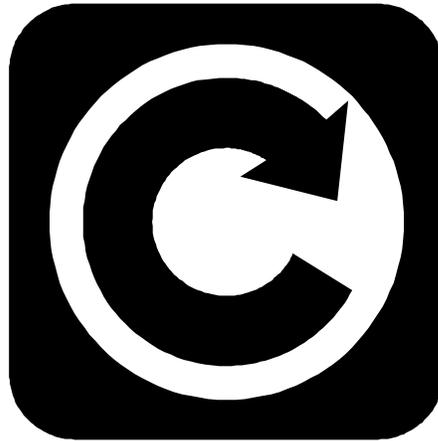


CENTROID™



M-SERIES Operator's Manual

Version 8.22

Rev. 030826

U.S. Patent #6490500
© 2004 Centroid Corp. Howard, PA 16841

CNC Control Information Sheet

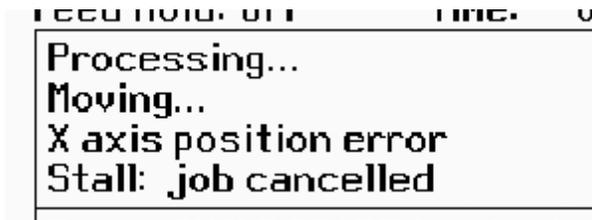
Fill out the following and fax back to Centroid Tech support **814-353-9265**. Date: _____
Company name: _____ Your name: _____
Address: _____ City: _____ State: _____ Zip: _____
Phone #: _____ Fax #: _____
Control Serial Number K _____ Software Version: _____ Approx. purchase date: _____
Dealer: _____ Machine Brand: _____

Machine type (check below)

- | | |
|------------------------------------|-------------------------------------------|
| <input type="checkbox"/> Knee Mill | <input type="checkbox"/> Machining Center |
| <input type="checkbox"/> Bed Mill | <input type="checkbox"/> Lathe |
| <input type="checkbox"/> Router | <input type="checkbox"/> Other |

Please indicate the messages (word for word) in the message window:

Message Window Example



Error Messages

- 1 _____
- 2 _____
- 3 _____
- 4 _____
- 5 _____

Describe the symptoms. What does the system do (or not do)?

System Voltages (requires an AC/DC voltmeter)

Source L1/L2 _____ VAC

Source L1/L3 _____ VAC

Source L2/L3 _____ VAC

Drive Voltage _____ VDC Measured at terminal 9 (GND) and 10 (+Vm) on the servo drive (E-Stop Released).

Phase Converter Yes ____ No ____

Control Parameters

Please fill in the parameter tables below. To get to the parameter screens:

1. Go to the Main screen of CNC7 software (this is the screen that appears when your system is first turned on).
2. Press <F1> to enter the Setup screen.
3. Press <F3> to enter the Configuration screen.
4. Type "137" in the window which asks for the password. Press <ENTER> to accept this.
5. Press <F2> Machine for Jog and Motor parameters.

Jog Parameters

Axis	Slow Jog (inches/minute)	Fast Jog (inches/minute)	Max Rate (inches/minute)	Dead Start (inches/minute)	Delta Vmax (inches/minute)
X					
Y					
Z					
4th					
5th					

Motor Parameters

Axis	Label	Motor revs/inch	Encoder counts/rev	Lash	Limit		Home		Direction Reversed	Travel
					-	+	-	+		
1										
2										
3										
4										
5										

PID parameters: Press ESC at the motor/jog parameters screen. Then press F4 PID

Axis	Kp	Ki	Kd	Limit	Kg	Kv1	Ka	Accel.	Max. Vel.
1									
2									
3									
4									
5									

To obtain the control configuration info, press ESC at the PID screen, then press F1 Control:

DRO display units: _____

Machine units: _____

Max spindle (high range): _____

Machine home at powerup: _____

Console type (Jog Panel Type): _____

Jog panel required: _____

Screen blank delay: _____

RS232 Drive & Directory: _____

Parameters: To obtain following parameter info, press ESC at the Control Configuration screen, and then press F3
Params:

Param #	Value										
21		23		29		40		56		58	
22		24		30		41		57		59	

TABLE OF CONTENTS

CHAPTER 1 - Introduction

Window Description	1-1
Conventions	1-3
Machine Home	1-4
Keyboard Operation	1-5

CHAPTER 2 - CNC7 Main Screen

Option Descriptions	2-1
Canceling and Resuming Jobs	2-6
Canceling a Job in Progress	2-7
Resuming a Canceled Job	2-7
M-Series CNC G-Code Editor Description	2-8

CHAPTER 3 - Part Setup (F1 from Setup)

Operation Description	3-2
Part Setup Examples	3-4
Work Coordinate Systems Configuration	3-5
Coordinate Systems Rotation	3-7

CHAPTER 4 - Tool Setup (F2 from Setup)

Offset Library	4-1
Automatic Tool Measurement	4-3
Setting up Tool Height Offsets	4-3
Tool Library	4-4

CHAPTER 5 - Power Feed (F4 from Setup)

CHAPTER 6 - The Utility Menu

F1 - Format	6-1
F2 – Update, F3 - Backup	6-2
F4 - Restore	6-3
F5 - File Ops, F6 - PLC Diag.	6-4
F7 – Report, F8 – Options, F9 – Log	6-4

CHAPTER 7 – Digitize (F9 from Main Menu)

Grid Digitize	7-2
Radial Digitize	7-6
Contour Digitize	7-10

CHAPTER 8 - Probing

Part Setup with Probing	8-1
Calibrating the Probe Tip Diameter	8-2
Probing Cycles	8-2
Probe Parameters	8-6

CHAPTER 9 - Engraving

Quick Start Engraving Software Tutorial	9-1
-----------------------------------------	-----

CHAPTER 10 - Intercon Software

Intercon Main Screen	10-1
Insert Operation	10-8
Graphics	10-42
Math Help	10-44
Intercon Tutorial #1	10-52
Intercon Tutorial #2	10-58

CHAPTER 11 - CNC Program Codes

Miscellaneous CNC Program Symbols	11-1
-----------------------------------	------

CHAPTER 12 - G-codes

G-Code Quick Reference	12-1
------------------------	------

CHAPTER 13 - M functions

Macro M functions	13-1
-------------------	------

CHAPTER 14 - Operator Panels

M-400 Operator Panel / M-39 Jog Pendant	14-1
Keyboard Jog Panel	14-6

CHAPTER 15 – Configuration

Password	15-1
Control Configuration	15-2
Machine Configuration	15-5
Machine Parameters	15-8
PID Configuration	15-26
Handwheel Configuration	15-28

CHAPTER 16 - CNC7 Messages

CHAPTER 1

Introduction

Window Description

The CNC7 display screen is separated into five areas called windows. A sample screen is shown below for reference. The five windows are the DRO display window, the status window, the message window, the options window, and the user window. The information that each window displays is described in detail in the following sections.

<i>DRO Display Window</i>					
WCS #1 (G54)	Current Position (Inches)	Job Name: C_ROD.CNC		<i>Status Window</i>	
X	+0.6541	Tool: T---H1	Program #10000		
Y	+0.6541	Feedrate: 24 ipm	Part Cnt: 0	<i>Message Window</i>	
Z	+0.1246	Spindle: 0	A Part #: 0		
W	+0.0000	FLOOD	M Time: 0:00:14	<i>User Window</i>	
B	+0.0000	Tool change: T1	Waiting for dwell time		
Dist To Go		Processing...		<i>Options Window</i>	
		Moving...			
		Press ESC to cancel			
X	+0.0000	17. M7		<i>User Window</i>	
Y	+0.0000	18. G4 P3.00 ; pause for dwell			
Z	-0.0246	19. G43 D1			
W	+0.0000	20. ; --- Drill BHC ---			
B	+0.0000	21. N0030 X0.6541 Y0.6541 Z0.5 H1			
		22. G81 X0.6541 Y0.6541 Z-0.51 R0.1 F2.0			
		23. X-0.4199 Y0.8242			
		24. X-0.9136 Y-0.1447			
		25. X-0.1447 Y-0.9136			
		26. X0.8242 Y-0.4199			
		27. G80			
Feed -1% F1	Feed +1% F2	Repeat On F3	/Skips On F4	Feed Hold F7	Rapid Off F9

DRO display

The DRO display contains the digital read out of the current position of the tool. The display is configurable for number of axes and desired display units of measure (see Chapter 15). The bars under each axis are the load meters and represent the amount of power being supplied to the drive for that axis. The display of axis load meters is configured by machine parameter 143 – see Chapter 15 for specific information. See also “Hot Keys” later in this chapter.

Distance to Go DRO

The distance to go DRO is located below the main DRO. This display shows the distance to go to complete the current movement. The display of distance to go is controlled by parameter 143. See Chapter 15 for details. See also “Hot Keys” later in this chapter.

Status window

The first line in the status window contains the name of the currently loaded job file. Below the job name are the Tool Number, Program Number, Feedrate Override, Spindle Speed, and Feed Hold indicators. The Feedrate Override indicator displays the current override percentage set on the Jog Panel. The Feedrate label will turn RED if the rapid override is turn off. If your machine is equipped with a variable frequency spindle drive (inverter), the Spindle indicator will display the current spindle speed. The Feed Hold indicator displays the current status (on/off) of FEED HOLD. See Chapter 14 for descriptions of the Feed Hold Button, Feedrate Override Knob, and Spindle controls. For a description of the Program Number see G65 in Chapter 12 or M98 in Chapter 13.

The Part Cnt and Elapsed Time indicators appear when CYCLE START is pressed while a job is running. The Part Count indicator displays the number of times the currently loaded job has been run. They count increments by one after the completion of a run. If a job is canceled prematurely, the part count will not be incremented. The Part # counter shows the how many parts have been run, with an up/down arrow displayed to indicate the counting direction. See the run menu for more information on the Part Cnt and Part # setting.

The Part Time indicator displays how much time has passed since the CYCLE START button was pressed. The indicator will help you to determine how long it takes to mill a particular part. The timer will not stop until the job is canceled. It will continue to count for optional stops, tool changes, FEED HOLD, etc.

Message window

The message window is divided into a message section and a prompt section. The prompt section of the window is the lowest text line in the window and will display prompts to the user. For example, the prompt 'Press CYCLE START to start job' is displayed on the prompt line after power-up.

The message section is the top four text lines of the message window. This section will display warnings, errors, or status messages. The newest messages always appear on the lowest of the four lines. Old messages are shifted up until they disappear off the top of the message window. See Chapter 17 for a description of the CNC7 error and status messages.

Options window

Options are selected by pressing the function key indicated in the box. For example, on the main screen, pressing the function key <F5> selects the CAM option.

User window

The information contained in this window is dependent upon on the operation the user is performing on the control. If no action is being taken, the window is empty.

For instance, when the CYCLE START button is pressed and a job is processed correctly, up to 11 lines of G codes will be displayed in this window for the user to observe during the Run of the part. All of the part zeros, the tool library setup, and the Digitizing/Probing information are entered in by the user in this window.

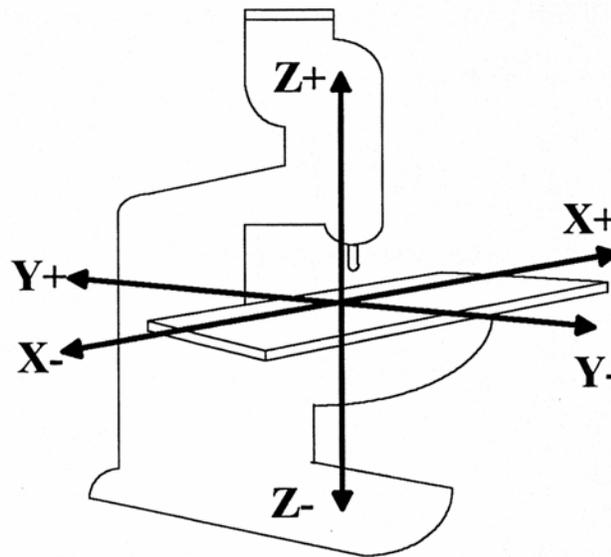
Conventions

*Keystrokes are represented by enclosing the capitalized name of the key in "less than" and "greater than" symbols. For example, the A key is written as <A> and the enter key is written as <ENTER>. The "Escape" key is written as <ESC>. Key combinations such as <ALT-D> mean that you should press and hold <ALT> then press <D>.

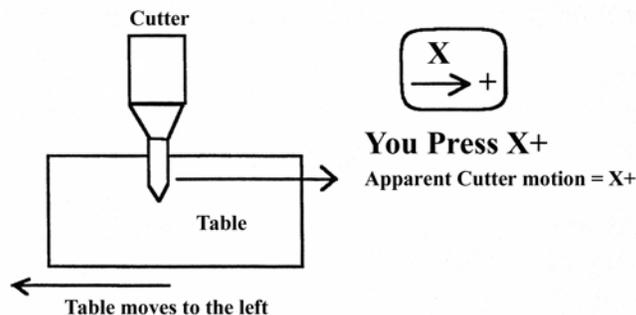
*All data entry screens in the M-Series Control use <F10> to save changes.

*Any menu in the M-Series Control can be exited by pressing <ESC>. This will take you back to the previous menu. This also usually discards any changes you have made in that menu.

*All program examples and software use the standard Cartesian coordinate system (see the figure below). If you are facing the mill, the X-axis is defined positive to your right; the Y-axis is defined positive to the mill; and the Z-axis is defined positive upward, perpendicular to the XY plane.



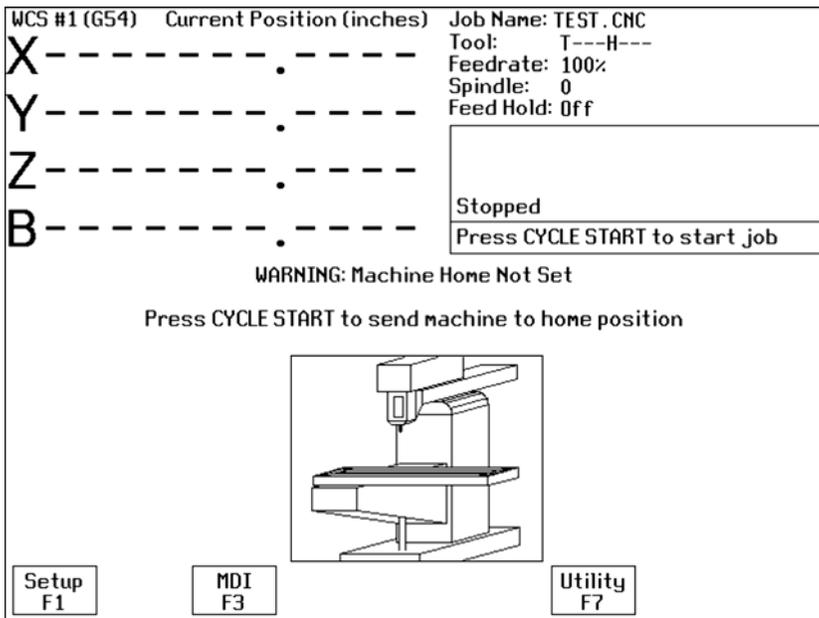
*The direction of motion is defined by the CUTTER motion, not the TABLE motion.



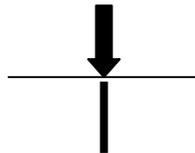
*CW stands for clockwise, and CCW stands for counterclockwise.

Machine Home

When the M-Series control is first started, the Main screen will appear as below.

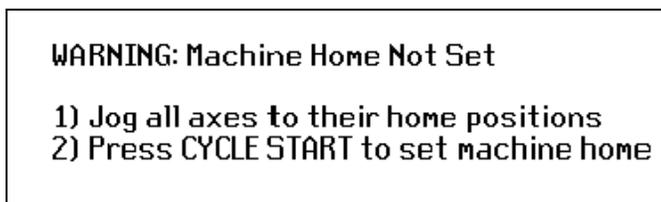


Before you can run any jobs, you must set the machine home position. If your machine has home/limit switches, reference marks or safe hard stops, the control can automatically home itself. If your machine has reference marks, jog the machine until the reference marks are lined up, (see below), before you press CYCLE START to begin the automatic homing sequence. The control will execute the G-codes in a file called CNC7.HOM in the C:\CNC7 directory. By default, this file contains commands to home Z in the plus direction, then X in the minus and Y in the plus direction.



Typical Reference Marks

If your machine does not have home/limit switches or safe hard stops, the following message will appear instead.



In this case you must move the machine to its home position yourself, using either the jog keys or the handwheels. Once all axes are at their home positions, press CYCLE START to set machine home.

Keyboard Operation

A computer style keyboard is supplied with most systems. This keyboard can be used as a jog panel. See Chapter 14, “Operator Panels” for more information. The keyboard jog panel has many “hot keys”. Hot keys are keys that can be used at almost any time, with few exceptions. (Some menus may prohibit their use.) CNC7 has many other hot keys in addition to the jog panel hot keys. The hot keys are listed below.

Hot Keys

Hot Key	Action
<ALT A>	Spindle auto/manual*
<ALT B>	Screen blanker on
<ALT C>	Flood coolant on/off*
<ALT D>	Switch between current position and machine position
<CTRL D>	Switch DRO between position and distance to go
<ALT E>	Mist coolant on/off*
<ALT F>	Displays available system memory
<ALT H>	Feed hold on/off*
<ALT I>	PLC diagnostics
<ALT J>	Enables keyboard jogging*
<ALT K>	Displays current ATC tool bin location
<ALT M>	MDI
<ALT O>	Tool check*
<ALT P>	Live PID display
<CTRL_P>	Clear max and min error display
<ALT Q>	Spindle on/off counter-clockwise*
<ALT R>	Spindle on/off clockwise*
<ALT S>	Cycle start
<ALT T>	Displays current motor temperature estimates
<ALT V>	Displays current software version #
<ALT W>	MPG on/off*
<ALT +> <ALT ->	Selects next WCS, cycles through WCS 1-18**
<ALT 1> - <ALT 0>	Selects WCS 1 – WCS 10**
<CTRL F1> - <CTRL F12>	Executes Aux function 1 – 12*

Notes:

- * This is a keyboard jog panel function. See Chapter 14 for details.
- ** Not available during jobs, in jog panel or while handwheels are engaged.

Mill M and G Codes

This is a summary list of M and G codes. See Chapters 12 – 13 for more information.

M00	Stop for operator	G00	Rapid Positioning
M01	Optional Stop for operator	G01	Linear Interpolation
M02	Restart Program	G02	Circular or Helical Interpolation CW
M03	Spindle on CW	G03	Circular or Helical Interpolation CCW
M04	Spindle on CCW	G04	Dwell
M05	Spindle off	G09	Exact Stop
M06	Start Tool Change	G10	Parameter Setting
M07	Mist Coolant on	G17	Circular Interpolation Plane Selection XY
M08	Flood Coolant on	G18	Circular Interpolation Plane Selection ZX
M09	Coolant off	G19	Circular Interpolation Plane Selection YZ
M10	Clamp on	G20	Select Inch Units
M11	Clamp off	G21	Select Metric Units
M15	Unclamp tool, air on	G28	Return to Reference Point
M16	Unclamp tool, air off	G29	Return from Reference Point
M18	Home tool changer	G30	Return to Secondary Reference Point
M19	Orient spindle	G40	Cutter Compensation Cancel
M20	Pick up tool	G41	Cutter Compensation Left
M21	Move head up	G42	Cutter Compensation Right
M22	Move head to ATC level	G43	Tool Length Compensation (+)
M23	Rotate carousel	G44	Tool Length Compensation (-)
M24	Start tool put back	G49	Tool Length Compensation Cancel
M25	Move to Z home	G50	Scaling/Mirroring Off (Optional)
M26	Set axis home	G51	Scaling/Mirroring On (Optional)
M39	Air drill	G52	Offset Local Coordinate System Origin
M50	Index tool plus	G53	Rapid Position in Machine Coordinates
M51	Index tool minus	G54	Select Work Coordinate System #1
M80	Carousel in	G55	Select Work Coordinate System #2
M81	Carousel out	G56	Select Work Coordinate System #3
M91	Move to minus home	G57	Select Work Coordinate System #4
M92	Move to plus home	G58	Select Work Coordinate System #5
M93	Release motor power	G59	Select Work Coordinate System #6
M94	Turn on input X	G61	Exact Stop Mode
M95	Turn off input X	G64	Cutting Mode
M98	Call subprogram	G65	Call Macro
M99	Return from subprogram	G68	Rotate
M100	Wait for input to open	G69	Cancel Rotate
M101	Wait for input to close	G73	High Speed Peck Drilling
M102	Restart program	G74	Counter Tapping
M103	Programmed action timer	G80	Canned Cycle Cancel
M104	Cancel programmed action timer	G81	Drilling and Spot Drilling
M105	Move minus to switch	G82	Drill with Dwell
M106	Move plus to switch	G83	Deep Hole Drilling
M107	Output BCD tool number	G84	Tapping
M108	Enable override controls	G85	Boring
M109	Disable override controls	G89	Boring with Dwell
M115	Protected probing move	G90	Absolute Positioning Mode
M116	Protected probing move	G91	Incremental positioning Mode
M120	Open data file (overwrite existing file)	G92	Set Absolute position
M121	Open data file (append to existing file)	G98	Initial Point Return
M122	Record position(s) and/or comment in data field	G99	R Point Return
M123	Record value and/or comment in data field	G117	Rotation of Plane Selection XY
M125	Protected probing move	G118	Rotation of Plane Selection ZX
M126	Protected probing move	G119	Rotation of Plane Selection YZ

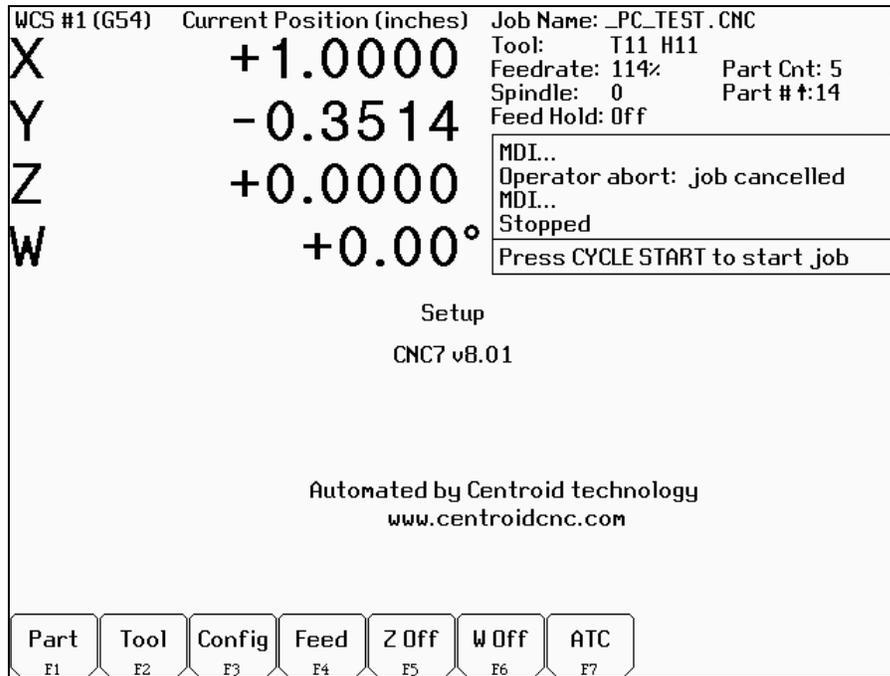
CHAPTER 2

CNC7 Main Screen

Option Descriptions

F1 – Setup

This will bring up the Setup menu as shown below.



F1 - Part

This key displays the Part Setup menus which are explained in Chapter 3.

F2 - Tool

This key displays the Tool Setup menus which are explained in Chapter 4.

F3 - Config

This key displays the Configuration menu which is explained in Chapter 15.

F4 - Feed

This key displays the Feed menu which is discussed in Chapter 5.

F5 – 3rd Axis Toggle

This key will only be displayed if Machine parameter 130 is set. See Chapter 15 for the various settings.

F6 – 4th Axis Toggle

This key will only be displayed if Machine parameter 131 is set. See Chapter 15 for the various settings.

F7 – ATC

This key will only be displayed if Machine parameter 6 is set to 1.0. It has the same effect as the <F7> ATC key in the Tool menus, which is to prompt for a tool number and then perform the actions required for an automatic tool change cycle.

F2 – Load Job

This option allows you to specify the file name of the CNC program that you want to run next. On the Load Job screen, the available keys are:

<F1>	change to the Floppy drive (A:\ directory)
<F2>	change to the Hard Drive (C:\CNC7\NCFILES directory)
<F3>	change to an attached computer's drive via RS232 port or network connection
<F10>	load the selected file
<Page Up>	move the cursor back one page. (A page is 32 files)
<Page Down>	move the cursor forward one page.
<END>	select the last file in the list.
<HOME>	select the first file in the list.
Arrow Keys	move the cursor in the selected direction

When the Load Job screen is first displayed, the initial list of files will come from the controller's hard drive. Press <F1> to switch to the controller's floppy drive, or press <F3> to switch to the drive of a computer attached via an Remote null modem cable. Press <F2> to switch back to the Controller's hard drive. You can use the arrow keys to move the cursor to the file you want to load. Once the job file name you wish to load is displayed on the "Job to load" line, press <F10>.

If you wish to use the Remote feature with an RS-232 null modem cable, you should run the INTERSVR program (supplied with IBM DOS) on the attached computer. If you wish to use the feature with a network connection the server should be a suitable DOS compatible LAN server. See Remote Drive and Directory in Chapter 6 if you need to set up a default drive and directory for the Remote feature.

Subdirectories are shown at the end of the list with square brackets, "[" and "]", around the name. If the current directory is not the root directory, a parent directory reference is shown as the last item of the list, signified by an up arrow next to the name.

Advanced users:

The "Job to load" line can perform functions similar to the DOS commands DIR and CD. See the examples below:

If you type:	The screen will
*.CNC	display all files in the current directory that have a .CNC extension
F*.*	display all files in the current directory that begin with F
..	move up one directory and display all files located in that directory
A:	change to the last selected directory on the A: floppy drive and display all files located in that directory
\	change to the root directory of the current drive and display all the files located in that directory
C:\ICN	change to the C:\ICN directory and display all the files located in that directory
A:\G*.CNC	change to the A: floppy drive root directory and display all files beginning with G that have a CNC extension
TEST?.CNC	display all files beginning with TEST that have one more character (TESTA, TEST1, etc.) and have the

	CNC extension
--	---------------

Using this ability is similar to using DIR and CD in DOS but leaving off the DIR or CD. If you can only remember part of the file name or it is located in another directory, these commands make it easier to locate. (See the DIR and CD commands in your PC-DOS manual for further information).

Do NOT load non G-code files and attempt to run or edit them. This can cause serious damage to the machine and controller, destroy the file, or cause personal injury.

F3 - MDI

Press <F3> to directly enter M and G-codes one block at a time. Enter one line of M or G-codes and then press <CYCLE START>. The controller will execute the command. It will then prompt you for another line. When you are finished entering commands, press <ESC>. The Rapid Override function key <F9> appears while in MDI mode (see below).

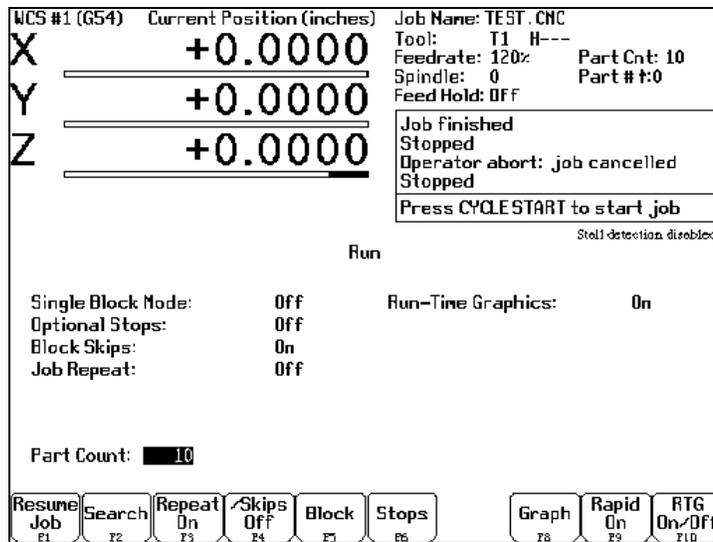
Examples:

- Block? G92X0Y0 ; Set the current XY position to 0,0
- Block? M26 ; Set the current Z position as Z home

All of the functions of Part Setup and Power Feed can be performed in MDI mode by typing the appropriate G and M-codes.

F4 - Run

Press <F4> to change the way your part program will run. An example screen appears below:



F1 - Resume Job

The Resume Job feature allows you to resume a previously canceled job at or near the point of interruption. See the section in this chapter titled "**Canceling and Resuming Jobs**" for a further detailed explanation.

F2 - Search

Invoking this option will bring you to the "Search and Run" menu. This menu will allow you to specify the program line, block number, or tool number at which execution of a program is to begin. Program lines are numbered from the top of the file down with the first line numbered 1. To enter a block number place an "N" in front of the number. To enter a tool number place a "T" in front of the number. Pressing CYCLE START from here would start the program at the point you specified.

An extra option unique to the “Search and Run” screen is the <F1> “Do Last Tool Change” function. This key toggles the tool change option as shown on screen. A "YES" tells the control to perform a tool change so that the tool specified for the line or block has the tool indicated in the program. A "NO" uses the currently loaded tool, regardless of what tool is specified for the line or block being searched.

F3 – Repeat On/Off

This key toggles the repeat feature for part counting. When part counting is in effect and Repeat is on, the job will be automatically run again until the specified number of parts have been run. The On or Off label indicates the state to which the repeat feature will toggle to when pressed. It does not indicate the current state. The current state is indicated in the user window above.

The **Part Count:** prompt is used to set the Part count. Positive values set the part counter to count up and negative values configure the part count to count down. For example, if 10 is entered in the Part Count prompt, the Part Cnt in the status window changes to 10 and the Part # changes to 0 with an upward arrow indicator. When a job is run and then completes, the Part # will increment to 1. If repeat is on, the job will automatically start again and keep running until the Part # has reached the Part Cnt. If a -10 is entered in the Part Count prompt, the Part Cnt in the status window changes to 10 and the Part # changes to 10 with a downward arrow indicator. When a job completes, the Part # will be decremented to 9. If repeat is on, the job will automatically start again and keep running until the Part # has reached 0.

F4 - /Skips On/Off

This function toggles the block skip feature. When block skipping is on, G-code lines that start with a forward slash character ‘/’ are skipped, i.e., they are not processed. The On or Off label indicates the state to which the /Skips feature will toggle to when pressed. It does not indicate the current state. The current state is indicated in the user window above.

F5 - Block Mode

Turns single block mode on and off. This is similar to pressing AUTO/BLOCK. If single block mode is on, CNC7 will stop after each block in your part program and wait for you to press CYCLE START. The current state is indicated in the user window above.

F6 - Optional Stops

Turns optional stops on and off. If optional stops are on, any M1 codes that appear in your program will cause a wait for CYCLE START (just like M0). If optional stops are off, M1 codes will be ignored. The current state is indicated in the user window above.

F8 - Graph

Graphs the part. For more information, see the "<F8> - Graph" as described later in this chapter. If this feature is invoked from the Run and Search screen or the Resume Job screen, then the graphics will show exactly where the searched line or block begins. Dotted lines indicate the portion of the part that is skipped. Solid lines indicate the portion of the part that will be machined.

F9 – Rapid On/Off

This function key toggles Rapid Override. The On or Off label indicates the state to which the Rapid Override feature will toggle to when pressed. It does not indicate the current state. It has the same effect as the Rapid Over key discussed in Chapter 14.

F10 – RTG On/Off

This function key toggles the Run-Time Graphics option. If the option is turned on, Run-Time Graphics automatically starts when the CYCLE START button is pressed. This option must be turned on for Run-Time Graphics to be used. If the option is turned off, Run-Time Graphics cannot be started while a job is running.

F5 - CAM

Choose <F5> from the Main Menu to load an installed CAM software package. Currently, the default CAM system is Intercon (**I**nteractive **C**onversational) software. Your dealer can install other CAD/CAM packages. If more than one CAD/CAM program or on line software package has been installed, a menu will appear that allows you to choose the appropriate program. When you exit the CAD/CAM software, you will return to the M-Series Control Main Screen. A part created in Mastercam, Engraving, or ICN will automatically be loaded into the CNC7 main program. The part program must be stored in one of the following directories in order for it to be automatically loaded:

Engraving - Loads files from **C:\CNC7\NCFILES** directory
Mastercam - Loads files from **C:\NC** directory
ICN - Loads files from **C:\CNC7\NCFILES** directory

Contact your dealer if you wish to change these directories or if you want to add third party software.

F6 - Edit

The edit function from the Main Screen loads a text editor so that you may edit CNC files. Press <F6> to load the current job file. When you exit the text editor, you will return to the CNC7 Main Screen.

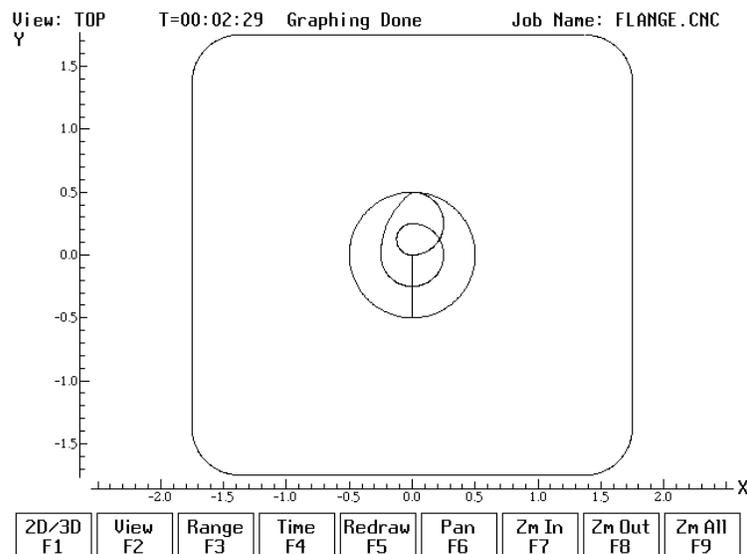
Attempting to edit files that contain non-printable characters may cause unexpected results. **DO NOT** edit the CNC7 files CNC7.CFG, CNC7.PRM, CNC7.JOB, CNC7.TL, CNC7.OL, and CNC7.WCS. These files will be destroyed and all information lost if they are edited.

F7 - Utility

Press <F7> to bring up the Utility Screen. This screen gives you several options from diagnostics to file functions. See Chapter 6 for a detailed description of the utility operations.

F8 – Graph

This option plots the tool path of the current program loaded. Canned drilling cycles are shown in gray. Rapid traverse movements are shown in red. Feedrate movements are shown in yellow.



The screen shows the following keys:

<F1> - Press this key to view your part isometrically (3D). An axis pointer indicates the current direction of the view. To return back to the tri-planar view, press <F1> again.

<F2> - Press this key to rotate your part. Use the keyboard arrow keys to rotate any direction.

OR

<F2> - Press this key to change the planar **view** of your part. The view is indicated by a TOP, RIGHT, or FRONT shown at the top of the screen.

<F3> - Press this key to set the range of line numbers or block numbers to graph.

<F4> - Press this key to estimate the time needed to create part. It takes into account accelerations and decelerations, but neglects tool change times.

<F5> - Press this key to redraw the part at any time.

<F6> - Press this key to move the part around the screen. Once pressed, use the crosshairs to pick a location of the part that will redraw at the center of the screen. Once a section is selected, press <F6> again to continue panning.

<F7> - Press these keys to zoom into the part relative to the center of the screen.

<F8> - Press these keys to zoom away from the part relative to the center of the screen.

<F9> - Press this key to view the entire part fit inside the screen.

Turn the FEEDRATE OVERRIDE knob to control the speed of the graphing. To pause the tool path, turn the knob counter-clockwise until it stops. Turn the knob clockwise to resume drawing.

F9 - Digitize

Press <F9> to bring up the Digitize screen. This screen allows you to set up and run touch probe digitizing. See Chapter 7 for a detailed description of the digitizing operation.

F10 - Park

Press <F10> to park the machine at the end of the day for quicker machine homing at startup. Once <F10> is selected, The Cycle Start key must be press to start machine movement. The park feature homes each axis, at the maximum rate, to ¼ motor revolution from its home position. The Z axis is moved first, then all the other axis are done.

CYCLE START (or START)

Press this key to run a job from this screen.

ALT-S

The <ALT-S> option is for those operators who have no Jog Panel. Pressing <ALT-S> is the same as pressing the CYCLE START button on the operator panel. See Chapter 15 for a description of the CYCLE START button.

Canceling and Resuming Jobs

The control provides several ways for the operator to cancel jobs in progress. The control also allows the operator several ways to resume a canceled job. The information in this section does not apply to digitizing.

Canceling a Job in Progress

There are three conventional ways to cancel a currently running job (CNC program). When a job is canceled using any of the following methods, the job's progress will be recorded. This allows the user to restart the job using the Resume Job option or the Search and Run option.

CYCLE CANCEL

Pressing this key while a job is running will cause the control to abort the job currently being run. The control will stop movement immediately, clear all M-functions, and return to the main screen. Hitting the escape key on the keyboard is equivalent to hitting "CYCLE CANCEL."

TOOL CHECK

Pressing this key while a job is running will cause the control to stop the normal program movement. In addition, the Z-axis will be pulled to its home position and all M-functions will be cleared. The control will automatically go to the resume job screen.

EMERGENCY STOP (E-Stop)

Pressing the EMERGENCY STOP key while a job is running will cause the control to abort the job currently being run. The control will stop movement immediately, clear all M-functions, and return to the main screen. Also, the power to all axes will be released.

Resuming a Canceled Job

If a job is canceled using one of the methods described above, it can be resumed in one of 2 ways.

Resume Job Screen – F1 from the Run Screen

Access the resume job screen by pressing <F4> on the main screen to go to the run screen, and then pressing <F1> in the run screen to go to the resume job screen. If the job was canceled by pressing Tool Check, the control will go to the resume job screen automatically. From this screen, the user can modify tool offsets and the tool library, turn block mode on and off, turn optional stops on or off, graph the partially completed job, or start the partially completed job.

The resume job option is not always available. The following situations will cause the resume job option to be unavailable:

- Loading a new job.
- Running a job to completion.
- Parse errors in the job.
- Editing or reposting the job file.
- Loss of power while a job is running.

Search and Run Screen – F2 from the Run Screen

The search and run screen can also be used to restart a job. Search and run allows the user to specify at which line, block, or tool number the job should be resumed. You cannot search into a subroutine.

M-Series CNC G-Code Editor Description

This is a detailed description of the <F6> Edit option invoked from the Main screen.

Usage

If the editor is invoked from the DOS command line, a file may be loaded into the editor by either specifying a name on the command line, or by entering the editor and selecting the Load File (F9) option.

Examples:

<code>C:\CNC7\NCFILES>cnc7edt</code>	Invoking editor from command line
<code>C:\CNC7\NCFILES>cnc7edt cnc40.nc</code>	Invoking editor and loading a file from the command line

To edit a G-code program, press "F6 Edit" from the main screen. The G-code of the current job will be loaded.

Editor Screen

The editing screen will have a status line across the top of the screen, while the bottom line of the screen will show some of the available editor functions. The status line displays the current cursor line and column, the current typing mode (Insert/Overwrite), a "modified" message if the file has been modified since the last time it was saved, and the name of file currently being edited. Below is a sample editing screen:

Line 1	Column 1	Insert	EAGLE2.DIG						
;Remove colon from in front of G51 for scaled size needed.									
;For full size DONOT remove any colons from any g51									
;digitized area full size x25.8 y9 z1.45			1/4" ball endmill						
;digitized area .603 size x15.56 y5.427 z.874			5/32" ball endmill						
;g51 x0y0z0 p.603									
;digitized area half size x12.9 y4.5 z7.25			1/8" ball endmill						
;g51 x0y0z0 p.500									
;digitized area .375 size x9.675 y 3.375 z.544			3/32" ball endmill						
;g51 x0y0z0 p.375									
;digitized area x8.100 y3.750 z.500			1/8" ball endmill						
;g51 x0y0z0 i.3140 j.417 k.3448									
;digitized area x9.675 y3.375 z.4375									
g51 x0y0z0 i.375 j.375 k.30172									
m3s4200									
M25 G0 X0.0000 Y0.0000 ; NO DATA									
g43 h47 z.100 m8									
G1 F5.0000									
X0.0000 Y0.0000 Z-1.4500									
X25.8000 Y0.0000 Z-1.4500 F10.0									
X25.8000 Y0.0200 Z-1.4500									
X0.0000 Y0.0200 Z-1.4500 F25.									
Help F1	Save F2	Find F3	Findnxt F4	Replace F5	Goto F6	PgUp F7	PgDn F8	New F9	Quit F10

Pressing the <F1> key will display a complete list of editor functions and the key(s) that activate them. Press any key to return to editor screen.

Editor Functions

The following table contains a list of all available editor functions, the keys that activate them, and brief descriptions of their effects:

Editor function	Key(s)	Comments
Insert/Typeover mode	Insert	Typeover cursor is an underline; insert cursor is a block.
Move cursor left, right, up, down	Arrows	
Move cursor to beginning of line	Home	
Move cursor to end of line	End	
Scroll up one screen	Page Up	
Scroll down one screen	Page Down	
Move to beginning of file	Ctrl + Page Up	
Move to end of file	Ctrl + Page Down	
Delete character under cursor	Del	Deleting at end of line joins with next line.
Delete character in front of cursor	Backspace	Deleting at beginning of line joins to preceding line.
Delete current line	Ctrl + Y	
Help	F1	Displays list of all editor commands. Pressing any key returns to the editor.
Load file	F9	Load a file for editing. If a file name is specified for a file that does not exist, a new file will be created.
Save file	F2	

Exit	F10	See user dialog table below. Save current file to disk. See user dialog table below. Quit using editor. See user dialog table below.
Search forward Search forward again Replace	F3 F4 F5	Specify string to search for; this is a case-sensitive search Replace all occurrences of one text string with another string. See user dialog table below.
Escape	Esc	Cancel current dialog sequence.

Table 1 - Editor Function Descriptions

The table below describes the dialog sequences involved in using editor functions:

Function	Condition	Question
Save file	A file with the current name already exists. You answer N to the "replace" question. You choose a file name that already exists.	"Do you want to replace the original file <current file name>? Y/N" "Specify a new file name" "Do you want to replace the original file <selected file name?> Y/N"
Load file	You have made changes to the current file and have not saved them. You answer Y to the "save" question. You answer N to the "save" question, or you complete the above save process.	"Do you want to save changes in the file <current file name>? Y/N" Perform the save file process above. "Specify file name to be loaded"
Exit editor	You have made changes to the current file and have not saved them. You answer Y to the "save" question.	"Do you want to save changes in the file <current file name>? Y/N" Perform the save file process above.
Replace text	The file is modified in memory and on disk.	"Pattern" (string to search for) "Replacement" (string to substitute) Perform the save file process above.

Table 2 - User Dialogs

CHAPTER 3

Part Setup (F1 from Setup)

General

WCS #1 (G54)	Current Position (inches)	Job Name: C_ROD.CNC
X	+1.0000	Tool: T1 H1
Y	+2.0000	Feedrate: 120%
Z	+3.0000	Spindle: 0
		Feed Hold: Off
MDI..		
Operator abort: job cancelled		
MDI..		
Stopped		

Set Part 0/Position

- 1) Select Axis with F1
- 2) Jog to Touch Off on Part
- 3) Edit the Value if Necessary
- 4) Press F10 to Set Position

Axis	Part Position	Edge Finder Diameter	Approach From
X	0.0000	0.0000	Right(+)

Next Axis F1	Auto F4	Probe F5	Prev WCS F6	Next WCS F7	CSR F8	WCS Table F9	Set F10
-----------------	------------	-------------	----------------	----------------	-----------	-----------------	------------

The Part Setup menu is used to set the part location or the coordinate system origin for the part.

<F1> will display the position for the next axis. If changes were made to the current axis, but not yet accepted, they will be discarded.

<F6> will select the previous work piece coordinate system. The position that will be set only affects the currently selected coordinate system.

<F7> will select the next work piece coordinate system. The position that will be set only affects the currently selected coordinate system.

<F8> can be used to automatically detect coordinate system rotation. This function key appears only when the software option for Coordinate System Rotation is unlocked.

<F9> will open the Work Coordinate System (WCS) Configuration screen. See the Work Coordinate System Configuration section later in this chapter for a complete description.

<F10> will accept the position for the current axis, correcting for edge finder diameter based on the approach direction if appropriate. It will not automatically advance to the next axis.

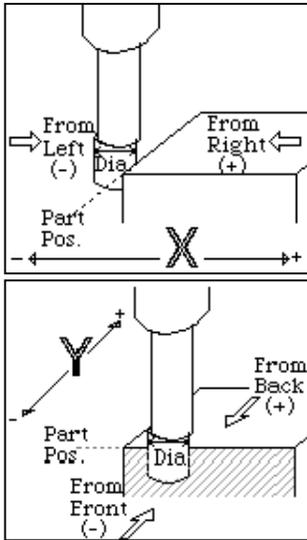
For description of <F4> and <F5>, see Chapter 8.

Operation Description

Setting the part zeros establishes a coordinate system with an origin at the part zero.

The <F1> Next Axis option selects the axis to be defined next. This field toggles between axis X, Y, Z and the fourth axis (if you have a 4-axis system). For each axis you will see a graphic description of the parameters to be entered, as well as the corresponding fields.

Setting up X or Y AXIS



Set Part 0/Position

- 1) Select Axis with F1
- 2) Jog to Touch Off on Part
- 3) Edit the Value if Necessary
- 4) Press F10 to Set Position

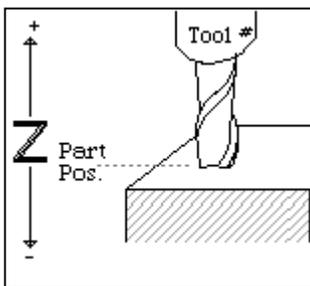
Axis	Part Position	Edge Finder Diameter	Approach from
X	0.0000	0.0000	Left (-)

Part Position: enter the offset you want between the position of the edge finder and the desired position of the origin.

Edge Finder Diameter: enter the diameter of the tool, piece, or edge finder you are using to determine the part zero. The value entered is stored.

Approach From: Toggle the direction the edge finder or probe is approaching the part from.

Setting up the Z AXIS



Set Part 0/Position

- 1) Select Axis with F1
- 2) Jog to Touch Off on Part
- 3) Edit the Value if Necessary
- 4) Press F10 to Set Position

Axis	Part Position	Tool Number
Z	0.0000	0

Part Position: enter the offset you want between the position of the edge finder and the desired position of the origin.

Tool Number: enter the tool number from the Tool Library that corresponds to the tool being used. When the Tool Number field is set to a value other than zero, the controller uses the Height Offset for that tool from the Tool Library to calculate the actual position.

Example 1 (You are using the reference tool to find the Z-axis part zero):

Set Tool Number to 0. Setting the Tool Number equal to zero tells the controller that you are using the reference tool.

Example 2 (You are using a tool other than the reference tool, and not a ball nose cutter):

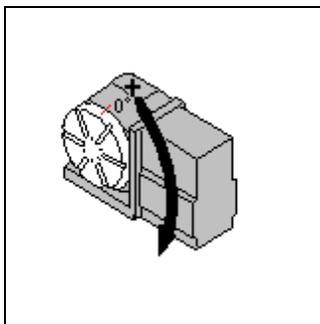
Set Tool Number to the number this tool is assigned in the tool library.

Example 3 (You are using a ball nose cutter, other than the reference tool):

Set Part Position to the position of the surface plus the nose radius of the ball nose cutter; set Tool Number to the number this tool is assigned in the tool library.

The Tool and Offset libraries must be up to date before setting the Z-axis Part Zero.

Setting up the 4th or 5th AXIS



Set Part 0/Position

- 1) **Select Axis with F1**
- 2) **Jog to Touch Off on Part**
- 3) **Edit the Value if Necessary**
- 4) **Press F10 to Set Position**

Axis	Part Position	Standoff Distance	Approach from
B	0.0000	0.0000	+

Position: enter the offset you want between the position of the edge finder and the desired position of the origin.

Standoff Distance: this field is a generic parameter. Its physical meaning will depend on the specific nature of your machine's fourth axis. It is the distance between the center of the tool and the point at which the tool is touching the part surface.

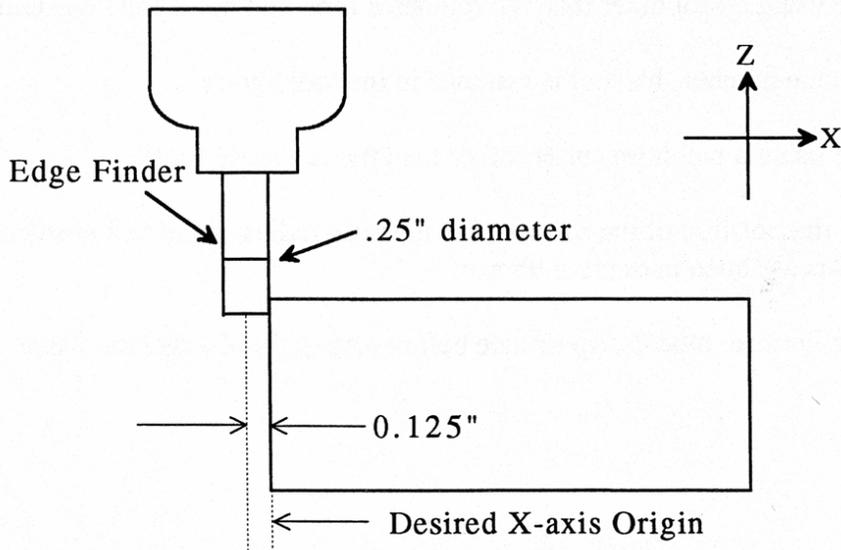
Approach from: enter the direction the edge finder is approaching the part from. Enter the correct direction given the nature of your 4th-Axis.

Using Multiple Coordinate Systems

If you will be using multiple coordinate systems, you must set the part position separately for each coordinate system. Follow the instructions above to set the position for each axis in the first coordinate system, then press <F2> to select the previous work coordinate system or <F3> to select the next work coordinate system. Move to the next fixture and repeat the process. The currently selected coordinate system is displayed below the axis picture on the Part Setup screen. It is also displayed above the DRO at all times. For a description on setting up each coordinate system, see the Work Coordinate System Configuration section later in this chapter.

Part Setup Examples

Example 1: Setting the X-axis Part Zero with no offset (See diagram below)



If you wanted the left edge of the part to be the origin for the X-axis:

1. Move the Edge Finder to the left edge of the part
2. Press <F1> until the Axis label displays 'X'
3. Move the cursor to the Edge Finder Diameter field
4. Type .25 and press <ENTER>
5. Press <SPACE> until Left (-) is displayed
6. Press <F10> to accept the values

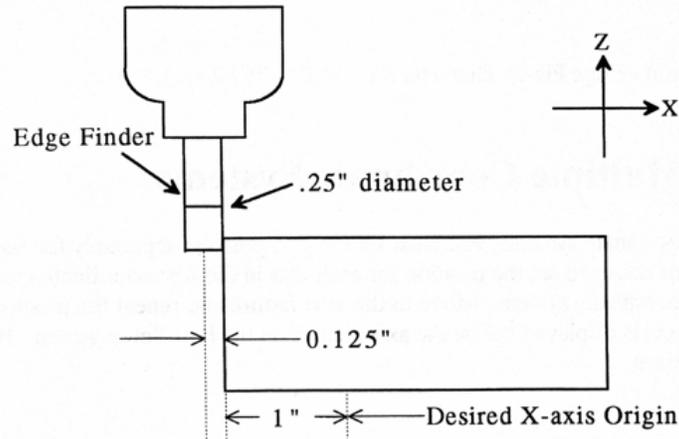
Axis	Position	Edge Finder Diameter	Approach From
X	0	0.25	Left (-)

Since no offset is being applied, Position is zero. The Edge Finder is approaching the part from the -X direction and has a diameter of .25 inches. Once this data is entered and <F10> is pressed, the X-axis DRO display will read -0.125. This means the center of the Edge Finder is sitting to the left (minus) of the part by 0.125 inches (half of the Edge Finder Diameter).

This value is computed by: Position (Approach from) Edge Finder Diameter / 2.

Where (Approach from) is the sign of the approach direction. In other words, if the approach direction is minus, then the value is: Position - Edge Finder Diameter / 2 = 0.0 - .25 / 2 = -0.125

Example 2: X-Axis origin offset into part 1 inch.



If you wanted the origin offset 1 inch into the part:

1. Move the Edge Finder to the left edge of the part
2. Press <F1> until the axis field displays 'X'
3. Move the cursor to the Position field
4. Type -1 and press <ENTER>
5. Type .25 and press <ENTER>
6. Press <SPACE> until Left (-) is displayed
7. Press <F10> to accept the value

Axis	Position	Edge Finder Diameter	Approach from
X	-1	0.25	Left (-)

The Position value is relative to the current position of the Edge Finder. Position equals -1.0 since the Edge Finder is positioned 1 inch to the left (minus direction) of where you want the X-axis origin.

Another way to view the Position value is to assume the origin is already set at 1 inch into the part. In this case, the Edge Finder would have to move -1 inches from where the origin is to get to the left edge of the part.

The Edge Finder is approaching the part from the -X direction and has a diameter of .25 inches. Once this data is entered and <F10> is pressed, the X-axis DRO display will read -1.125. This means the center of the Edge Finder is sitting to the left (minus) of the origin by 1.125 inches. The X-axis origin is now 1 inch into the part.

This value is computed by: Position (Approach from) Edge Finder Diameter / 2.

Where (Approach from) is the sign of the approach direction. In other words, if the approach direction is minus, then the value is:

$$\text{Position} - \text{Edge Finder Diameter} / 2 = -1.0 - .25 / 2 = -1.125$$

Work Coordinate Systems (WCS) Configuration

Press <F9> from the Part Setup screen to display the Work Coordinates System (WCS) menu. The Work Coordinate Systems screen provides access to soft travel limits, reference points, and coordinate system origins. Make sure your Home position has been set properly. Otherwise, the positions of each coordinate system will not be in the appropriate position.

When you enter the Work Coordinate System Configuration screen, the DRO display will automatically switch over to machine coordinates as an aid to entering numbers. All the values on this screen are represented in machine coordinates.

F1 - Travel Limits and Reference Return Points

The <F1> key is used to set the soft travel limits and the reference return points for the machine.

The travel limits are measured from machine home. If machine home is at the minus end of the axis, then a positive value should be entered, indicating how far the machine can move in the positive direction until it reaches the other end of its travel. If machine home is at the plus end of the axis, then a negative value should be entered, indicating how far the machine can move in the negative direction until it reaches the other end of its travel. The machine will never be allowed to move beyond the machine home position.

In the example above, the home positions for the X and Y axes are at the minus limit, and the home position for the Z axis is at the plus limit.

The reference return points are used with the G28 and G30 codes (see Chapter 12). They are specified in machine coordinates. The Z coordinate of the first reference point is also used as a Z home position by the M2, M6, and M25 codes (see Chapter 13).

F2 - Origin

Use the <F2> key to specify the locations (in machine coordinates) of the origins of the *first six* work coordinate systems. This option is a convenience and is not an absolute necessity for setting work coordinate system origins. If you want to set the origins for the rest of coordinate systems, you must use the Part Setup menu.

If the software option Coordinate System Rotation is unlocked, the CSR angle for each of the first six work coordinate systems can also be set.

All coordinate systems are relative to Home position that is set during control power up. Note that the DRO while in this screen shows the actual machine position relative to Home, not the location relative to the WCS origin.

Coordinate System Rotation (CSR)

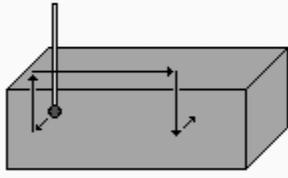
Coordinate System Rotation saves you time when setting up your part. Rather than clamping your part and indicating the edge of the material to square it with the machine axes, you can use CSR to automatically rotate the coordinate system to the angle of the part or fixture that was probed. This allows you to compensate for different orientations.

Simply clamp your part, then probe two points along either the X or Y axis of the material using the process described below.

WCS #1 (G54) Current Position (inches)		Job Name: TEST .CNC	
X	-0.0001	Tool: T1 H---	
Y	+0.0004	Feedrate: 120%	Part Cnt: 10
Z	+0.0002	Spindle: 0	Part # ↑:0
		Feed Hold: Off	
		Waiting for PLC operation Stopped	

Coordinate System Rotation

1. Select orientation with F1
2. Position probe at one end of vise jaw
3. Enter distance to other end of jaw
4. Enter Z clearance amount (incr. from start)
5. Select automatic or manual move to other end
6. Press CYCLE START to start



Distance: **1.0000**

Clearance Amount: 0.5000

Movement Between Points: Auto

WCS #1 (G54)

Orient
F1

Manual
F2

Zero
Cur
F3

Zero
All
F4

Prev
WCS
F6

Next
WCS
F7

WCS
Table
F9

<F1> is used select the orientation for the CSR measurement. There are four possible orientations, which are: from the front (pictured above), the back, and the left and right sides.

<F2> is used to determine the CSR angle without probing. The user jogs an edge finder to two positions along one wall. These positions will be used for computing the CSR angle.

<F3> is used to set the CSR angle for the current WCS to zero.

<F4> is used to set all CSR angles to zero.

<F6> and <F7> are used to cycle through the available WCS systems.

<F9> is a shortcut to the Work Coordinate System Configuration Screen described above.

The instructions on how to perform a CSR measurement are numbered on the screen.

Distance: The distance the X axis (in front or back orientation) or Y axis (in right or left side orientation) will move to probe the second point. If the distance is negative, the axis will be moved in the negative direction.

Clearance Amount: The distance the Z axis will be moved upward when moving between the first probe point and the second probe point. The clearance move will only be made when using the “Auto” option of the Movement Between Points.

Movement Between Points can be toggled between Jog and Auto modes. In Auto mode, the clearing moves are made automatically as well as the move to the second point. In Jog mode, a prompt will be displayed in the center of the screen after the first point is probed.

CHAPTER 4

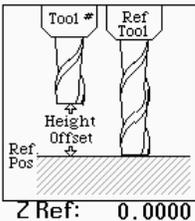
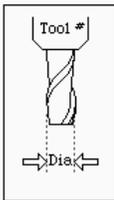
Tool Setup

(F2 from Setup)

Tool Setup allows you to specify information about the tools you will be using. Press <F1> to edit the Offset Library Height Offset and Diameter (H and D) values, or <F2> to edit the Tool Library tool descriptions.

Offset Library

The Offset Library file contains the values for the Height Offset and Diameter Numbers. For example, if entry H01 has a value of -.25, a height offset of -.25 is applied when height offset 01 is referenced. If entry D01 shows a value of 1.5, the diameter offset 01 has a diameter of 1.5 associated with it.

WCS #1 (G54)	Current Position (inches)	Job Name: F .CNC
X	+0.0000	Tool: T0 H---
Y	+0.0000	Feedrate: 100%
Z	+0.0000	Spindle: 0
B	+0.0000	Feed Hold: Off
		Jog Panel Offline Check Jog Panel cable Stopped
Offset Library		
Height Offset		Diameter
	H001 -0.1235	D001 0.0000
	H002 -0.2345	D002 0.0000
	H003 -0.4560	D003 0.0000
	H004 -1.2345	D004 0.0000
	H005 0.0000	D005 0.0000
	H006 0.0000	D006 0.0000
	H007 0.0000	D007 0.0000
	H008 0.0000	D008 0.0000
	H009 0.0000	D009 0.0000
	H010 0.0000	D010 0.0000
		
Z Ref: 0.0000		
Z Ref F1	Manual F2	Auto F3
Batch F4	+ .001 F5	- .001 F6
ATC F7	Save F10	Cancel ESC

Press <F1> to set the Z reference height. Press <F2> to manually measure tools. If you purchased the Tool Length Probing option, press <F3> to automatically measure tool lengths. Press <F5> or <F6> to adjust the selected offset. If you have an automatic tool changer installed, press <F7> to change tools. Press <F10> to save changes and exit, or <ESC> to exit without saving changes. If you have both purchased the Tool Length Probing option **and** also have an automatic tool changer installed, then you can press <F4> to perform batch tool measuring, by entering a list of multiple tool numbers.

You can inspect and change any of the 200 Height Offset (H) values, and any of the 200 Diameter (D) values. In most cases you will use the automatic tool length measurement features described below to set H values, and you will enter D values manually, based on the known or measured diameters of your tools.

Note that H01 and D01, H02 and D02, H03 and D03, etc. are displayed together on the same line for convenience only. The Height and Diameter Offset Numbers can be used independently; associations are made only in the Tool Library.

Height Offset

This is the distance the control adjusts Z-axis positions when tool length compensation (G43 or G44) is used with a particular H value. For example, if H001 is -1.0 and the job contains G43 H1, then CNC7 will shift all Z-axis positions down 1.0 to compensate for the shorter tool.

To edit the Height Offset entries move to the desired height offset number with the arrow keys, <Page Up>, <Page Down>, <HOME>, and <END>. You can choose to manually edit or automatically measure the value.

Height Offsets values are measured using the Z Reference position. The Z Reference position is the Z-axis position when the tip of the reference tool is touching the work surface. The reference tool should always be the longest tool.

The Height Offset value for end mills and drills is the difference between the Z-axis position when the tip of the tool is touching the work surface and the Z Reference position. The Height offset value for ball nose and bull nose cutters is the difference between the Z-axis position when the center of the tool is at the work surface and the Z reference position. Because it is not possible to position the tool in this way, you must instead move the tip of the tool to the work surface, and then manually edit the value to subtract the tool nose radius.

To manually edit a Height Offset value, simply type the desired value and press <ENTER>.

To manually measure Height Offset values, use the following procedure:

Establishing the Z reference position

Press <F1> to select the Z Reference setting function.

Insert the longest tool into the tool holder (you can use the jog keys or the TOOL CHECK key to assist you).

Jog the tip of the tool down to the top of the work surface.

Press <F10> to save this Z Position as the Reference Position.

* NOTE: The parts Z zero must be set before setting the Z reference.

Measuring each tool height (Z position for tool minus Z position for Reference tool)

Insert the desired tool into the tool holder (Jog keys or the TOOL CHECK key can be used to assist you).

Jog the tip of the tool down to the top of the work surface.

If the tool is a drill or end mill, press <F2> to measure the height.

If the tool is a ball nose or bull nose cutter, press <F2> to measure the height, and then subtract the tool nose radius.

After a tool height is measured, the next Height Offset entry is automatically selected.

When the edit is complete, press <F10> to save the Offset Library and Exit.

Examples (assuming Z Reference = -1.5):

If the tool position is -1.75, then the tool height = -0.25

If the tool position is -1.75 and nose radius is .25, then the tool height = -0.50

If the tool position is -2.25, then the tool height = -0.75

If the tool position is -2.75 and nose radius is .125, then the tool height = -1.375

Diameter

This field tells the control the distance to adjust when cutter diameter compensation (G41 or G42) is used with a particular D value. For example, if D001 is 0.5 and the job contains G41 D1, CNC7 will adjust all X-Y positions 0.25 (half the tool diameter) to the left of the programmed tool path.

To edit the Diameter entries move to the desired diameter offset number with the arrow keys, <Page Up>, <Page Down>, <HOME>, and <END>. You must manually edit the Diameter Offset value. Type the desired value and then press the <ENTER> key.

You can make small adjustments to Height Offsets and Diameters using <F5> and <F6>. Use the arrow keys to highlight the value to be adjusted. Press <F5> to increase the offset value by 0.001" (or 0.02 mm in Metric mode). Press <F6> to decrease the offset by the same amount. If the cut parts are undersized, use <F5> to cut less material. If the cut parts are oversized, use <F6> to cut more material.

Automatic Tool Measurement

Z-minus single-surface probing, using the TT-1 tool touch-off post, is available in the Tool Offset Library.

NOTE: Make sure the proper parameters are set as per Chapter 8, and the detector is plugged in and is at the correct location on the table!

WARNING: When first testing the TT-1, hold the TT-1 in hand and touch the unit off the tool to confirm correct setup. Incorrect setup may cause damage to the machine, tool and/or operator.

Setting the Z Reference:

Using the longest tool for the job to be run or the designated reference tool, press <F1>, then <F3> and then CYCLE START. The Z-axis will then move down until the tool touch-off is detected. The Z reference will be set at that position.

Setting the Tool Height Offsets:

Pressing <F3> and then CYCLE START at the prompt will cause the Z-axis to move down until the tool touch-off is detected; the resulting tool length will be entered in the table (same as with <F2> Manual). The Z-axis then returns to its home position.

If Parameter 17 has been set to the number of a valid return point (1 or 2), the <F3> option will move the X and Y axes to that return point before moving Z down. Return point 1 is the G28 position from the Work Coordinate System Configuration screen (see Chapter 6). Return point 2 is the G30 position on that screen. If Parameter 17 is zero (0), the X and Y-axes will not move before Z moves down. In this case you must be careful to jog the machine directly over the detector before pressing <F3>.

Note: <SHIFT+F3> can be used to override any return point movement in cases where parameter 17 is set to use it. This is helpful for measuring tools wherein the height measurement is not taken from the center point of the tool.

Batch Tool Height Offset Measurement Process:

If you have both purchased the Tool Length Probing option **and** also have an automatic tool changer installed, then you can press <F4> to measure multiple tools in one process. After pressing <F4>, you will be prompted with the following dialogue box:

Enter the list of tools to measure.
Example: 1-4, 6, 15
----> █

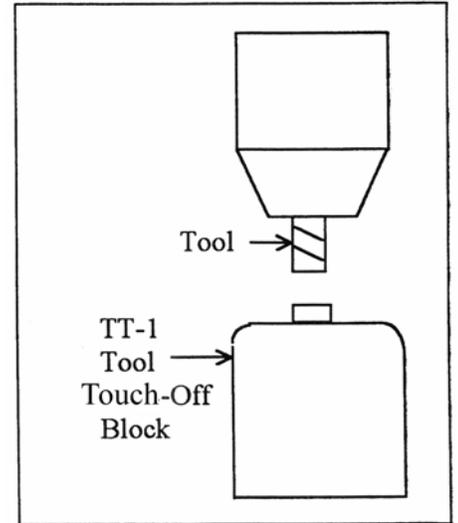
After entering a list of tool numbers, you can press CYCLE START to perform the batch tool measurement process. This process is similar to the single tool height offset measurement (accessed via <F3>) but will do multiple tools in one shot.

Setting up Tool Height Offsets

WARNING: Before manually jogging any probe to a position, make sure the machine Feedrate is slow (less than 10 in/min) or damage to the probe may result!!!

Using a Probe as the Reference Tool

Before you set the Z Reference, make sure the probe Tool # is entered into Parameter 12 on the Machine Parameters screen. Make sure that Parameter 17 on the Machine Parameters screen contains a 0. Follow these steps to probe Z Reference:



1. Load the probe into the machine.
2. Jog the probe over the desired reference surface and press <F1>.
3. Press <F3> and then CYCLE START; the probe will find the Z Reference.

At this point, the Z Reference is now entered into the Offset Library and is the reference height for all other tools. Remove the probe and measure any other tool offsets manually as described earlier in this chapter.

Measuring Each Tool Offset Using a Fixed Detector

Before measuring any tool height, make sure you enter the probe or reference tool-measuring location. Do this by entering a reference point number (1 or 2) into Parameter 17 and entering the detector position as the corresponding Reference Return Point on the WCS Configuration screen. Otherwise, the machine may traverse to a location that could damage the probe or reference tool. Also remember that if Parameter 17 is zero (0), the X and Y-axes will not move before Z moves down. Now that a permanent location has been set, do the following:

Load reference tool (preferably the longest tool) and highlight its corresponding Height Offset # using the up or down arrow keys.

Press <F1>, then <F3> and then CYCLE START to set the Z reference using this tool. The X and Y-axes will traverse to the preset location, then Z will move down until the tool is detected and the Z reference will be set.

Load the next tool.

Highlight the desired Height Offset # on screen using the up or down arrow keys.

Press <F3> and then CYCLE START. The X and Y-axes will traverse to the preset location, then Z will move down until the tool is detected. Once the detector is triggered, the tool offset will show on the screen. A negative offset means the tool is shorter than the reference tool.

Once all of the tool offsets have been measured, press <F10> to save them. Otherwise, press <ESC> to cancel any changes.

Tool Library

WCS #1 (G54)		Current Position (inches)		Job Name: ISLANDS.CNC			
X		+0.0000		Tool: T1 H---			
Y		+0.0000		Feedrate: 120%			
Z		+0.0000		Spindle: 0			
				Feed Hold: Off			
				Waiting for PLC operation			
				Processing...			
				Job finished			
				Stopped			
Tool Library							
Tool	Bin	Ht.	Dia.	Coolant	Spindle	Speed	Description
T001	000	H001	D001	FLOOD	CW	350	.325 end mill
T002	002	H002	D002	FLOOD	CW	1200	
T003	003	H003	D003	FLOOD	CW	550	
T004	004	H004	D004	FLOOD	CW	1234	
T005	005	H005	D005	OFF	OFF	0	
T006	006	H006	D006	OFF	OFF	0	
T007	007	H007	D007	OFF	OFF	0	
T008	008	H008	D008	OFF	OFF	0	
T009	009	H009	D009	OFF	OFF	0	
T010	010	H010	D010	OFF	OFF	0	Probe
Tool in spindle: 1							
Clear Bin F1		Clear All F2		Init F3		Save F10	

The definitions in the Tool Library associate tool (T) numbers with height offset (H) and diameter (D) numbers, the default coolant type, spindle direction, and spindle speed for the tool, and a text description of the tool. This information is used by the Intercon programming package (described in Chapter 10) to provide defaults whenever a tool change is selected. For enhanced ATC features, the (T) numbers are also associated with bin numbers. See Chapter 15 for more information about enhanced ATC features (parameter 160).

Note: If enhanced ATC features are not on, the cursor cannot be moved into the bin column and the message “Bin fields are locked.” will appear where the tool in spindle display is located. In addition, the F1, F2, and F3 keys only appear if enhanced ATC features are on.

You can inspect and change any of the 200 tool definitions. To edit a Tool Library definition move to the desired tool number with the arrow keys, <Page Up>, <Page Down>, <HOME>, and <END>. To change Height Offset numbers, Diameter numbers, default spindle speed values and the tool description, type a new value into the field and then press <ENTER>. To change the default spindle direction and coolant type press <SPACE> to cycle through the possible values. When the changes are complete press <F10> to save the Tool Library and exit.

Bin

This field specifies which bin location, or ATC carousel position, that the tool is occupying. Valid values are -1 (shown as dashes “---”) through the maximum number of tools specified by machine parameter 161. A value of 0 indicates that the tool is in the spindle. The F1-F3 keys will work when the cursor is in the Bin column.

<F1> Clear Bin – places dashes “---“ into the bin field (same as entering -1).

<F2> Clear All – places dashes into every bin field.

<F3> Init – sets T1 to Bin 1, T2 to Bin 2, T3 to Bin3, and so on, all the way up to the maximum number of tools specified in machine parameter 161.

Note: For enhanced ATC applications, the bin numbers will be updated when tool changes are completed. For random, or arm type tool changers, tools in the spindle are placed into the bin where the next tool is picked up, and not necessarily from the bin which it was originally taken.

Height

This field specifies a default Height Offset (H) number to use with each tool. Possible values are 1 to 200. Intercon uses this information to provide a default H value at each tool change. CNC7 also uses this information to correct for the length of the tool that is used to establish the Z-axis position in Part Setup (see Chapter 4).

Diameter

This field specifies a default Diameter (D) number to use with each tool. Possible values are 1 to 200. Intercon uses this information to provide a default D value at each tool change. To change the value type a new number and press <ENTER>.

Coolant

This field specifies a default coolant type to use with each tool. Possible values are FLOOD, MIST, or OFF. Intercon uses this information to automatically insert M7 or M8 after a tool change. To change the value, press <SPACE> until the desired value is shown.

Spindle

This field specifies a default spindle direction to use with each tool. Possible values are CW, CCW, or OFF. Intercon uses this information to automatically insert M3 or M4 after a tool change. To change the value, press <SPACE> until the desired value is shown.

Speed

This field specifies a default spindle speed to use with each tool. Possible values are 0 to 500000. Intercon uses this information to automatically insert an S code after a tool change. To change the value type a new number and press <ENTER>.

Description

This field contains a text description of the tool. The description will appear in a prompt message on the screen when CNC7 reaches a tool change (M6).

CHAPTER 5

Power Feed

(F4 from Setup)

The Power Feed screen is used to command axis movement. All the operations available on the Power Feed screen may also be performed in MDI with the appropriate M and G codes.

F1 - Absolute Power Feed

Press <F1> to move an axis to an absolute position, at a specified feedrate.

F2 - Incremental Power Feed

Press <F2> to move an axis an incremental distance, at a specified feedrate.

F3 - Free XY

Press <F3> to release power to the X and Y motors, allowing you to use your machine manually for these two axes.

F4 - Power XY

Press <F4> to apply power to the X and Y motors, allowing you to use your machine with the jog panel for these two axes.

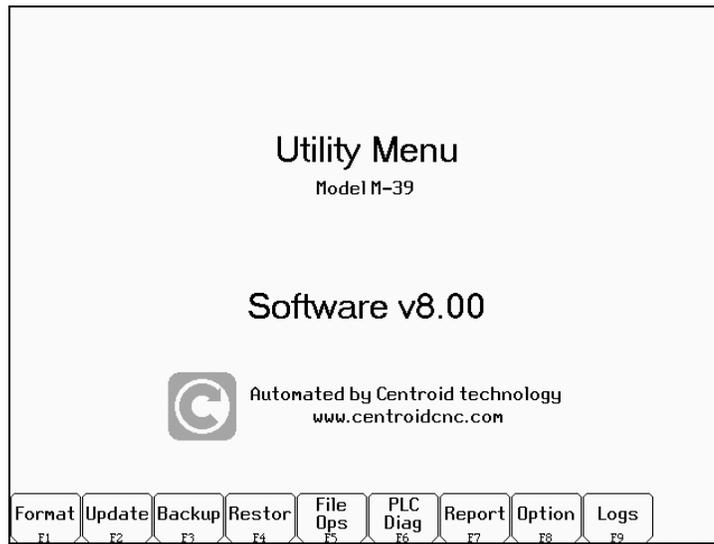
CHAPTER 6

The Utility Menu

To get to the Utility Menu, press <F7> at the CNC7 Main Screen. The model number will vary depending on your M-Series Control model.

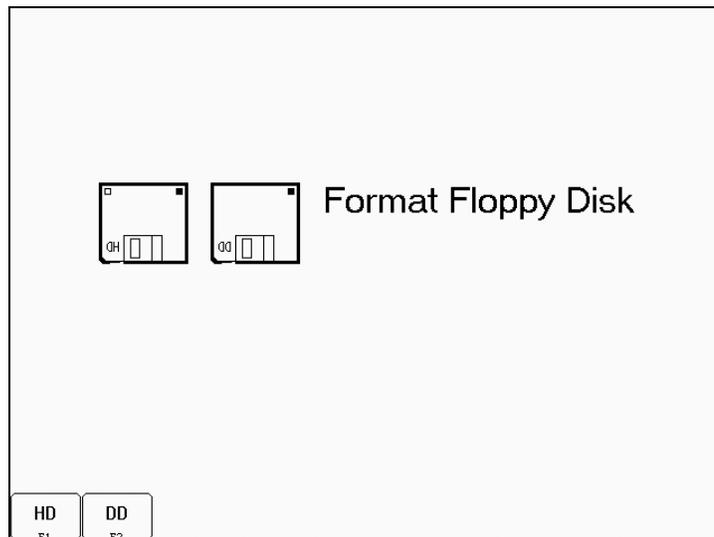
F1 - Format

Press <F1> to enter the Format Screen that gives you a choice of formatting either a high density or a low-density floppy disk. The marks that distinguish a high-density disk from a low-density disk are an extra hole and the letters "HD".



F1 - HD

Pressing <F1> at the Format Screen will display a prompt to press <ENTER>. If you press <ENTER>, the floppy disk will be formatted as high density (1.44M). If you do not want to format the disk, press <CTRL-C> to cancel the operation.



F2 - DD

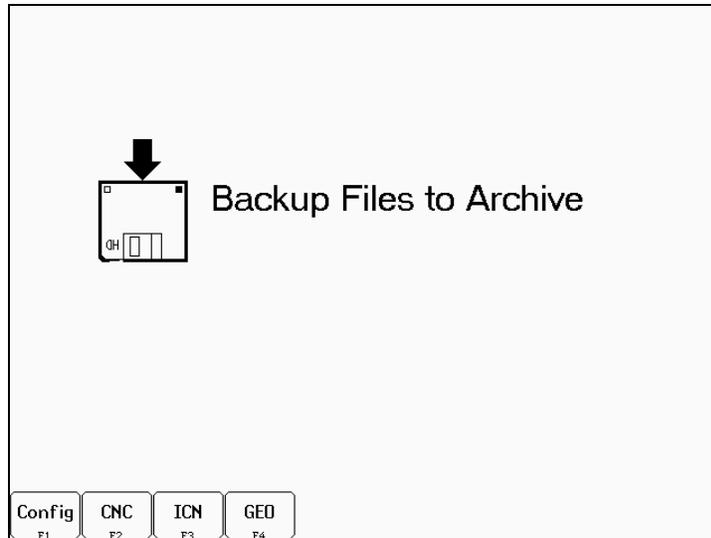
Pressing <F2> will perform the same function as above, except that the floppy disk will be formatted at double density (720K).

F2 - Update

To update your control software, put the update disk in the floppy disk drive and press <F2>. The new software will then be automatically loaded onto the hard drive. Once the new software is loaded, the controller must be powered down to use the new software. Failure to do this may cause unpredictable errors.

F3 - Backup

Press <F3> to enter the Backup Files screen. It is recommended that you back up the M-Series Control's files on a regular basis. Also, you should label your diskettes clearly after backing up.



F1 - Config

Press <F1> to backup the Control's configuration files to floppy disk.

*Hint: It's good to have a backup of your Configuration.

F2 - CNC

Press <F2> to list CNC files that are stored on the Controller (in the directory C:\CNC7\NCFILES). You can select the ones you want to backup with <F1> (or select all of them with <F2>), and then accept them with <F10>.

Follow the on-screen instructions. The selected files will be backed up to one or more floppy disks.

F3 - ICN

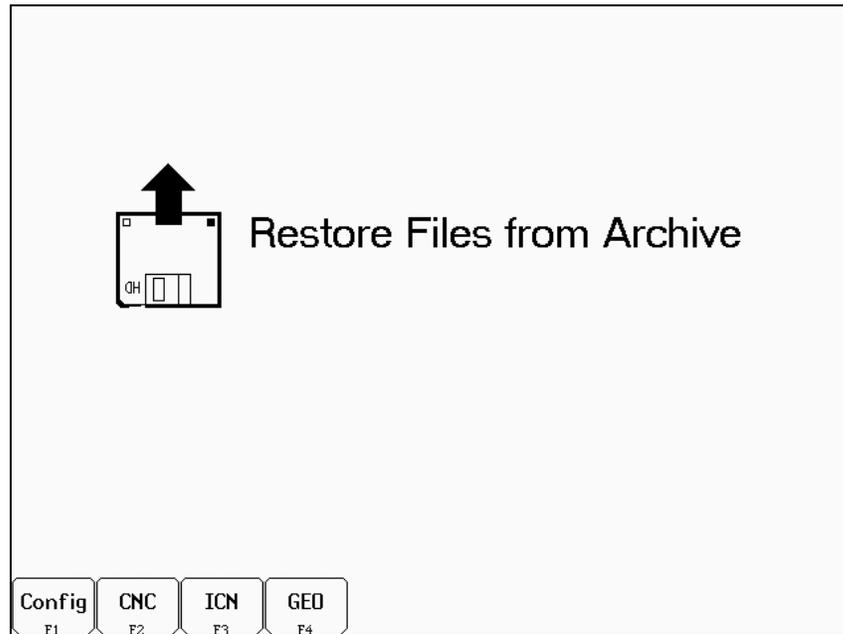
Pressing <F3> will do the same function as above, except that this will back up the .ICN files that are stored on the Controller (in the INTERCON directory specified in PATHM.INI).

F4 - GEO

Pressing <F4> will do the same as above, except that this will back up the Mastercam Geometry files that are stored on the Controller (in the directory C:\NC\GEO).

F4 - Restore

Press <F4> to restore files that were previously saved with the Backup (<F3>) option. When restoring files be sure to have the proper back up disk.



F1 - Config

Press <F1> on this screen to restore the Controller's configuration from a floppy disk backup.

F2 - CNC

Press <F2> on this screen to restore CNC files from a floppy disk backup. This will restore to the C:\CNC7\NCFILES directory.

F3 - ICN

Pressing <F3> will restore .ICN files from a floppy disk backup. This will restore to the INTERCON directory.

F4 - GEO

Press <F4> to restore Mastercam[®] Geometry files from a floppy disk backup. This will restore to the C:\NC\GEO directory.

F5 - File Ops

Press <F5> to access an additional file options screen. These options operate on the CNC files stored on the Controller's hard drive. (The controller's CNC files are stored in the C:\CNC7\NCFILES directory.)

F1 - Import

Press <F1> to list files on the floppy drive. Make sure that the files on the floppy disk are CNC files. You can select the ones you want to import with <F1> (or select all of them with <F2>), and then accept them with <F10>. The selected files will then be copied from the floppy drive to the Controller's hard drive.

F2 - Export

Press <F2> to list CNC files on the Controller's hard drive. You can select the ones you want to export with <F1> and then accept them with <F10>. The selected files will then be copied from the Controller's hard drive to the floppy drive.

F3 - View

Press <F3> to list CNC files on the Controller's hard drive. You can select the one you want to view with <F10> or <ENTER>. The first 19 lines of the file will be displayed. When you are done viewing the file, press <ESC>.

F4 - Delete

Press <F4> to list CNC files on the Controller's hard drive. You can select the ones you want to delete with <F1>, and then proceed with <F10>. The selected files will then be deleted from the Controller's hard drive.

F5 - Dig->CAD

Press <F5> to list Digitize data files on the Controller's hard drive (see Chapter 12). You can select the ones you want to translate to Mastercam's spline format with <F1>. Accept your selection with <F10>. The selected files will then be translated and placed in the CAD directory specified in PATHM.INI. The translated files will have an extension of. DOC.

F6 - PLC Diag

Runs PLC diagnostic program to check PLC I/O. Do not attempt to troubleshoot the PLC without the assistance of your dealer.

F7 - Report

Generates a backup of system configurations in a text file on the floppy drive. Your dealer may then use the disk for servicing and troubleshooting purposes.

F8 - Options

Shows the software options that you have purchased or added to your control.

F9 - Logs

Shows the messages and errors that have been logged by the control.

F1 - Errors

Displays the error/message log. Use <PgUp>, <PgDn>, <Home> & <End> to view. <Esc> to exit.

F2- Stats

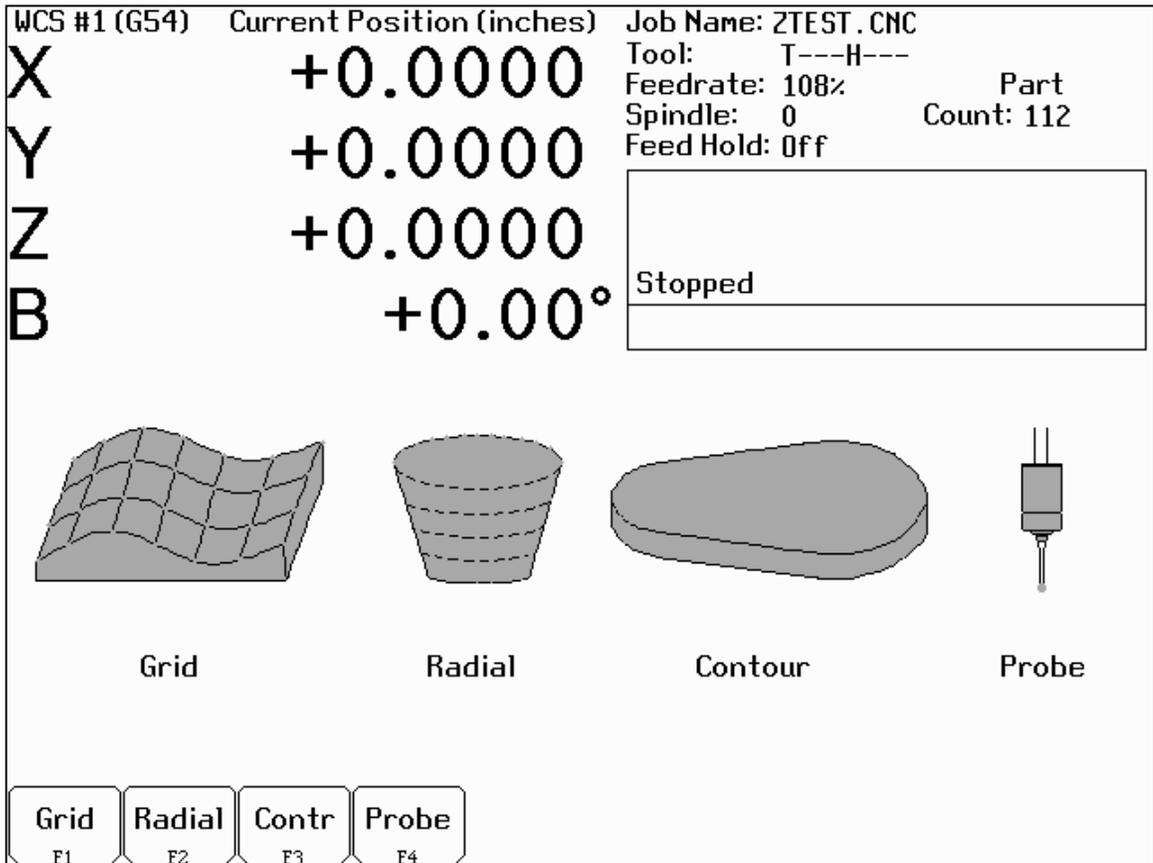
Displays counts of errors logged. Use <PgUp>, <PgDn>, <Home> & <End> to view. <Esc> to exit.

F3 - Export

Exports the log to a floppy disk. Insert a floppy and press <Enter>.

CHAPTER 7

Digitize (F9 from Main Menu)



The Digitize feature of CNC7 is used to digitize a rectangular surface area (grid), an inside circular bore area (radial), or a contour. The digitizing process creates a file with M & G codes that represent the digitized surface. If the digitizing probe tip is chosen to match the milling cutter size, the digitized file can be loaded and run to produce an exact copy of the digitized part.

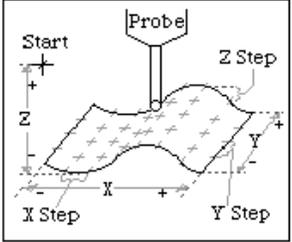
To digitize rectangular surface areas, press F1 (see grid digitize section). To digitize the inside of bores and wells, press F2 (see radial digitize section). To digitize the contour of a part press F3 (see contour digitize section). Press <F4> to select from the Probing Cycles (See Chapter 8 of this manual).

When using a continuity touch probe, clean the metallic surfaces you are digitizing using glass beading or some other suitable method. This allows for better contact and produces a more accurate digitizing.

Brushless motor note: If you experience excessive vibration on a brushless drive system, use Parameter 10 to select smooth deceleration in digitizing probing moves. See Chapter 15 for more information.

Grid Digitize

(F1 from Digitize Screen)

WCS #1 (G54)	Current Position (inches)	Job Name: E_Z_PART.CNC
X	+0.0000	Feedrate: 100%
Y	+0.0000	Spindle: 0
Z	+0.0000	Feed Hold: Off
Stopped		
Grid Digitizing		
1) Jog Probe Tip to Maximum Height	X Patch Length:	1.0000
2) Jog Probe Tip to Surface Corner	X Step Over:	0.0100
3) Edit the Digitize Parameters	Y Patch Width:	1.0000
4) Press CYCLE START to Begin	Y Step Over:	0.0100
	Z Maximum Depth:	1.0000
	Z Step Up:	0.1000
	Axis to Move First:	X
	Digitize File Name:	
	Replay Feedrate:	3.0000
	Multiple Patch:	No
	Replay Pattern:	Zig Zag
		
Save		Cancel
F10		ESC

Grid Digitize Run Setup

To set up a digitizing run, edit the parameters shown and then press CYCLE START. The control will move through the area to be digitized in a rectangular pattern. At each X-Y point in the pattern, it will measure the Z height of the sample surface, and record the coordinates in the data file.

Digitizing begins at the current tool position when the CYCLE START button is pressed. This position should be in one corner of the digitize area, at a Z position higher than any point on the surface.

Grid Digitize Parameters

X Patch Length: The length of the area to be digitized, along the X-axis. A positive value will cause digitizing to proceed in the X+ direction from the starting point; a negative value will cause digitizing to proceed in the X- direction. If the value is 0, then digitizing will collect just one stripe along Y.

X Step Over: The distance to move between points on the X-axis. A smaller value should be used for a fine digitize along the X-axis. A larger value should be used for a rough digitize along the X-axis. This distance should be a positive incremental value.

Y Patch Width: The width of the area to be digitized, along the Y-axis. A positive value will cause digitizing to proceed in the Y+ direction from the starting point; a negative value will cause digitizing to proceed in the Y- direction. If the value is 0, then digitizing will collect just one stripe along X.

Y Step Over: The distance to move between points on the Y-axis. A smaller value should be used for a fine digitize along the Y-axis. A larger value should be used for a rough digitize along the Y-axis. This distance should be a positive incremental value.

Z Maximum Depth: The maximum distance the Z-axis moves below the starting height. If the probe does not contact the surface at the maximum depth, that data point will be recorded as being *at* the maximum depth, and digitizing will proceed with the next point.

Z Step Up: The distance the Z-axis moves up after making contact, before the control attempts to move X or Y. A small value should be used when digitizing parts with gentle slopes; a larger value should be used when digitizing parts with many steep walls.

Axis to Move First: The axis (either X or Y) which moves all the way across the digitize area with each pass.

Digitize File Name: The base name of the file in which the digitize data is stored. The file has an extension of .DIG for CNC replay, and an extension of .DOC when translated for import into Mastercam[®] Level 2.

Replay Feedrate: The feedrate to include with the G1 command on the first line of the data file. If the data file is run as a CNC program, this is the feedrate at which the machine will retrace the digitized surface.

Multiple Patch: Indicates whether or not this digitizing is a continuation of an earlier digitizing. Choose NO if the current digitizing is the first or only digitize run for the part to be digitized. Choose YES if the current digitizing is not the first digitize run for the part. If Yes is selected, specify the name of a digitize file of a previous multiple patch.

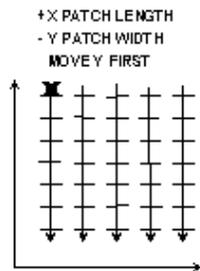
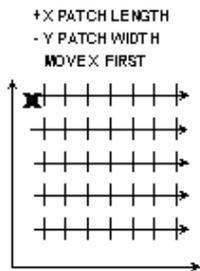
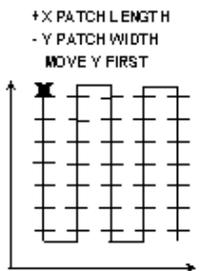
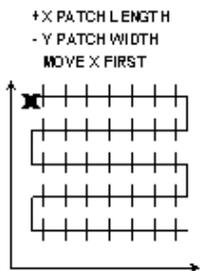
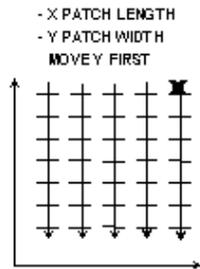
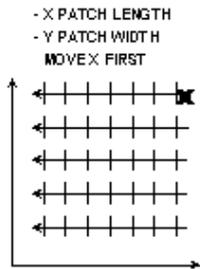
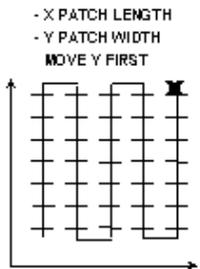
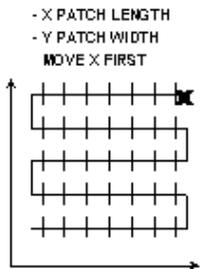
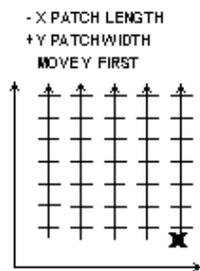
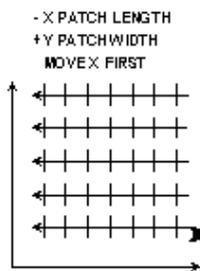
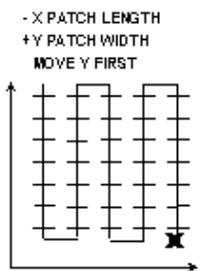
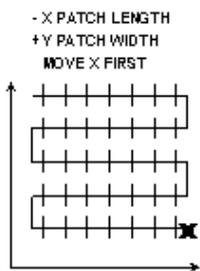
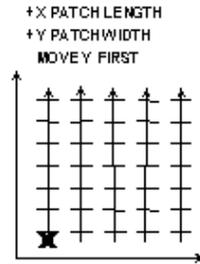
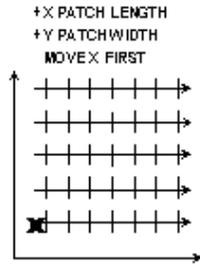
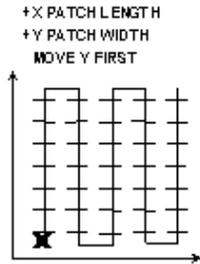
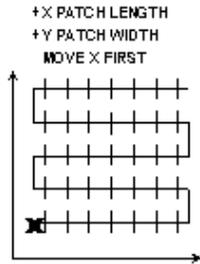
Replay Pattern: Indicates the replay movement pattern. If ZIG ZAG is selected, the replay pattern will alternate between positive and negative directions on each successive pass. If ONE WAY is selected, the replay pattern will maintain a constant "one-way" direction throughout the playback.

Grid Digitize Notes

1. A guide to the possible grid digitizing paths is as follows:

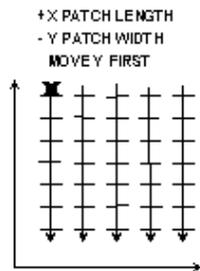
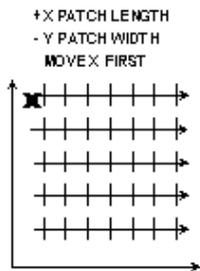
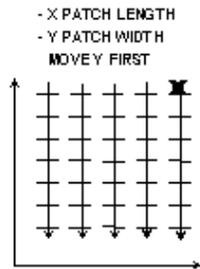
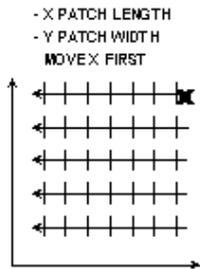
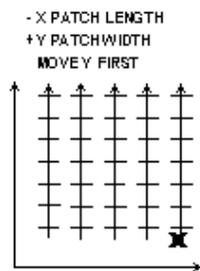
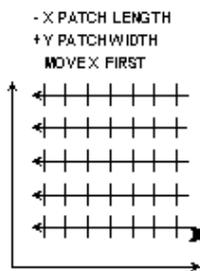
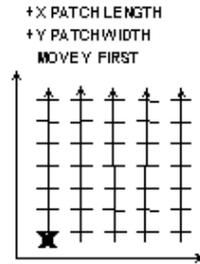
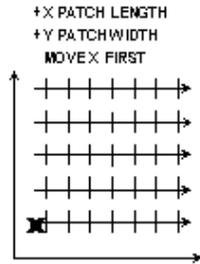
ZIG-ZAG REPLAY PATTERN

X INDICATES START POINT

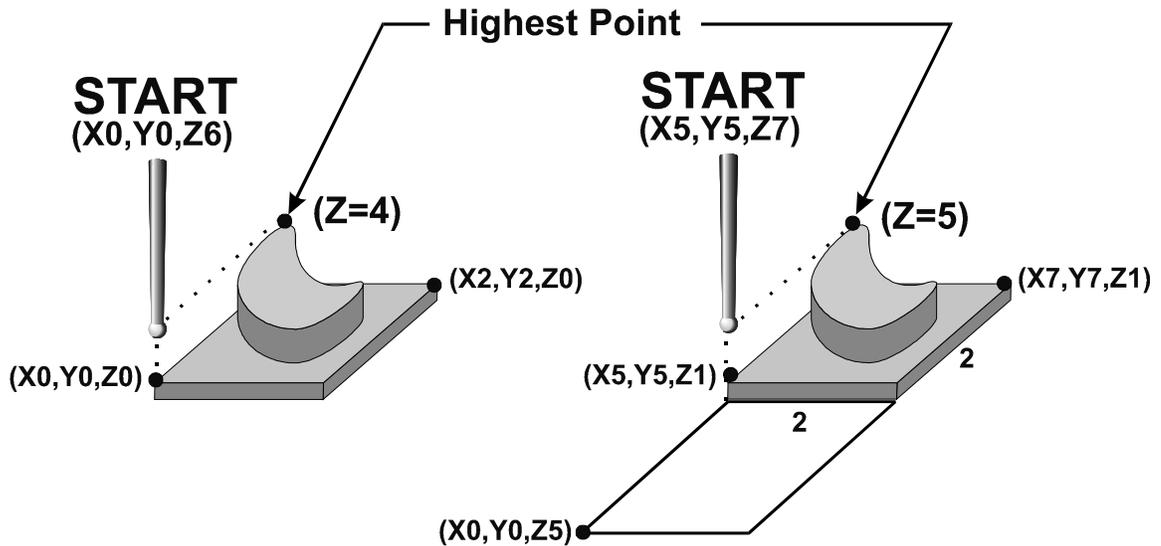


ONE-WAY REPLAY PATTERN

X INDICATES START POINT

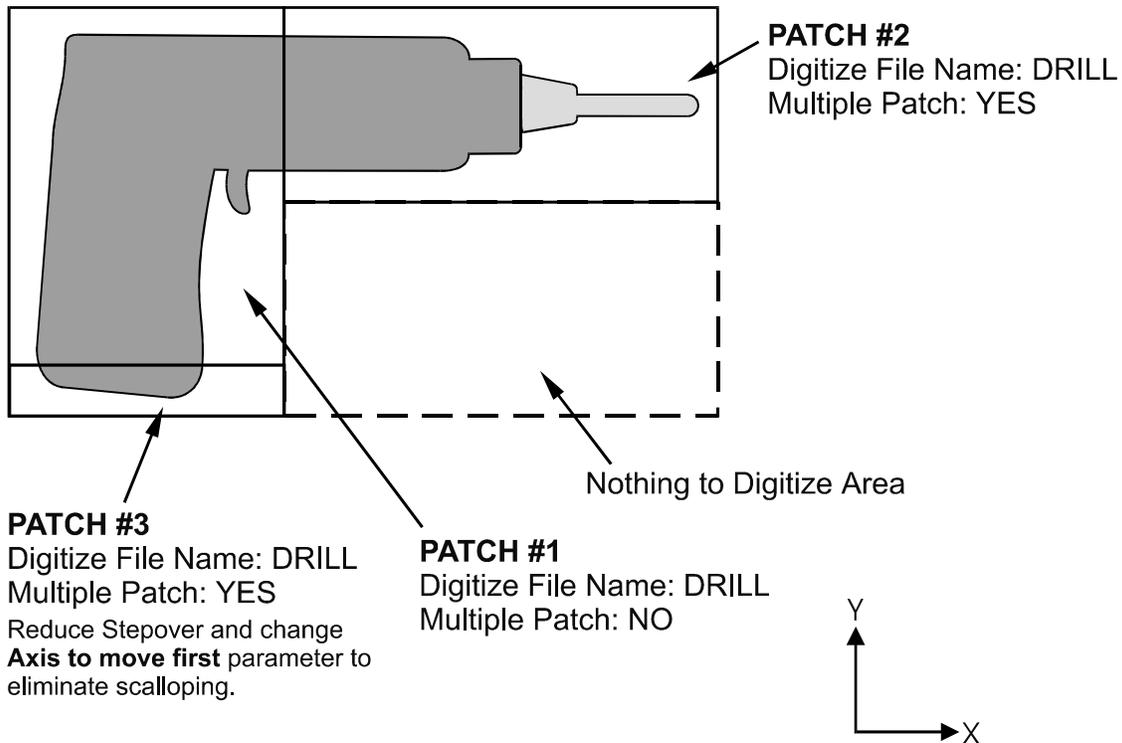


2. A digitizing patch can be located anywhere in the coordinate system. The digitizing starting point is referenced from the part zero. For example, setting up digitizing, as shown in the figure on the right below, will record the first point at (X5, Y5, Z1) and the last point at (X7, Y7, Z1). If the digitizing replay starting point is desired to be at the part zero, be sure to set the part zero equal to the digitizing start point, as shown in the figure on the left below. This orientation will record the first point at (X0, Y0, Z0) and the last point at (X2, Y2, Z1). For more information on part setup see Chapter 3.



3. A good technique for calculating Z maximum depth is to touch off the lowest surface of the part to be digitized and set the part zero's Z value to Z0. Then jog the probe tip to a point higher than the highest surface of the part to be digitized. Note the displacement in the Z-axis. Again, set this Z height to Z0 and use the noted displacement for the Z maximum depth.

4. Multiple patches are useful in the following situations: completing a canceled digitize run, digitizing parts with large areas that contain nothing to be digitized (shown below), and patching vertical walls to eliminate scallops caused by the cutting tool.



The drill shown in the previous example is L-shaped. Therefore, it can be digitized faster and more efficiently using three rectangular patches than digitizing the complete area with a single patch.

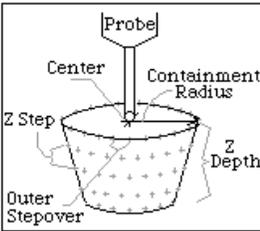
Digitizing the entire part and then adding multiple small patches along the walls can avoid vertical wall scalloping. A small rectangular patch extending the length of the vertical wall with the axis to move first set to the *opposite* axis along which the length of the wall extends is suggested (i.e. If a vertical wall extending along the X axis needs to be cleared of scallops, a small patch running the length of the wall with the Axis to Move First parameter set to Y would clear the scallops).

Radial Digitize (F2 from CNC7 Digitize Screen)

WCS #1 (G54)	Current Position (inches)	Job Name: E_Z_PART . CNC
X	+0.0000	Feedrate: 100%
Y	+0.0000	Spindle: 0
Z	+0.0000	Feed Hold: Off
		Stopped

Radial Digitizing

1) Jog Probe Tip to Starting Height	Containment Radius: 1.0000
2) Jog Probe Tip to Center of Well	Z Patch Depth: 1.0000
3) Edit Radial Digitize Parameters	Z Step: 0.0100
4) Press CYCLE START to Begin	Outer Stepper: 0.0100
	Replay Pattern: Zig Zag
	Replay Feedrate: 3.0000
	Digitize File Name:
	Containment Angle: Full
	Multiple Patch: No
	Move Between Levels: Clearance
	Clearance Height: 0.0100



Center	X: Not
Y: Set	

Center F1	Partial F2	Save F10	Cancel ESC
--------------	---------------	-------------	---------------

Setting up a Radial Digitize Run

To set up a digitize run, edit the parameters shown. Jog the probe tip to the starting height and to the center of the well to be digitized. Then press F1 to define the center position for digitizing. This center position will be used as the center of all radial digitizing runs until you leave the radial digitize menu or redefine the center. If you are using a full angle, you can now press CYCLE START to begin digitizing. If you have specified partial angle, press F2 to define the partial angle (see setting the partial angle section in this chapter). After defining the partial angle, pressing CYCLE START will start the digitize run.

WARNING: The probe must be able to retreat to the center from any position on the digitize surface. If the digitize surface contains features that do not allow for the probe to exit after entering, a probe crash may occur! See radial digitize note 2.

Radial Digitize Parameters

Containment Radius: The maximum distance from the center position to look for a digitize data point. This parameter is used to contain the probe within a circle with this radius centered at the center position. If the probe does not contact the surface before reaching the maximum radius, that data point will be recorded as being *at* the maximum radius, and digitizing will proceed with the next point.

Z Patch Depth: The depth of the patch to be digitized, along the Z-axis. A positive value will cause digitizing to proceed in the Z+ direction from the starting point; a negative value will cause digitizing to proceed in the Z- direction.

Z Step: The distance to move between points on the Z-axis. A smaller value should be used for a fine digitize along the Z-axis. A larger value should be used for a rough digitize along the Z-axis. This distance should be a positive incremental value.

Outer Stepover: The distance to move between points on one contour. A smaller value should be used for a fine digitize along any one contour. A larger value should be used for a rough digitize along any one contour. This distance should be a positive incremental value.

Replay Pattern: Indicates the replay movement pattern. If Zigzag is selected, the replay pattern will alternate between positive and negative angle directions (CW and CCW) on each successive contour. If CW or CCW is selected, the replay pattern will maintain a constant angle direction throughout the playback.

Replay Feedrate: The feedrate to include with the G1 command on the first line of the data file. If the data file is run as a CNC program, this is the feedrate at which the machine will retrace the digitized surface.

Digitize File Name: The base name of the file in which the digitize data is stored. The file has an extension of .DIG for CNC replay.

Containment Angle: Indicates whether or not the digitizing is to follow a full circle or a partial sector. Choose Full if 0 to 360 degrees is desired. Choose Partial if some other angles are needed. These partial angles can then be changed later (see setting the Partial Digitizing Sector Setup section that follows).

Multiple Patch: Indicates whether or not this digitizing is a continuation of an earlier digitizing. Choose No if the current digitizing is the first or only digitize run for the part to be digitized. Choose Yes if the current digitizing is not the first digitize run for the part. If Yes is selected, specify the name of a digitize file of a previous multiple patch.

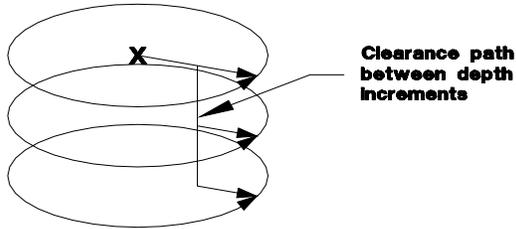
Move Between Levels: This field is enabled only if Partial and CCW or CW option is selected. It indicates the move between Z levels on replay of a partial sector radial digitize file. This move may now be done in three different ways: Clearance, which goes to the clearance height as in previous versions, Center, which goes to the digitizing center and then to the Z level of the next pass, and Direct, which goes directly to the starting point of the next pass.

Clearance Height: This field is enabled only if Partial, CCW or CW replay pattern, and Clearance Move type option is selected. This distance indicates the clearance height needed to move the cutter from the end of one contour to the beginning of the next contour. This distance should be a positive value.

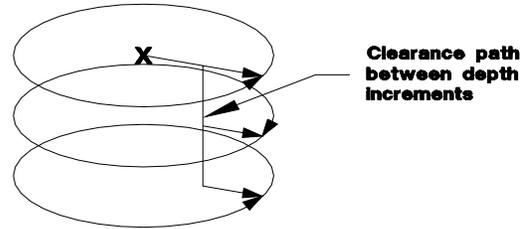
Radial Digitize Notes

1. A guide to possible radial digitizing paths is as follows:

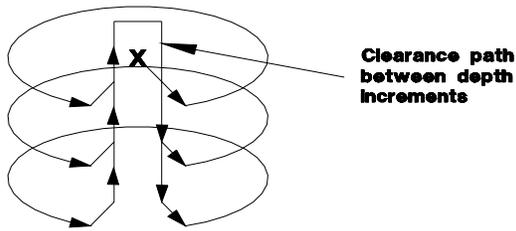
**FULL CONTAINMENT ANGLE
ONE WAY REPLAY PATTERN**



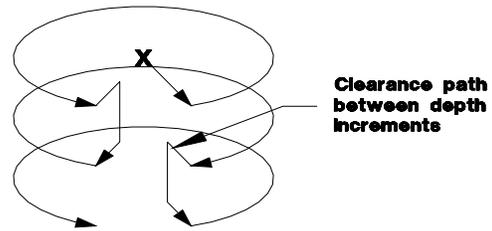
**FULL CONTAINMENT ANGLE
ZIG ZAG REPLAY PATTERN**



**PARTIAL CONTAINMENT ANGLE
ONE WAY CLEARANCE REPLAY PATTERN**

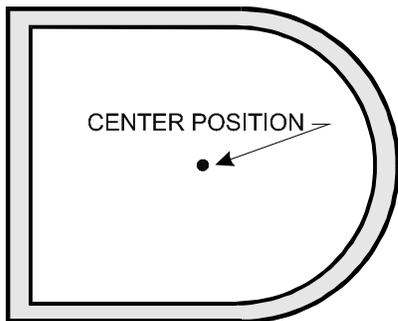


**PARTIAL CONTAINMENT ANGLE
ZIG ZAG DIRECT REPLAY PATTERN**

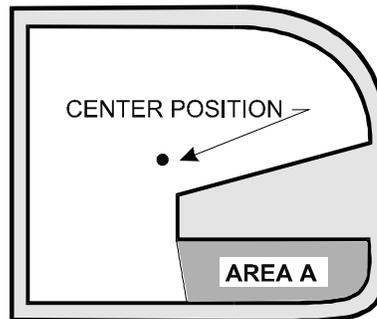


2. When radial digitizing, make sure the probe can fully retract to the center position without obstructions. Observe the two parts below. The cross section on the left has no obstructions that could keep the probe from full retraction to the center position. The cross section on the right does not allow the digitizing to retract to the center in **Area A**. This area will cause a probe crash; single patch digitizing of parts such as this should be avoided. Use 2 or more patches to digitize the part on the right (in this case, you could divide the part in half horizontally, and do each half separately).

IDEAL PART

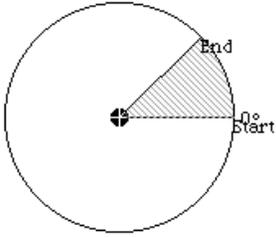


PROBLEM PART



Partial Digitizing Sector Setup

If you set the Radial Digitize Containment Angle to "Partial" then you must set up the Digitizing Setup by pressing <F2> from CNC7 Radial Digitize Screen.

WCS #1 (G54)	Current Position (inches)	Job Name: E_Z_PART .CNC
X	+0.0000	Feedrate: 100%
Y	+0.0000	Spindle: 0
Z	+0.0000	Feed Hold: Off
		Stopped
<p>Partial Digitizing Sector</p> <p>1) Edit the angle fields on the right, OR Jog to the start or end point of the sector, then use Start (F1) or End (F2) to set the start or end points.</p> <p>2) Press F10 to Save changes or ESC to exit.</p> <p>Start Angle: 0.00 End Angle: 45.00</p>		
		
Start F1	End F2	Save F10 Cancel ESC

The partial sector can be by one of two methods:

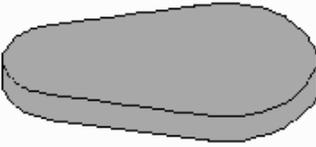
One method is by editing the start and end angles directly. The start angle is referenced from zero degrees and defines the beginning of the digitizing sector. The end angle is referenced from zero degrees and defines the end of the digitizing sector.

The second method involves jogging the probe tip and touching off the digitize surface. By moving the probe tip to positions on the digitizing surface, one can set the angles. To set the start angle, jog the probe tip to the position on the digitizing surface where the digitizing is to begin and press <F1> to define this as the start angle. Notice that the picture of the sector and the start angle's value change to reflect these settings. To define the end angle, follow the procedure above but press <F2> instead to set the end angle.

Regardless of the method used to define the start and end angles, pressing <F10> saves the angles and exits back to the radial digitize menu. Pressing <ESC> will return to the radial digitize menu without making changes to the start and end angles.

Contour Digitize

(F3 from CNC7 Digitize Screen)

WCS #1 (G54)	Current Position (inches)	Job Name: ZTEST.CNC
X	+0.0000	Tool: T---H---
Y	+0.0000	Feedrate: 108%
Z	+0.0000	Spindle: 0
B	+0.00°	Feed Hold: Off
		Part Count: 112
		Stopped
Contour Digitizing		
1) Jog probe tip to center of part.	Copy Type	CAM
2) Press F1 to define center of part.	X Patch Length:	0.0000
3) Jog probe to start pt.	Y Patch Width:	0.0000
4) Press CYCLE START to Begin	Axis Step	0.0100
	Digitize File Name:	
	Replay Feedrate:	10.0000
	Plunge Rate	10.0000
	Z Surface Height	0.0000
	Z Clearance	0.1000
	Z Depth	0.0000
	Z Depth Increment	0.0000
	Probe Diameter:	0.1250
	Center	
	X: Not	
	Y: Set	
	Press SPACE to change	
Center F1		Save F10

Contour Digitize Run Setup

To set up a digitizing run, jog the probe tip to the center of the part and hit <F1> to assign that as your center point.

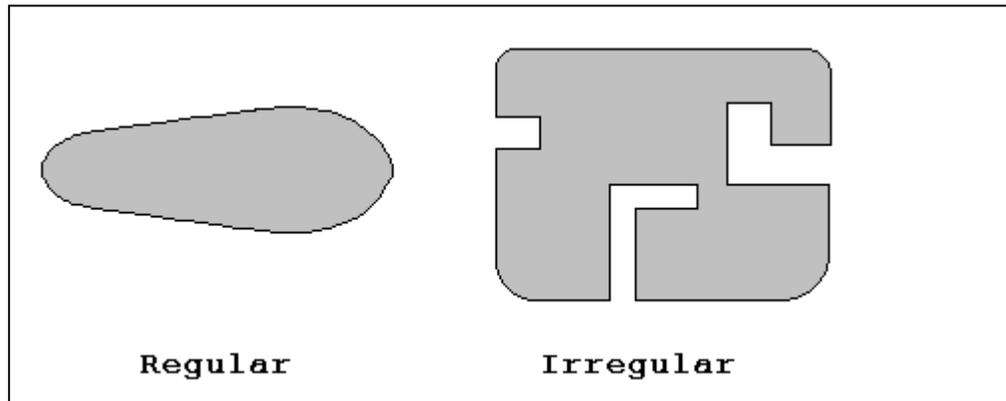
Select CAM for a true CAM shape contour, or Wall for irregular shapes for wall following. Enter the rest of the parameters for the part and digitizing job as shown.

Jog the probe to a starting point and press CYCLE START. The control will move the probe toward the center line in the +/- X direction until it comes into contact with the part. At each point of contact, the X and Y coordinates will be recorded in the data file.

The probe will continue around the contour until it returns to the starting point to complete the cycle. Based on the starting point, and the first point of contact, the digitize software will determine if the contour is internal or external.

Contour Digitize Parameters

Copy Type: Toggle between CAM or Wall. Use CAM for precise replication of regular shapes and use Wall following for contours with irregular shapes.



X Patch Length: The length of the contour to be digitized, along the X-axis.

Y Patch Width: The width of the contour to be digitized, along the Y-axis.

Axis Step Over: The distance to pull back away from the surface in the X and/or Y direction between . A larger value should be used for a rough digitize along the Y-axis. This distance should be a positive incremental value.

Digitize File Name: The base name of the file in which the digitize data is stored. The file has an extension of .CAM for CNC replay and is stored in the C:\CNC7\NCFILES directory.

Replay Feedrate: The feedrate to include with the G1 command on the first line of the data file. If the data file is run as a CNC program, this is the feedrate at which the machine will retrace the digitized surface.

Plunge Rate: The feedrate the Z axis plunges between successive depth passes.

Z Surface Height: Surface height of material for reproduction of digitized parts.

Z Clearance: Clearance to rapid to above surface of part during replay.

Z Depth: Depth of the part as measured from the surface height.

Z Incremental Depth: Depth of cut for each Z step of the part.

Probe Diameter: Diameter of probe tip used to digitize the part.

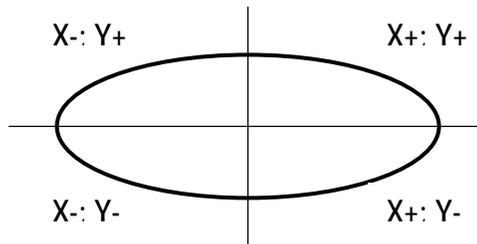
Contour Digitize Notes

Contour digitizing creates an M&G code file with a .CAM extension. The structure of the .CAM file starts with a header of comments indicating some of the parameters used when digitizing the contour, followed by the contour itself which is output as a subprogram. Specifically, the M&G codes for the contour are preceded by an O9800 (start of subprogram) and followed by an M99 (end of subprogram). The end of the .CAM file contains the initial positioning moves and a call to the contour subprogram (G65 P9800).

Probing direction: When starting the digitizing cycle, choose a starting point where the X travel will contact a point on the cam on the Y axis. The probe tip will move toward the center line in the X direction until it contacts the cam, then will move either clockwise or counterclockwise around the cam, depending on which quadrant you started the cycle (see Table 1).

Table 1 – Probe direction by starting quadrant

X	Y	Probe travels
-	-	CW
-	+	CCW
+	+	CW
+	-	CCW
-	0	CW
+	0	CCW



If the probe bypasses a contact point on the Y axis, it will continue moving in the X direction across the center line until it reaches the patch length limit and faults out.

Canceling a job: Unlike grid or radial digitizing, if you cancel a contour before it is completed, you will not be able to backplot to the point of interruption to continue the cycle. You will need to start over.

Before running a job: Before running a job created by contour digitizing, you will need to add some information to the file to define any required tool change, cutter compensation and height offset commands.

1. Do a search in the G code for the phrase “Add Comp Here.”
2. Refer to the descriptions of G40, G41 and G42 in Chapter 12.
3. Add the proper G code to the file after the “Add Comp Here” prompt.
4. Save the file, and run your job.

CHAPTER 8

Probing

NOTE: Refer to the Probe Parameters section at the end of this chapter before using any probe.

Part Setup with Probing

Single axis, single surface probing is available on the Set Part 0/Position screen using the Auto (<F4>) key. This allows you to probe various surfaces to define the part coordinate system. Multi-axis and multi-surface probing cycles are available on a separate screen, accessible from Set Part 0/Position with the Probing (<F5>) key. These allow you to locate the center points and corners of differently shaped parts. To enter Part Setup from the Main Screen, press <F1> and then <F1> again.

Brushless motor note: If you experience excessive vibration on a brushless drive system, use Parameter 10 to select smooth deceleration in probing moves. See Chapter 15 for more information.

WCS #1 (G54)	Current Position (inches)	Job Name: A. CNC
X	+0.0000	Tool: T1 H---
Y	+0.0000	Feedrate: 100%
Z	+0.0000	Spindle: 0
B	+0.00°	Feed Hold: Off

MDI...
 Operator abort: job cancelled
 MDI...
 Stopped

Set Part 0/Position

- 1) Select Axis with F1
- 2) Jog to Touch Off on Part
- 3) Edit the Value if Necessary
- 4) Press F10 to Set Position

Axis	Part Position	Edge Finder Diameter	Approach From
X	0.0000	0.0000	Left (-)

WCS #1 (G54)

Press SPACE to change

Next Axis F1	Auto F4	Probe F5	Prev WCS F6	Next WCS F7	WCS F9	Set F10
-----------------	------------	-------------	----------------	----------------	-----------	------------

WARNING: Before manually jogging any probe, make sure the machine Feedrate is slow (less than 10 in/min) or damage to the probe may result!

Automatically Setting Part 0

Part zero can be found using the probe. The Probe Tool Number is defined in Machine Parameter 12. Make sure the probe length offset and diameter are set properly for this tool in the Offset Library. The Edge Finder Diameter will be set automatically.

To set part 0 using the probe:

1. Select the current work coordinate system by pressing <F6> or <F7>. Then select the axis you want to probe by pressing <F1>. <F1>, <F6> and <F7> are described in Chapter 4 (Part Setup).
2. Manually jog the probe about 1/2 inch away from the surface you wish to define. Make sure the approach direction to the part is set properly. Probe the selected axis by pressing <F4>. When the surface is found, the control will assume this point to be the new axis 0.
3. If you want this probed surface to be something different than 0, enter the value by the using arrow keys to highlight Part Position. Type in the value and press <F10>.

Repeat steps 1-3 to set the remaining axes using the probe. Any previously entered Edge Finder Diameter or Tool Number value will be discarded.

Finding Center/Corner Points

You can locate a point in the center of a bore, boss, slot, web, or channel using the probing cycles. You can also find corner points even when they are not at right angles. No Edge Finder Diameter need be entered, since these cycles place the probe directly over the center or corner of the part. To enter the Probing Cycles screen, press <F5> from the Set Part 0/Position screen.

Calibrating the Probe Tip Diameter

You can calibrate the probe tip diameter to compensate for pre-travel (the amount that the probe deflects before it actually trips). Simply enter a probe tip diameter of zero, probe out a precision bore with a known diameter, and enter the difference between the reported bore diameter and the found bore diameter as the actual probe tip diameter.

Probing Cycles

You can enter the Probing Cycles screen from either the Set Part 0/Position screen or the Digitize menu. The Probing Cycles screen is shown below:

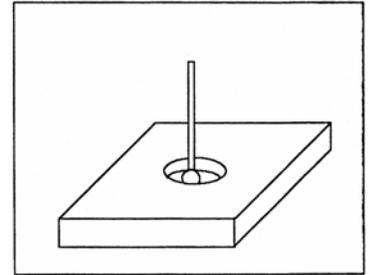
WCS #1 (G54)	Current Position (inches)	Job Name: TEST.CNC
X	+0.0000	Tool: T---H---
Y	+0.0000	Feedrate: 100%
Z	+0.0000	Spindle: 0
B	+0.0000	Feed Hold: Off
		Stopped
Probing Cycles		
Probe diameter (Tool #101): 0.2000		
Bore F1	Boss F2	Slot F3
Web F4	In Cnr F5	Out Cnr F6
1 Axis F7	Exit ESC	

The probing cycles will report the location and dimensions, as applicable, of the probed feature in a floating dialog box. The dimensions are adjusted to compensate for the probe tip diameter, entered in the Offset Library (see Parameter 12). For your convenience, you can edit the probe diameter on this screen, as long as the Tool Number, as set in Parameter 12, is not 0. During the probing cycles, the probe will move at speeds specified in Parameters 14 and 15. Refer to the Probe Parameters section later in this chapter for more information.

F1 – Bore

Press <F1> to enter the Bore screen. A picture similar to the one shown at right will appear, with instructions. Follow these steps:

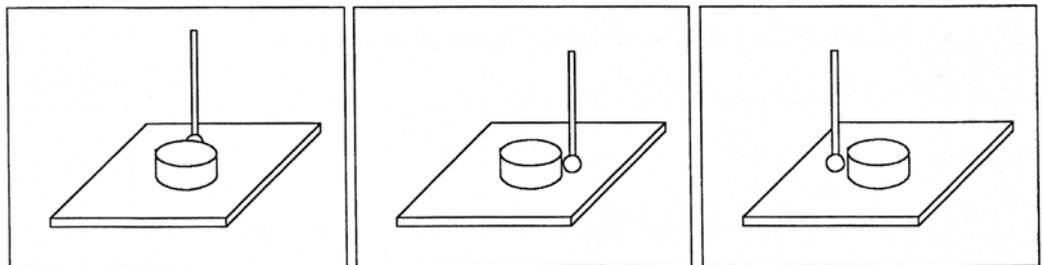
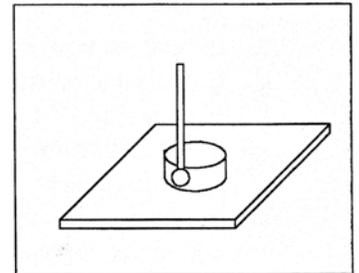
1. Make sure the probe is clear of any obstacles.
2. Manually jog the probe inside the hole. The probe tip should be just below the top edge of the surface.
3. Press <F10> to start the probing.
4. At the end of probing, the probe will be positioned at the center of the bore, and the X and Y positions will be shown on the screen.
5. Press <ESC> to return to the Set Part 0/Position screen.



F2 - Boss

Press <F2> to enter the Boss screen. A picture similar to the following will appear, with instructions and two input fields. Follow these steps:

1. Press <F1> to select the orientation of the probe with respect to the Boss. You will see one of the screens shown below.
2. Slowly jog the probe to the approximate orientation as shown in the picture. Be sure to give enough space for the probe tip to clear any obstacles during the jog.
3. Enter the approximate Boss diameter.
4. Highlight the Z clearance amount by pressing the down arrow key.
5. Enter approximate distance (in the Z direction) the probe must move to lift up over the Boss.
6. Press <F10> to start the probing cycle.



If the Z clearance you entered is too small, the probe will stop and show an error message. Correct the problem by repeating the previous steps. If the approximate diameter you entered is too small, the probe will bounce by 10 percent of its diameter across the top surface until: 1) It finds the correct edge; 2) The additional distance searched is equal to Parameter 16; or, 3) The travel limit is reached.

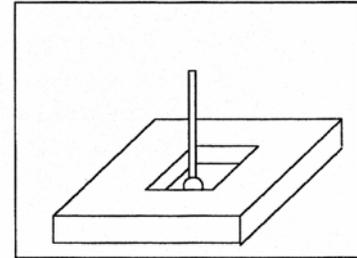
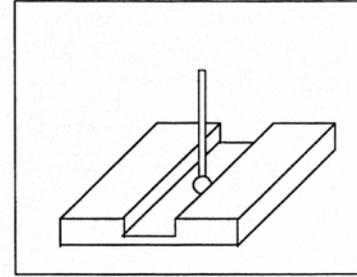
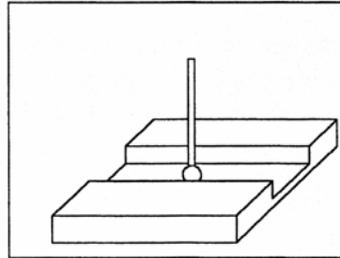
Once the probing cycle is complete, the probe will be positioned at the center of the boss, at the Z clearance level entered. Press <ESC> to return to the Set Part 0/ Position screen.

F3 - Slot

Press <F3> to enter the Slot screen. A picture similar to the ones shown will appear along with instructions:

1. Press <F1> to select the orientation of the probe with respect to the Slot.
2. Slowly jog the probe to the approximate position shown in the picture. During this jog, make sure you have enough space between the probe and the part and the part.
3. Press <F10> to begin the probing cycle.

Once the cycle is finished, the probe will be located at the center of the slot. Press <ESC> to return to the Set Part 0/ Position screen.

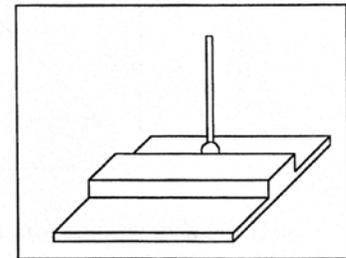
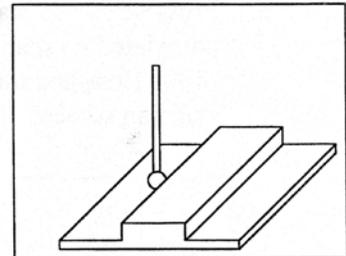
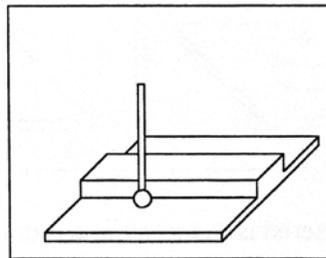
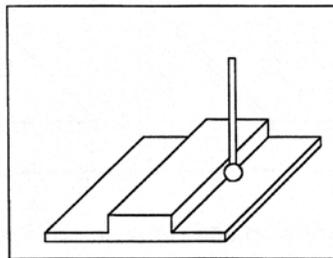


F4 - Web

Press <F4> to enter the Web screen. A picture similar to the following will appear, with instructions and two input fields. Follow these steps:

1. Press <F1> to select the orientation of the probe with respect to the Web. You will see one of the screens shown below.
2. Slowly jog the probe to the approximate position shown in the picture. During this jog, be sure to give enough space between the probe and the part.
3. Enter the approximate Web width.
4. Highlight the Z clearance value using the up or down arrow key. Enter approximate distance the probe has to travel in order to lift up over the Web.

Press <F10> to start the probing cycle. Once the probe has completed its search, it will automatically position to the center line of the web. Press <ESC> to return to the Set Part 0/ Position screen.

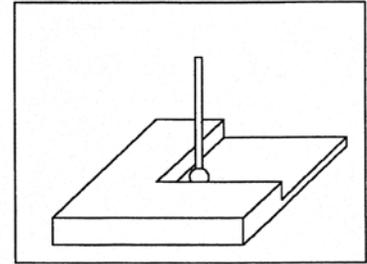
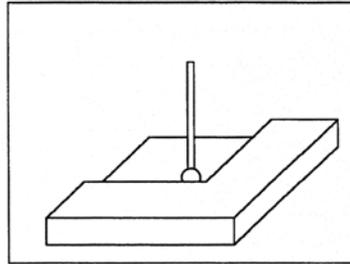
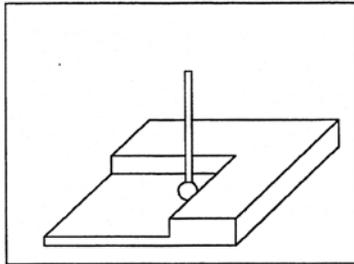
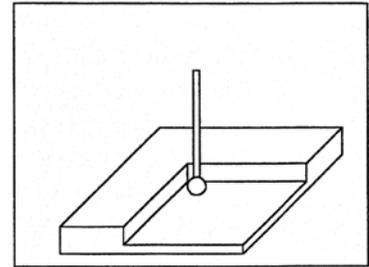


If the Z clearance you entered is too small, the probe will stop and show an error message. Correct the problem by repeating the previous steps. If the approximate width you entered is too small, the probe will bounce by 10 percent of its width across the top surface until: 1) It finds the correct edge; 2) The additional distance searched is equal to Parameter 16; or, 3) The travel limit is reached.

Once the probing cycle is complete, the probe will be positioned at the center of the web, at the Z clearance level entered. Press <ESC> to return to the Set Part 0/ Position screen.

F5 - In Cnr (Inside Corner)

Press <F5> to enter the Inside Corner screen. A picture similar to the following will appear, with instructions and an input field. The steps for this cycle are similar to that of a slot cycle. The main difference is that you need to enter a clearance amount. This clearance amount is the approximate distance upward on the Z axis that the probe needs to move to clear the corner. Press <F1> and the screen will cycle through one of the probe orientations shown here. If the corner is rounded, jog the probe far enough away for it to miss the curved area during the probing cycle (at least twice the corner radius).



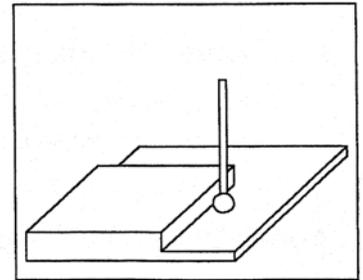
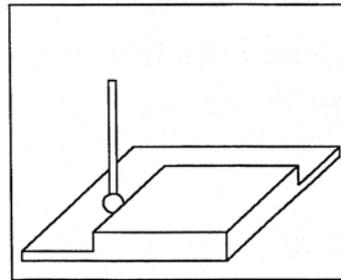
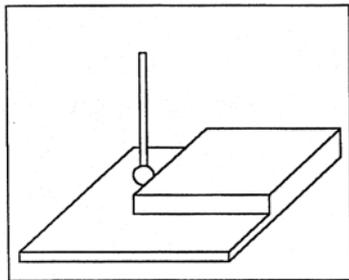
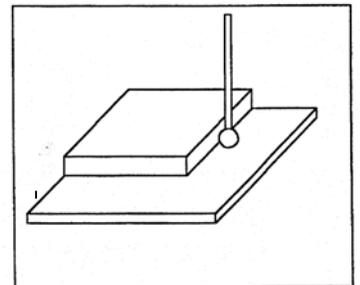
At the end of the probing cycle, the probe will be positioned above the corner at the Z clearance level entered. Press <ESC> to return to the Set Part 0/ Position screen.

F6 - Out Cnr (Outside Corner)

Press <F6> to enter the Outside Corner screen. A picture similar to the following will appear, with instructions and two input fields.

Follow these steps:

1. Press <F1> to select the orientation of the probe with respect to the Corner. You will see one of the pictures shown.
2. Press <F2> to select which side of the corner the probe will be positioned near. You will see the screen change.
3. Slowly jog the probe to the approximate position as shown in the picture. Be sure to give enough probe clearance.
4. Select the Z clearance field using the arrow keys. Enter the approximate distance the probe has to travel in order to lift up over the corner.
5. Select the Distance to Corner amount using the up or down arrow keys. Enter the approximate distance from the corner the probe is along the X or Y axis.
6. Press <F10> to start the probing cycle.

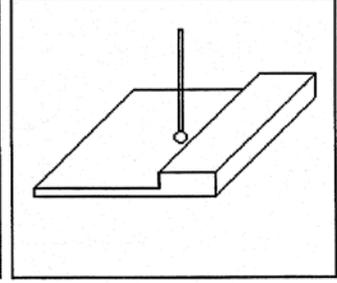
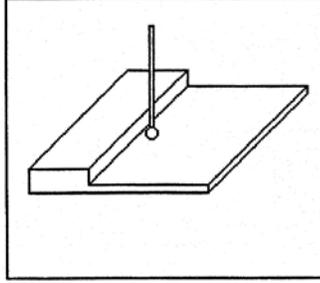
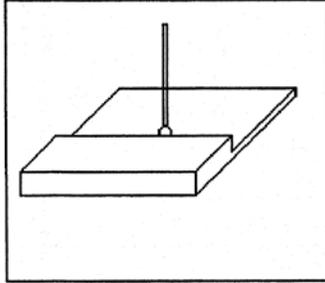
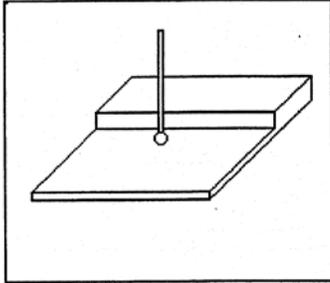
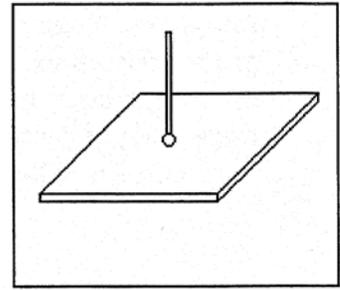


Once the probe has completed its search, it will be positioned above the corner at the Z clearance you specified. Press <ESC> to return to the Set Part 0/ Position screen.

F7 - 1 Axis (Single Axis)

Press <F7> to enter the Single Axis screen. Follow these steps:

1. Press <F1> to select the orientation of the probe. You will see one of the screens shown below.
2. Slowly jog the probe to the approximate position as shown in the picture.
3. Be sure to give enough probe clearance.
4. Press <CYCLE START> to start the probing cycle.



Once the probing cycle is complete, the probe will move away from the surface by the amount entered in Parameter 13 (Recovery Distance). The dimension where it found the surface will be shown on the control. Press <ESC> to return to the Set Part 0/ Position screen.

Probe Parameters

Various probing parameters can be set on the Machine Parameters screen (see Chapter 15). Make sure you enter these parameters before you begin using the probe. If these parameters are not entered properly, damage to the probe may result.

PLC Input Number and Contact State (Parameter 11) - A single value, +/-1 through 240. A positive number indicates Closed on contact; a negative number indicates Open on contact.

Touch Probe Tool Number (Parameter 12) - A single value, 0 through 200, used to look up the length offset and tip diameter of the probe in the Offset Library.

Recovery Distance (Parameter 13) - The additional distance the probe moves off of a surface after contact is broken, before attempting to traverse parallel to the surface.

Fast Probing Rate (Parameter 14) - Used for positioning moves and initial surface detection, this parameter is determined by machine response and permitted probe deflection as well as desired tolerance. The default setting is 25 inches/min.

Slow Probing Rate (Parameter 15) - Used for final measuring moves, this parameter is determined by a speed/accuracy tradeoff. The default setting is 3.5 inches/min.

Maximum Probing Distance (Parameter 16) - The maximum distance that a probing cycle "searches" for a surface in a given direction if no travel limits have been entered. The default is 10 inches. A larger value should be entered for the hole and slot cycles if you are measuring very large features.

Detector Location Return Point (Parameter 17) - A non-zero value specifies the number of the reference return point (entered into the WCS Configuration) directly above a permanently mounted tool detector. There are two separate return points available; enter 1 or 2. A Zero (0) indicates that the tool detector is not permanently mounted; automatic tool measurement will be performed without XY movement.

Chapter 9

Engraving

Introduction

Centroid offers an optional engraving package, which uses engraving software called Millwrite. Purchase of this option includes a user manual for the software.

To launch the engraving software, press <F5> from the main screen, and select Millwrite from the options shown on the screen.

To operate the engraving software, you can use the mouse and keyboard on conjunction with each other, or you can use the keyboard alone.

Since many operators do not use a mouse, these instructions are for keyboard only operation of the engraving software.

In general, the engraving software is a conversational-style programming format that prompts you for dimensional information about the letters or symbols that you want to engrave. Simply type in the letters you want to engrave and define where and how big you want them: the engraving software will create a G-code program to do just that!

When using the keyboard, there are two important keys that are the "key" to operation:

The <ESC> key will display a menu of choices. Use the highlighted letter to select that choice. For example, select Q for **Quit**, F for **File**, etc.

The Control Key <Ctrl> will display a choice of options that will be displayed at the bottom of the screen. For example, You can directly see a graphic representation of the information you have entered by pressing <Ctrl> and holding it down and pressing the letter V (for View). Press and hold down <Ctrl> and press the letter G to Generate a G-Code program from the dimensional information you have typed in.

Quick Start Engraving Software Tutorial

Follow the instructions on the Offline Millwrite Engraving Demo disk to start the program.
Follow the steps below after Millwrite has started.

Step 1: Enter some text

Begin by typing a few keys or a short message. For instance:

```
Reduce, Re-Use, Recycle
```

When you begin typing, an editor box will appear in the middle of the screen and the letters you type will appear there. Press <Enter> when you are finished entering text. Press the down arrow to begin another line of text.

Step 2: Select a symbol from the symbol libraries

A. Press <CTRL-C> (Choose symbol) Use the arrow keys or mouse to select "Recycling symbols".

Look for the keyboard assignment of the symbol you wish to select, in this case "V", or select it by double-clicking on the symbol with the mouse.

B. Use the right arrow key or mouse to move the box over to the font attribute section (right side) of the screen. Select the "Y" and enter a value of -1.000. This moves the "Y" center of this line text to Y -1.000.

Step 3: View the tool path

Press <CTRL-V>. This will display the text in graphics mode. Press <ESC> and choose <Q>uit graphics. If you used the mouse go to B, otherwise press <ESC> then <ENTER> and type (without quotes) "V". Press <ENTER>.

Step 4: Scale the symbol

Drag the mouse to the right then to the bottom of the screen. Select Scale. Place the cross hairs on the symbol and left click and hold. Dragging the mouse to the right increases the size and to the left decreases it. Increase the size to approx. 1" x 1" and release the mouse.

Step 5: Wrap the text

Drag the mouse to the right and then to the bottom of the screen. Select Radius. Place the crosshairs on the text "Reduce, Re-use, Recycle" and left click and hold. Dragging the mouse to the left decreases the radius, to the right increases it. Select a radius of approx. 1.1". (The radius is seen in the bottom center of the screen, labeled "New Radius".)

Step 6: Change the font

Right click on the "Reduce, Re-use, Recycle" text. When the font attribute box pops up select "Dutch outline italic". Press <ESC>.

Step 7: Generate the G-Code

Type <CTRL-G> to generate the G-code. Since you are running the demo, the G-code generated will contain B's.

Step 8: Save this G-code

Press <ESC> and then <F> for file, <S> to save. The default directory should be c:\cnc7\ncfiles and the default extension .cnc. Press <ENTER> after a filename is typed.

Step 9: Exit Millwrite

Press <ESC> and <Q>uit to return from the G-code editor. Press <ESC> again and E<X>it to DOS.

You will be prompted to choose to save the file. Choose no and you will be returned to the main screen of your controller.

Step 10: Load the G-code

Press <F2> to load the file that you saved in Step 4. At this point you may <F8> Graph to view a backplot of the G-code file, or, if your part and tool setup has been set, you may begin to engrave.

To learn more about Millwrite, run it again and load a sample file (<ESC>, <F>ile, <O>pen). Press [>] or move the mouse to the right. Make changes to the line of text such as the Height, Slant, Angle, and Wrap radius. View these effects as you did in Step 2.

CHAPTER 10

Intercon Software

Introduction

Centroid's Intercon Conversational Software allows you to quickly create part program right at the control without having to be a G-code expert. Intercon will prompt you to enter values from your print that describes the geometry of the part. Intercon will display graphics of the part as you are creating it, helping you quickly proceed through part programming.

Intercon generates a G-code program from the geometric information you have entered. This is an advantage in several ways:

1. The G-code program generated by Intercon can be edited using the built-in Centroid G-code editor (<F6>)
2. Intercon programs can be interrupted and restarted even in the middle of a canned cycle.

You can purchase an offline version of the Intercon software for use on your desktop PC. You will need to purchase a hardware key which will allow the offline version to run. Simply plug the key into the computer, install the required drivers, and run it.

Intercon Main Screen

When you access Intercon through the <F5> CAM option in the CNC7 Main screen, the part program will be displayed if the current job loaded in CNC7 had an associated Intercon program. If the job file in CNC7 did not have an associated Intercon program, the <F1> File menu will be displayed.

Intercon Mill v8.00				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	
0010	:Demo program				
0020	Rapid	1.0000	1.0000	1.0000	
0030	Rapid	4.0000	2.0000	1.0000	
0040	Line	7.0000	3.0000	1.0000	
0050	Arc CW	10.0000	3.0000	1.0000	
0060	Tool #1	0.0000	0.0000	Home	
0070	Tap	0.0000	0.0000	0.1000	
0080	Face	3.0000	6.0000	0.0000	
0090	Rect Poc	4.0000	-8.0000	0.0000	
0100	Circ Poc	4.0000	-8.0000	0.0000	
0110	Frame	1.7500	-6.5000	0.0000	
0120	Thread	4.0000	-8.0000	0.1000	
0130	M41 ; Select Low Gear Range				
0140	Rotary	4.0000	-8.0000	0.1000	Status Tool Number : 0 Diameter : 0.0000 Length : 0.0000 Cutter Comp : Off Feedrate : 0.0000 Coolant Type : Off (M9) Spindle Speed : 0 Spindle Dir. : Off (M5)
0150	Comp Left	4.0000	-8.0000	0.1000	
0160	Depth Rpt	3.0000	6.0000	-0.4000	
0170	End Prog	3.0000	6.0000	Home	

File (F1)
Modify (F2)
Insert (F3)
Cut (F4)
Paste (F5)
Copy (F6)
Copy Menus... (F7)
Graph (F8)
Setup (F9)
Post (F10)

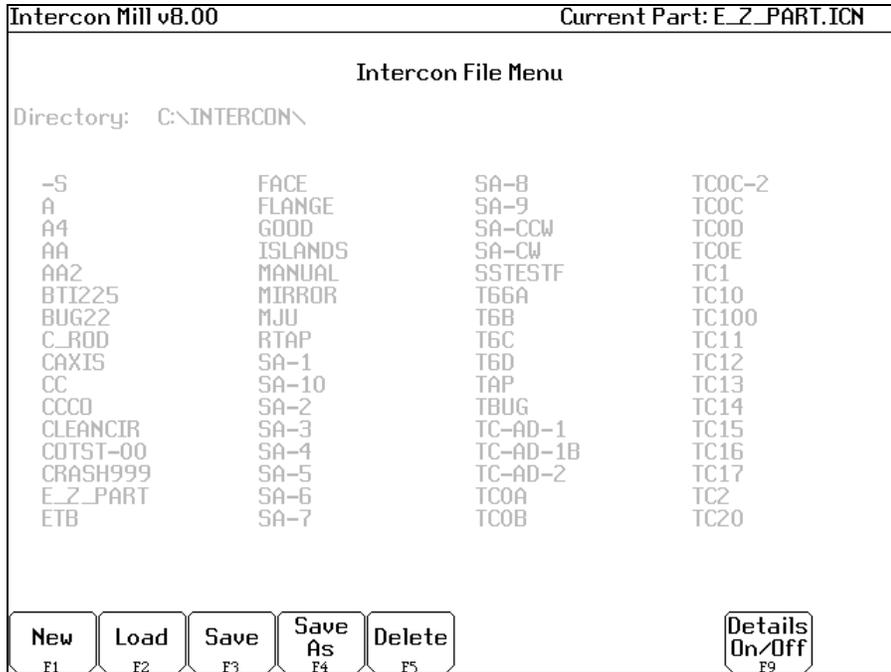
While in this mode, different operations can be navigated and highlighted for additional actions by using the arrow keys, and the <HOME>, <END>, <PAGE UP> and <PAGE DOWN> keys.

Teach Mode

The X, Y and Z keys will fill in a field with the current position for the related axis. This feature works when editing most fields in an operation. Press <F9> when editing an operation to display a DRO.

F1 - File

Choosing <F1> File will display the screen below. Intercon stores part designs in files identified with the extension **.ICN**. For example, if you specify the name of a given part design as E_Z_CAM, this part design will be saved on disk in a file called E_Z_CAM.ICN. The ICN files are only readable by Intercon.



F1 - New

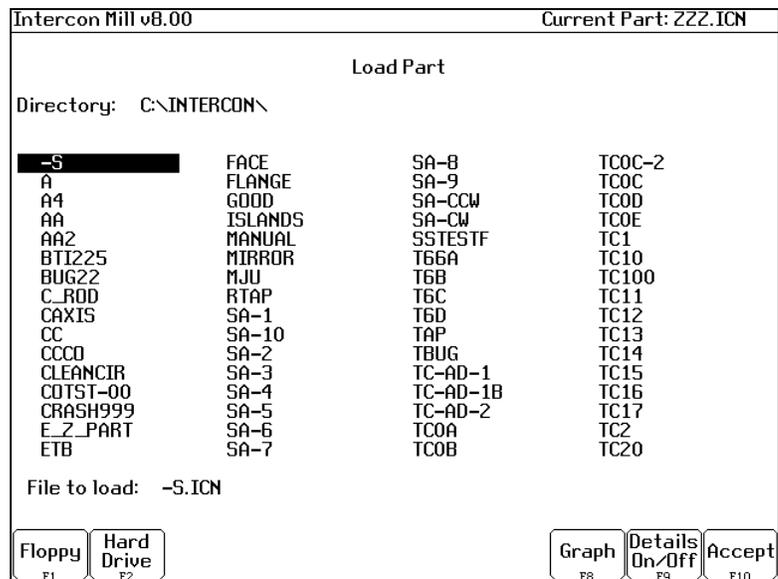
Choosing <F1> New will display the “New file:” prompt above the function keys. The name of the new program can be typed, followed by the <F10> or <ENTER> key to accept the new name. After accepting the new name, the program header information can be entered.

F2 - Load Part Program from Disk

When you press <F2>, the screen at right is displayed.

The program to be loaded is highlighted. The arrow keys can be used to move the cursor around and highlight the file to be loaded. The <HOME>, <END>, <PAGE UP> and <PAGE DOWN> keys can be used to navigate the list of files. Names that are bracketed, for example [...], are the names of directories in the current directory, which is displayed at the top of the screen.

It is also possible to start typing the name of the program to be loaded. When typing has started,



the characters appear in the “File to load:” prompt above the function keys. Different drives and directories can be accessed by typing in the path at the “File to load:” prompt, or by pressing <F10> or <ENTER> on a bracketed directory name.

The <F1> Floppy key will change to the “A:” drive and the <F2> Hard Drive option will change the directory to C:\INTERCON.

Choosing <F8> Graph will graph the highlighted Intercon file.

The <F9> Details On/Off changes the format of the display such that each file or directory is on a separate line and there are columns displayed for Programmer, Description, and Date Modified, i.e., the information that is contained in the program header operation.

When loading a new file, a prompt will be displayed asking whether to save the existing file if there was one.

F3 - Save

Press <F3> to save the current part program. The current program will be saved under the specified name.

F4 - Save As

Press <F4> to save the current program with a different name. Type the new name into the “Save part as:” prompt that appears above the function keys. If the new name already exists, a prompt will be displayed as a warning and will give the option to overwrite the existing file or return to enter a different name.

F5 - Delete

Press <F5> to delete a file. After <F5> is pressed, the screen will appear as in the <F2> Load option where the same keys can be used to navigate the files. A yes/no prompt will appear after accepting a file for deletion for final confirmation.

F2 - Modify

Choosing <F2> from the Intercon main menu will allow the currently highlighted operation to be modified. When an operation is modified, the fields for that operation are displayed on the right hand side. When modifying an operation, the <PAGE UP> and <PAGE DOWN> keys can be used to move up and down through the Intercon operations listed on the left hand side of the screen.

F3 - Insert

Choosing <F3> will insert a new operation before the operation that is currently highlighted, unless the highlighted operation is the first operation in which case the inserted operation will be inserted as the second operation.

F4 - Cut

Choosing <F4> will cut (remove) the highlighted operation from the program. The operation that is cut is placed onto the clipboard stack.

F5 - Paste

Choosing <F5> will paste the last operation that was cut or copied into the clipboard stack into the current program line that is before the highlighted operation. The number of operations that are currently in the clipboard stack is indicated by a number on the second line of the Paste key. As long as you stay in Intercon, the clipboard stack will remain intact. You may cut and copy operations from one program and paste them into a different program.

F6 - Copy

Choosing <F6> will copy the highlighted operation into the clipboard stack and advance the cursor to the next operation.

F7 – Copy Menus...

Choosing <F7> will display these options:

<F1> Copy Menu - allows a range of operations to be copied. Specify the Start Block, End Block, and Destination in the prompts that appear in the Copy Menu. The range of operations is copied into a location that precedes the destination block.

<F2> Move Menu - allows a range of operations to be moved. Specify the Start Block, End Block, and Destination in the prompts that appear in the Move Menu. The range of operations is moved into a location that precedes the destination block.

<F3> Cut, <F4> Paste, <F5> Copy perform the same actions as described above.

<F9> Clear Clipbrd - removes all operations in the clipboard stack.

F8 – Graph

Choosing <F8> will graph the current program. The graph is the same as what would be produced if the current program were translated into G-codes and graphed from CNC7 software. See Chapter 2 for more information about the Graph menu.

F9 - Setup

Choosing <F9> Setup will display the Setup menu where certain options can be set. The Setup menu appears as below.

Intercon Mill v8.20		Current Part: E_Z_PART.ICN	
Intercon Setup			
Comment Generation	:	Enabled	
Clearance Amount	:	0.1000	
Spindle/Coolant Delay	:	3.00	
Corner Feedrate Override	:	50.00%	
Modal Linear	:	No	
Modal Arc	:	No	
Modal Drill/Bore/Tap	:	No	
Rotary 4th Axis	:	No	
DRO Units	:	Inches	
Machine Units	:	Inches	
Help Icons always on	:	No	

Toggle Accept

F1 F10

Use the up and down arrow keys to move from field to field. Clearance Amount, Spindle/Coolant Delay, and Corner Feedrate Override require a value to be typed in; the other fields have fixed values which may be toggled by using the <F3> or <SPACE> keys.

Comment Generation: When this field is set to **Enabled**, Intercon will put a comment describing the operation type before each block. Disabling Comment Generation will make the CNC files generated by Intercon smaller.

Clearance Amount: This is the distance that Intercon raises the Z-axis above the programmed surface height in pockets, facing and frame mills when traveling across the work piece.

Spindle/Coolant Delay: Set this delay to the time in seconds you want Intercon to wait for the spindle to get up to speed and the coolant to begin flowing.

Corner Feedrate Override: This is the percent feedrate that will be used in the corners of rectangular pockets and inside frame mills. The default value is 50%.

Modal Operations: These options specify whether to automatically insert the same operation after the first has been accepted. Once modal insert mode has begun, press <ESC> to insert a different operation.

Rotary 4th Axis: This option specifies whether 4th axis movement fields appear in Linear and Rapid moves and whether or not the Intercon program will post 4th axis information. This option affects the value in machine parameter 94.

DRO Units: Specifies the Units used for the DRO in CNC7. It affects the corresponding field in the Control Configuration of CNC7.

Machine Units: Specifies the Units used for machining. It affects the corresponding field in the Control Configuration of CNC7. The posted G-code will contain a G20 for Inches mode and a G21 for Metric mode.

Help Icons always on: Toggle between yes or no. Selecting “yes” means that help information will always be displayed when editing operations. “No” means that you will have to press a key to get help. Whether set to “yes” or “no”, help screens can always be toggled on or off by pressing the <F5> key when editing an operation.

F10 - Post

Choosing <F10> will post the current program. Posting is the process of converting the operations into G-codes. When the posting process is completed, Intercon is exited. The Intercon program is also saved as part of the posting.

Insert Operation

When you press <F2> from the Edit Operation screen, or when you choose New Part from the Main Screen, you will see the Insert screen:

Intercon Mill v8.11					Current Part: E_Z_PART.ICN
Operation #	Type	X	End Y	Z	
0010	;Demo Program				Select operation to insert...
0020	End Prog	0.0000	0.0000	Home	

Rapid F1	Linear F2	Arc F3	Tool F4	Cycles F5	Other F6	Cutter Comp F7	Subpgm F9
-------------	--------------	-----------	------------	--------------	-------------	-------------------	--------------

The new operation will be inserted right before the currently highlighted one. The block number of the new operation is shown on the right side of the screen. The operation types that you can insert are listed across the bottom of the screen.

F1 - Rapid Traverse

Press <F1> from the Insert Operation screen to insert a Rapid Traverse. You may see the following screen:

Intercon Mill v8.11					Current Part: E_Z_PART.ICN
Operation #	Type	X	End Y	Z	
0010	;Demo Program				N0020 Rapid Traverse End X: 0.0000 Y: -5.0000 Z: 0.1000 Angle : 270.00° Length : 5.0000
0020	Rapid				
0030	End Prog	0.0000	0.0000	Home	

Abs Inc F1	Z Home F2	Math Help F6	Graph F8	Teach Mode F9	Accept F10
---------------	--------------	-----------------	-------------	------------------	---------------

End: When you first access the rapid traverse screen, the cursor will be highlighting the first field, End X. This is the X coordinate of where the cutter will be after the rapid traverse has been completed. Similarly, Y and Z represent the coordinates of the cutter after the rapid traverse is completed. The angle and length fields will be computed if you choose to enter the end point of the move.

Angle: The destination may also be specified in terms of a counterclockwise angle from the three o'clock position. When combined with a length for the current move, the corresponding X and Y coordinates for the destination will be calculated and placed in the correct fields. The Z destination will remain unchanged, however.

Length: The length of the rapid traversal. When combined with the angle of the current move, the corresponding X and Y coordinates for the destination will be calculated and placed in the correct fields. The Z destination will remain unchanged, however.

The <F1> key toggles between incremental and absolute positioning modes in any of the fields where a positional dimension is needed. For example: X, Y, or Z-axis dimensions can all be in incremental or absolute coordinates, or a mixture of both. The length and angle fields cannot be incremental. These fields are absolute values.

The <F2> key may be used on the Z destination field to tie the ending Z coordinate to the Z home position. This means that no matter what your Z home value is at the time that you run your program, the final Z position will be the Z home position.

When you are finished entering all of the dimensions for the rapid move, press <F10> to accept the operation and return to the Insert Operation screen.

NOTE: When making rapid moves, if a Z destination higher than the current cutter position is specified, the cutter will first be raised to the destination Z position, and then move linearly in X and Y to arrive at the destination. If a Z destination lower than the current cutter position is specified, the cutter will move linearly in X and Y first and then plunge Z to the destination Z position.

NOTE: The Rapid traverse operation can have rotary fields, if you have a rotary fourth axis. The rotary field descriptions are the same as that of the Linear Mill operation. See "F2 - Linear Mill" below.

F2 - Linear Mill

If you press <F2> for LINEAR from the Insert Operation screen, a screen similar to the following appears:

Intercon Mill v8.11			Current Part: E_Z_PART.ICN		
Operation #	Type	X	End Y	Z	
0010	;Demo Program				N0060 Linear
0020	Rapid	0.0000	-5.0000	0.1000	End X: 7.0000
0030	Tool #1	0.0000	0.0000	Home	Y: 3.0000
0040	Line	4.0000	0.0000	1.0000	Z: 1.0000
0050	Line	4.0000	2.0000	1.0000	Angle : 18.43°
0060	Line				Length : 3.1623
0070	EndProg	4.0000	2.0000	Home	Connect Radius : 0.0000
					Feedrate : 12.0000

Toggle
F3

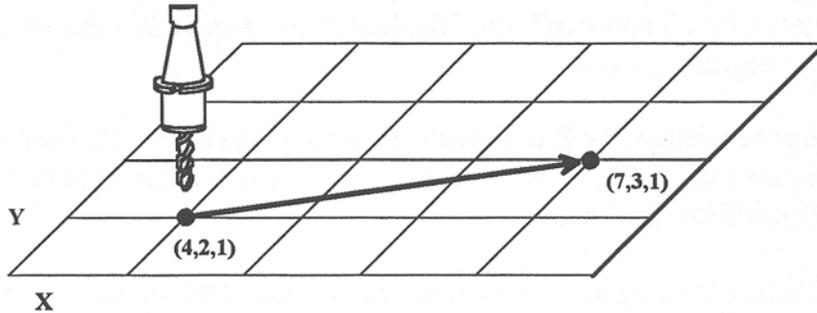
Math Help
F6

Graph
F8

Teach Mode
F9

Accept
F10

The numbers in the different fields on the screen correspond to the following Linear Mill example shown here graphically:



End: When you first access the linear mill screen, the cursor will be highlighting the first field, End X. This is the X coordinate of where the cutter will be after the linear move has been completed. Similarly, Y and Z represent the coordinates of the cutter after the linear move is completed. The angle and length fields will be computed if you choose to enter the end point of the move.

Angle: The destination may also be specified in terms of a counterclockwise angle from the three o'clock position. When combined with a length for the current move, the corresponding X and Y coordinates for the destination will be calculated and placed in the correct fields. The Z destination will remain unchanged, however.

Length: The length of the linear mill. When combined with the angle of the current move, the corresponding X and Y coordinates for the destination will be calculated and placed in the correct fields. The Z destination will remain unchanged, however.

Connect Radius: If you are performing two linear mill operations and you wish to have a rounded 'corner' between them instead of a sharp peak, you may enter the radius of the 'corner' and Intercon will insert an arc between the linear mill operations. This connect radius also works for blending a line into an arc operation.

Feedrate: Speed at which the cutter moves.

The feedrate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the feedrate field. If the feedrate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol shown below. The slave feedrate has no symbol and is set to the last modal feedrate set in the program, when the modal feedrate changes all the following slave feedrates change until the next modal feedrate is encountered.



If you have a fourth axis installed and it is rotary, additional fields are shown for Linear Mill operations after the feedrate field. This screen is illustrated above:

Degrees: The number of degrees you want to move the rotary axis. This value can be positive or negative and the movement of the rotary axis will depend on the orientation of the axis.

Minutes: The number of minutes you want to move the rotary axis. Values for this field are between 0 and 59.

Seconds: The number of seconds you want to move the rotary axis. Values for this field are between 0 and 59.

Decimal degrees: This is another method of entering the number of degrees. If you choose to enter the movement of the rotary axis with the fields listed above, the value of this field will be calculated automatically. If you choose to enter the number of degrees with this field or make changes to it, then the degrees, minutes and seconds will be calculated or changed automatically. Values for this field can be positive or negative.

Rotary movement defaults to zero degrees, incremental. To enter an absolute (rather than incremental) rotary position, you must press F1 (Abs/Inc) to toggle to absolute.

F3 - Arc Mill

If you press <F3> for ARC MILL from the Insert Operation screen, you will see the following screen:

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0050 Arc
0010	;Demo Program				Arc Type : EP & R
0020	Rapid	0.0000	-5.0000	0.1000	Mid X: 8.5000
0030	Rapid	4.0000	2.0000	1.0000	Mid Y: 4.5000
0040	Line	7.0000	3.0000	1.0000	Mid Z: 1.0000
0050	Arc CW				End X: 10.0000
0060	End Prog	7.0000	3.0000	Home	End Y: 3.0000
					End Z: 1.0000
					Center X: 8.5000
					Center Y: 3.0000
					Center Z: 1.0000
					Angle : 180.00°
					Radius : 1.5000
					Plane : XY
					Direction : CW
					Connect Radius : 0.0000
					Feedrate : 20.0000
					Angle <= 180 : Yes

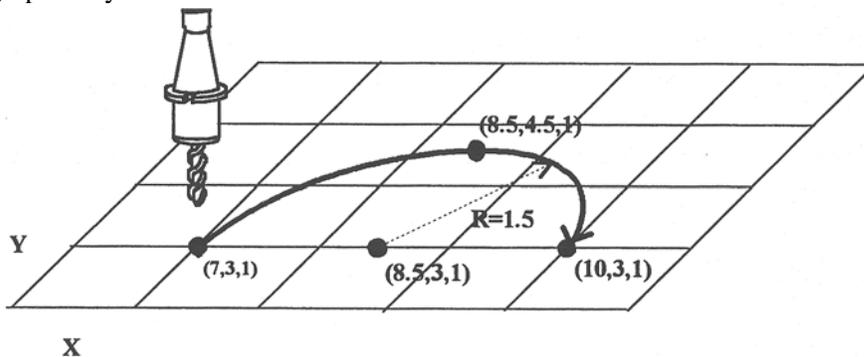
Math Help
F6

Graph
F8

Teach Mode
F9

Accept
F10

The numbers in the different fields on the screen correspond to the following Arc Mill example shown here graphically:



Operation Type: There are four ways to specify your ARC: using an endpoint and a radius (EP&R), using a center point and an angle (CP&A), using a center point and an end point (CP&EP), or using a mid-point and an end point (Three Point). The Three Point arc is designed to be used in conjunction with Teach Mode. When specifying a particular kind of arc, you will not be able to modify certain fields. For example, if you are specifying an endpoint and a radius, you will not be able to modify the mid point, center point and angle fields. This is because Intercon calculates the correct values for these fields.

Mid: The X, Y, and Z coordinates of a point on the arc path somewhere between the start point and end point of the arc. You will be able to modify this field only when specifying a Three Point arc. Also, the coordinate that does not lie in the plane of the arc cannot be edited; it is automatically calculated.

End: The X, Y and Z coordinates of where the cutter will be once the arc move is complete. You will not be able to edit this field if you are specifying a center point and angle (CP&A) arc.

Center: This is the X, Y and Z position of the center of the arc. You will not be able to edit this field if you are specifying an end point and radius (EP&R) arc or a Three Point arc. Also, the coordinate that does not lie in the plane of the arc cannot be edited; it is automatically carried forward from the last operation.

Angle: Number of degrees through which the cutter will travel. This value must lie between 0 and 360 degrees. You will be able to edit this field only if you are specifying a center point and angle (CP&A) arc.

Radius: Distance from the center of the arc to its edge. This value must be greater than 0. You will only be able to edit this value if you are specifying an end point and radius (EP&R) arc.

Plane: This determines whether the arc is to be milled in the XY-, ZX- or YZ-plane. If any of the Z coordinate values are tied to the Z home position, only XY-plane arcs may be selected.

Direction: Determines whether the arc moves clockwise (CW) or counterclockwise (CCW). Note that the direction of XZ arcs is judged looking Y+ (i.e. from the front of the machine). This is natural, but it is opposite of the way arcs are specified in G codes. Intercon automatically makes this translation when it generates CNC codes.

Connect Radius: This field works like the Linear Mill connect radius. It allows for the blending of an arc into the next line or arc operation.

Feedrate: Speed at which the cutter moves.

The feedrate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the feedrate field. If the feedrate is modal then it will have the “M” symbol or if it is fixed it will have the “F” symbol shown below. The slave feedrate has no symbol and is set to the last modal feedrate set in the program, when the modal feedrate changes all the following slave feedrates change until the next modal feedrate is encountered.



Angle <= 180°: For end point and radius (EP&R) arcs, this field determines whether the arc is to be less than (YES) or greater than (NO) 180 degrees.

F4 - Tool Functions

When you select the tool functions by pressing <F4> the following screen appears:

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0060 Tool Change
0010	;Demo Program				Tool Number : 1
0020	Rapid	0.0000	-5.0000	0.1000	Description
0030	Rapid	4.0000	2.0000	1.0000	[.325 end mill]
0040	Line	7.0000	3.0000	1.0000	Position X: 0.0000
0050	Arc CW	10.0000	3.0000	1.0000	Y: 0.0000
0060	Tool #0				Tool H Offset : 1
0070	End Prog	10.0000	3.0000	Home	Tool Height : 0.0000
					Tool D Offset : 1
					Tool Diameter : 0.3250
					Spindle Speed : 3000
					Spindle Dir. : CW (M3)
					Coolant Type : Flood (M8)
					Actual Change? : Yes

Toggle
F3

Math Help
F6

Graph
F8

Teach Mode
F9

Accept
F10

The following parameters for this tool change is as follows:

Tool Number: Number of the tool (between 1 and 200) to use. Entering this value pulls the current settings for this tool from the CNC7 tool library. You may then edit the length offset, diameter offset and diameter values if you wish to redefine your tool. The length value is not editable.

Description: Description of the tool selected above, from the tool library.

Position: X and Y coordinates for the place at which the tool change will occur. This should be a place at which the current tool can be removed from the quill and the new tool can be inserted.

Tool H Offset: Index in the offset library (between 0 and 200) of the actual tool height offset.

Tool Height: Tool height associated with the H offset selected above. This field is not editable.

Tool D Offset: Index in the offset library (between 0 and 200) of the actual tool diameter.

Tool Diameter: Tool diameter associated with the D offset selected above.

Spindle Speed: Speed at which the spindle will rotate when the spindle is started after the tool change.

Spindle Direction: Direction in which the spindle will turn after the tool change. If this is set to CW or CCW, the spindle will be started automatically after the tool change. Press <F3> or <SPACE> to toggle between CW, CCW, and Off.

Coolant Type: Type of coolant to activate after the tool change. If this is set to Flood or Mist, the selected coolant system will be started automatically after the tool change. Press <F3> or <SPACE> to toggle between Flood, Mist, and Off.

Actual Tool Change: Determines whether an M6 code is generated (answer Yes) during the tool change. If you do not want to remove the current tool, but instead want to alter its diameter or length offsets (e.g. for doing a finish pass while using cutter compensation, you may want to use a diameter offset which is slightly larger than the actual tool for the first passes, then use the actual tool diameter for the finish pass), answer No to this question. Spindle and coolant will not be automatically turned off if you answer No here.

Press <F10> when you are finished to accept these values. If you have changed any field other than the Tool Number of the Actual Tool Change field or position, you will actually make changes to the CNC7 Tool Library. At the end of the program, Intercon always turns off the spindle and coolant and returns the Z-axis to the home position. These codes do not need to be entered at the end of your program.

If you answer Yes, the Tool Change operation will be accepted and the new tool library values will be applied. If you answer No, all changes to the tool library will be discarded; however, modified values for the Tool H Offset and Tool D Offset fields will be retained in the current Tool Change operation. The Tool Change operation will be accepted. At the end of a program, Intercon always turns off the spindle and coolant, and returns the Z-axis to the home position. These codes do not need to be entered at the end of your program.

F5 - Canned Cycles

When you choose the Canned Cycle operation by pressing <F5>, the following screen appears:

Intercon Mill v8.11				Current Part: E_Z_PART.ICN
Operation #	Type	X	End Y	Z
0010	;Demo Program			
0020	Rapid	0.0000	-5.0000	0.1000
0030	Rapid	4.0000	2.0000	1.0000
0040	Line	7.0000	3.0000	1.0000
0050	Arc CW	10.0000	3.0000	1.0000
0060	Tool #1	0.0000	0.0000	Home
<hr/>				
0070	End Prog	0.0000	0.0000	Home
Select operation to insert...				
<div style="display: flex; justify-content: space-around; border-top: 1px solid black; padding-top: 5px;"> Drill ... F1 Bore ... F2 Tap ... F3 Face F4 Rect. Pocket F5 Circ. Pocket F6 Frame F7 Thread F8 Clean- out F9 </div>				

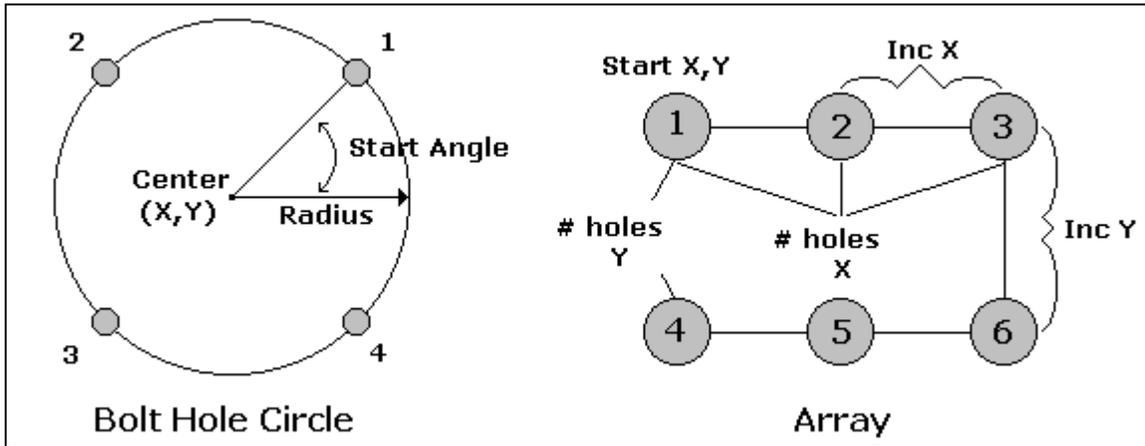
Canned Cycle Introduction #1: Using Pattern and Repeat (Drilling, boring, tapping)

Selecting the Drill, Bore or Tap canned cycle will give you four choices, selecting <F1> Drill is shown below. Bore and Tap will have the same menu selections as drill except with Drill is replace with Bore or Tap:

Intercon Mill v8.10				Current Part: DRILL_R.ICN
Operation #	Type	X	End Y	Z
0010	Header			
0020	Tool #1	0.0000	0.0000	Home
0030	Drill	1.1250	-1.1250	0.1000
0040	Drill	1.8750	-4.1250	0.1000
<hr/>				
0050	End Prog	1.8750	-4.1250	Home
Select operation to insert...				
<div style="display: flex; justify-content: space-around; border-top: 1px solid black; padding-top: 5px;"> Drill F1 Drill BHC F2 Drill Array F3 Drill Repeat F4 </div>				

All canned cycle operations using the Drill BHC (Bolt Hole Circle) or Drill Array are identical to their equivalent using the <F1> Drill single hole selection. The use of the Drill BHC or Drill Array, however, offers the option to drill more than one hole in a pattern dictated by the new fields in the menu. <F4> Drill Repeat allows the user to repeat a set of single holes with a different type drilling, boring or tapping operation with out re-entering the X,Y coordinates.

The bolt hole circle and array patterns are explained graphically in the following figure:



Canned Cycle Introduction #2: Linear Repetition Of Operations (Drilling, Boring, Tapping)

If you want to perform one operation several times in a linear pattern, simply define Position X, Y or both as incremental values. To do this, use the <F1> (Change positioning mode) function. This key will toggle the Position value mode between incremental and absolute. If you define X and/or Y as incremental values, a new field will appear asking for the number of holes:

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0070 Drill
0010	;Demo Program				Cycle Type : Drilling
0020	Rapid	0.0000	-5.0000	0.1000	Position X: 4.0000 INC
0030	Rapid	4.0000	2.0000	1.0000	Y: -8.0000 INC
0040	Line	7.0000	3.0000	1.0000	Surface Height : 0.1000
0050	Arc CW	10.0000	3.0000	1.0000	Clearance Height : 0.1000 INC
0060	Tool #1	0.0000	0.0000	Home	"Rapid to" Depth : 0.1000 INC
0070	Drill				Depth: Total: 0.5000 INC
0080	End Prog	0.0000	0.0000	Home	Plunge Rate : 20.0000
					Dwell Time : 1.00
					Number of Holes : 5

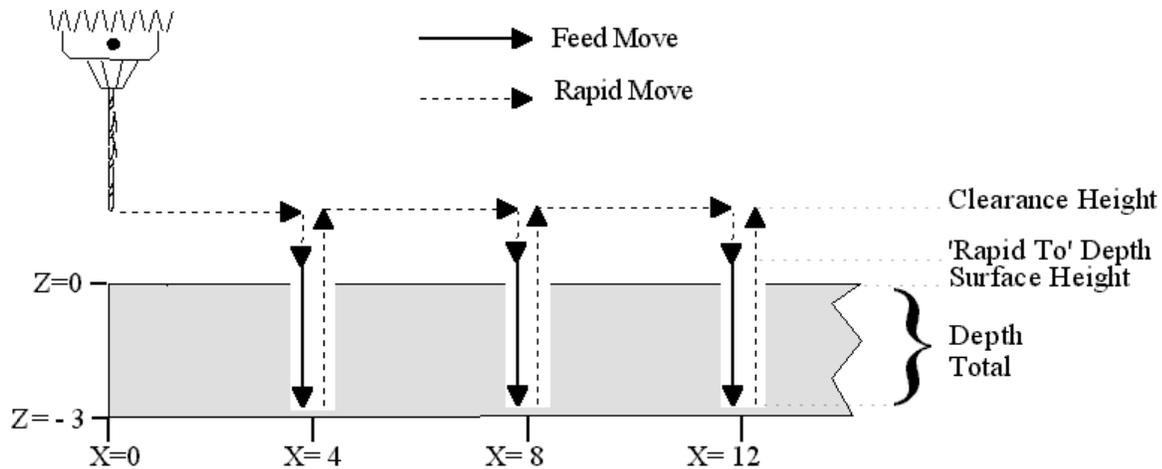
Math Help
F6

Graph
F8

Teach Mode
F9

Accept
F10

The numbers in the fields on the screen correspond to the following example, shown here graphically:



Drilling (F1 in the Canned Cycle Menu: option #1)

If you press <F1> (Drill) from the Canned Cycle Menu, you will gain access to three types of drilling operations: Drilling, Chip Breaking, and Deep Hole drilling. The current drilling operation in use is reflected in the field Cycle Type and pressing <F3> or <SPACE> toggles between all three. In this section we will examine the first option: Drilling.

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0070 Drill
0010	;Demo Program				Cycle Type : Drilling
0020	Rapid	0.0000	-5.0000	0.1000	Position X: 20.0000
0030	Rapid	4.0000	2.0000	1.0000	Y: -40.0000
0040	Line	7.0000	3.0000	1.0000	Surface Height : 0.1000
0050	Arc CW	10.0000	3.0000	1.0000	Clearance Height : 0.1000 INC
0060	Tool #1	0.0000	0.0000	Home	"Rapid to" Depth : 0.1000 INC
0070	Drill				Depth: Total: 0.5000 INC
0080	End Prog	0.0000	0.0000	Home	Plunge Rate : 20.0000
					Dwell Time : 1.00

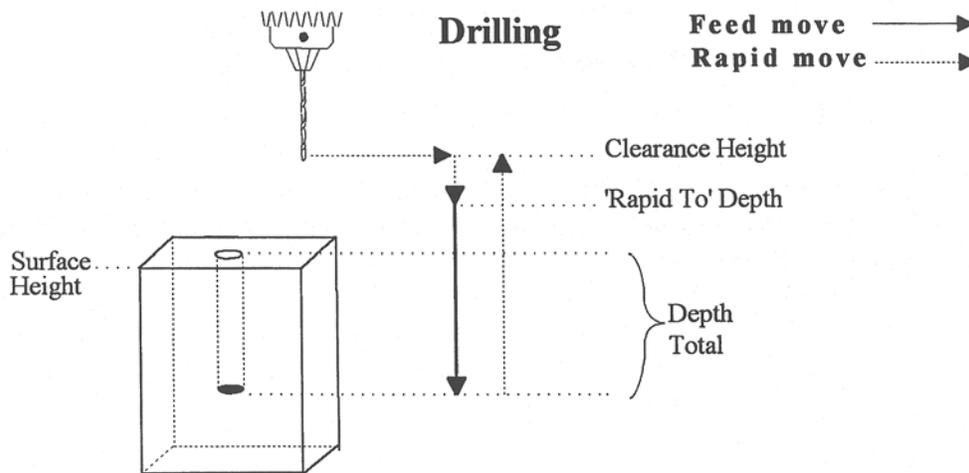
Math Help
F6

Graph
F8

Teach Mode
F9

Accept
F10

The numbers in the fields on the screen correspond to the following example, shown here graphically:



Where:

Cycle Type: Selects one of three drilling operations: Drilling, Chip Breaking, or Deep Hole drilling. Press <F3> or <SPACE> to toggle between the three choices.

Position: Specifies the X and Y coordinates where the drilling will take place. If either the X or Y coordinate is an incremental value, you will have the option to drill multiple holes in a linear pattern (See Canned Cycle Introduction #2).

Surface Height: Absolute Z-axis position from where each incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height used when performing rapid moves to the position of each hole being drilled.

'Rapid To' Depth: The depth (below the Clearance Height but above the Surface Height) to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate.

Depth: Total: Depth of hole (incremental) as measured from Surface Height.

Plunge Rate: Z-axis speed of descent during drilling.

The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol. The slave plunge rate has no symbol and is set to the last modal plunge rate set in the program, when the modal plunge rate changes all the following slave plunge rates change until the next modal plunge rate is encountered.

Dwell Time: Delay at bottom of hole before starting ascent.

Drilling provides a rapid to the hole position at the Clearance Height, followed by a rapid Z down to the 'Rapid To' Depth. Next is a feedrate down to the specified depth. If a Spot facing cycle is desired, enter a value in the dwell time field and the cutter will wait the desired amount of time before performing a rapid move up to the Clearance Height.

Chip Breaking (F1 in the Canned Cycle Menu: option #2)

If you press <F1> (Drilling) from the Canned Cycle Menu you will gain access to three types of drilling operations: Drilling, Chip Breaking, and Deep Hole drilling. The current drilling operation in use is reflected in the field Cycle Type, and pressing <F3> or <SPACE> toggles between all three. In this section we will examine the second option: Chip Breaking.

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0070 Drill
0010	Demo Program				Cycle Type : Chip Breaking
0020	Rapid	0.0000	-5.0000	0.1000	Position X: 20.0000
0030	Rapid	4.0000	2.0000	1.0000	Y: -40.0000
0040	Line	7.0000	3.0000	1.0000	Surface Height : 0.1000
0050	Arc CW	10.0000	3.0000	1.0000	Clearance Height : 0.1000 INC
0060	Tool #1	0.0000	0.0000	Home	"Rapid to" Depth : 0.1000 INC
0070	Drill				Depth: Total: 0.5000 INC
0080	End Prog	0.0000	0.0000	Home	Increment : 0.0000
					Peck Clearance : 0.0500
					Plunge Rate : 20.0000

Toggle
F3

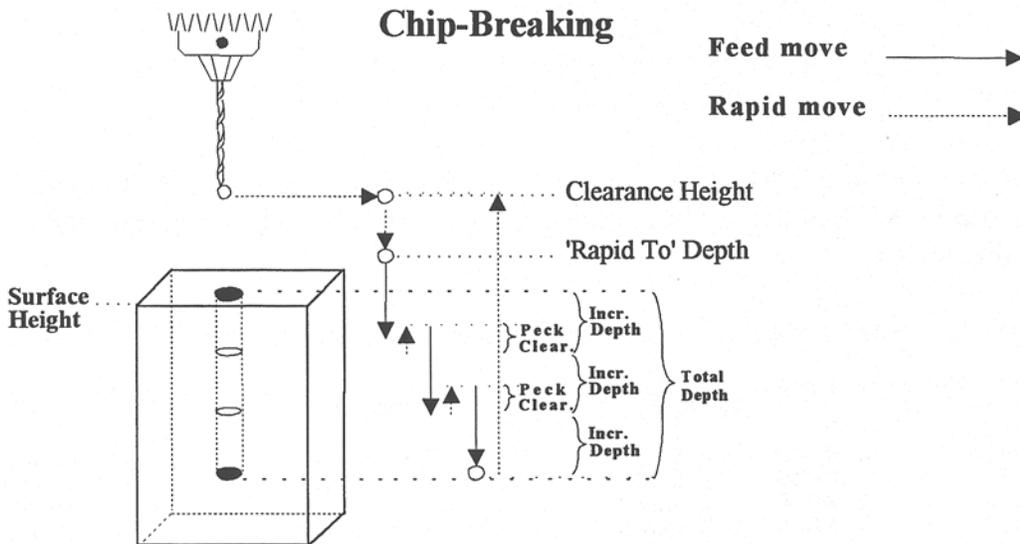
Math Help
F6

Graph
F8

Teach Mode
F9

Accept
F10

The numbers in the fields on the screen correspond to the following example, shown here graphically:



Where:

Cycle Type: Selects one of three drilling operations: Drilling, Chip Breaking, or Deep Hole drilling. Press <F3> or <SPACE> to toggle between the three choices.

Position: Specifies the X and Y coordinates where the drilling will take place. If either the X or Y coordinate is an incremental value, you will have the option to drill multiple holes in a linear pattern (See Canned Cycle Introduction #2).

Surface Height: Absolute Z-axis position from where each incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height used when performing rapid moves to the position of each hole being drilled.

'Rapid To' Depth: The depth (below the Clearance Height but above the Surface Height) to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate.

Depth: Total: Depth of hole (incremental) as measured from Surface Height.

Depth: Increment: Depth of each individual peck.

Peck Clearance: Distance the tool retracts before drilling the next peck.

Plunge Rate: Z-axis speed of descent during drilling.

The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the “M” symbol or if it is fixed it will have the “F” symbol . The slave plunge rate has no symbol and is set to the last modal plunge rate set in the program, when the modal plunge rate changes all the following slave plunge rates change until the next modal plunge rate is encountered.

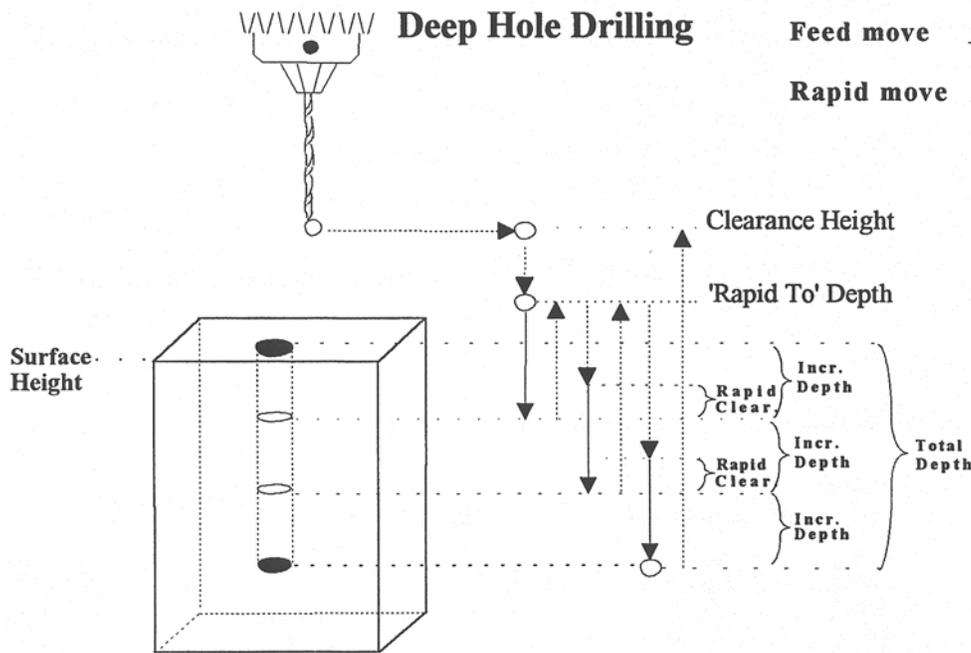
Deep Hole Drilling (F1 in the Canned Cycle Menu: option #3)

If you press <F1> (Drilling) from the Canned Cycle Menu you will gain access to three types of drilling operations: Drilling, Chip Breaking, and Deep Hole drilling. The current drilling operation in use is reflected in the field Cycle Type, and pressing <F3> or <SPACE> toggles between all three. In this section we will examine the third option: Deep Hole drilling.

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0070 Drill
0010	;Demo Program				Cycle Type : Deep Hole
0020	Rapid	0.0000	-5.0000	0.1000	Position X: 20.0000
0030	Rapid	4.0000	2.0000	1.0000	Y: -40.0000
0040	Line	7.0000	3.0000	1.0000	Surface Height : 0.1000
0050	Arc CW	10.0000	3.0000	1.0000	Clearance Height : 0.1000 INC
0060	Tool #1	0.0000	0.0000	Home	"Rapid to" Depth : 0.1000 INC
0070	Drill				Depth: Total: 0.5000 INC
0080	End Prog	0.0000	0.0000	Home	Increment : 0.0000
					Rapid Clearance : 0.0500
					Plunge Rate : 20.0000

Toggle F3	Math Help F6	Graph F8	Teach Mode F9	Accept F10
--------------	--------------------	-------------	---------------------	---------------

The numbers in the fields on the screen correspond to the following example, shown here graphically:



Where:

Cycle Type: Selects one of three drilling operations: Drilling, Chip Breaking, or Deep Hole drilling. Press <F3> or <SPACE> to toggle between the three choices.

Position: Specifies the X and Y coordinates where the drilling will take place. If either the X or Y coordinate is an incremental value, you will have the option to drill multiple holes in a linear pattern (See Canned Cycle Introduction #2).

Surface Height: Absolute Z-axis position from where each incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height used when performing rapid moves to the position of each hole being drilled.

'Rapid To' Depth: The depth (below the Clearance Height but above the Surface Height) to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate.

Depth: Total: Depth of hole (incremental) as measured from Surface Height.

Depth: Increment: Depth of each individual step of the drilling.

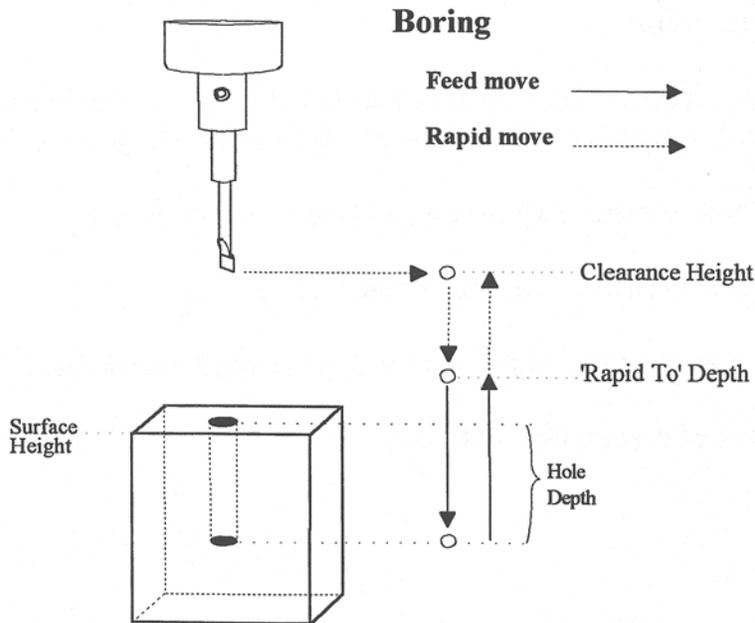
Rapid Clearance: Distance from the last incremental depth drilled that the tool rapid moves before starting the next plunge.

Plunge Rate: Z-axis speed of descent during drilling.

The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol. The slave plunge rate has no symbol and is set to the last modal plunge rate set in the program, when the modal plunge rate changes all the following slave plunge rates change until the next modal plunge rate is encountered.

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0070 Bore
0010	;Demo Program				Position X: 4.0000
0020	Rapid	0.0000	-5.0000	0.1000	Y: -8.0000
0030	Rapid	4.0000	2.0000	1.0000	Surface Height : 0.1000
0040	Line	7.0000	3.0000	1.0000	Clearance Height : 0.1000 INC
0050	Arc CW	10.0000	3.0000	1.0000	"Rapid to" Depth : 0.1000 INC
0060	Tool #1	0.0000	0.0000	Home	Hole Depth : 0.5000 INC
0070	Bore				Plunge Rate : 20.0000
0080	End Prog	0.0000	0.0000	Home	Dwell Time : 1.00

The numbers in the fields on the screen correspond to the following example, shown here graphically:



Where:

Position: Specifies the X and Y coordinates where the boring will take place. If either the X or Y coordinate is an incremental value, you will have the option to bore multiple holes in a linear pattern. (See Canned Cycle Introduction #2)

Surface Height: Absolute Z-axis position from where each incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height used when performing rapid moves to the position of each hole being drilled.

'Rapid To' Depth: The depth (below the Clearance Height but above the Surface Height) to which the cutter rapidly moves before beginning to drill the hole at the specified Plunge Rate.

Hole Depth: Depth of hole (incremental) as measured from Surface Height.

Plunge Rate: Z-axis speed of descent during drilling.

The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the “M” symbol or if it is fixed it will have the “F” symbol . The slave plunge rate has no symbol and is set to the last modal plunge rate set in the program, when the modal plunge rate changes all the following slave plunge rates change until the next modal plunge rate is encountered.

Dwell Time: Delay at bottom of hole before starting ascent.

Tapping (F3 in the Canned Cycle Menu)

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	End X Y	Z	N0070 Tap	
0010	;Demo Program			Tap Head Type	: Rigid
0020	Rapid	0.0000 -5.0000	0.1000	Position X:	0.0000
0030	Rapid	4.0000 2.0000	1.0000	Position Y:	0.0000
0040	Line	7.0000 3.0000	1.0000	Surface Height	: 0.0000
0050	Arc CW	10.0000 3.0000	1.0000	Clearance Height	: 0.1000 INC
0060	Tool #1	0.0000 0.0000	Home	"Rapid to" Depth	: 0.1000 INC
0070	Tap	0.0000 0.0000	0.1000	Depth: Total:	0.5000 INC
0080	End Prog	0.0000 0.0000	Home	Increment	: 0.0000
				Threads / Inch	: 12.0000
				Thread Pitch	: 0.0833
				Dwell Time	: 0.10
				(Spindle Dir.	: CW (M3))
				(Spindle Speed	: 350)

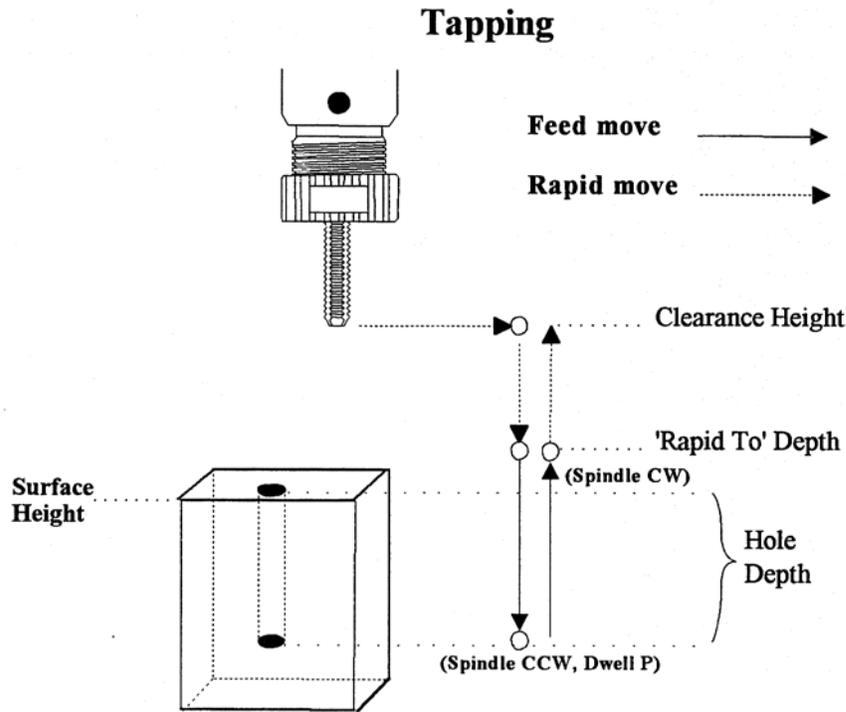
Math Help
F6

Graph
F8

Teach Mode
F9

Accept
F10

The numbers in the fields on the screen correspond to the following example, shown here graphically:



Where:

Tap Head Type: Without rigid tapping, this selects either Floating tap head or Reversing tap head. If rigid tapping is enabled, you can select either rigid or reversing.

Spindle Direction: Shows the current spindle direction. The spindle direction should be CW for right-hand tapping, and CCW for left-hand tapping. The spindle speed and direction appropriate for the tapping tool should be set in the tool change in which the tapping tool was loaded. This field will be hidden if a reversing tap head is used.

* **WARNING:** The tap must be rotating in the correct direction before performing this operation.

Position: Specifies the X and Y coordinates where the tapping will take place. If either the X or Y coordinate is an incremental value, you will have the option to tap multiple holes in a linear pattern. (See Canned Cycle Introduction #2)

Surface Height: Absolute Z-axis position from where each incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height used when performing rapid moves to the position of each hole being drilled.

'Rapid To' Depth: The depth (below the Clearance Height but above the Surface Height) to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate.

Depth Total: Depth of hole (incremental) as measured from Surface Height.

Depth Increment: (available only on rigid tapping.) This sets the length of each progressive "peck" down the hole.

Threads / Unit: Number of threads on each inch/mm of the tap. Used in conjunction with the Spindle Speed to calculate the appropriate plunge rate ($\text{Plunge Rate} = \text{Spindle Speed} / \text{Threads per Unit}$).

Spindle Speed: Rate at which the spindle rotates. Used in conjunction with the Threads / Unit to calculate the plunge rate.

* **WARNING:** The spindle speed must be set before performing this operation.

Dwell Time: Delay at bottom of hole before starting ascent. This is used for a floating tap to allow the spindle time to reverse direction at the bottom of the hole. A default value of 0.1 seconds is suggested. This field will be hidden if a reversing tap head is used; the tap head will reverse direction when the quill begins ascending.

* **NOTE:** When using low gear for tapping, the spindle may turn opposite the direction specified. The operator is responsible for setting the correct spindle speed and direction.

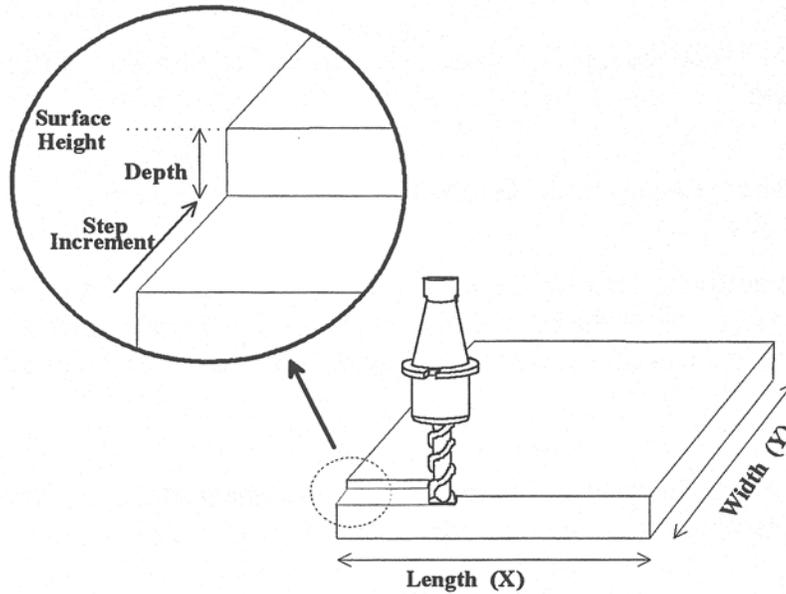
Facing (F4 in the Canned Cycle Menu)

If you press <F4> (Facing) at the Canned Cycle Selection Menu, the following screen is displayed:

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	End X	Y	Z	N0080 Face
0010	;Demo Program				Start X: 3.0000
0020	Rapid	0.0000	-5.0000	0.1000	Y: 3.0000
0030	Rapid	4.0000	2.0000	1.0000	Surface Height : 0.0000
0040	Line	7.0000	3.0000	1.0000	Length X: 3.0000 INC
0050	Arc CW	10.0000	3.0000	1.0000	Width Y: 3.0000 INC
0060	Tool #1	0.0000	0.0000	Home	Depth : 0.5000 INC
0070	Tap	0.0000	0.0000	0.1000	Step Increment : 0.1000
0080	Face				Feedrate : 30.0000 <input type="checkbox"/>
0090	End Prog	0.0000	0.0000	Home	Plunge Rate : 10.0000 <input type="checkbox"/>

Toggle F3	Math Help F6	Graph F8	Teach Mode F9	Accept F10
--------------	--------------------	-------------	---------------------	---------------

The parameters in the previous screen correspond to the following dimensions:



Start: X and Y coordinates of the starting corner of the area to be faced.

Surface Height: Z coordinate of the top of the area to be faced.

Length: X-axis dimension of the area to be faced. If a negative value is entered for the length, the facing will occur in the negative X-axis direction from the X-axis start position; otherwise, facing will occur in the positive X-axis direction from the X-axis start position.

Width: Y-axis dimension of the area to be faced. If a negative value is entered for the width, the facing will occur in the negative Y-axis direction from the Y-axis start position; otherwise, facing will occur in the positive Y-axis direction from the Y-axis start position.

Depth: Incremental amount of material to be removed from Surface Height.

Step Increment: Distance that the cutter will step over in the Y direction for each pass.

Plunge Rate: Z-axis speed of descent during facing.

The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the “M” symbol or if it is fixed it will have the “F” symbol . The slave plunge rate has no symbol and is set to the last modal plunge rate set in the program, when the modal plunge rate changes all the following slave plunge rates change until the next modal plunge rate is encountered.

Feedrate: Speed of the cutter during facing.

The feedrate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the feedrate field. If the feedrate is modal then it will have the “M” symbol or if it is fixed it will have the “F” symbol shown below. The slave feedrate has no symbol and is set to the last modal feedrate set in the program, when the modal feedrate changes all the following slave feedrates change until the next modal feedrate is encountered.

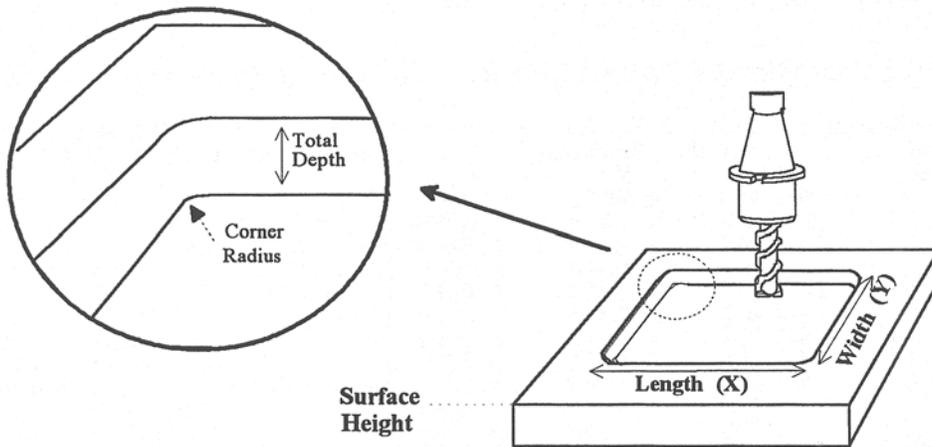


Rectangular Pocket (F5 in the Canned Cycle Menu)

Pressing <F5> (Rectangular Pocket) from the Canned Cycle Selection Menu displays the following screen:

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0090 Rectangular Pocket
0010	:Demo Program				Center X: 4.0000
0020	Rapid	0.0000	-5.0000	0.1000	Y: -8.0000
0030	Rapid	4.0000	2.0000	1.0000	Surface Height : 0.0000
0040	Line	7.0000	3.0000	1.0000	Length X: 5.0000 INC
0050	Arc CW	10.0000	3.0000	1.0000	Width Y: 3.0000 INC
0060	Tool #1	0.0000	0.0000	Home	Corner Radius : 0.2500
0070	Tap	0.0000	0.0000	0.1000	Depth: Total: 0.5000 INC
0080	Face	3.0000	6.0000	0.0000	per Pass : 0.1000
0090	Rect Poc				Plunge Rate : 10.0000
0100	End Prog	3.0000	6.0000	Home	Plunge Type : Ramped
					Plunge Angle : 0.00°
					Rough Cuts : Conventional
					Stepover : 0.1950
					Feedrate : 30.0000
					Finish Pass : Climb
					Amount : 0.0200
					Feedrate : 30.0000

The parameters on the screen correspond to the following dimensions:



Where:

Center: X and Y coordinates of the center of the RECTANGULAR POCKET.

Surface Height: Z-axis position from which each incremental depth is measured.

Length: X-axis dimension of the rectangular pocket.

Width: Y-axis dimension of the rectangular pocket.

Corner Radius: Radius of curvature of the corners. It cannot be smaller than the current cutter radius.

Depth: Total: Total depth of the rectangular pocket.

Depth: Per Pass: Depth of each individual pass.

Depth: Plunge Rate: Z-axis speed of descent.

Depth: Plunge Type: Straight or Ramped. Straight plunge does a vertical Z plunge with no X, Y movement. Ramped plunge does a zigzag plunge limited by the Plunge Angle entered below.

Depth: Plunge Angle: The maximum limit angle allowed for a ramped plunge. A special value of 0 means that there is no limit angle. Note that this field means nothing if the Plunge Type is Straight.

Rough Cuts: Selects type of rough cut: conventional or climb. Use <SPACE> to toggle between them.

Rough Cuts: Stepmover: Amount of material removed by cutter during each pass around the pocket.

Rough Cuts: Feedrate: Speed at which cutter performs rough cuts.

Finish Pass: Selects type of finish pass: climb, conventional or none at all. Use <F3> or <SPACE> to toggle between them.

Finish Pass: Amount: Amount of material to be removed on the finish pass.

Finish Pass: Feedrate: Speed at which cutter performs finish pass.

The feedrate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the feedrate field. If the feedrate is modal then it will have the “M” symbol or if it is fixed it will have the “F” symbol shown below. The slave feedrate has no symbol and is set to the last modal feedrate set in the program, when the modal feedrate changes all the following slave feedrates change until the next modal feedrate is encountered.



Circular Pocket (F6 in the Canned Cycle Menu)

When you press <F6> (Circular Pocket) from the Canned Cycle Selection Menu, this screen is displayed:

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0100 Circular Pocket
0010	;Demo Program				Center X: 4.0000
0020	Rapid	0.0000	-5.0000	0.1000	Y: -8.0000
0030	Rapid	4.0000	2.0000	1.0000	Surface Height : 0.0000
0040	Line	7.0000	3.0000	1.0000	Diameter : 2.0000
0050	Arc CW	10.0000	3.0000	1.0000	Cleanout : Yes
0060	Tool #1	0.0000	0.0000	Home	Depth: Total: 0.5000 INC
0070	Tap	0.0000	0.0000	0.1000	per Pass : 0.1000
0080	Face	3.0000	6.0000	0.0000	Plunge Rate : 10.0000
0090	Rect Poc	4.0000	-8.0000	0.0000	Plunge Type : Ramped
0100	Circ Poc				Plunge Angle : 0.00°
0110	End Prog	4.0000	-8.0000	Home	Rough Cuts : Conventional
					Stepover : 0.1950
					Feedrate : 30.0000
					Finish Pass : Climb
					Amount : 0.0200
					Feedrate : 30.0000

Abs
Inc
F1

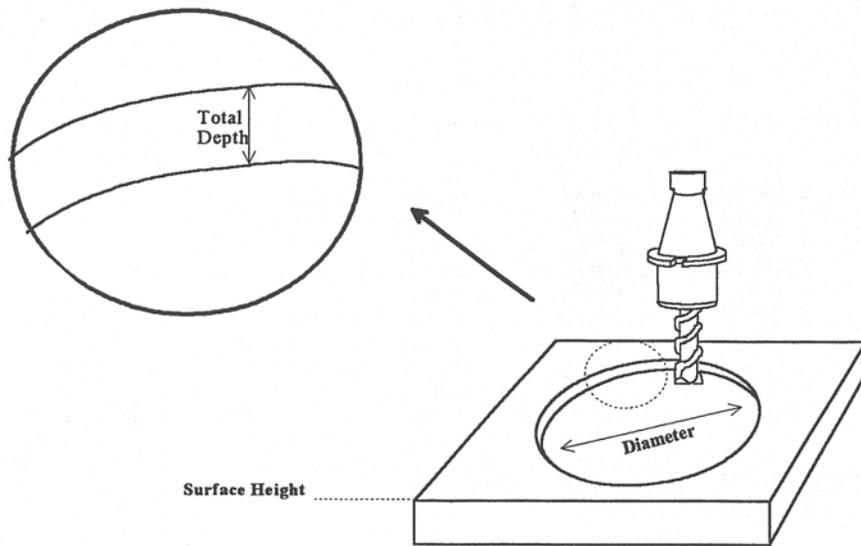
Math
Help
F6

Graph
F8

Teach
Mode
F9

Accept
F10

The parameters on the screen correspond to the following dimensions:



Where:

Center: X and Y coordinates of the center of the circular POCKET.

Surface Height: Z-axis position from which each incremental depth is measured.

Diameter: Diameter of circular pocket.

Cleanout: If cleanout is Yes, all the material in the pocket will be removed. If cleanout is No, a circular frame mill will result, with the cutter starting in the pocket center and arcing its way out and then going around the frame.

Depth: Total: Total depth of the circular pocket.

Depth: Per Pass: Depth of each individual pass.

Depth: Plunge Rate: Z-axis speed of descent.

Depth: Plunge Type: Straight or Ramped. Straight plunge does a vertical Z plunge with no X, Y movement. Ramped plunge does a zigzag plunge limited by the Plunge Angle entered below.

Depth: Plunge Angle: The maximum limit angle allowed for a ramped plunge. A special value of 0 means that there is no limit angle. Note that this field means nothing if the Plunge Type is Straight.

Rough Cuts: Selects type of rough cut: conventional or climb. Use <F3> or <SPACE> to toggle between them.

Rough Cuts: Stepper: Amount of material removed by cutter during each pass around the pocket.

Rough Cuts: Feedrate: Speed at which cutter performs rough cuts.

Finish Pass: Selects type of finish pass: climb, conventional or none at all. Use <F3> or <SPACE> to toggle.

Finish Pass: Amount: Amount of material to be removed on the finish pass.

Finish Pass: Feedrate: Speed at which cutter performs finish pass.

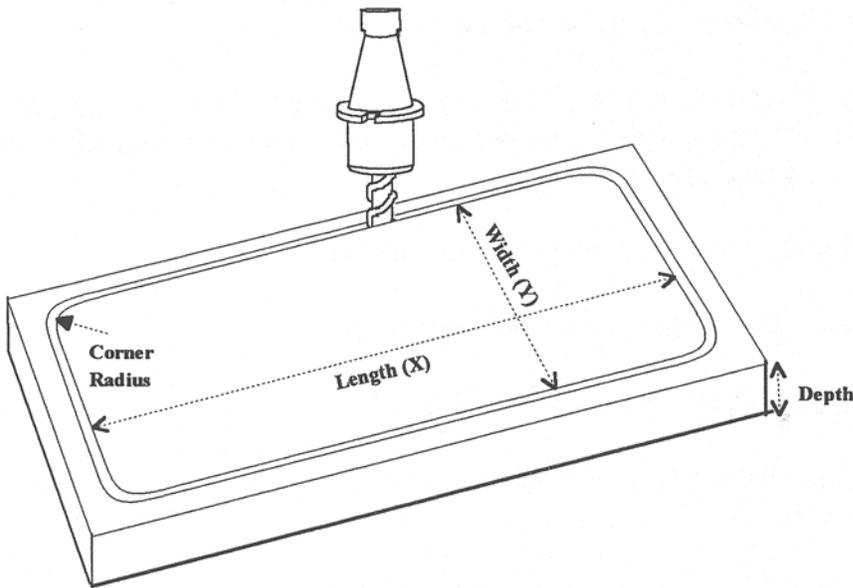
Rectangular or Circular Frame Milling (F7 in the Canned Cycle Menu)

When you press <F7> (Frame Milling) from the Canned Cycle Selection Menu, the following screen is displayed:

Intercon Mill v8.20				Current Part: TEST.ICN	
Operation #	Type	Z	End X	A	N0040 Frame
0010	Header				Frame Type : Inside Rect
0020	Face	0.0000	-5.0000	-0.9000	Center Z: 0.0000
0030	Circ Poc	0.0000	0.0000	-0.9000	Center X: 0.0000
0040	Frame				Surface Height : -1.0000
0050	End Prog	0.0000	0.0000	Home	Length Z: 0.0000 INC
					Width X: 0.0000 INC
					Corner Radius : 0.0000
					Depth: Total: 1.0000 INC
					per Pass : 0.1000
					Plunge Rate : 1.0000
					Plunge Type : Ramped
					Plunge Angle : 0.00°
					Cut Type : Conventional
					Feedrate : 1.0000

Toggle F3	Help F5	Math Help F6	Graph F8	Teach Mode F9	Accept F10
--------------	------------	-----------------	-------------	------------------	---------------

The parameters on the screen correspond to the following dimensions (rectangular frame):



Where:

Frame Type: Selects Inside Rectangle, Outside Rectangle, Inside Circle, and Outside Circle. Press <F3> or <SPACE> to toggle between them.

Center: X and Y coordinates of the center of the frame mill.

Surface Height: Z-axis position from where each incremental depth is measured.

Length: X-axis dimension of the frame mill. (Rectangular frame only.)

Width: Y-axis dimension of the frame mill. (Rectangular frame only.)

Diameter: Diameter of the frame mill. (Circular frame only.)

Corner Radius: Radius of curvature of the corners. On an Inside frame, corner radius must be greater than the current cutter radius. (Rectangular frame only.)

Depth: Total: Total depth of the frame mill.

Depth: Per Pass: Depth of each individual pass.

Plunge Rate: Z-axis speed of descent.

Plunge Type: Straight or Ramped. Straight plunge does a vertical Z plunge with no X, Y movement. Ramped plunge does a zigzag plunge limited by the Plunge Angle entered below.

Plunge Angle: The maximum limit angle allowed for a ramped plunge. A special value of 0 means that there is no limit angle. Note that this field means nothing if the Plunge Type is Straight.

Cut type: Selects type of cut: conventional or climb, use <F3> or <SPACE> to toggle between them.

Feedrate: Speed at which the cutter performs frame mill.

NOTE: To make a circular frame mill of radius R, specify R as the Corner Radius and set the Length and Width parameters equal to 2 x R.

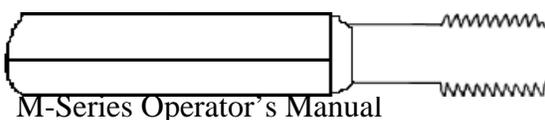
Thread Milling (F8 in the Canned Cycle Menu)

When you press <F8> (Thread Milling) from the canned cycle menu, the following screen is displayed:

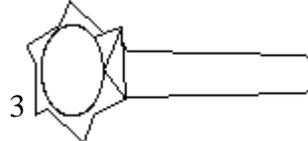
Intercon Mill v8.11				Current Part: E_Z_PART.ICN		
Operation #	Type	X	End Y	Z	N0120 Thread Mill	
0010	:Demo Program				Center	X: 4.0000
0020	Rapid	0.0000	-5.0000	0.1000		Y: -8.0000
0030	Rapid	4.0000	2.0000	1.0000	Diameter	: 2.0000
0040	Line	7.0000	3.0000	1.0000	Threads / Inch	: 20.0000
0050	Arc CW	10.0000	3.0000	1.0000	Thread Pitch	: 0.0500
0060	Tool #1	0.0000	0.0000	Home	Thread Type	: Right-hand
0070	Tap	0.0000	0.0000	0.1000	Thread Direction	: Top-to-Bottom
0080	Face	3.0000	6.0000	0.0000	Tool Type	: Single Point
0090	Rect Poc	4.0000	-8.0000	0.0000	Thread Approach	: Internal
0100	Circ Poc	4.0000	-8.0000	0.0000	Feedrate	: 30.0000
0110	Frame	1.7500	-6.6625	0.0000	Surface Height	: 0.0000
0120	Thread				Clearance Height	: 0.1000 INC
0130	End Prog	1.7500	-6.6625	Home	"Rapid to" Depth	: 0.1000 INC
					Depth: Total	: 0.5000 INC
					Plunge Rate	: 10.0000
					Number of Passes	: 1

Math Help (F6) Graph (F8) Teach Mode (F9) Accept (F10)

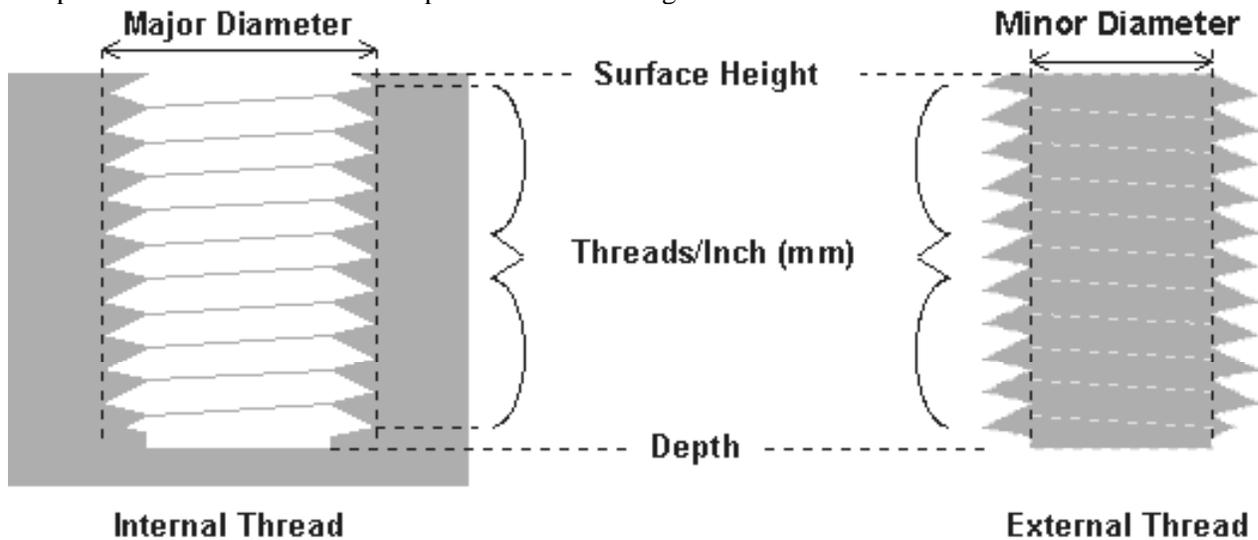
Multiple Thread Mill



Single Thread Mill



The parameters on the screen correspond to the following:



Where:

Center: X and Y coordinates of the center of the thread mill operation.

Diameter: Major diameter of thread for external thread milling and minor diameter for internal thread milling.

Thread / Unit: Number of threads per inch or mm. Used to calculate thread pitch.

Thread Pitch: Thread pitch calculated from threads/unit field. This field cannot be modified.

Thread Type: Specifies right or left hand threads.

Thread Direction: Specifies whether to start at the bottom of the hole and work up or start at the top of the hole and work down.

Tool Type: Single point or full form threading tool.

Thread Approach: Internal or external thread.

Clearance Amount: Used for external thread milling only. Specifies the diameter of the lead-in arc. Minimum clearance is 0.050 inches.

Clearance Angle: Used for external thread milling only. Specifies the angle from which the lead in arc will start.

Feedrate: Cutting feed rate.

Surface Height: Absolute Z-axis position from position from where the incremental depth is measured.

Clearance Height: This parameter specifies the Z axis height used when performing rapid moves to the position of each hole being thread.

'Rapid to' Depth: The depth (below the Clearance Height but above the Surface Height) to which the cutter rapid moves before beginning to thread mill at the specified Plunge Rate.

Depth: The total depth of the thread.

Number of Passes: Number of times the thread mill is to be done on the same hole.

Cleanout (F9 in the Canned Cycle Menu)

The cleanout cycle performs a horizontal zigzag pocket cleanout of a profile composed of lines and arcs. When you press <F9> (Cleanout) from the canned cycle menu, the following screen is displayed:

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0130 Pocket Cleanout
0010	Demo Program				Rough Cuts : Conventional
0020	Rapid	0.0000	-5.0000	0.1000	Stepover : 0.1250
0030	Rapid	4.0000	2.0000	1.0000	Feedrate : 50.0000 <input type="checkbox"/>
0040	Line	7.0000	3.0000	1.0000	Finish Pass : Climb
0050	Arc CW	10.0000	3.0000	1.0000	Amount : 0.0300
0060	Tool #1	0.0000	0.0000	Home	Feedrate : 20.0000 <input type="checkbox"/>
0070	Tap	0.0000	0.0000	0.1000	Tool Number : 2
0080	Face	3.0000	6.0000	0.0000	Surface Height : 0.0000
0090	Rect Poc	4.0000	-8.0000	0.0000	Clearance Height : 0.1000 INC
0100	Circ Poc	4.0000	-8.0000	0.0000	"Rapid to" Depth : 0.1000 INC
0110	Frame	1.7500	-6.6625	0.0000	Depth: Total: 1.0000 INC
0120	Thread	4.0000	-8.0000	0.1000	per Pass : 0.2500
0130	Cleanout				Plunge Rate : 20.0000 <input type="checkbox"/>
0140	End Cleanout	4.0000	-8.0000	0.1000	
0150	End Prog	4.0000	-8.0000	Home	

Toggle
F3

Math
Help
F6

Graph
F8

Teach
Mode
F9

Accept
F10

Where:

Rough Cuts: Selects type of rough cut: conventional or climb. Use <F3> or <SPACE> to toggle between them.

Rough Cuts: Stepover: The distance between zigzag lines in the pocket cleanout. This value should be less than the tool diameter to ensure all material is removed.

Rough Cuts: Feedrate: Speed at which cutter performs rough cuts.

Finish Pass: Selects type of finish pass: climb, conventional or none at all. Use <F3> or <SPACE> to toggle between them.

Finish Pass: Amount: Amount of material to be removed on the finish pass.

Finish Pass: Feedrate: Speed at which cutter performs finish pass.

Finish Pass: Tool Number: Tool number to be used for the finish pass.

Surface Height: The Z-axis position from where the incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height to which the tool is retracted before moving to different segments during a pocket cleanout.

'Rapid To' Depth: The depth to which rapid positioning moves will be made to when moving the Z axis downward.

Depth: Total: The total depth of the pocket measured as an incremental depth from the surface height.

Depth: per Pass: The depth amount of cut to be taken to reach the total depth. This value must be greater than 0.0 and cannot exceed the total depth.

Depth: Plunge Rate: The feedrate at which the Z axis is moved when plunging to a lower depth.

After the cleanout parameters are accepted, the a screen similar to the following appears:

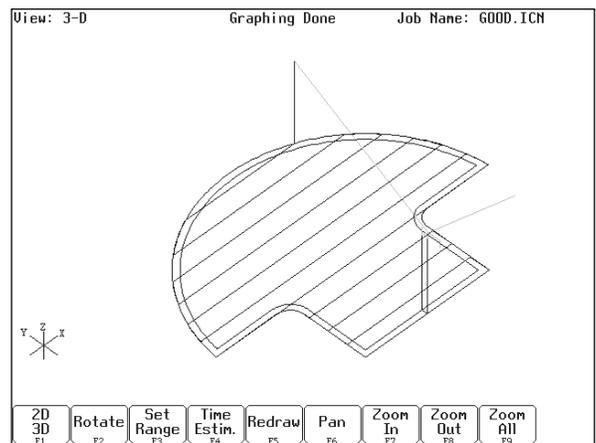
Intercon Mill v8.00					Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	Select operation to insert...	
0010	;Demo program					
0020	Line	0.0000	0.0000	5.0000		
0030	Tool #1	0.0000	0.0000	Home		
0040	Cleanout	0.0000	0.0000	0.2000		
0050	Line	0.0000	0.0000	0.2000		
0060	Line	3.0000	0.0000	0.2000		
0070	Line	0.0000	3.0000	0.2000		
0080	Line	0.0000	0.0000	0.2000		
0090	End Cleanout	0.0000	0.0000	0.2000		
0100	End Prog	0.0000	0.0000	Home		

Linear
F2

Arc
F3

To complete the contour to be cleaned out, choose <F2> or <F3> to insert line or arc segments that define the pocket contour. The first segment should be a linear move to a point on the pocket contour; it cannot be an arc. If the profile contour does not end at the start point, a linear segment is automatically inserted to close the contour. The last line of the contour will not include a connecting radius to the starting point.

At right is an example of a completed cleanout backplot.



F6 - Other

Press <F6> to add a comment to your program, control flood, mist, spindle, and clamp; you can also send the tool to Z home or enter in any G code or M function available on your control. Pressing <F6> from the Insert Operation screen shows:

Intercon Mill v8.00				Current Part: E_Z_PART.ICN
Operation #	Type	X	End Y	Z
0010	;Demo program			
0020	Rapid	1.0000	1.0000	1.0000
0030	Rapid	4.0000	2.0000	1.0000
0040	Line	7.0000	3.0000	1.0000
0050	Arc CW	10.0000	3.0000	1.0000
0060	Tool #1	0.0000	0.0000	Home
0070	Tap	0.0000	0.0000	0.1000
0080	Face	3.0000	6.0000	0.0000
0090	Rect Poc	4.0000	-8.0000	0.0000
0100	Circ Poc	4.0000	-8.0000	0.0000
0110	Frame	1.7500	-6.6625	0.0000
0120	Thread	4.0000	-8.0000	0.1000
0130	End Prog	4.0000	-8.0000	Home

Select operation to insert...

Com
ment
F1

Spindle
F2

Coolant
F3

Clamp
F4

Z Home
F5

M&G
Code
F6

Rotary
F7

Press <F1> to enter a comment, up to 35 characters long, which will be displayed in the generated CNC program. Press <F2> to change the actual state of the spindle. Press <F3> or <SPACE> to toggle between CW, CCW, and OFF.

Press <F3> to change the actual state of the coolant. Press <F3> or <SPACE> to toggle between FLOOD, MIST, and OFF.

Press <F4> to turn the Clamp ON and OFF. Press <F3> or <SPACE> to change the clamp state.

Press <F5> to send the Z-axis to its home position.

Pressing <F6> from the OTHER screen displays the following screen:

Intercon Mill v8.11				Current Part: E_Z_PART.ICN
Operation #	Type	X	End Y	Z
0010	;Demo Program			
0020	Rapid	0.0000	-5.0000	0.1000
0030	Rapid	4.0000	2.0000	1.0000
0040	Line	7.0000	3.0000	1.0000
0050	Arc CW	10.0000	3.0000	1.0000
0060	Tool #1	0.0000	0.0000	Home
0070	Tap	0.0000	0.0000	0.1000
0080	Face	3.0000	6.0000	0.0000
0090	Rect Poc	4.0000	-8.0000	0.0000
0100	Circ Poc	4.0000	-8.0000	0.0000
0110	Frame	1.7500	-6.6625	0.0000
0120	Thread	4.0000	-8.0000	0.1000
0130				
0140	End Prog	4.0000	-8.0000	Home

N0130 M & G Codes

M41 ; Select Low Gear Range

Warning: Any M & G codes entered here will be unrecognizable by Intercon. Careful consideration must be taken before using this function.

Math
Help
F6

Graph
F8

Teach
Mode
F9

Accept
F10

This operation lets you directly enter M & G codes into your Intercon part program. Great care must be taken when using this function, as you could cause unpredictable results in the controller if you accidentally changed positioning modes in your program, or perhaps turned the spindle off during a cut.

Rotary - Rapid Move

If you have a fourth axis and it is rotary, then the OTHER screen will allow you to make rotary moves by pressing <F7>. The fields are identical to the fields in the Linear Mill operation as shown below, but the resulting move is a G0 (Rapid) moving only the rotary axis.

Intercon Mill v8.11					Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0140 Move Rotary Axis	
0010	;Demo Program				Rotary Axis B	
0020	Rapid	0.0000	-5.0000	0.1000	Degrees	: 25°INC
0030	Rapid	4.0000	2.0000	1.0000	Minutes	: 10 INC
0040	Line	7.0000	3.0000	1.0000	Seconds	: 10 INC
0050	Arc CW	10.0000	3.0000	1.0000	Decimal Degrees	: 25.1694°INC
0060	Tool #1	0.0000	0.0000	Home		
0070	Tap	0.0000	0.0000	0.1000		
0080	Face	3.0000	6.0000	0.0000		
0090	Rect Poc	4.0000	-8.0000	0.0000		
0100	Circ Poc	4.0000	-8.0000	0.0000		
0110	Frame	1.7500	-6.6625	0.0000		
0120	Thread	4.0000	-8.0000	0.1000		
0130	M41 ; Select Low Gear Range					
0140	Rotary					
0150	End Prog	4.0000	-8.0000	Home		

Abs Inc
F1

Math Help
F6

Graph
F8

Teach Mode
F9

Accept
F10

F7 - Cutter Compensation

When you press <F7> from the Insert Operation screen, you will see the following screen:

Intercon Mill v8.11					Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0150 Cutter Comp	
0010	;Demo Program				Cutter Comp : Left	
0020	Rapid	0.0000	-5.0000	0.1000		
0030	Rapid	4.0000	2.0000	1.0000		
0040	Line	7.0000	3.0000	1.0000		
0050	Arc CW	10.0000	3.0000	1.0000		
0060	Tool #1	0.0000	0.0000	Home		
0070	Tap	0.0000	0.0000	0.1000		
0080	Face	3.0000	6.0000	0.0000		
0090	Rect Poc	4.0000	-8.0000	0.0000		
0100	Circ Poc	4.0000	-8.0000	0.0000		
0110	Frame	1.7500	-6.6625	0.0000		
0120	Thread	4.0000	-8.0000	0.1000		
0130	M41 ; Select Low Gear Range					
0140	Rotary					
0150	Comp Off					
0160	End Prog	4.0000	-8.0000	Home		

Toggle
F3

Math Help
F6

Graph
F8

Teach Mode
F9

Accept
F10

You can press <F3> or <SPACE> to select cutter compensation Left, Right, or Off. Cutter compensation may be used with Linear Mill, Frame Mill, and Rapid Traverse operations. For details on using cutter compensation, see Chapter 2 of the M-Series operator's manual.

The Rectangular Pocket, Circular Pocket, and Frame Mill canned cycles perform cutter compensation automatically. If compensation left or right was selected before the canned cycle, it will be turned off.

F9 - Subprograms

Intercon subprograms allow you to make additional copies of a programmed contour. The copies may be repeated vertically (to depth), horizontally or radially, or may be a mirror image of the original. To create a subprogram, first define the operations that will compose the contour. Any type of program operation (rapid, linear mill, arc mill, canned cycle, subprogram, etc.) may be included in the contour. These operations must be programmed at the Z depth at which the first pass will occur. When you are finished doing this, return to the Program Edit Menu. Move to the place in the program where you want to repeat these operations and press the Insert Operation (<F2>) key. The operations will be performed once before the repeat operation occurs; therefore the operations to compose the contour should be defined at the place in the program where they should first occur.

When you press <F9> from the Insert Operation screen you will see the Insert Subprogram screen:

Intercon Mill v8.00		Current Part: E_Z_PART.ICN		
Operation #	Type	X	End Y	Z
0010	;Demo program			
0020	Rapid	1.0000	1.0000	1.0000
0030	Rapid	4.0000	2.0000	1.0000
0040	Line	7.0000	3.0000	1.0000
0050	Arc CW	10.0000	3.0000	1.0000
0060	Tool #1	0.0000	0.0000	Home
0070	Tap	0.0000	0.0000	0.1000
0080	Face	3.0000	6.0000	0.0000
0090	Rect Poc	4.0000	-8.0000	0.0000
0100	Circ Poc	4.0000	-8.0000	0.0000
0110	Frame	1.7500	-6.6625	0.0000
0120	Thread	4.0000	-8.0000	0.1000
0130	M41 ; Select Low Gear Range			
0140	Rotary	4.0000	-8.0000	0.1000
0150	Comp Left	4.0000	-8.0000	0.1000
0160	End Prog	4.0000	-8.0000	Home

Select operation to insert...

Depth Repeat F1	Repeat F2	Mirror F3	Rotate F4
--------------------	--------------	--------------	--------------

You may now select the type of subprogram desired. A typical subprogram screen appears as follows:

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0160 Repeat
0010	;Demo Program				Start Block : N0020
0020	Rapid	0.0000	-5.0000	0.1000	End Block : N0080
0030	Rapid	4.0000	2.0000	1.0000	Increment (X) : 2.0000
0040	Line	7.0000	3.0000	1.0000	Increment (Y) : 1.0000
0050	Arc CW	10.0000	3.0000	1.0000	Clearance Height : 0.1000
0060	Tool #1	0.0000	0.0000	Home	Plunge Rate : 10.0000
0070	Tap	0.0000	0.0000	0.1000	Number of Copies : 10
0080	Face	3.0000	6.0000	0.0000	
0090	Rect Poc	4.0000	-8.0000	0.0000	Skip list:
0100	Circ Poc	4.0000	-8.0000	0.0000	003 005 007 --- --- ---
0110	Frame	1.7500	-6.6625	0.0000	--- --- --- --- --- ---
0120	Thread	4.0000	-8.0000	0.1000	--- --- --- --- --- ---
0130	M41 ; Select Low Gear Range				
0140	Rotary	4.0000	-8.0000	0.1000	
0150	Comp Left	4.0000	-8.0000	0.1000	
0160	Repeat				
0170	End Prog	4.0000	-8.0000	Home	

Math Help
F6

Graph
F8

Teach Mode
F9

Accept
F10

All subprogram operations contain the following fields:

Start Block: Selects the first operation in the block of operations to repeat. This operation must lie before the place in your program where you are trying to repeat operations.

End Block: Selects the last operation in the block of operations to repeat. Again, this operation must lie prior to place in your program where you are trying to repeat operations, but not precede the start block.

Clearance Height: This field determines the Z height at which the tool is moved over the workpiece before being repositioned at the start of the contour. This value must meet or exceed the maximum Z height of all operations contained within the contour. If any operation places the tool at the Z home position, then you **must** tie this value to the home position (<F2> - Tie Z coordinate to home).

Plunge Rate: This is the speed at which the tool is repositioned on the Z-axis when moving to the **beginning** of the first move of the contour. This has no effect on a plunge that you have programmed into the contour; however, this has the effect of providing a vertical plunge for you in the event that you do not program your own plunge into the contour.

Other fields specific to the various subprogram operations are described in the next few pages.

Repeat to Depth (F1 in the Insert Subprogram Menu)

Intercon Mill v8.11				Current Part: E_Z_PART.ICN	
Operation #	Type	X	End Y	Z	N0160 Repeat to Depth
0010	:Demo Program				Start Block : N0020
0020	Rapid	0.0000	-5.0000	0.1000	End Block : N0080
0030	Rapid	4.0000	2.0000	1.0000	Total Depth : 0.5000 INC
0040	Line	7.0000	3.0000	1.0000	Depth Increment : 0.1000 INC
0050	Arc CW	10.0000	3.0000	1.0000	Clearance Height : 0.1000
0060	Tool #1	0.0000	0.0000	Home	Plunge Rate : 10.0000
0070	Tap	0.0000	0.0000	0.1000	
0080	Face	3.0000	6.0000	0.0000	Warning!
0090	Rect Poc	4.0000	-8.0000	0.0000	Operator must program first pass
0100	Circ Poc	4.0000	-8.0000	0.0000	at depth increment. Plunge must
0110	Frame	1.7500	-6.6625	0.0000	be programmed into this pass.
0120	Thread	4.0000	-8.0000	0.1000	
0130	M41 ; Select Low Gear Range				
0140	Rotary	4.0000	-8.0000	0.1000	
0150	Comp Left	4.0000	-8.0000	0.1000	
0160	Depth Rpt				
0170	End Prog	4.0000	-8.0000	Home	

Toggle
F3

Math
Help
F6

Graph
F8

Teach
Mode
F9

Accept
F10

The Repeat to Depth feature is useful for repeating a part contour when the material being machined is too thick to cut in just one pass. The contour formed by these operations may either be a closed contour or an open one. If a non-vertical plunge to the start of the contour is desired, it must be programmed into the contour (a vertical plunge between passes will be provided if one is not programmed).

Total Depth: Indicates how deep the final depth pass is to be. This is a positive value. Note that because the contour has been programmed at a depth of one depth increment below the workpiece surface, the final depth assumes that one depth pass has already been performed, and, thus, subtracts one depth increment from the total depth.

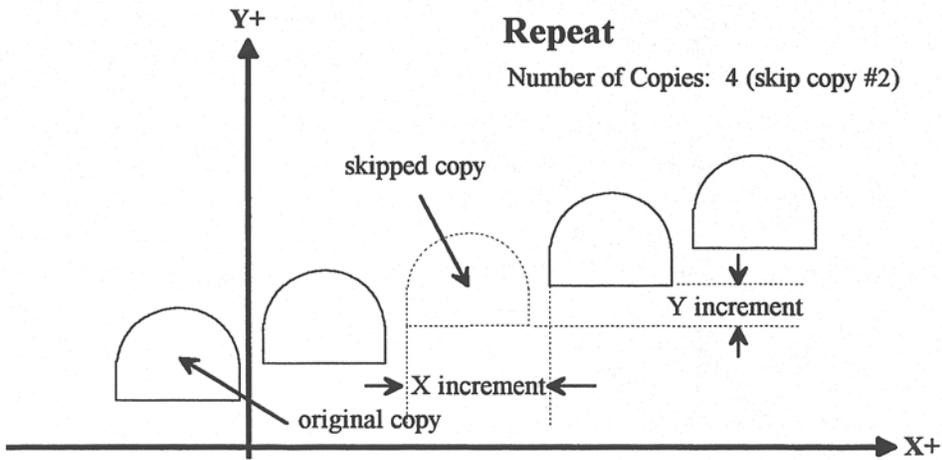
Depth Increment: Specifies the distance to drop each time the contour is repeated. This is a positive value that may not exceed the total depth of the operation.

When you have finished entering the required parameters, press <F10> to accept them. An operation labeled >D Rpt< will be inserted into your program in front of the highlighted operation. You may now edit this operation just as you would edit any other operation (use the cursor keys to highlight the >D Rpt< operation, then press <ENTER>).

* NOTE: If you wish to change the amount of the depth increment per pass after the contour has been programmed, you must also change the Z depth of all the operations inside the contour to correspond to the new increment.

Linear Repeat (F2 in the Insert Subprogram Menu)

The Linear Repeat feature is useful for repeating a part contour multiple times along a straight line. The contour formed by these operations may either be closed or open.

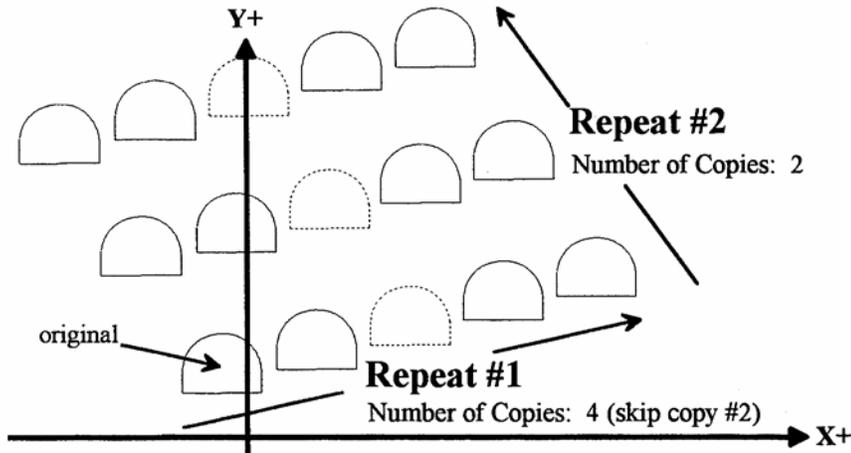


Increment: Specifies the X and Y distances between the start points of each copy of the contour.

Number of Copies: The number of times to repeat the contour.

Skip Copy: Prompt at which the list of skips may be modified. Entering positive integers adds skips to the list, while entering negative integers removes skips from the list. Multiple entries can be processed at the same time by separating them with commas.

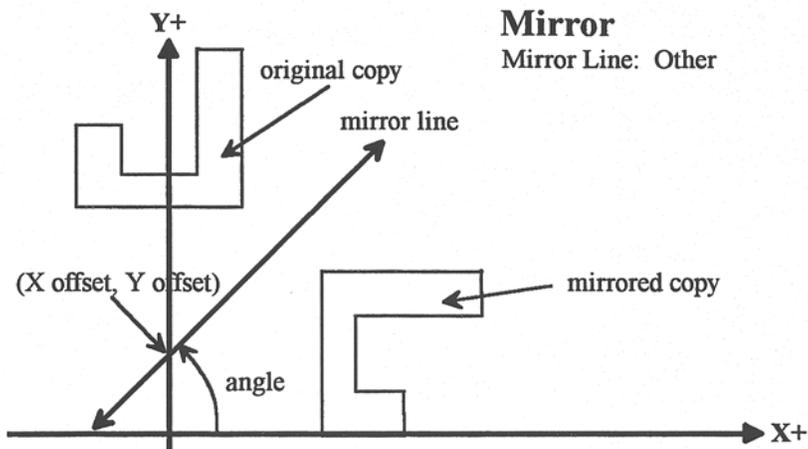
Skip List: List of skipped copies currently selected.



* NOTE: An array of repeats may be accomplished by doing a repeat of a repeat.

Mirror (F3 in the Insert Subprogram Menu)

The Mirror feature is useful for reflecting a part contour over a line. The contour formed by these operations may either be closed or open.



Mirror Line: Specifies the type of mirror line to use. Choices are Horizontal, Vertical and Other (user-defined).

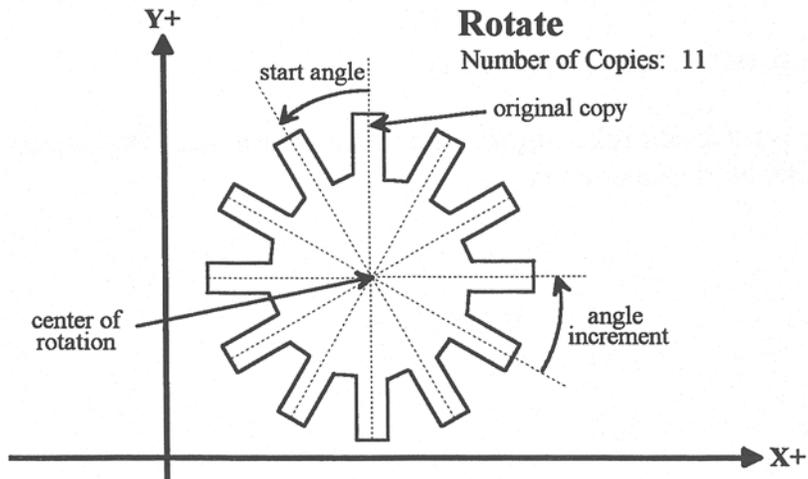
X Offset: Specifies the X coordinate of a point on the Mirror Line. This field will not be visible if a horizontal mirror line is being defined.

Y Offset: Specifies the Y coordinate of a point on the Mirror Line. This field will not be visible if a vertical mirror line is being defined.

Angle: Specifies the angle (from the three o'clock position) of the Mirror Line. This field will only be visible for a user-defined mirror line and is used in conjunction with the X Offset and Y Offset fields to define the mirror line.

Rotate (F4 in the Insert Subprogram Menu)

The Rotate feature is useful for rotating a part contour multiple times around a given point. The contour formed by these operations may either be closed or open.



Center: The XY location of the center of rotation.

Start Angle: The angle from the original copy at which the first copy will be placed. A positive angle indicates a counterclockwise rotation, while a negative angle indicates a clockwise rotation.

Angle Increment: The angle at which each copy after the first will be placed from the first copy. A positive angle indicates a counterclockwise, while a negative angle indicates a clockwise rotation.

Number of Copies: The number of times to rotate the contour.

End Angle: The angle at which the final rotated copy will start, **not the angle at which it will end**. A positive angle indicates a counterclockwise rotation, while a negative angle indicates a clockwise rotation.

Skip Copy: Prompt at which the list of skips may be modified. Entering positive integers add skips to the list, while entering negative integers remove skips from the list. Multiple entries can be processed at the same time by separating them with commas.

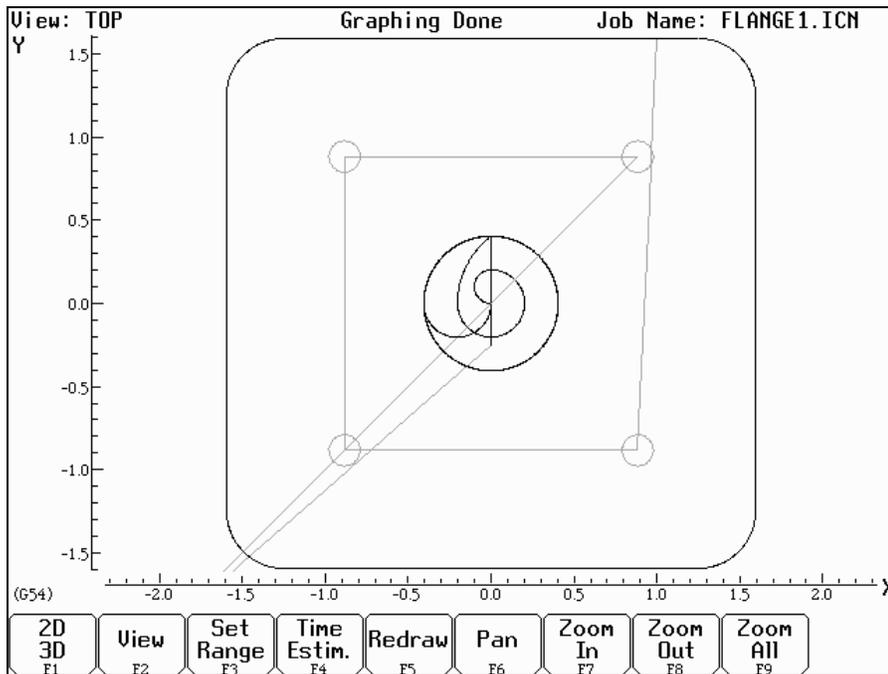
Skip List: List of skipped copies currently selected.

- NOTE: The user may enter the Start Angle, the Number of Copies and either the Angle Increment or the End Angle value, and Intercon will compute the rest.

Graphics

Intercon features three-dimensional previews of the tool path to be followed when milling the part. You may choose to display your project in one of two formats: a three-plane display, where the project is shown in each of the XY-, ZX-, and YZ-planes; an isometric display, which depicts the project three-dimensionally from an observer's point of view. To view the graphics, press <F8> from the Main Menu or from any Operation Edit screen.

The format of the display will be similar to the following:



The display will consist of arcs and/or lines that make up the tool path followed. Rapid (G0) moves will appear in color, while linear (G1) and arc (G2, G3) moves will be uncolored. Canned cycle operations (except the facing cycle) will also display a gray outline of the final result of the operation as the operation progresses. The type, start and end positions of the last displayed operation are shown in the lower left corner of the screen. The help bar that appears at the bottom of the screen shows which option will be activated upon pressing that key.

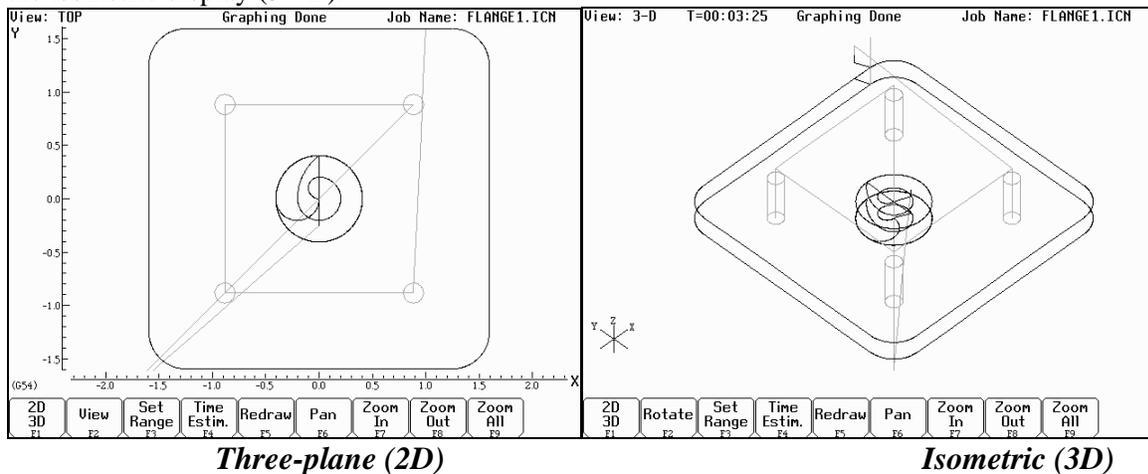
Many of the keys on the help bar work in conjunction with the arrow keys, which have the following functions:

Key	Pan & Zoom	Rotate
<Left Arrow>	Move crosshairs left	Rotate XY-plane left
<Right Arrow>	Move crosshairs right	Rotate XY-plane right
<Down Arrow>	Move crosshairs down	Rotate Z axis down
<Up Arrow>	Move crosshairs up	Rotate Z axis up
<ENTER>	Accept screen center	Accept axis orientation
<ESC>	Cancel Pan or Zoom	Cancel Rotate

Table 1 - Pan/Rotate Arrow Key Functions

F1 – 2D/3D

Pressing <F1> selects the format of the project display. This may take the form of the three-plane display (2 - D) or the isometric display (3 - D).



F2 – View/Rotate

In three-plane (2D) view, <F2> switches the point of view to a different plane. In isometric.(3D) view, <F2> enables the arrow keys to rotate the figure. The arrow keys actually rotate a larger version of the YZX axes figure that shows the orientation in which the part will be redrawn. Press <F2> to redraw without leaving rotation mode. Press <Enter> or <F5> to redraw and return the arrow keys to pan mode. Press <Esc> to cancel rotation.

F3 – Set Range

Press <F3> to specify the range of operations to draw. You will be prompted for a start block and an end block.

F4 – Time Estimate

Press <F4> to hide or display the time estimate.

F5 - Redraw

Pressing the <F5> key will cause the simulation to start again from the first operation (Redraw).

F6 - Pan

When using the pan feature, the project can be recentered in the display windows of the three-plane display, or rotated around the center of the isometric display screen. To enter pan mode, simply press the <F6> key or press one of the arrow keys. A set of crosshairs will appear. Adjust the center of the crosshairs to the new desired center. Press <Enter> or <F5> or <F6> to redraw the part with the new screen center point.

F7, F8 & F9: Zoom In, Zoom Out & Zoom All

The project can also be viewed in an enlarged or reduced state by pressing the <F7> or <F8> keys to activate Zoom In and Zoom Out respectively. Pressing <F9> redraws the project at its original size. Use the arrow keys to select the new screen center before zooming in or out.

1 – 9, 0, Space – Feed Rate Override & Hold

If no jog panel is attached (or “Keyboard” has been selected as the jog panel type) the number keys 1 – 9 and 0 choose feed rate overrides 10% - 90% and 100%, respectively. The space bar toggles feed hold on and off.

Math Help

Intercon provides a math assistance function to solve the trigonometric problems common in part drawings. To enter Math Help, press <F6> from any Edit Operation screen. The first time that you invoke Math Help, the following screen appears which shows all available *solvers*:

Intercon Mill v8.00 Current Part: E_Z_PART.ICN

N0020 Face

Math Help Chooser

F1 **Triangle: Right**

F2 Triangle: Other

F3 Tangent: Line Arc

F4 Tangent: Arc Arc

F5 Tangent: Line Arc Arc

F6 Tangent: Arc Arc Arc

F7 Intersection: Line Line

F8 Intersection: Line Arc

F9 Intersection: Arc Arc

The figures on the right are a graphical representation of the highlighted solver on the left. Pressing <ENTER> will display another menu that has various fields particular to the type of problem that is being solved. The graphic below displays the Right Triangle Calculator menu. The options that are available on the function keys are the same for every type of math help solver and perform the following operations:

Intercon Mill v8.00 Current Part: E_Z_PART.ICN

N0020 Face

Right Triangle Calculator

A X: **Unknown**
 Y: Unknown
 Angle: 90.000°

B X: Unknown
 Y: Unknown
 Angle: Unknown°

C X: Unknown
 Y: Unknown
 Angle: Unknown°

Length AB: Unknown
 BC: Unknown
 CA: Unknown

Prev Soln
F1

Next Soln
F2

Clear All
F3

Prev Solver
F4

Next Solver
F5

Hide Math
F6

Copy <<<
F7

Copy >>>
F8

Graphic On/Off
F9

F1 – Prev Soln

F2 – Next Soln

The Prev Soln and Next Soln options will cycle backward and forward, respectively, through the available solution sets for math solvers that may have multiple solutions. A status line near the bottom left of the screen appears once a valid solution has been found. The solution status line indicates the total number of solutions and the solution number that is currently represented by the graphic display on the right. For example, in an Arc Tangent Arcs math help, the display solution status may be “- Solution 1 of 8 -“. In this case, the Prev Soln and Next Soln can be used to cycle through all eight of the solutions.

F3 – Clear All

The Clear All option removes all solutions. It sets all fields for a particular solver to UNKNOWN.

F4 – Prev Solver

F5 – Next Solver

The Prev Solver and Next Solver options cycle backward and forward, respectively, through the various math help solvers. These options are shortcuts which have the same effect as pressing <ESC> to reach the main math help menu, navigating to the previous or next math help option, and then pressing <ENTER>.

F6 – Hide Math

The Hide Math option exits math help mode and returns to the operation edit menu. Pressing <F6> to invoke Math Help again will restore Math Help exactly as you left it. After copying values from Math Help, you can press <F6> to hide Math Help, then hit <F10> to accept the values entered.

F7 – Copy <<<

F8 – Copy >>>

The Copy <<< option will move the value from the selected edit operation field into the selected math help menu field and the Copy >>> operation will move the value from the selected math help menu field into the selected edit operation field. For both options, the selected fields in the math help menu and the operation edit menu are advanced. If the graphical display is visible when choosing one of these options, the effect is to turn off the graphics display. Only when the graphics display is off will the Copy operations actually copy values and advance field selections.

The currently selected fields have either a box drawn around them or are highlighted depending upon which menu is active. The active menu, which is either the math operation menu on the left hand side or the operation edit menu on the right hand side, depicts the selected field by highlighting the entire field. The non-active menu displays the active field with a box drawn around it. Use the arrow keys to select fields as described below.

F9 – Graphic On/Off

The Graphic On/Off option will toggle the graphical representation of the math help menu on the display.

↑ ↓ ← → (Arrow Keys) – Select Fields

The <LEFT> and <RIGHT> arrow keys are used to navigate between the math menu and the edit menu. The <UP> and <DOWN> arrow keys are used to navigate within a menu. To choose fields for the “Copy” option, above, use the <UP> and <DOWN> arrow keys to highlight the desired field in the menu and use the <LEFT> or <RIGHT> arrow keys to switch menus.

Other features common to all math help operations

In some math help operations, there will be an asterisk '*' character that appears immediately to the right of a field. This character marks the field as a "given" field, which means that the value of this field will be held constant in the process of solving the math equations.

F1 –Triangle:Right

F2 –Triangle:Other

Intercon Mill v8.00 Current Part: E_Z_PART.ICN

N0020 Face

Triangle Calculator

A X: Unknown
Y: Unknown

B Angle: Unknown°
X: Unknown
Y: Unknown

C Angle: Unknown°
X: Unknown
Y: Unknown

Length AB: Unknown
BC: Unknown
CA: Unknown

Prev Soln (F1) Next Soln (F2) Clear All (F3) Prev Solver (F4) Next Solver (F5) Hide Math (F6) Copy <<< (F7) Copy >>> (F8) Graphic On/Off (F9)

The screen will show **UNKNOWN** if the value of each parameter is not known. Math Help waits for known values to be entered, where:

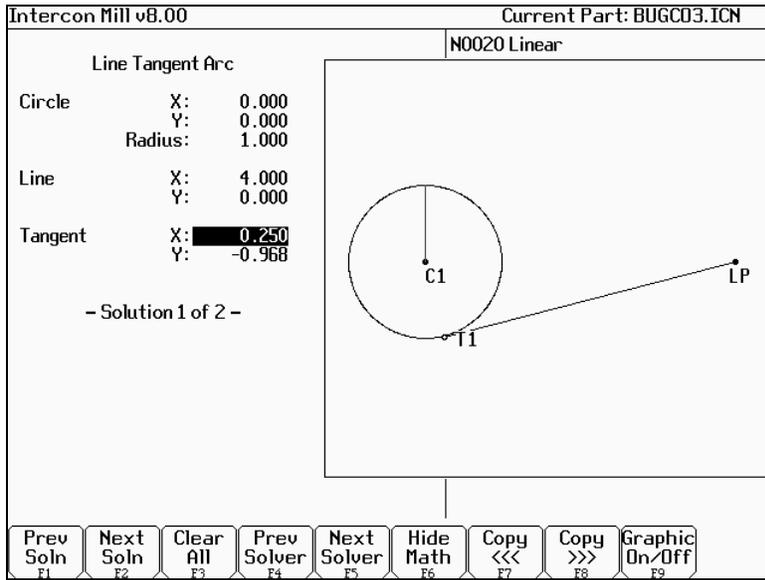
Point a, b, or c is the coordinate value for each corner of the triangle.

Angle A, B, or C is the angle at each point of the triangle.

Length of values are the distances between the points indicated.

Continue adding all the known parameters. Select parameters using the arrow soft keys. When Math Help solves the remaining unknown values, the screen will display them.

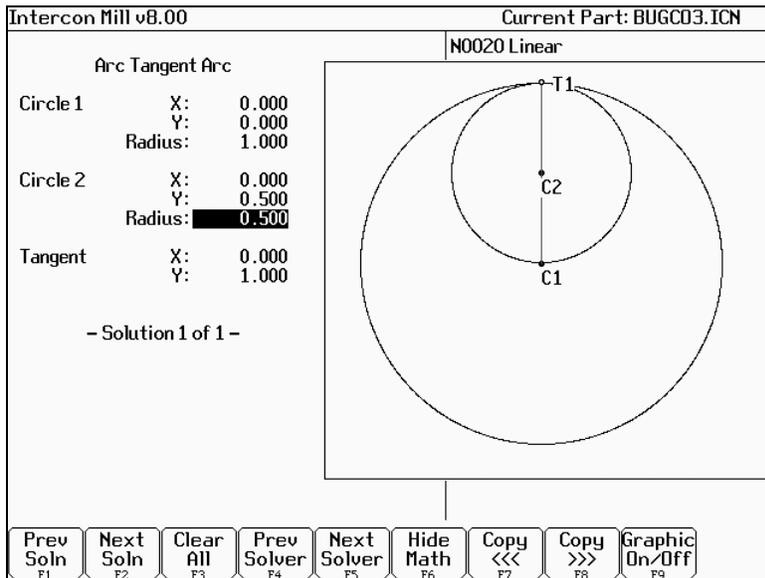
F3 – Tangent: Line Arc



Given the center (C1) and radius of an arc and 1 point (LP) on a line, find the lines tangent to the arc (defined by the tangent point (T1)).

You must enter the X and Y coordinates for the circle's center point, the circle's radius, and the X and Y coordinates for a point on the line.

F4 – Tangent: Arc Arc



Given the center points (CP1 and CP2) and radii (R1 and R2) of two arcs, find the point (T) at which they are tangent.

You must enter the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

F5 – Tangent: Line Arc Arc

Intercon Mill v8.00 Current Part: BUGC03.ICN

N0020 Linear

Line Tangent Arcs		
Circle 1	X:	0.000
	Y:	0.000
	Radius:	1.000
Circle 2	X:	4.000
	Y:	0.000
	Radius:	1.500
Tangent 1	X:	-0.125
	Y:	0.992
Tangent 2	X:	3.813
	Y:	1.488

- Solution 1 of 4 -

Prev Soln (F1) Next Soln (F2) Clear All (F3) Prev Solver (F4) Next Solver (F5) Hide Math (F6) Copy <<< (F7) Copy >>> (F8) Graphic On/Off (F9)

Given the center points (CP1 and CP2) and radii (R1 and R2) of two arcs, find the lines (defined by T1 - T8) tangent to both arcs.

You must enter the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

F6 – Tangent: Arc Arc Arc

Intercon Mill v8.00 Current Part: BUGC03.ICN

N0020 Linear

Arc Tangent Arcs		
Circle 1	X:	-2.000
	Y:	0.000
	Radius:	1.000
Circle 2	X:	4.000
	Y:	0.000
	Radius:	2.000
Circle 3	X:	0.250
	Y:	-3.307
	Radius:	3.000
Tangent 1	X:	-1.438
	Y:	-0.827
Tangent 2	X:	2.500
	Y:	-1.323

- Solution 1 of 4 -

Prev Soln (F1) Next Soln (F2) Clear All (F3) Prev Solver (F4) Next Solver (F5) Hide Math (F6) Copy <<< (F7) Copy >>> (F8) Graphic On/Off (F9)

Given the center points (C1 and C2) and radii of two arcs and the radius of a third arc, find the center point of the third arc and the tangent points (T1 and T2).

You must enter the radius of the tangent arc, the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

F7 – Intersection: Line Line

Intercon Mill v8.00		Current Part: BUGCD3.ICN	
N0020 Linear			
Line Intersection Line			
Line 1	X1:	0.000 *	
	Y1:	0.000 *	
	X2:	5.000 *	
	Y2:	5.000 *	
	Angle:	45.000	
Line 2	X1:	0.000 *	
	Y1:	5.000 *	
	X2:	8.000 *	
	Y2:	-1.000 *	
	Angle:	323.130	
Inter-section	X:	2.857	
	Y:	2.857	
* Given (Space to Toggle)			
- Solution 1 of 1 -			

Prev Soln F1	Next Soln F2	Clear All F3	Prev Solver F4	Next Solver F5	Hide Math F6	Copy <<< F7	Copy >>> F8	Graphic On/Off F9
-----------------	-----------------	-----------------	-------------------	-------------------	-----------------	----------------	----------------	----------------------

You must enter the X and Y coordinates for 1 point on each line, and also one of the following:

- * the X and Y coordinates for a second point
- * the X coordinate for a second point and the angle from horizontal
- * the Y coordinate for a second point and the angle from horizontal
- * the angle from horizontal only

F8 – Intersection: Line Arc

Intercon Mill v8.00 Current Part: BUGC03.ICN

N0020 Linear

Line Intersection Arc

Circle X: 0.000
 Y: 0.000
 Radius: 1.000

Line X1: -2.000 *
 Y1: -2.000 *
 X2: 2.000 *
 Y2: 0.309 *
 Angle: **30.000** *

Int. 1 X1: 0.956
 Y1: -0.293
Int. 2 X2: -0.224
 Y2: -0.975

* Given (Space to Toggle)
- Solution 1 of 1 -

Prev Soln F1
Next Soln F2
Clear All F3
Prev Solver F4
Next Solver F5
Hide Math F6
Copy <<< F7
Copy >>> F8
Graphic On/Off F9

Given the center (CP) and radius (R) of an arc, 1 point (LP1) and either a second point (LP2) or one coordinate (LP2 X or Y) and the angle from horizontal, find the intersection point(s) (I1 and I2).

You must enter the X and Y coordinates for the circle's center point, the circle's radius, the X and Y coordinates for one point on the line, and one of the following:

- * the X and Y coordinates of a second point on the line
- * the X coordinate of a second point and the angle from horizontal
- * the Y coordinate of a second point and the angle from horizontal

F9 – Intersection: Arc Arc

Intercon Mill v8.00 Current Part: BUGC03.ICN

N0020 Linear

Arc Intersection Arc

Circle 1 X: **0.000**
 Y: 0.000
 Radius: 3.000

Circle 2 X: 4.000
 Y: 0.000
 Radius: 4.000

Int. 1 X1: 1.125
 Y1: -2.781
Int. 2 X2: 1.125
 Y2: 2.781

Prev Soln F1
Next Soln F2
Clear All F3
Prev Solver F4
Next Solver F5
Hide Math F6
Copy <<< F7
Copy >>> F8
Graphic On/Off F9

Given the center points (CP1 and CP2) and the radii (R1 and R2) of two arcs, find the intersection point(s) (I1 and I2) of the arcs.

You must enter the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

Intercon Tutorial #1

This is a step-by-step instructional example of going from blueprint to part with Intercon. The tool path to be created is for the part shown in Figure 1. For instructional purposes, this part will be programmed to cut into stock held in 3 fixtures, 6 inches apart along the X-axis.

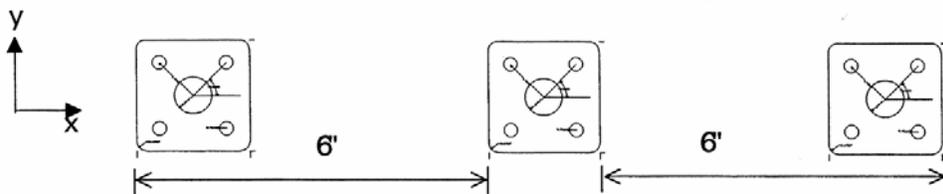
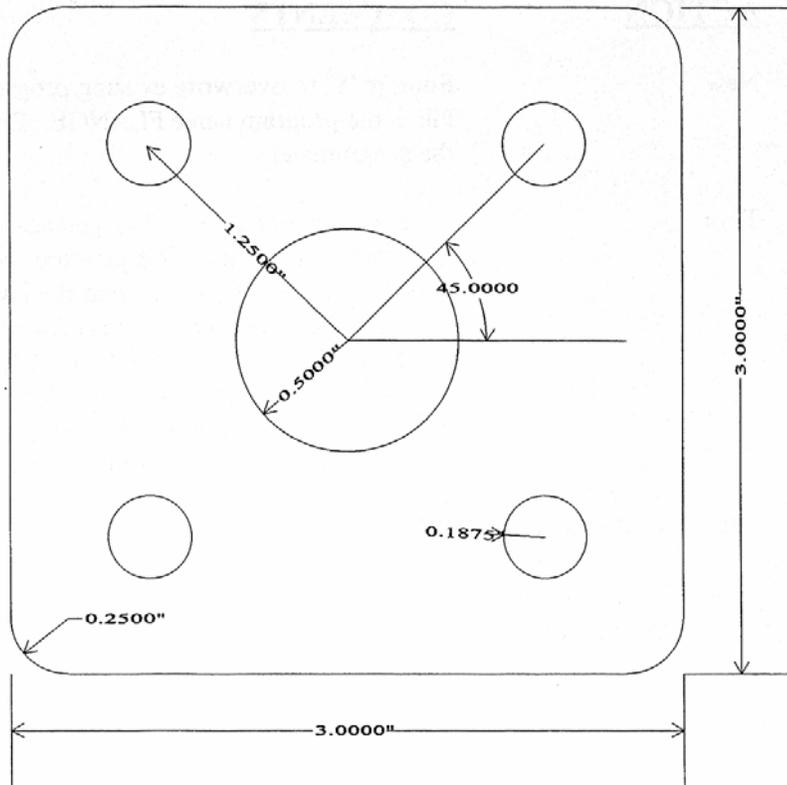


FIG. 1 Blueprint of flange part and the 3 fixtures.

Part Creation

Each feature of the part will become an operation in your program. Before beginning, decide where you want the **X0** and **Y0** reference. For this particular part, the center of the bolt hole pattern was selected. Now start the Intercon program (from the **CNC7 Main** screen, press <**F5**> for CAM). Beginning from the Intercon File Menu (press <**F1**> "File" if the file menu is not shown) the following series of keystrokes will describe the step-by-step process of designing the part shown in Figure 1.

PRESS	ACTION	COMMENTS
F1	New	Fill in the program name FLANGE. Enter your name in as the programmer. Enter the description as "Intercon Tutorial #1".
F4	Tool	Describe the tool below. The position values specify where to do the tool change. This position should be a point outside of the workpiece so that the last tool can be removed from the chuck and the new tool can be inserted. The Yes in the 'Actual Tool Change' field turns off the spindle and coolant upon reaching this spot. Use a 0.3750-inch diameter cutter. The length and diameter are updated based on the offsets. (The longest tool should have a 0.0000 length).

N0020 Tool change

Tool Number	:	1
Description	:	3/8" end mill
Position:	X	: -2.0000
Y	:	-2.0000
Tool H Offset	:	1
Tool Height	:	0.0000
Tool D Offset	:	1
Tool Diameter	:	0.3750
Spindle Speed	:	1000
Spindle Direction	:	CW (M3)
Coolant Type	:	Flood (M8)
Actual Tool Change	:	Yes

F10	Accept	Keep selected values.
F5	Cycles	Access the list of available Canned Cycles.
F6	C Pocket	Start with the 1.0000-inch diameter circular pocket. Enter the following values:

N0030 Circular pocket

Center:	X	:	0.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Diameter		:	1.0000	
Cleanout		:	Yes	
Depth:	Total	:	0.5000	INC
Per Pass		:	0.2500	
Plunge Rate		:	2.0000	[M]
Plunge Type		:	Ramped	
Plunge Angle		:	0.00°	
Rough Cuts		:	Conventional	
Stepover		:	0.2250	
Feedrate		:	20.0000	[M]
Finish Pass		:	Climb	
Amount		:	0.0020	
Feedrate		:	10.0000	[M]

- F10** Accept Keep selected values.
- F5** Cycles Access the list of available Canned Cycles.
- F1** Drill Select drilling cycles
- F2** Drill BHC Select the bolt hole circle type of drilling cycles

N0040 Drill bolt holes

Cycle Type		:	Drilling	
Center:	X	:	0.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Clearance Height		:	0.2500	INC
'Rapid To' Depth		:	0.1000	INC
Depth:	Total	:	0.5000	INC
Plunge Rate		:	2.0000	
Dwell Time		:	0.0000	
Number of holes		:	4	
Radius		:	1.2500	
Start angle		:	45.00°	

- F8** Graph Display a preview of the part up to this point. This preview can be used to detect problems that may occur if the part was cut now.

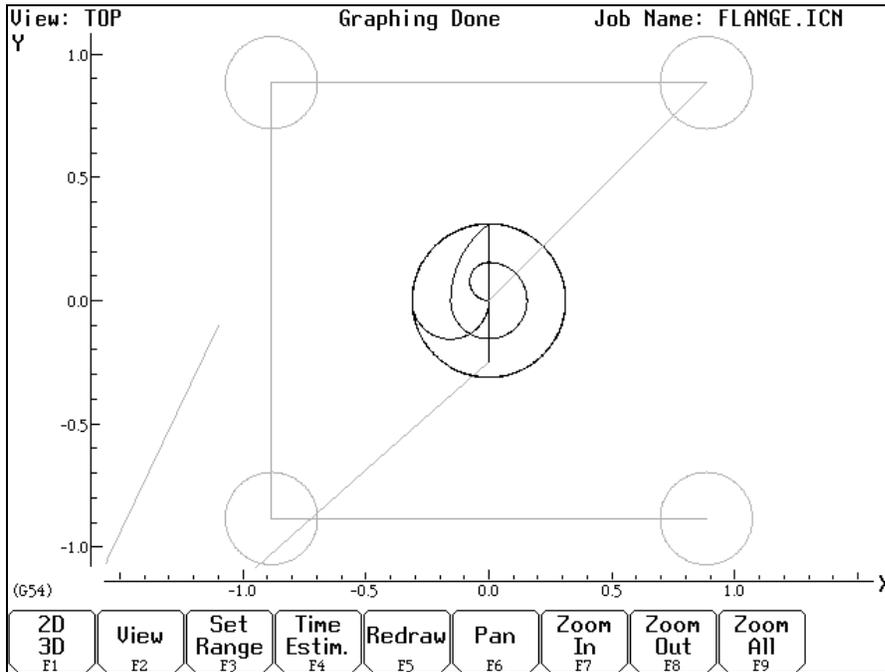


FIG. 2 - Graphics screen showing bolt holes and circular pocket

ESC/CANCEL	Cancel	Return to the editing screen.
F10	Accept	Keep selected values.
F5	Cycles	Access the list of available Canned Cycles.
F7	Frame	Now add an outside frame to cut the flange out of the material. The flange is 3.0000 inches long by 3.0000 inches wide, and has rounded corners with 0.2500-inch radii.

N0050 Frame mill

Frame	:	Outside	
Center:	X	: 0.0000	
	Y	: 0.0000	
Surface Height	:	0.0000	
Length	X	: 3.0000	INC
Width	Y	: 3.0000	INC
Corner Radius	:	0.2500	
Depth:	Total	: 0.5000	INC
per Pass	:	0.2500	
Plunge Rate	:	2.0000	
Plunge Type	:	Ramped	
Plunge Angle	:	0.00°	
Cut Type	:	Conventional	
Feedrate	:	10.0000	

F8	Graph	Display a preview of the part up to this point. This preview can be used to detect problems that may occur if the part was cut now.
-----------	-------	-------------------------------------------------------------------------------------------------------------------------------------

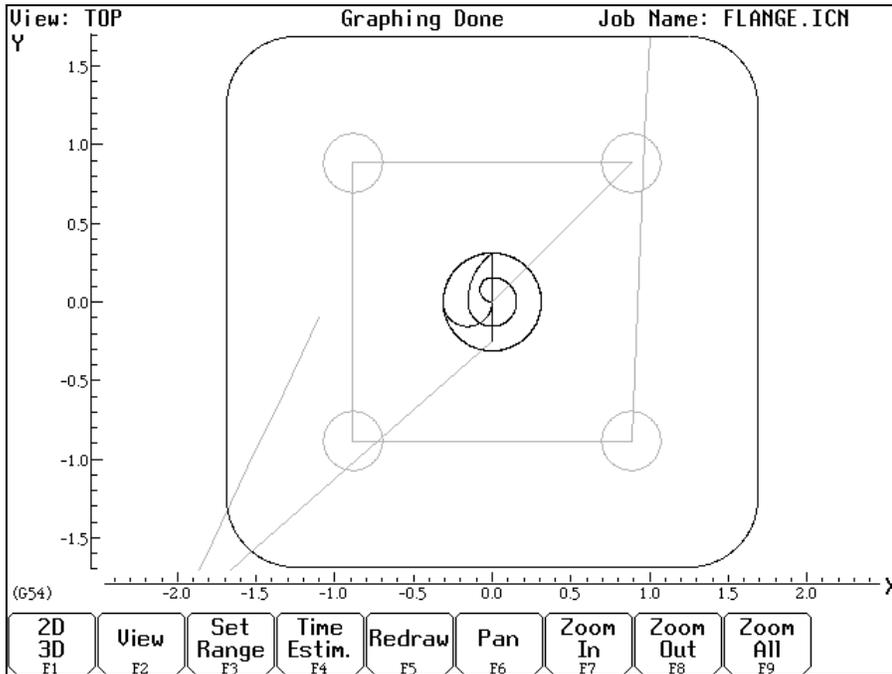


FIG. 3 - Graphics screen showing part with bolt holes and outer frame

ESC/CANCEL Cancel Return to the editing screen.

F10 Accept Keep selected values.

F9 Subpgm Access the **Insert Subprogram** screen.

F2 Repeat We programmed the part to cut one copy only. We now want to repeat the part 2 more times at an incremental distance of 6 inches along the X-axis. The part can now be cut into the stock mounted into the two other fixtures. The part begins with the circular pocket in operation #0030 and ends with the linear mill in operation #0050. Press <F2> to enter "Home" for "Clearance Height."

N0060 Repeat

Start Block	:	N0030
End Block	:	N0050
Increment	X :	6.0000
	Y :	0.0000
Clearance Height	:	Home
Plunge Rate	:	2.0000
Number of copies	:	2

F8 Graph Display a preview of the parts. This preview can be used to detect problems that may occur if the part was cut now.

ESC/CANCEL Cancel Return to Repeat Subprogram.

F10 Accept Keep selected values if you wish to cut these two extra parts.

If you do not wish to do this, press <ESC/CANCEL>.

ESC/CANCEL	Cancel	Creation of the part is complete. Intercon programs automatically turn the spindle and coolant off at the end.
F1	File	Press <F3> to save the part under its current name. Press <F4> to save it under a new name.
F10	Post	The CNC file needed to run this part on your mill will be generated at this time. The Intercon program displays the operation number of the part it is processing as it works through each operation in memory:

Generating CNC Program
Block 0050

As it processes each operation, it checks for values that, if used, will cause incorrect code to be produced. If such a value is found, a message will appear on the screen alerting you of the problem. For example, a problem with a Frame Mill may produce this message:

Message
Corner radius too small for Cutter...hit a key.

Changes to the part would then be required to allow proper code generation to proceed. If no problems are encountered during code generation, the following message appears:

Message
CNC code generation successful

You are now finished designing your part. In order to run your part, you now need to return to the CNC7 software.

Program Finished!

Milling The Part

Now that the part has been programmed, it is time to mill it. Take your material and clamp it to the table. Remember that the clamps must be positioned such that they do not interfere with the tool as it cuts. You may choose either to place the clamps around the edges of the material for the entire process and let the part drop through upon completion, or you may wish to pause after milling the circular pockets and place clamps through the holes to prevent the part from moving. The second option decreases the chance of the part being marred because it moved during milling.

Now you need to set your XYZ reference points. Insert your longest tool in the quill and follow the procedure listed below:

PRESS	ACTION	COMMENTS
JOG KEYS	Jog Axis	Jog the table so that your tool rests on the stock at the location that will represent X0 and Y0.
F1	Setup	Enter the main program (CNC7) Setup screen. We are going to establish the part XYZ zero at the current tool location.
F1	Part	Access the Part Setup options.
F10	Set	Set your X zero position at current tool location.
F1	Next Axis	Select the Y-axis next.
F10	Set	Set your Y zero position at current tool location.
F1	Next Axis	Select the Z-axis next.
F10	Set	Set your Z zero position at current tool location.
TOOL CHECK	Move tool to Tool Check position	Moves the quill to the Z home position if the home position has been set. Moves tool to Z+limit switch and sets home position if not.
ESC/CANCEL	Cancel	Leave Part Setup screen.
F2	Tool	Access Tool Library Editor. This is the place where we want to measure the actual heights of our tools (since we could not set the actual values in Intercon).
F1	Offsets	You need to make sure that the tool diameter and height offset values are the correct ones for the tools you are going to be using. Inspect the values for D001, H001, D002 and H002. D001 should be 0.1875, H1 should be 0.0000 (the two inch tool), D002 should be 0.2500 and H002 should be -1.0000 (the one inch tool). If any of these values are incorrect, use the arrow keys to select the incorrect values. Enter the new values in their places and press <ENTER> to accept them. <i>NOTE:</i> The tool heights used above are merely example heights. In order to accurately measure the heights of your tools, see the description of measuring tool heights on page 5 - 10 of this tutorial.
F10	Save	Keep the updated tool offset library values.

F2	Tools	Now you need to make sure that each tool uses the correct diameter and height offset values. Inspect the values for T001 and T002. T1 should use H001 and D001, while T002 should use H002 and D002. If any of these values are incorrect, use the arrow keys to select the incorrect values. Enter the new values in their places and press <ENTER> to accept them. You may also select spindle and coolant settings for your tools here, or enter a short description of the tool.
F10	Save	Keep the updated Tool Library values.
ESC/CANCEL	Cancel	Leave Tool Setup. Return to the CNC7 Setup Screen.
ESC/CANCEL	Cancel	Leave CNC7 Setup. Return to the CNC7 Main Screen.
CYCLE START	Run Program	The CYCLE START/START button is located on your jog panel. This key will cause the mill to begin cutting your part

Tutorial Complete!

Intercon Tutorial #2

This demonstration will show you how to create a tool path for a part from a blueprint using the Math Help function of Intercon. The tool path to be created is for the part shown in Figure 1 below.

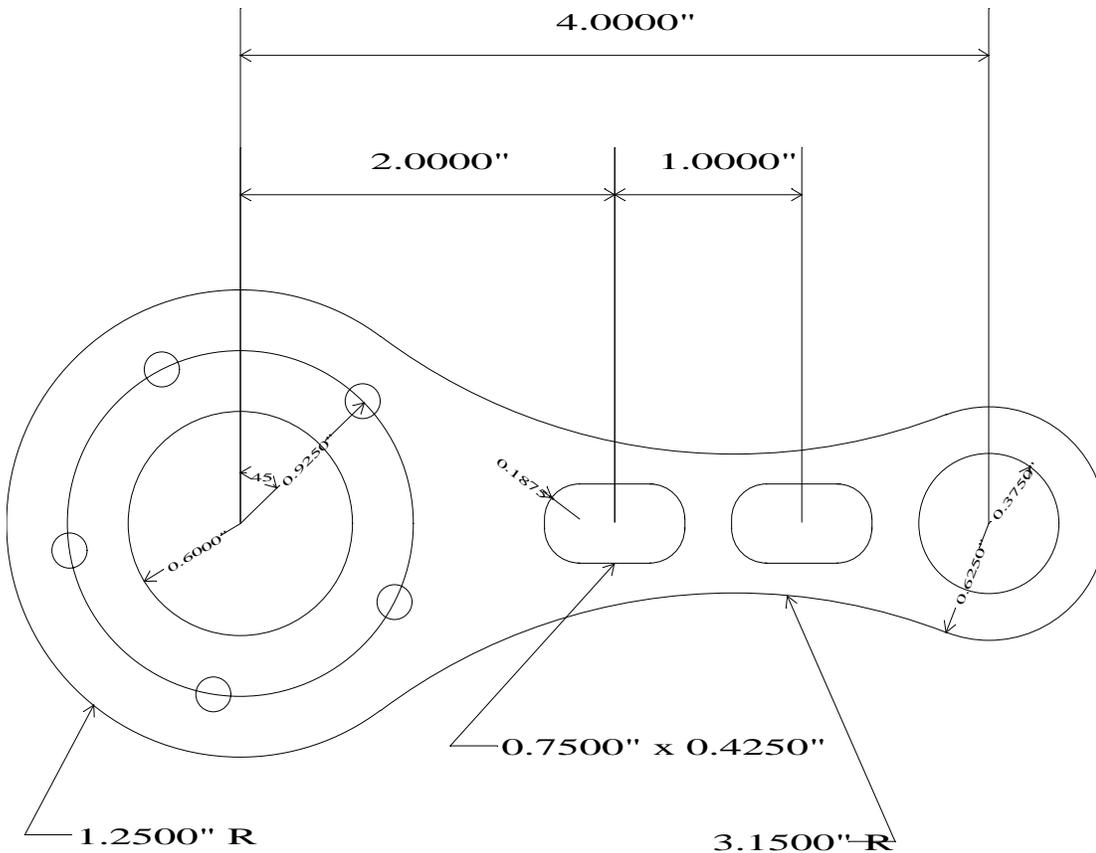


FIG. 1 - Part to be machined.

Part Creation

The process of creating a part is called part programming. Each feature of the part will become an operation in your program. Before beginning, decide where you want the **X0** and **Y0** reference. For this particular demo, the center of the Bolt Hole pattern was selected for convenience). Beginning from the Intercon File Menu (press <F1> "File" if the file menu is not shown) the following series of keystrokes will describe the step-by-step process of designing the part shown in Figure 1.

PRESS	ACTION	COMMENTS
F1	New	Create a new program by filling in the appropriate program name (we recommend C_ROD) and your name. Press Enter or <F10> to accept the new name. Enter "Intercon Tutorial #2" for the description. Press <F10> to accept.
F4	Tool (M6)	Describe the tool below. The position values specify where to do the tool change. The Yes in the 'Actual Tool Change' field turns off the spindle and coolant upon reaching this spot. Use a 2.0000 x 0.1875 inch cutter. The height and diameter are updated based on the offsets. (The longest tool should have a 0.0000 height offset). If this tool does not have the desired spindle (CW) and coolant (Flood) settings, you should also select these values to match your particular machine setup.

N0020 Tool change

Tool Number	:	1
Description	:	Tool #1 H001 D001
Position:	X :	0.0000
	Y :	0.0000
Tool H Offset	:	1
(Tool Height	:	0.0000)
Tool D Offset	:	1
Tool Diameter	:	0.1875
Spindle Speed	:	1000
Spindle Direction	:	CW (M3)
Coolant Type	:	Flood (M8)
Actual Tool Change	:	Yes

Notice for this particular screen, the Tool height shows '0.0000', since it has the same tool height as the Reference tool. However, your screen may differ since Intercon cannot change the Reference tool height in the Tool Library. This will change when you run this program. Refer to the Measuring Tool Heights section on page 6-24 for more details.

F10	Accept	Keep selected values.
F5	Cycles	Access the list of available Canned Cycles.
F1	Drill	Select drilling cycles
F2	Drill BHC	Select a bolt hole circle operation.

The clearance height is the Z height from which the downward rapid traverse begins before each hole. It is also the Z height to which the tool returns upon completion of drilling the hole.

The 'Rapid To' depth is the Z height to which the tool rapid traverses before drilling a hole.

BOLT HOLE CIRCLE

Number of holes: 5

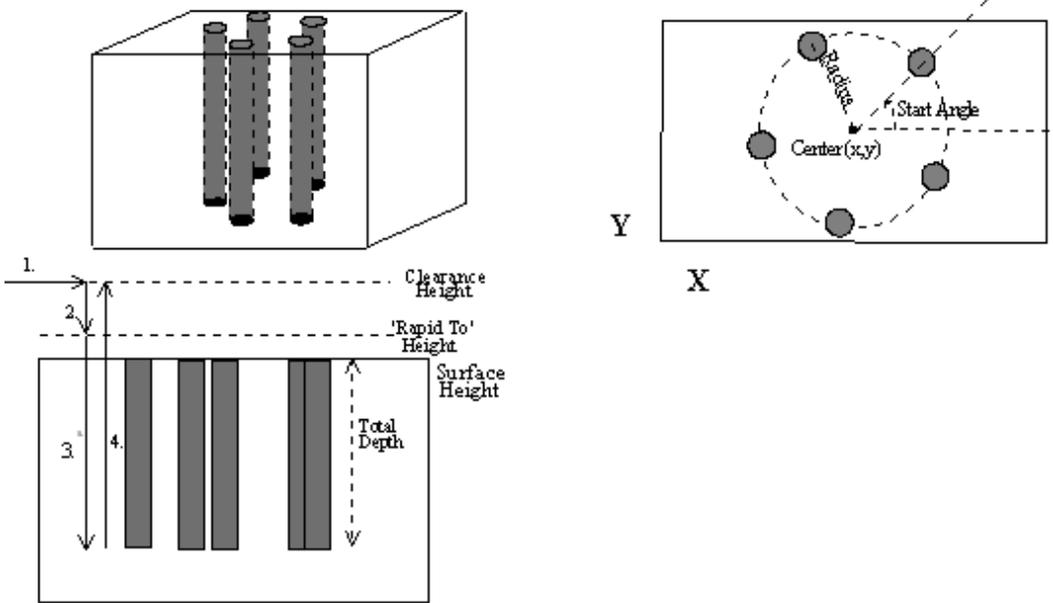


FIG. 2 - Bolt Hole Circle

N0030 Drill bolt holes

Cycle Type	:	Drilling	
Center:	X	: 0.0000	
	Y	: 0.0000	
Surface Height	:	0.0000	
Clearance Height	:	0.5000	INC
'Rapid To' Depth	:	0.1000	INC
Depth:	Total	: 0.5100	INC
Plunge Rate	:	2.0000[M]	
Dwell Time	:	0.0000	
Number of holes	:	5	
Radius	:	0.9250	
Start angle	:	45.00°	

F10 Accept Keep selected values.

F4 Tool (M6) Use a 1.0000 x 0.2500 inch cutter now. Notice that the tool height shown below is a negative value. This value represents the difference in height between this tool and the longest tool being used. The longest tool used (in this case, operation N0020 above) has a height of 0.0000. Again, do not be alarmed if the Tool Height is not -1 for operation N0040. If this tool does not have the desired spindle (CW) and coolant (Flood) settings, you should also enter values specific to your machine setup.

N0040 Tool change

Tool Number	:	2
Description	:	Tool #2 H002 D002
Position:	X	: 0.0000
	Y	: 0.0000
Tool H Offset	:	2
(Tool Height	:	your tool)
Tool D Offset	:	2
Tool Diameter	:	0.2500
Spindle Speed	:	1000
Spindle Direction	:	CW (M3)
Coolant Type	:	Flood (M8)
Actual Tool Change	:	Yes

F10	Accept	Keep selected values.
F5	Cycle	Access the list of available canned cycles.
F6	C Pckt	Start with 1.2000 inch diameter Pocket.

N0050 Circular pocket

Center:	X	:	0.0000
	Y	:	0.0000
Surface Height	:		0.0000
Diameter	:		1.2000
Cleanout	:		Yes
Depth:	Total	:	0.5100 INC
	Per Pass	:	0.2500
	Plunge Rate	:	2.0000 [M]
	Plunge Type	:	Ramped
	Plunge Angle	:	0.00 °
Rough Cuts	:		Conventional
	Stepover	:	0.2000
	Feedrate	:	2.0000 [M]
Finish Pass	:		Climb
	Amount	:	0.1000
	Feedrate	:	2.0000 [M]

F10	Accept	Keep selected values.
F5	Cycle	Access the list of available canned cycles.
F6	C Pckt	Repeat above pocket cycle. The center X value 4.0000 and the diameter is 0.7500 inches.

N0060 Circular pocket

Center:	X	:	4.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Diameter		:	0.7500	
Cleanout		:	Yes	
Depth:	Total	:	0.5100	INC
	Per Pass	:	0.2500	
	Plunge Rate	:	2.0000	
	Plunge Type	:	Ramped	
	Plunge Angle	:	0.00	°
Rough Cuts		:	Conventional	
	Stepover	:	0.2000	
	Feedrate	:	2.0000	
Finish Pass		:	Climb	
	Amount	:	0.1000	
	Feedrate	:	2.0000	

- F10** Accept Keep selected values.
- F5** Cycle Access the list of available Canned Cycles.
- F5** R Pckt Cut the first rectangular pocket.

N0070 Rectangular pocket

Center:	X	:	2.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Length	(X)	:	0.7500	INC
Width	(Y)	:	0.4250	INC
Corner Radius		:	0.1875	
Depth:	Total	:	0.2500	INC
	Per Pass	:	0.2500	
	Plunge Rate	:	2.0000	
	Plunge Type	:	Ramped	
	Plunge Angle	:	0.00	°
Rough Cuts		:	Conventional	
	Stepover	:	0.1000	
	Feedrate	:	2.0000	
Finish Pass		:	None	
	Amount	:	0.0000	
	Feedrate	:	2.0000	

- F10** Accept Keep selected values.
- F5** Cycle Access the list of available Canned Cycles.
- F5** R Pckt Repeat above Pocket cycle. The center X value lies at 3.0000.

N0080 Rectangular pocket

Center:	X :	3.0000	
	Y :	0.0000	
Surface Height	:	0.0000	
Length	(X) :	0.7500	INC
Width	(Y) :	0.4250	INC
Corner Radius	:	0.1875	
Depth:	Total :	0.2500	INC
Per Pass	:	0.2500	
Plunge Rate	:	2.0000	
Plunge Type	:	Ramped	
Plunge Angle	:	0.00	°
Rough Cuts	:	Conventional	
Stepover	:	0.1000	
Feedrate	:	2.0000	
Finish Pass	:	None	
Amount	:	0.0000	
Feedrate	:	2.0000	

F10 Accept Keep selected values.

F7 Cutter Comp Hit <Space> until **Left** cutter compensation is selected. The tool must move outside of the part outline at a distance at least equal to its radius so the part outline is the correct size. Cutter compensation should be turned on before a rapid to maintain proper line and arc travel.

N0090 Comp left

F10 Accept Keep selected values.

F1 Rapid (G0) Move to a location outside the part. The purpose of this move is to prepare to use cutter compensation on the tool.

N0100 Rapid traverse

End:	X :	5.0000
	Y :	0.5000
	Z :	0.1000
Angle	:	14.04°
Length	:	2.0616

F10 Accept Keep selected values.

F3 Arc (G2) Mill up to the edge of the part to cut the first arc. This is called a lead-in move. The cutter compensation selected above needs a lead-in move in order to position the cutter before milling the actual part.

N0110 Arc

Arc Type:	:	EP&R
Mid	X :	
	Y :	
	Z :	
End	X :	4.625
	Y :	0.0000
	Z :	-0.0500
Center:	X :	
	Y :	
	Z :	
Angle	:	
Radius	:	.5
Plane	:	XY
Direction	:	CCW
Connect Radius	:	0.0000
Feedrate	:	10.0000
Angle <= 180	:	Y

You will see that after you enter in these values, the other points and arcs will be entered in automatically.

F10 Accept Keep selected values.

F3 Arc (G2 & G3) The first arc to be cut is labeled as ARC 1 in Figure 3 below. The start point, labeled P1, is the end point of the previous move. The end point of the arc will be generated with Math Help. We will be using end point and radius (EP&R) arcs.

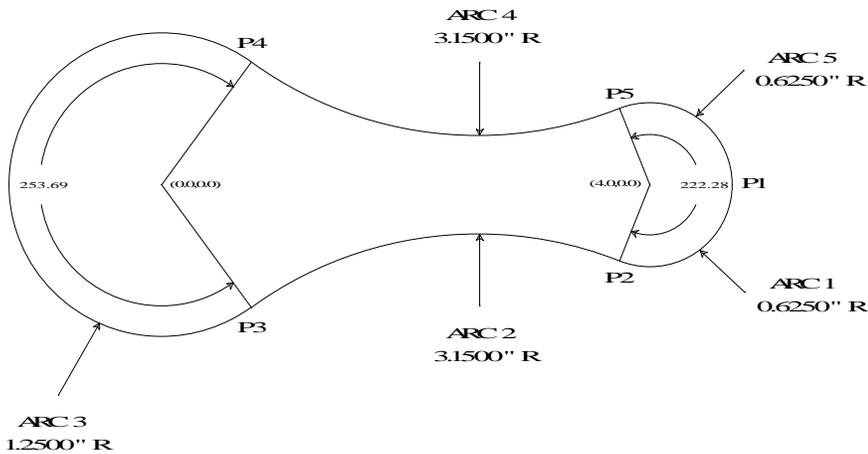


FIG 3. - Tangent point and arc reference.

N0120 Arc

Arc type	:	EP&R
Mid	X :	
	Y :	
	Z :	
End:	X :	4.6250
	Y :	0.0000
	Z :	-0.0500
Center:	X :	
	Y :	
	Z :	
Angle	:	
Radius	:	.625
Plane	:	XY
Direction	:	CW
Connect Radius	:	0.0000
Feedrate	:	10.0000
Angle <= 180°	:	Yes

- F6** Math Help We are trying to find end points for the arcs that make up the outside edge of the part. Note: the main Math Help menu should appear listing all available Math Help solvers. If it does not appear, press <Esc/Cancel>.
- F6** Tangent Arc Arc Arc This scenario will generate tangent points P2 - P5 of Figure 3. Enter the values as shown below:

Arc Tangent Arcs:

Circle 1:	CP X :	4.0000
	Y :	0.0000
	Radius :	0.6250
Circle 2:	CP X :	0.0000
	Y :	0.0000
	Radius :	1.2500
	Radius :	3.1500

Intercon will calculate the missing values for this scenario.

- F2** Next Soln. Find scenario that corresponds to the actual arcs being milled. Observe Figure 4. Point T1 is the one needed.

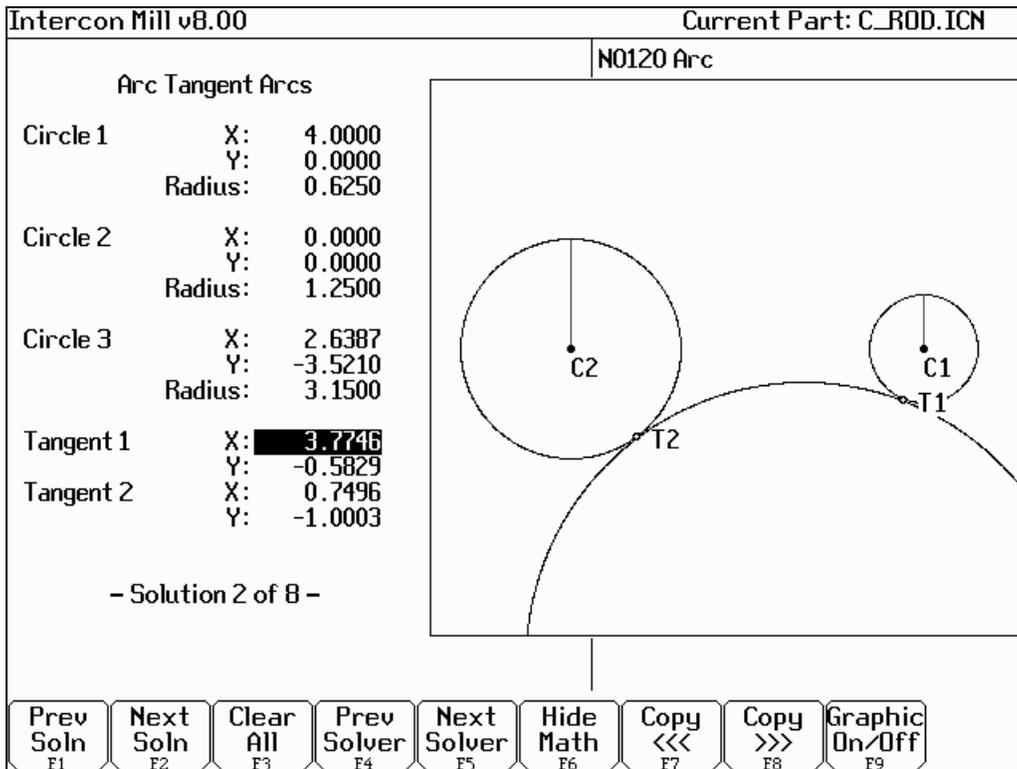


FIG. 4 - Screen showing Math Help Arc Tangent Arc solutions

- | | | |
|------------------|----------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ARROWS | Move Cursor | If necessary, move the block cursor to the Tangent 1 X field as shown above. Note: Use only $\langle \uparrow \rangle$ and $\langle \downarrow \rangle$. If you press the right arrow, press the left arrow to get back to the Math Help fields. |
| F9 | Graphic On/Off | Press to hide the graphical display and reveal the arc operation behind it. |
| → (ARROW) | Move Cursor | Move the cursor to the arc operation. The solid block cursor on the left side of the screen will be replaced by an outlined rectangle and the solid block will appear in the arc operation on the right. |
| ARROWS | Move Cursor | Move the block cursor to the End X field of the arc operation. As before, use only $\langle \uparrow \rangle$ and $\langle \downarrow \rangle$. |
| F8 | Copy >>> | Transfer the tangent point T1 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically. |
| F8 | Copy >>> | Transfer the tangent point T1 value for Y into the end point Y coordinate. |
| ARROWS | Move Cursor | Move down to the radius field and enter the radius of the arc labeled as ARC 1 in Figure 3. (This radius is 0.6250 in.). |

Arc type	:	EP&R
Mid	X :	
	Y :	
	Z :	
End	X :	3.7746
	Y :	-0.5829
	Z :	-0.0500
Center:	X :	
	Y :	
	Z :	
Angle	:	
Radius	:	0.6250
Plane	:	XY
Direction	:	CW
Feedrate	:	10.0000
Angle <= 180°	:	Yes

- F6** Hide Math Hide Math Help temporarily. (We will return later to pick up the other tangent points.)
- F8** Graph Observe Figure 5. The graphics show a preview of the part up to this point. This preview can be used to detect problems that may occur if the part was cut now.

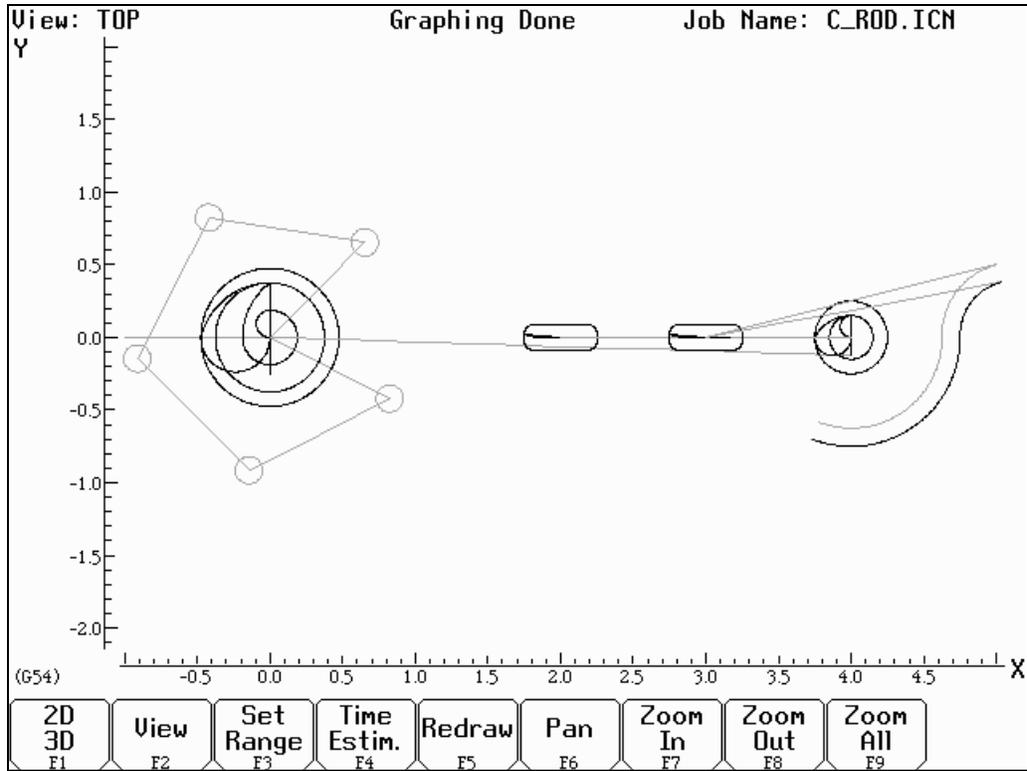


FIG. 5 - Draw screen showing Bolt Holes, Pockets and first arc of part

ESC/CANCEL Cancel Return to the editing screen.

- F10** Accept Keep selected values. The other arc values were calculated for you.
- F3** Arc (G2 & G3) The next arc to be cut is labeled as ARC 2 in Figure 3. The start point is labeled P2, the end point of the last arc.

N0130 Arc

- ↑↓(UP/DOWN)** Move Cursor Move down to the End X field. This selects End X as the destination of the Math Help solution.
- F6** Math Help Redisplay the Math Help values calculated for the last arc. The screen will look like figure 6, below.

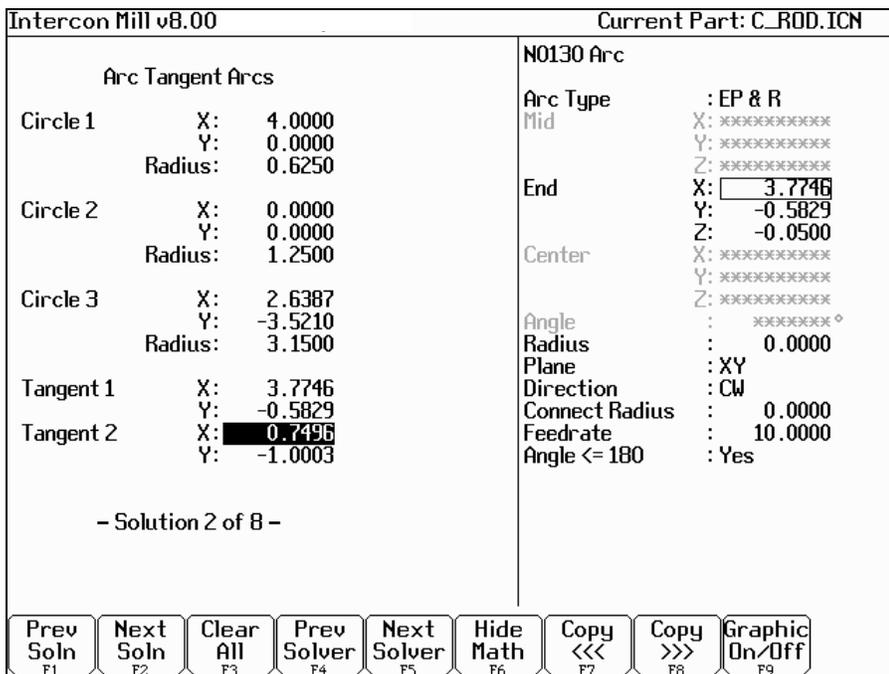


FIG. 6 – New arc 2 entry screen shown with solution for arcs 1 and 2 of Figure 3.

- ↑↓(UP/DOWN)** Move Cursor If necessary, move the block cursor to the Tangent 2 X field as shown above. The rectangle at End X shows that it will be the destination of the copy.
- F8** Copy >>> Transfer the tangent point T2 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically.
- F8** Copy >>> Transfer the tangent point T2 value for Y into the end point Y coordinate.
- F6** Hide Math Hide Math Help temporarily.

↑↓(UP/DOWN)

Move Cursor

Move down to the radius field and enter the radius of the arc labeled ARC 2 in Figure 3 (this radius is 3.1500 inches). Set the direction to “CCW”.

Arc type	:	EP&R
Mid	X :	
	Y :	
	Z :	
End:	X :	0.7496
	Y :	-1.0003
	Z :	-0.0500
Center:	X :	
	Y :	
	Z :	
Angle	:	
Radius	:	3.1500
Plane	:	XY
Direction	:	CCW
Feedrate	:	10.0000
Angle <= 180°	:	Yes

F10

Accept

Keep selected values.

F3

Arc (G2 & G3)

The third arc to be cut is labeled as ARC 3 in Figure 3. The start point is labeled P2, the end point of the previous arc. The end point of the arc will be generated with Math Help.

N0140 Arc

↑↓(UP/DOWN)

Move Cursor

Move down to the End X field. This selects End X as the destination of the Math Help solution.

F6

Math Help

Redisplay the Math Help values calculated for the last arcs.

F9

Graphic On/Off

Redisplays the diagram of the scenario selected to calculate arcs 1 and 2 on Figure 3.

F1

Next Soln.

Continue pressing F1 until you arrive at the scenario showing arcs 3 and 4 in Figure 3 (in this case, solution #1 is the appropriate one)

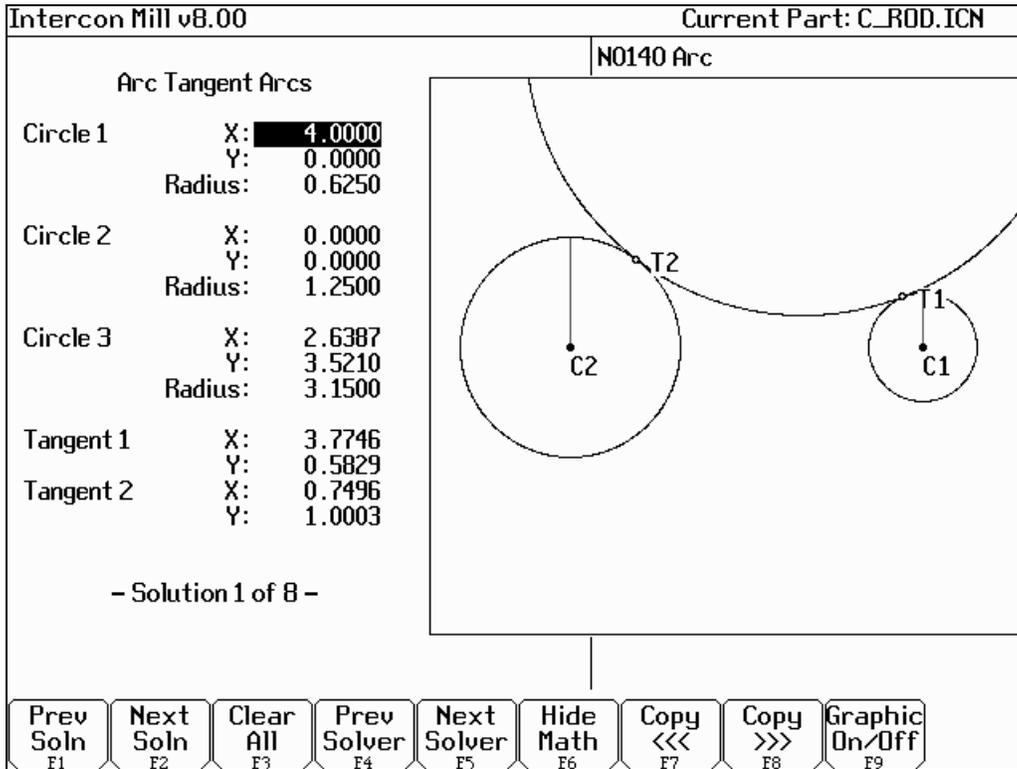


FIG. 7 - Math Help solution for arcs 3 and 4.

- | | | |
|------------|-------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ↑↓(ARROWS) | Move Cursor | Press to highlight the needed tangent point X coordinate in Math Help. Tangent point T2 is the one you want this time. |
| → (ARROW) | Move Cursor | Press to remove the graphic display and move the cursor to the arc operation. (This shortcut saves you from pressing <F9> to hide the graphics each time.) The solid block cursor on the left side of the screen will be replaced by an outlined rectangle and the solid block will appear in the arc operation on the right. |
| ↑↓(ARROWS) | | Move the block cursor to the End X field of the arc operation. |
| F8 | Copy >>> | Transfer the tangent point T2 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically. |
| F8 | Copy >>> | Transfer the tangent point T2 value for Y into the end point Y coordinate. |
| ↑↓(ARROWS) | Move Cursor | Move down to the radius field and enter the radius of the arc labeled ARC 3 in Figure 3. (This radius is 1.2500 inches). Also, enter in No for the angle of this arc, since it is greater than 180°. |

```

Arc type      : EP&R
Mid:          X :
              Y :
              Z :
End:          X : 0.7496
              Y : 1.0003
              Z : -0.0500
Center:       X :
              Y :
              Z :
Angle         :
Radius        : 1.2500
Plane         : XY
Direction     : CW
Feedrate      : 10.0000
Angle <= 180° : No

```

F10 Accept Keep selected values.

F3 Arc (G2 & G3) The fourth arc to be cut is labeled as ARC 4 in Figure 3. The start point, labeled P3, is the end point of the previous arc. The end point of the arc will be generated with Math Help.

N0150 Arc mill

↑↓(ARROWS) Move Cursor Move down to the End X field. This selects End X as the destination of the Math Help solution.

F6 Math Help Re-display the Math Help values calculated for the last arc.

Intercon Mill v8.00		Current Part: C_ROD.ICN						
<p>Arc Tangent Arcs</p> <p>Circle 1 X: 4.0000 Y: 0.0000 Radius: 0.6250</p> <p>Circle 2 X: 0.0000 Y: 0.0000 Radius: 1.2500</p> <p>Circle 3 X: 2.6387 Y: 3.5210 Radius: 3.1500</p> <p>Tangent 1 X: 3.7746 Y: 0.5829</p> <p>Tangent 2 X: 0.7496 Y: 1.0003</p> <p>- Solution 1 of 8 -</p>		<p>N0150 Arc</p> <p>Arc Type : EP & R</p> <p>Mid X: xxxxxxxxxxxx Y: xxxxxxxxxxxx Z: xxxxxxxxxxxx</p> <p>End X: 0.7496 Y: 1.0003 Z: -0.0500</p> <p>Center X: xxxxxxxxxxxx Y: xxxxxxxxxxxx Z: xxxxxxxxxxxx</p> <p>Angle : xxxxxxxx °</p> <p>Radius : 0.0000</p> <p>Plane : XY</p> <p>Direction : CW</p> <p>Connect Radius : 0.0000</p> <p>Feedrate : 10.0000</p> <p>Angle <= 180 : Yes</p>						
Prev Soln F1	Next Soln F2	Clear All F3	Prev Solver F4	Next Solver F5	Hide Math F6	Copy <<< F7	Copy >>> F8	Graphic On/Off F9

FIG. 8 - New arc 4 entry screen shown with solution for arcs 3 and 4 of Figure 3.

↑↓(UP/DOWN)	Move Cursor	Highlight the needed tangent point X. Tangent point T1 is the one you want this time.
ARROWS	Move Cursor	If necessary, move the cursor to the arc operation and select the End X field.
F8	Copy >>>	Transfer the tangent point T1 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically.
F8	Copy >>>	Transfer the tangent point T1 value for Y into the end point Y coordinate.
F6	Hide Math	Hide Math Help.
ARROWS	Move Cursor	Move down to the radius field and enter the radius of the arc labeled ARC 4 in Figure 3. (This radius is 3.1500 inches). Be sure to set the direction to CCW.

Arc type	:	EP&R
Mid:	X :	
	Y :	
	Z :	
End:	X :	3.7746
	Y :	0.5829
	Z :	-0.0500
Center:	X :	
	Y :	
	Z :	
Angle	:	
Radius	:	3.1500
Plane	:	XY
Direction	:	CCW
Feedrate	:	10.0000
Angle <= 180°	:	Yes

F10	Accept	Keep selected values.
F3	Arc (G2 & G3)	Mill the arc labeled as ARC 5 in Figure 3 back to point P1.

N0160 Arc mill

Operation type	:	EP&R
Mid:	X :	
	Y :	
	Z :	
End:	X :	4.6250
	Y :	0.0000
	Z :	-0.0500
Center:	X :	
	Y :	
	Z :	
Angle	:	
Radius	:	0.6250
Plane	:	XY
Direction	:	CW
Feedrate	:	10.0000
Angle <= 180°	:	Yes

F10 Accept Keep selected values.

F3 Arc (G2 & G3) Move tool away from the edge of the part after the last arc.

N0170 Arc mill

Arc type	:	EP&R
Mid:	X :	
	Y :	
	Z :	
End:	X :	5.0000
	Y :	-0.5000
	Z :	0.1000
Center:	X :	
	Y :	
	Z :	
Angle	:	
Radius	:	0.5000
Plane	:	XY
Direction	:	CCW
Feedrate	:	10.0000
Angle <=180°	:	Yes

F10 Accept Keep selected values.

F9 Subpgm Access the **Subprogram** screen.

F1 DpthRpt We programmed the outer contour of the part so that our tool would only penetrate a small portion of the material per pass. We now want to repeat the outer contour operations until the tool has cut the entire way through the material (the assumed material thickness is 0.5 inches). The outer contour begins with the Plunge in operation N0110 and ends with the Linear Mill in operation N0170.

N0180 Repeat to Depth

Start Block	:	0110	
End Block	:	0170	
Total Depth	:	0.5100	INC
Depth Increment	:	0.0500	INC
Clearance Height	:	0.2500	
Plunge Rate	:	5.0000	

F10 Accept Keep selected values.

F7 Comp Hit the space bar until cutter compensation is turned 'Off'. It is no longer needed.

N0190 Comp off

F10 Accept Keep selected values.

F1 Rapid (G0) Move the tool away from the part. This is called a lead-out move. When cutter compensation is turned off, the compensation is removed during the next move. This must be done to allow the CNC7 software to correct its position.

N0200 Rapid traverse

End:	X	:	0.0000
	Y	:	0.0000
	Z	:	3.0000
Angle	:	174.29°	
Length	:	5.0249	

F10 Accept Keep selected values.

ESC/CANCEL Cancel Creation of the part is complete. Intercon programs automatically turn the spindle and coolant off at the end.

F8 Graph Display a preview of the finished part. Just make sure that the finished part is going to look the way you want it to. The display shown in Figure 7 has rulers placed around the various view windows that are scaled to the same size as the part displayed to allow visual inspection of the part. Remember, this preview shows where the center of the current tool will move (cutter

compensation is not represented except in pocket and frame displays).

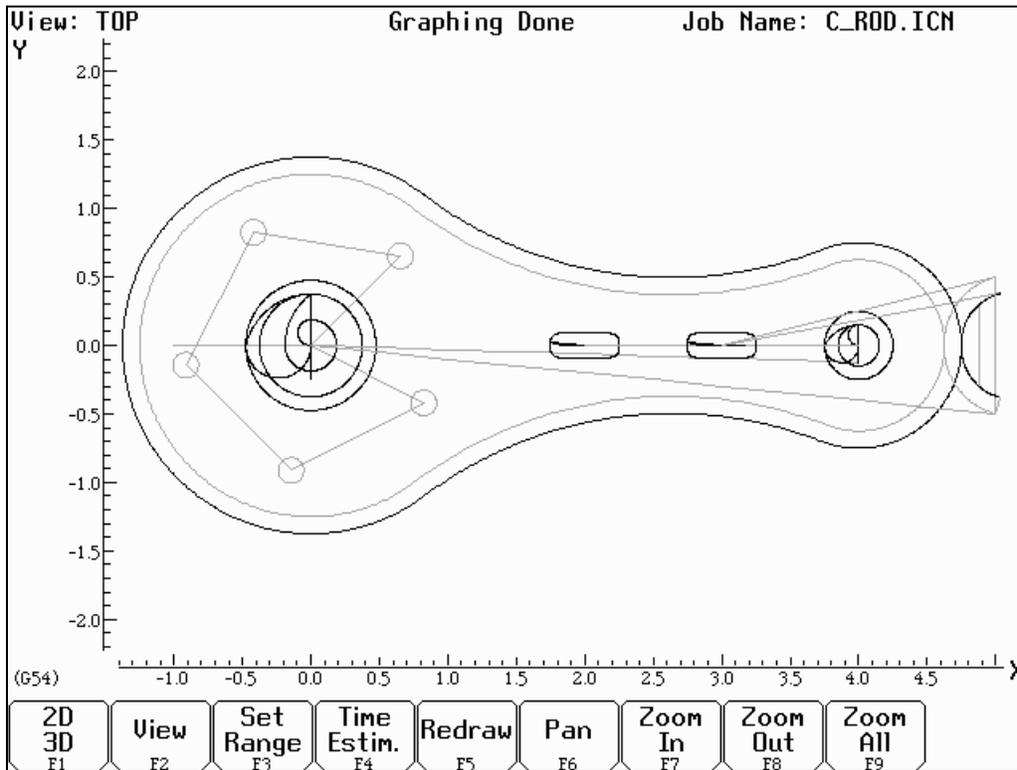


FIG. 9 - Draw screen showing complete part

- | | | |
|-------------------|--------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ESC/CANCEL | Cancel | Return to Main screen. |
| F1 | File | Go to the File Menu. Press <F3> to save under the current file name or press <F4> to save the program under a different name. |
| F10 | Post | The CNC file needed to run this part on your mill will be generated at this time. The Intercon program displays the operation number of the part it is processing as it works through each operation in memory: |

Generating CNC Program

Block 0050

As it processes each operation, it checks for values that, if used, will cause incorrect code to be produced. If such a value is found, a message will appear on the screen alerting you of the problem. For example, a problem with a rectangular pocket may produce this message:

Message

Corner radius too small for Cutter...hit a key.

Changes to the part would then be required to allow proper code generation to proceed. If no problems are encountered during code generation, the following message appears:

Message

CNC code generation successful

You are now at the main menu again.

You are now finished designing your part. In order to run your part, you now need to return to the CNC7 software.

Program Finished!

Milling The Part

Now that the part has been programmed, it is time to mill it. Take your material and clamp it to the table. Remember that the clamps must be positioned such that they do not interfere with the tool as it cuts. You may choose either to place the clamps around the edges of the material for the entire process and let the part drop through upon completion, or you may wish to pause after milling the circular pockets and place clamps through the holes to prevent the part from moving. The second option decreases the chance of the part being marred because it moved during milling.

Now you need to set your **XYZ** reference points. Insert your longest tool in the quill and follow the procedure listed below:

PRESS	ACTION	COMMENTS
JOG KEYS	Jog Axis	Jog the table so that your tool rests on the stock at the location that will represent X0 and Y0.
F1	Setup	Enter the CNC7 Setup screen. We are going to establish the part XYZ zero at the current tool location.
F1	Part	Access the Part Setup options.
F10	Set	Set your X zero position at current tool location.
F1	Next Axis	Select the Y-axis next.
F10	Set	Set your Y zero position at current tool location.
F1	Next Axis	Select the Z-axis next.
F10	Set	Set your Z zero position at current tool location.
TOOL CHECK	Move tool to Tool Check position	Moves the quill to the Z home position if the home position has been set. Moves tool to Z+limit switch and sets home position if not.
ESC/CANCEL	Cancel	Leave Part Setup screen.
F2	Tool	Access Tool Library Editor. This is the place where we want to measure the actual heights of our tools (since we could not set the actual values in Intercon).
F1	Offsets	You need to make sure that the tool diameter and height offset values are the correct ones for the tools you are going to be using. Inspect the values for D001, H001, D002 and H002. D001 should be 0.1875, H1 should be 0.0000 (the two inch tool), D002 should be 0.2500 and H002 should be -1.0000 (the one inch tool). If any of these values are incorrect, use the arrow keys to select the incorrect values. Enter the new values in their places and press <ENTER> to accept them.

NOTE: The tool heights used above are merely example heights. In order to accurately measure the heights of your tools, see the description of measuring tool heights on page 6 - 24 of this tutorial.

F10	Save	Keep the updated tool offset library values.
F2	Tools	Now you need to make sure that each tool uses the correct diameter and height offset values. Inspect the values for T001 and T002. T1 should use H001 and D001, while T002 should use H002 and D002. If any of these values are incorrect, use the arrow keys to select the incorrect values. Enter the new values in their places and press <ENTER> to accept them. You may also select spindle and coolant settings for your tools here, or enter a short description of the tool.
F10	Save	Keep the updated Tool Library values.
ESC/CANCEL	Cancel	Leave Tool Setup . Return to the CNC7 Setup Screen.
ESC/CANCEL	Cancel	Leave CNC7 Setup . Return to the CNC7 Main Screen.
CYCLE START	Run Program	The CYCLE START/START button is located on your jog panel. This key will cause the mill to begin cutting your part.

Tutorial Complete!

Measuring Tool Heights

The following is a brief description of the method used to measure tool height values (offsets). You will need to insert a reference tool into the quill before beginning.

PRESS	ACTION	COMMENTS
F1	Setup	From the main screen enter the Setup.
F2	Tool	Enter tool screen.
F1	Offsets	Enter the tool offsets screen.
JOG ARROWS	Move tool	You need to jog your reference tool down so it touches the top of some surface.
F1	Z Ref	Set your Z reference position. This is the value that appears on the DRO when the reference tool touches the top of the surface.
TOOL CHECK	Move tool to Tool Check position	Move the quill up to the Z home position. Insert the first tool to measure.
ARROWS	Select Height Offset	Select height offset which holds the height of the first tool.
JOG ARROWS	Move tool	Jog the tool down until it touches the same surface as did the reference tool.
F2	Manual	Record the height of the first tool. Now repeat the last four steps above (from TOOL CHECK to F2) for each additional tool to measure.
F10	Save Offsets	Store modifications to offset library of your tools.

Chapter 11

CNC Program Codes

General

The next three chapters contain a description of the CNC program codes and parameters supported by the M-Series Control. The M-Series Control has some G codes and parameters that are modal, and some that are "one shots." The G codes and parameters that are modal will stay in effect until a new G code or parameter is issued. One shots are effective for the current line only. For example, a movement command of G1, which is modal, will remain in effect until a different movement command is issued, such as G0, G2, G3, etc.

Miscellaneous CNC Program Symbols

D - Tool Diameter Offset Number

D is used to select the Tool Diameter Offset from the offset library. The D code values are stored in the Offset Library. Tool Diameter Offsets can be specified anytime before Cutter Comp is turned on (G41 or G42). Once specified, the offset amount is stored and will only be changed when another D code is entered. The Tool Diameter Offset (D) can be placed on a line by itself or on a line with other G-codes.

Example:

```
X0Y0F10      ;  
G41 D2 ; Enables cutter comp left with diameter D2.  
G1X0Y0 ;  
X1Y1.25      ; Cutter compensated moves  
X2Y1.4 ;  
G40      ; Cutter compensation off  
G42      ; Enables cutter comp. right (still using D2)
```

E - Select Work Coordinate System

E1 through E6 select among the six work coordinate systems. They are equivalent to G54 through G59.

F - Feedrate

The F command is used to set the cutting feedrate. The feedrate is expressed in units/minute. The programmed feedrate may be modified by the feedrate override knob (2% - 200%). The default feedrate is 3.0 units/minute. Units may be inches or millimeters.

Example:

```
G90 G1 X1.0 F50 ; linear mill to X1 at 50 units/minute
```

H - Tool Length Offset Number

H is used to select the Tool Length Offset Number. The H code offset amounts are stored in the file Offset Library. Tool Length Offsets can be specified anytime before a G43 or G44 is issued. Once specified the offset amount is stored and will only be changed when another H code is entered. The Tool Length Offset (H) can be placed on a line by itself or on a line with other G-codes. H00 is always a 0.0 length offset.

Example:

```
G43 H1 Z3 ; Selects offset corresponding to H1, moves to Z3
G1X0Y1 ;
H3 ; Selects offset corresponding to H3.
X1Y1.25 ;
G0H5 ; Selects offset corresponding to H5.
```

* NOTE: See Tool/Offset Library Edit (Chapter 4) for editing instruction for the offset library. For information on length compensation functions see G43 and G44 in Chapter 12.

N - Block Number

Block numbers are used to identify CNC program lines. Block numbers are optional, but can be used as the destinations of GOTO statements (see Advanced Macros in this chapter) and targets of the Search Function (See Main Screen Search option in Chapter 2). Block numbers also can make reading the NC files easier.

Example:

```
N1 G90 G17 M25
N2 G0 X0 Y0 Z0
```

O - Program Number

The O program number allows you to identify your program with a certain number.

If the program number is 9100-9999, the G codes from the O number through the next M99 will be extracted and placed in a separate subprogram/macro file. The lines will not be executed until the resulting file is called with M98 or G65.

Example:

```
O1521
N1 G90 G17 M25
N2 G0 X0 Y0 Z0
```

P - Parameter

P can correspond to Dwell Time, subprogram number, or a general parameter in canned cycles. This is used as a variable for any of those values in the NC file.

Examples:

```
G4 P1.32 ; Pause execution for 1.32 seconds
G10 P73 R.1 ; Set parameter #73 (G73 retract) to .1 inches
```

Q - Parameter

Q is used as a depth parameter in canned drilling cycles.

Example:

```
G73 X1.5 Y2.0 Z-.75 R.25 Q.25 F5 ; Q Sets the depth cut at .25
```

R - Radius, Return Point, Parameter

R can represent the radius, a return point, or a general parameter. This is used as a variable for any of those values in the NC file. R is similar to the P parameter.

Examples:

```
G10 D5 R.5 ; set D5=0.5  
G81 X0 Y0 Z-.5 R.1 F15 ; drill to Z-.5 with return height of .1
```

S - Spindle Speed Setting

Specifying a spindle speed causes the automatic spindle speed setting to be immediately updated. Setting the spindle speed does not cause the spindle to start. The maximum spindle speed is used to compute the output value to the spindle speed control circuit.

Example:

```
S1400 M3 ; Starts the spindle CW at 1400 RPM
```

* NOTE: The Spindle Speed is used in conjunction with the maximum spindle speed to determine the actual spindle speed output to the PLC. Also, this only works when a VFD (Variable Frequency Drive) spindle drive is connected.

T - Select Tool

Prompts the operator to insert the proper tool or change tools, when M6 is encountered.

Example:

```
T1 M6 ; Prompt operator to load tool number 1  
T2 ; no action  
G0X0Y0 ; move to X0 Y0  
M6 ; prompt operator to load tool number 2
```

: - Visible Comment Identifier

The colon (:) is used to indicate the start of a comment line within a CNC program. The colon must be the first character on the line.

Example:

```
: select absolute positioning
G90
: XY plane
G17
```

:Visible comments will be displayed on screen with the G-codes.

; - Internal Comment Identifier

The semicolon (;) is used to indicate the start of an internal comment within a CNC program line. All characters after the semicolon are ignored when the program is run. Internal comments are used to document NC programs or temporarily omit the remainder of a line.

Example:

```
G90 ; select absolute positioning
G17 ; XY plane
G1 X1 Y1 F10
G0 ; X0 Y0 ; G0 selected with no movement
```

[] – Numerical Expression

The left bracket '[' and right bracket ']' are used to delimit a numerical expression. Numerical expressions can contain floating point numbers or user and system variables in combination with mathematical operators and functions. The left parenthesis '(' or bracket '[' and right parenthesis ')' or bracket ']' can be used between the first left bracket and last right bracket to force operator precedence or associatively. A bracketed numerical expression can be used anywhere a number would be used. Comparison operators ('eq', 'ne', etc.) have built in rounding specified by parameter 144. Without this rounding, 'eq' would usually return "false" when comparing two numbers calculated in different ways. Comparison operators and logical operators ('!', '&&', '||') return 1.0 for "true" and 0.0 for "false".

The mathematical operators and functions are:

+	Addition (or unary positive)	eq, ==	Equals
-	Subtraction (or unary negative)	ne, !=	Not equals
*	Multiplication	ge, >=	Greater than or equals
/	Division	gt, >	Greater than
^	Exponentiation	le, <=	Less than or equals
mod, %	Modulo	lt, <	Less than
abs	Absolute value	not, !	Logical not
sin	Sine (degrees)	&&	Logical and
cos	Cosine (degrees)		Logical or
tan	Tangent (degrees)	and	Bit-wise and
sqrt	Square root	xor	Bit-wise exclusive or
#	Variable access	or	Bit-wise or
		~	Bit-wise complement

Examples:

```
G91 X[13/64] Z[1+3/8] ; move the X axis 13/64 (0.2031) units
; and the Z axis 1 3/8 (1.375) units incrementally
X[SQRT[ABS[SIN[#101]-COS[#102]]]] ; Move X as a function of #101 and #102
```

#, = - User or System Variable Assignment

The '#' character is used to reference a macro or a user or system variable. For variables that can be written, the '=' is used to assign to them.

Index	Description	Returns	R/W	
1-3	Macro arguments A-C	The floating point value if defined by a G65 call, 0.0 otherwise.	R/W	
4-6	Macro arguments I-K (1st set)		R/W	
7-9	Macro arguments D-F or 2nd set of I-K		R/W	
10	3rd I (G is invalid)		R/W	
11	Macro argument H or 3rd J		R/W	
12	3rd K (L is invalid)		R/W	
13	Macro argument M or 4th I		R/W	
14	4th J (N is invalid)		R/W	
15	4th K (O is invalid)		These can be used as private, local variables in any program or subprogram. (See examples.)	R/W
16	5th I (P is invalid)			R/W
17-18	Macro argument Q-R or 5th J-K			R/W
19-21	Macro arguments R-T or 6th set of I-K			R/W
22-24	Macro arguments U-W or 7th set of I-K			R/W
25-27	Macro arguments X-Z or 8th set of I-K			R/W
28-30	9th set of I-K		R/W	
31-33	10th set of I-K		R/W	
100 - 149	User variables	Floating point value. Initialized to 0.0 at start of job processing	R/W	
150 - 159	Nonvolatile user variables	Floating point value saved in CNC7.JOB file.	R/W	
2400, 2401-2418	Active WCS, WCS #1-18 CSR angles	Floating point value	R/W	
2500, 2501-2518	Active WCS, WCS #1-18 Axis 1 values (X)		R/W	
2600, 2601-2618	Active WCS, WCS #1-18 Axis 2 values (Y)		R/W	
2700, 2701-2718	Active WCS, WCS #1-18 Axis 3 values (Z)		R/W	
2800, 2801-2818	Active WCS, WCS #1-18 Axis 4 values (W)		R/W	
3901	Parts Cut (Part #)		R/W	
3902	Parts Required (Part Cnt)		R/W	
4001	Move mode		0.0 (rapid) or 1.0 (feed)	R
4002	Constant surface speed mode (lathe only)	96.0 (on) 97.0 (off)	R	
4003	Positioning mode	90.0 (abs) or 91.0 (inc)	R	
4005	Feedrate mode (lathe only)	98.0 (units per min) or 99.0 (units per rev)	R	
4006	Units of measure	20.0 (imp) or 21.0 (metric)	R	
4014	WCS	54.0-71.0 (WCS#1-18)	R	
4109	Feedrate (F)	Floating point value	R	
4119	Spindle Speed (S)		R	
4120	Tool Number (T)		R	
4121	Mill: Current height offset number (H) Lathe: Current offset ("oo" in "Tttoo")		R	
4122	Current diameter offset number (D, mill only)		R	
4201	Job processing state	0 = normal, 1 = graph	R	
4202	Search mode (0 = search mode off)	0 = search mode off	R	
5021-5024	Machine Position (X=5021, Y=5022, etc.)	Floating point value	R	
5041-5044	Current Position (X=5041, Y=5042, etc.)		R	
6001-6080	PLC Inputs 1 - 80	Least significant bit is lowest numbered PLC bit. 0 = closed, 1 = open	R	
6900-6909	PLC Inputs, eight at a time.		R	
7001-6080	PLC Outputs 1 - 80		R	
7900-7909	PLC Outputs, eight at a time.		R	
8001-6080	PLC Memory bits 1 - 80		R	
8900-8909	PLC Memory bits, eight at a time.		R	

9000-9199	Parameter values 0 – 199	See Chapter 15.	R
-----------	--------------------------	-----------------	---

Index	Description	Returns	R/W
10000	Mill: Height offset amount, active H	Floating point value	R/W
10001-10200	Mill: Height offset amount, H001 – H200	Floating point value	R/W
11000	Mill: Diameter offset amount, active D	Floating point value	R/W
11001-11200	Mill: Diameter offset amount, D001 – D200	Floating point value	R/W
12000	Mill: Tool H number, active tool (T)	0 - 200	R/W
12001-12200	Mill: Tool H number, tools 1 - 200	0 - 200	R/W
13000	Mill: Tool D number, active tool (T)	0 - 200	R/W
13001-13200	Mill: Tool D number, tools 1 - 200	0 - 200	R/W
14000	Mill: Tool coolant, active tool (T)	7, 8, 9	R/W
14001-14200	Mill: Tool coolant, tools 1 - 200	7, 8, 9	R/W
15000	Mill: Tool spindle direction, active tool (T)	3, 4, 5	R/W
15001-15200	Mill: Tool spindle direction, tools 1 - 200	3, 4, 5	R/W
16000	Mill: Tool spindle speed, active tool (T)	Floating point value	R/W
16001-16200	Mill: Tool spindle speed, tools 1 - 200	Floating point value	R/W
17000	Mill: Tool bin number, active tool (T)	Floating point value	R/W
17001-17200	Mill: Tool bin number, tools 1 - 200	Floating point value	R/W
18000	Mill: Tool putback, active tool (T)	Floating point value	R/W
18001-18200	Mill: Tool putback, tools 1 - 200	Floating point value	R/W

Examples:

```

#100 = #5041          ; set user variable #100 to the X axis current position
G90 X[#5041+1+7/32] ; move the X axis 1 7/32 units (1.2188) incrementally
#2501 = #5021        ; set WCS#1 X value to the current X position
#2703=[#2703+1/8]   ; add 1/8 units (.125) to the WCS#3 Z value

; Subroutine parameter and local variable access.
G1 X#A Y#B Z#C F#F   ; move to the coordinates passed as parameters
#[Q] = #F * .10     ; Assign local variable #Q to 10% of #F
#17 = #7 * .10      ; Same statement as previous using number references.
#[C] = 0.05         ; Reassign #C. (Value passed as parameter is lost.)

```

Advanced Macro Statements (Optional)

Warning: Branching and conditional execution are extremely powerful tools that, combined with access to system variables, allow you to do many things that would otherwise be impossible. Nevertheless, using branching and conditional execution can introduce undesirable and even unpredictable behavior into your programs. Undesirable effects can occur simply by graphing a program. The least of these undesirable effects could be entering an endless loop, failing to draw anything, or wiping out all the information in your tool library or WCS settings. It is your responsibility to make sure that undesirable things do not happen in your programs. You must monitor the job processing and search modes in your program, if necessary, and take appropriate action. Until you are confident of the actions of your program, you should step through it one block at a time to confirm your program logic.

GOTO - Branch Execution

To branch to another line within the same program or subprogram, use the statement

```
GOTO <expression>
```

where <expression> is any expression that evaluates to a valid block number in the program. GOTO causes an immediate branch to the specified destination. Program codes preceding a GOTO on the same line will be executed normally. Any program codes following GOTO on the same line will cause an error.

If fast branching is disabled (parameter 145 = 0) then CNC7 searches forward in the program for the first matching block number and resumes searching, if necessary from the top of the program. For this reason when fast branching is disabled, backward branches take longer than forward branches and backward branch times depend on the total program size. If the program is sufficiently large, use of the GOTO statement could introduce temporary pauses.

When fast branching is enabled (parameter 145 = 1) then CNC7 remembers the locations of block numbers as it finds them during program execution. Backward branches always take place immediately. The first forward branch to a block not yet encountered will take additional time as CNC7 searches forward for the block number; however, subsequent forward branches to that block number will take place immediately. The trade-off for using fast branching is that all line numbers at a given level of program or subprogram must be unique and programs will use more memory (approximately 16kilobytes of memory for every 1000 block numbers in the program.)

IF THEN ELSE - Conditional Execution

Program symbols, G codes, M codes and GOTO commands may be executed conditionally using the IF statement. The general form of the IF statement is:

```
IF <expression> THEN <execute if true> ELSE <execute if false>
```

where <expression> is any valid expression, <execute if true> is one or more program codes to execute if <expression> evaluates to “true” (non-zero) and <execute if false> is one or more program codes to execute if <expression> evaluates to “false” (zero). All parts of the IF statement must appear on the same line. The “ELSE <execute if false>” part of the statement is optional and may be omitted. The “THEN” may be omitted; however, <expression> must be enclosed in brackets ([]). The IF statement may follow other program codes on the same line. Compound conditionals are possible but they cannot be nested. The first THEN always pairs with the first IF. ELSE always pairs with the first <expression> that evaluates to “false”. All program codes executed are executed as part of the same block.

Examples:

```
; Branch to N200 if machine position is okay, otherwise go to N300
N100 IF #5041 LE 5.0 THEN GOTO 200 ELSE GOTO 300
; Force subprogram parameter #D to be within range.
IF [#D LE 0.005] #[D] = 0.005
; Compound conditionals
IF [#A LE 0.0] GOTO 100 ELSE IF [#A LE 2.5] GOTO 200 ELSE GOTO 300
IF [#A GT 0.0] IF [#D/#A GE 0.0] #[C] = SQRT[#D/#A]
```

INPUT – Prompt Operator for Input

The INPUT macro prompts the operator for numeric input. The general form of the INPUT statement is:

```
INPUT "<prompt>" <variable>
```

where <prompt> is the message prompt for the operator and <variable> is the variable in which to store the input. CNC7 will display a dialog with the given prompt and space for the operator response. The operator may enter any numeric expression (see above) including variables as a response. The operator must press CYCLE START or Alt-S to dismiss the dialog. Pressing Esc will cancel the job.

CNC7 parses well ahead of the current execution to maximize throughput and efficiency. For this reason, an INPUT macro may prompt the operator for input immediately even though the INPUT macro is located in the middle or near the end of the job. Parsing pauses while the dialog is displayed. Any statements parsed prior to the INPUT macro will have been queued and will continue to execute in the background while the prompt is displayed. Job processing will pause only if all queued statements have been executed before the operator supplies a response.

INPUT macros will not graph. If you must graph the job, first set the input variable to a default value and use a conditional to execute the INPUT only if the job is being run normally.

Use search mode cautiously with INPUT macros. To have search work properly, you may have to supply exactly the same input during the search as you did during the last actual run.

Examples:

```
; Ask operator for pocket depth. Store result in #101
; Note: this will not graph.
INPUT "Enter pocket depth" #101

; Allow job with INPUT statements to be graphed.
#101 = 0.5 ; Supply a default value for graphing
; Ask for operator input only if not graphing.
IF NOT #4201 THEN INPUT "Enter pocket depth" #101
```

CHAPTER 12

CNC Program Codes: G-codes

G-code	Group	Description
G00	* A	Rapid Positioning
G01	A	Linear Interpolation
G02	A	Circular or Helical Interpolation CW
G03	A	Circular or Helical Interpolation CCW
G04	B	Dwell
G09	B	Exact Stop
G10	B	Parameter Setting
G17	* C	Circular Interpolation Plane Selection XY
G18	C	Circular Interpolation Plane Selection ZX
G19	C	Circular Interpolation Plane Selection YZ
G20	* K	Select Inch Units
G21	L	Select Metric Units
G28	B	Return to Reference Point
G29	B	Return from Reference Point
G30	B	Return to Secondary Reference Point
G40	* D	Cutter Compensation Cancel
G41	D	Cutter Compensation Left
G42	D	Cutter Compensation Right
G43	E	Tool Length Compensation (+)
G44	E	Tool Length Compensation (-)
G49	* E	Tool Length Compensation Cancel
G50	* M	Scaling/Mirroring Off (Optional)
G51	M	Scaling/Mirroring On (Optional)
G52	B	Offset Local Coordinate System Origin (Optional)
G53	B	Rapid Position in Machine Coordinates (Optional)
G54	L	Select Work Coordinate System #1 (Optional)
G55	L	Select Work Coordinate System #2 (Optional)
G56	L	Select Work Coordinate System #3 (Optional)
G57	L	Select Work Coordinate System #4 (Optional)
G58	L	Select Work Coordinate System #5 (Optional)
G59	L	Select Work Coordinate System #6 (Optional)
G61	F	Exact Stop Mode
G64	* F	Cutting Mode
G65	J	Call Macro (Optional)
G68	N	Coordinate Rotation on
G69	* N	Coordinate Rotation off
G73	G	High Speed Peck Drilling
G74	G	Counter Tapping (Optional)
G80	* G	Canned Cycle Cancel
G81	G	Drilling and Spot Drilling
G82	G	Drill with Dwell

G83	G	Deep Hole Drilling
G84	G	Tapping (Optional)
G85	G	Boring
G89	G	Boring with Dwell
G90	* H	Absolute Positioning Mode
G91	H	Incremental positioning Mode
G92	B	Set Absolute position
G98	* I	Initial Point Return
G99	I	R Point Return
G117	* C	Rotation of Plane Selection XY
G118	C	Rotation of Plane Selection ZX
G119	C	Rotation of Plane Selection YZ

NOTES:

- *All the default G-codes have been marked with the symbol " * ".
- *A given line of a program may contain more than one G-code.
- *If several G-codes from one group are used in the same line, only the G-code specified last will remain active.
- *G-codes from group B are of "one shot" type (active only in the line in which they are specified). All other G-codes are modal (active until another G-code of the same group is specified).
- *If a G-code from group A is used in a canned cycle mode, the canned cycle will be canceled. Canned cycle G-codes, however, have no effect on G-codes from group A.

G00 - Rapid Positioning

G0 moves to the specified position at the maximum motor rate. The coordinates may be either absolute positions (G90) or incremental positions (G91). G0 is modal and remains in effect until another positioning mode (G1, G2, G3 etc.) is commanded. G0 is the default-positioning mode.



When the Z axis is commanded to move in the + direction, the Z axis will move up to its new position first, then the other axes will move to their new position along a straight line.

When the Z axis is commanded to move in the - direction, all axes but the Z axis will move to their new position along a straight line, then the Z axis will move down to its new position.

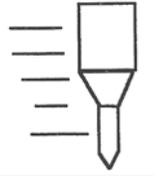
Example:

```
G0 X0.0 Y0.0 Z0.0
```

* NOTE: G0 moves are only affected by the feedrate override knob if rapid override (AUX1) is ON.

G01 - Linear Interpolation

G1 moves to the specified position at the programmed feedrate. The coordinates may be either absolute positions (G90) or incremental positions (G91). The movement will be along a straight line. G1 is modal and remains in effect until another positioning mode (G0, G2, G3 etc.) is commanded.



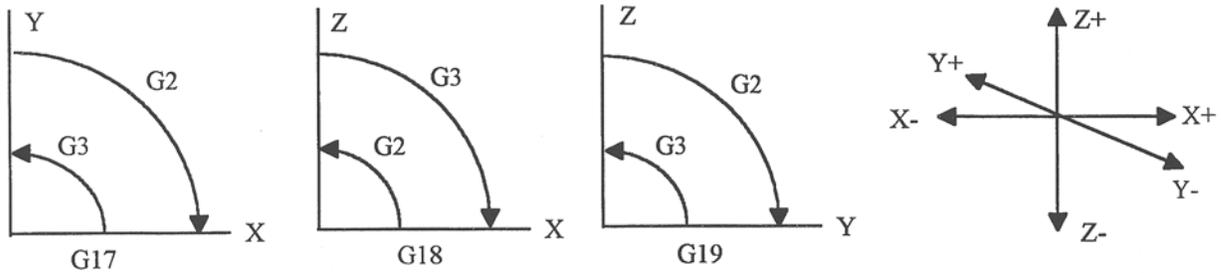
Example:

```
G01 X2 Y3 Z4 W5 F10
G91 X6 Y7
Z3 W4 F20
```

G02 & G03 - Circular Or Helical Interpolation



G2 moves in a clockwise circular motion, and G3 moves in a counterclockwise circular motion. This clockwise and counterclockwise motion is relative to your point of view, however. See the diagram below. The X, Y or Z position specified in the G2 or G3 command is the end position of the arc, and may be an absolute position (G90) or an incremental position (G91). G2 and G3 are modal and remain in effect until another positioning mode (G0, G1, etc.) is commanded.



View facing machine
looking Z-

View facing machine
looking Y+

View facing machine
looking X-

* **NOTE:** When using G18, the G2 command moves in a counterclockwise direction in the XZ plane.

The axes included in the currently selected circular plane (G17, G18, or G19) will move in a circular motion. Any other axes specified will move along a straight line (helical movement). The programmed feedrate is used for the interpolated motion along the movement of all axes.

Helical and circular motion can be programmed in two different ways: specifying the final point and the radius of the arc, or specifying the final point and the parameters I, J, K (center point of the arc as incremental values from the start position).

* **NOTE:** For closed circles (arc of 360 degrees), use method 2: specify final point and parameters I, J and K. Method 1 (specify final point and radius) will not work.

METHOD 1: USING FINAL POINT AND RADIUS

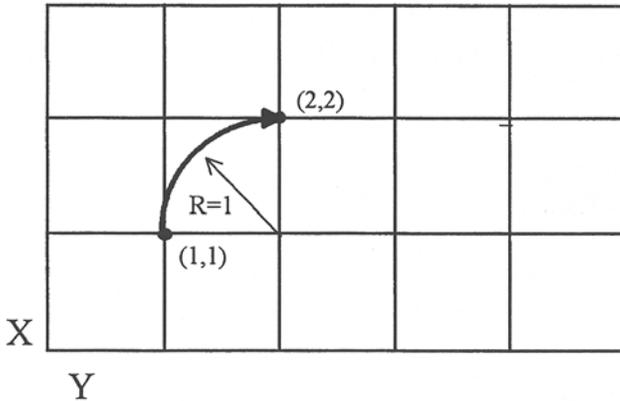
The commands G2 and G3 will have the following structure:

```
G2 Xa Yb Zc Rd
G3 Xa Yb Zc Rd
```

where a , b , and c will be the X, Y, and Z coordinates of the final point of the arc, and d will be the radius. In most cases there will be two possible arcs of the same radius connecting two given points. This occurs because the center of the arc is not specified. To choose the bigger arc, make the radius negative. To choose the smaller arc, make the radius positive. See examples 1 and 2 for graphical explanations of this concept.

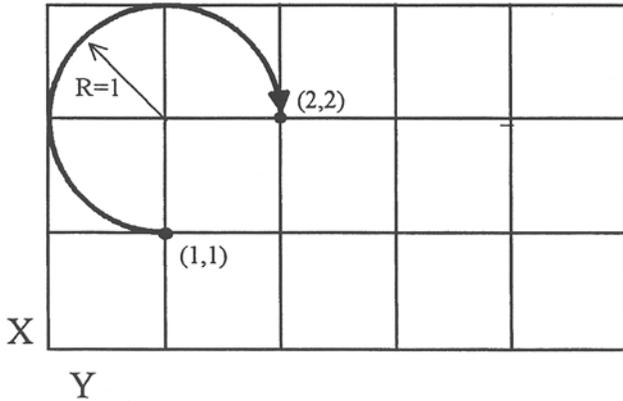
Example 1 (small arc solution: positive radius):

```
G17 G90 F25           ;select XY plane and absolute positioning
G00 X1.0 Y1.0 Z0      ;rapid to start position x1, y1, Z0
G02 X2 Y2 Z0 R1       ;arc to X2 Y2 Z0 with radius of 1
                       ; (small arc solution)
```



Example 2 (big arc solution: negative radius):

```
G17 G90 F25           ;select XY plane and absolute positioning
G00 X1.0 Y1.0 Z0      ;rapid to start position x1, y1, Z0
G02 X2 Y2 Z0 R -1     ;arc to X2 Y2 Z0 with radius of 1
                       ; (big arc solution)
```



METHOD 2: USING FINAL POINT AND PARAMETERS I, J, K

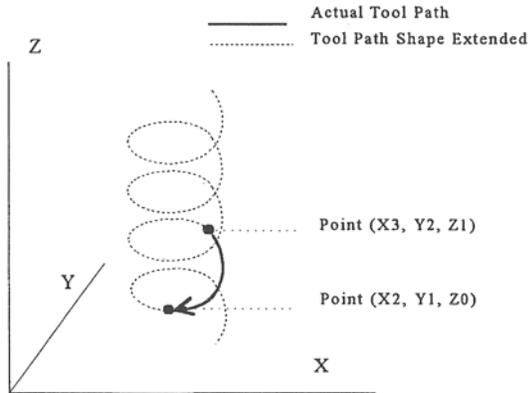
Another way to specify a helical or circular operation is using the parameters I, J, K instead of the radius R. The parameters I, J, and K are the **incremental** distances from the start point to the center of the arc.

- I = X center - X start (valid for G17 & G18)
- J = Y center - Y start (valid for G17 & G19)
- K = Z center - Z start (valid for G18 & G19)

Examples:

Circular motion (See graph in method 1, example 2)

```
G17 G90 F25 ;select XY plane and absolute positioning
G00 X1.0 Y1.0 Z0 ;rapid to start position x1, y1, Z0
G02 X2 Y2 Z0 J1 ;arc to X2 Y2 Z0 with radius of 1
```



Helical motion

```
G17 G90 F30 ; select XY plane and absolute positioning
G00 X3.0 Y2.0 Z1.0 ; rapid to start position X3, Y2, Z1
G02 X2.0 Y1.0 I-1.0 J0.0 Z0.0 ; CW XY arc from X3,Y2 to X2,Y1.
; Center at X2, Y2
; Helical Z move from 1 to 0
```

G04 - Dwell

G4 causes motion to stop for the specified time. The P parameter is used to specify in seconds to delay. G4 causes the block to decelerate to a full stop.



the time

The minimum delay is 0.01 seconds and the maximum is 327.67 seconds. The dwell time is performed after all motion and M functions on the line. If the P parameter is not specified, X will be used instead. If neither P nor X is specified, the default dwell time of 0.01 seconds will be used.

Example:

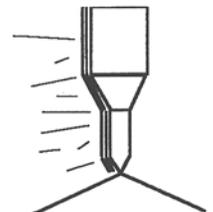
```
G0 X1 Y1 ; rapid to X1 Y1
G4 P2.51 ; pause for 2.51 seconds
X2 Y2
```

G09 - Exact stop

G9 causes motion to decelerate to a stop. G9 is equivalent to G4 P0.01. G9 is not modal; it is only effective for the block in which it appears. See G61 (exact stop mode).

Example:

```
G9 G0 X1 Y1 ; rapid to X1 Y1 and stop
X2 Y2 ; continue to X2 Y2
```



G10 - Parameter Setting

G10 allows you to set parameters for different program operations.

Examples:

```
G10 P73 R.05      ; Sets the peck drilling retract amount to .05
G10 P83 R.05      ; Sets the deep drill rapid down clearance to .05
G10 P81 R15       ; sets G81 to use M15 instead of Z movement
G10 H5 R-1.3      ; Sets tool length offset #5 to -1.3 in the
                  ; Offset Library
G10 D3 R.25       ; Sets tool diameter offset #3 to .25 in the
                  ; Offset Library
```

G17, G18, G19 - Circular Interpolation Plane Selection

G17, G18, and G19 select the plane for circular interpolation commands (G02 & G03). G17 is the default plane. See figure under G2 and G3.

G17 is the XY plane
G18 is the ZX plane
G19 is the YZ plane

G20 - Select Inch Units

G20 selects inch units, affecting the interpretation of all subsequent dimensions and feedrates in the job file. G20 does not change the native machine units as set on the control setup menu.

G21 - Select Metric Units

G21 selects metric units, affecting the interpretation of all subsequent dimensions and feedrates in the job file. G21 does not change the native machine units, as set on the control setup menu.

G28 - Return to Reference Point

G28 moves to the first reference point, by way of an intermediate point. The location of the reference point, in machine coordinates, may be set in Work Coordinate System Configuration. The intermediate point is specified in the local coordinate system, and may be at the current location (resulting in a move directly to the reference point). If an intermediate point is specified, only those axes for which positions are specified will be moved. If no axes are specified, all axes will be moved. The location of the intermediate point is stored for later use with G29.

Examples:

```
G28 G91 Z0                ; move Z axis directly to reference point
                          ; (X and Y don't move)

G28 G91 X-.5 Y0 Z0        ; move X -.5, then move all three axes to
                          ; reference point

G28 G90 X2 Y4 Z.1         ; move all axes to (2,4,0.1), then to
                          ; reference point

G28                       ; move all axes to the reference point
                          ; (no intermediate point)
```

* NOTE: As with G0 positioning moves, the Z-axis will move separately. If Z is moving up (the usual case) Z will move first, then the other axes. If Z is moving down, the other axes will move first, then Z. Because of this, it is rarely necessary to specify an intermediate point different from the current position.

G29 - Return from Reference Point

G29 moves all axes to the intermediate point stored in a preceding G28 or G30 command. It may be used to return to the workpiece. If a position is specified, the machine will move to that position (in local coordinates) after reaching the intermediate point. G29 may only be specified after G28 or G30, though there may be intervening moves.

Examples:

```
G29      ; move all axes back from reference point to  
      ; intermediate point
```

```
G29 X1 Y2  ; move all axes to intermediate point, then move to X1 Y2
```

* NOTE: As with G0 positioning moves, the Z-axis will move separately. If Z is moving up, Z will move first, then the other axes. If Z is moving down (the usual case for G29), the other axes will move first, then Z.

G30 - Return to Secondary Reference Point

G30 functions exactly like G28, except that by default it uses the second reference point from the Work Coordinate System Configuration table, and the P parameter may be used to request either reference point.

Examples:

```
G30 G91 Z0  ; move Z axis directly to second reference point
```

```
G30 P1 ; move all axes to first reference point
```

NOTE: G30 P1 is equivalent to G28.

G40, G41, G42 -Cutter Compensation

G41 and G42 in conjunction with the selected tool diameter (D code) apply cutter compensation to the programmed toolpath.

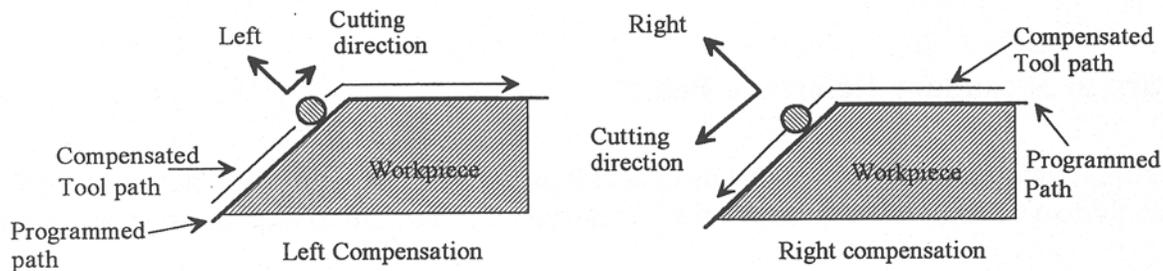
G41 offsets the cutter tool one half of the tool diameter selected with a D code, to the left of the workpiece, relative to the direction of travel.

G42 offsets the cutter tool one half of the tool diameter selected with a D code, to the right of the workpiece, relative to the direction of travel.

G40 cancels G41 and G42.

Example:

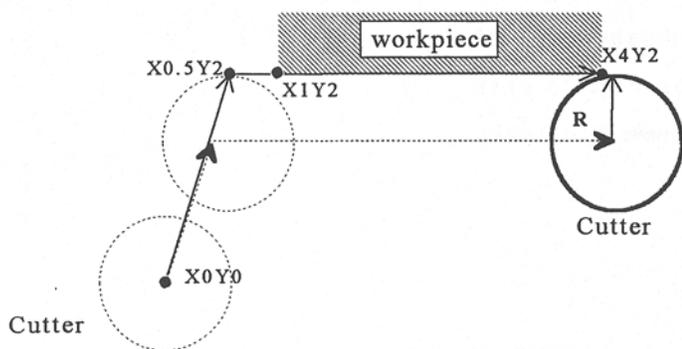
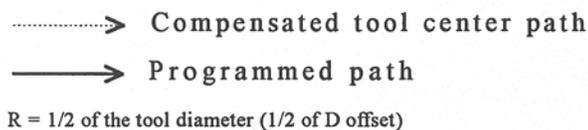
```
G41 D03      ; Tells the machine to compensate left half of the  
      ; Diameter of the amount that corresponds to D03 in the  
      ; Tool Library
```



Whenever cutter compensation is applied, the following factors must be taken into account in order to obtain proper results.

1. The cutter diameter compensation function (G41, G42) must be implemented before the cutter tool reaches the starting cutting point.

Example 1:



```
G0X0Y0 ;Rapid tool to X0, Y0
G42 D3 ;Turn cutter compensation on, with a diameter of D3
G0X.5Y2 ;Rapid to X0.5, Y2
G1X4.1Y2 ;Linear cut to X4.1, Y2.
;Cut to X4.1 to clear material.
G40 ;Turn cutter compensation off.
G0X5Y0 ;Rapid to X5, Y0.
```

You may want to add .1 or .05 inches on the final position for the last cut to clear the material.

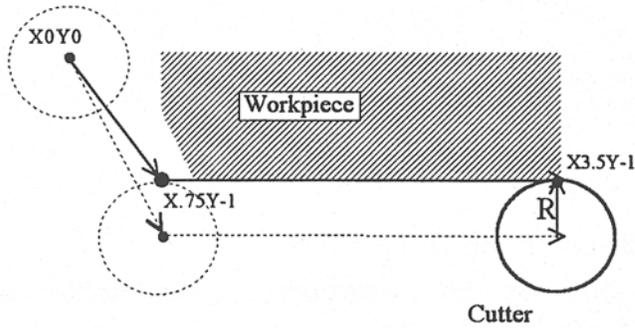
- NOTE: The diameter compensation statement G42 is placed before G0 X.5 Y2. As a result, the compensation is applied before the cutter reaches the starting cutting point X.5 Y2.

2. If the cutter is down, then the cutter compensation lead-in must always come from an appropriate direction. Otherwise, the workpiece will be incorrectly cut, and the cutter tool could be damaged. One way to avoid this problem is by always keeping the cutter above the workpiece whenever a transition is being made to a new starting cutting point. If for some reason this was not possible, then the G-code program should be written so that the cutter compensation lead-in paths do not interfere with the space occupied by the workpiece. Example 2 illustrates a possible harmful outcome of programming an inappropriate lead-in direction.

Example 2:

WRONG WAY

.....> Compensated tool center path
——> Programmed path
R = 1/2 of the tool diameter (1/2 of D offset)



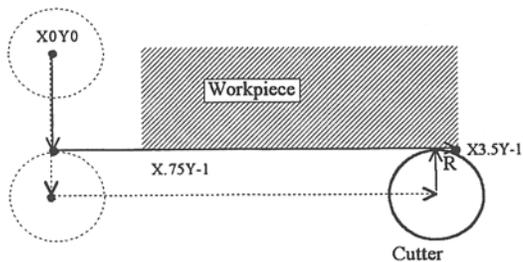
```
G0 X0Y0  
G42 D5  
G1 X.75Y-1 F5  
X3.6  
G40  
G0 X4Y-2
```

* NOTE: This problem could have been avoided by selecting a transitional point between X0 Y0 and X.75 Y-1. A transitional point such as X-1 Y-1 would properly modify the lead-in path, keeping the cutter from damaging the corner of the workpiece. Example 3 shows the correct way of performing this operation.

Example 3:

CORRECT WAY

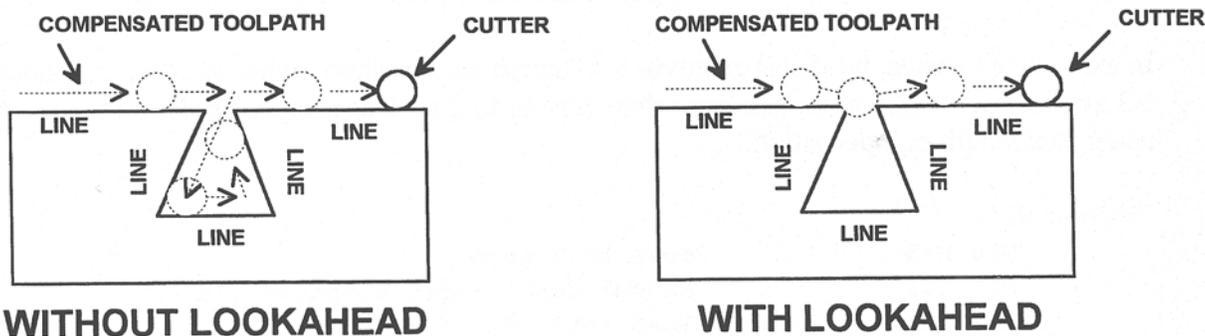
.....> Compensated tool center path
——> Programmed path
R = 1/2 of the tool diameter (1/2 of D offset)



```
G0X0Y0
```

```
G42D5
G0X0Y-1
G1X.75Y-1
X3.6
G40
```

3. *Lookahead*. When the control machines any rapid traverse (G0), line (G1), or arc (G2, G3) with tool diameter compensation enabled, the program will look up to 10 consecutive events ahead of the current event in order to anticipate toolpath clearance problems. Lookahead ensures that compensated tool paths don't overlap in programmed part sections where there is not enough clearance for the tool. The figure below shows a compensated tool path, and the actual toolpath after Lookahead corrects the clearance problem:



The number of Lookahead events the control scans is preset to 10. You can change the number of consecutive events from 1 to 10 by changing parameter 99 (refer to Chapter 16 for more information).

G43, G44, G49 - Tool Length Compensation

G43 and G44 apply tool length compensation to a selected tool to allow the control to utilize multiple tools in a single CNC program.

G43 applies positive compensation (from Z zero up). Work from part surface up.

G44 applies negative compensation (from Z zero down), used only when there is an absolute machine home. The spindle face is considered a zero length tool and all offsets are from there down.

G49 cancels tool length compensation (also canceled by issuing G43 H00).

Example:

```
G43 H01 ; tells the machine to offset the amount that
; corresponds to H01 in the Offset Library
```

G50, G51 - Scaling / Mirroring (Optional)

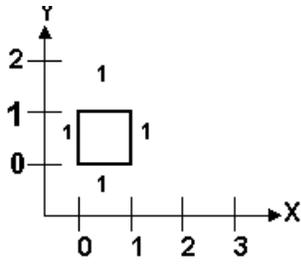
G50 and G51 scales program G-codes relative to a scaling center point defined as position (X, Y, Z).

A G51 applies scaling/mirror to all positions, lines, and arcs following this G-code until a G50 is entered. Specify scaling factors with a value I, J, K. The X, Y, and Z parameters are the coordinates of the scaling center. If the scaling center is not specified, the default scaling center is the current cutter position as shown on the DRO. To mirror, enter a negative value for the scaling factor.

Example, Scaling:

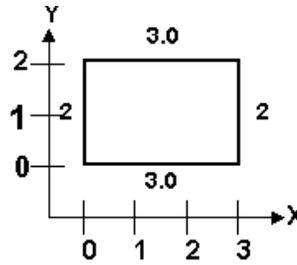
```
G51 X0.0 Y0.0 Z0.0 I3.0 J2 K1 ;turn scaling on
G00 X0.0 Y0.0 Z1.0 ;rapid to x0, y0, z1
G01 X1.0 Y0.0 Z1.0 ;line to X1, Y0, Z1
G01 X1.0 Y1.0 Z1.0 ;line to X1, Y1, Z1
G01 X0.0 Y1.0 Z1.0 ;line to X0, Y1, Z1
G01 X0.0 Y0.0 Z1.0 ;line to X0, Y0, Z1
G01 X0.0 Y0.0 Z0.0 ;line to X0, Y0, Z0
```

G50



Original Box

;cancel scale

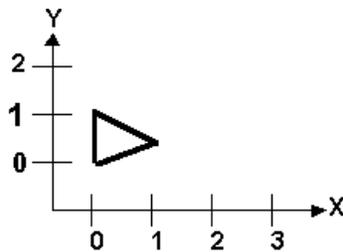


Scaled Box

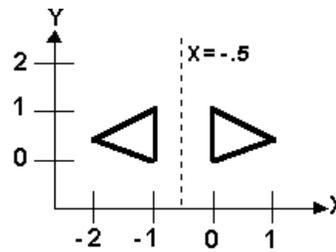
For this G51, the following program lines were scaled 3:1 in the X direction, 2:1 in the Y direction, and 1:1 in the Z direction. If no scale factor is specified, the default is 1:1 for all axes.

Example, Mirroring:

```
G51 X-0.5 Y0.0 Z.0      I-1 J1 K1      ;turn mirror on.
G00 X0.0  Y0.0  Z1.0    ;rapid traverse to X0, Y0, Z1
G01 X1.0  Y0.5  Z1.0    ;line to X1, Y.5, Z1
G01 X0.0  Y1.0  Z1.0    ;line to X0, Y1, Z1
G01 X0.0  Y0.0  Z1.0    ;line to X0, Y0, Z1
G50                                     ;cancel scale
```



Original Triangle



Mirrored Triangle

If scaling factors are the same for all the axes, parameter P can be used.

Example:

```
G51 X1.0 Y2.0 Z0.0      P2.5 ;scale all axes a factor of 2.5.
```

If an arc is scaled with uneven scaling factors, the result will depend on how the arc center and radius were specified:

1. If the arc radius was specified with R, the radius will be scaled by the larger of the two circular plane scale factors. The result will be a circular arc between the scaled arc start and the scaled arc end.
2. If the arc center was specified with I, J, and/or K, the centers will be scaled by the appropriate axis scale factors. The result will be a circular arc from the scaled arc start, around the scaled center, and usually with a line from the end of the circular arc to the scaled arc end.
3. In no case can an ellipse be generated using scaling.

G52 - Offset Local Coordinate System (Optional)

G52 shifts the local coordinate system origin by a specified distance. Multiple G52 codes are not cumulative; subsequent shifts replace earlier ones. The G52 shift may therefore be canceled by specifying a shift of zero. If you are using multiple coordinate systems, the G52 shift amount will affect all coordinate systems.

Example:

```
G0 X0 Y0      ; move to origin
M98 P9100     ; call subprogram
G52 Y4 ; shift coordinate system 4 inches in Y
G0 X0 Y0      ; move to new origin
M98 P9100     ; call subprogram again with new coordinates
G52 Y0 ; restore unshifted coordinate system
```

G53 - Rapid Positioning in Machine Coordinates (Optional)

G53 is a one shot code that performs a rapid traverse using machine coordinates. It does not affect the current movement mode (G0-G3) or coordinate system (G54-G59). G53 may only be used with absolute positioning (G90).

Example:

```
G53 X15 Y4 Z0      ; move to 15,4,0 in machine coordinates
```

G54 - G59 - Select Work Coordinate System (Optional)

G54 through G59 select among the six work coordinate systems. Subsequent absolute positions will be interpreted in the new coordinate system.

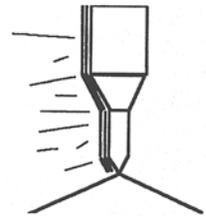
Example:

```
G54 G0 X0 Y0 Z0    ; select first WCS, move to origin
G2 X1 I.5 Z-.5     ; mill something...
G0 Z.1
G55 X1 Y1          ; select second WCS, move to 1,1
```

Using Extended Work Coordinate Systems: There are actually total of 18 workpiece origins. The extra workpiece origins are **not** accessible on the WCFG menu; they can only be set using Set Part 0/Position. In G-codes, the 12 additional workpiece origins may be selected with either “G54 P1” through “G54 P12” or “E7” through “E18.”

G61 - Exact stop mode

G61 invokes the exact stop mode. This forces deceleration to an exact stop at the end of each block (equivalent to G9 in each block). G61 is modal and remains in effect until canceled with G64 (cutting mode).

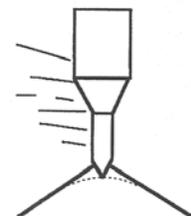


Example:

```
G0 X0 Y0
G61 X2      ; exact stop mode on: decel to stop at X2
X4          ; move to X4 and stop
X5          ; move to X5 and stop
```

G64 - Cutting mode (continuous, without exact stop)

G64 invokes cutting mode and cancels exact stop mode. No exact stops are performed at the end of each block. However, acceleration and deceleration is still performed. G64 is modal and remains in effect until exact stop mode (G61) is selected. Cutting mode is the default at the start of each program.



Example:

```
G0 X0 Y0
G64 X2      ;   exact stop mode off:  no stop at X2
X4          ;   continue to X4 without stop
X5          ;   continue to X5 without stop
```

G65 - Call Macro (Optional)

G65 calls a macro with user-specified values. A macro is a subprogram that executes a certain operation (e.g. drill pattern, contours, etc.) with values assigned to variable parameters within the operation.

Calling methods:

G65 Pxxxx Lrrrr *Arguments*

or

G65 "program.CNC" Lrrrr *Arguments*

Where *xxxx* is the macro number (in file O*xxxx*.CNC, 0000-9999 allowed, leading zeros required in filename), *rrrr* is the repeat value, "program.CNC" is the name of the macro file, and *Arguments* is a list of variable identifiers and values.

Arguments to macro calls are specified by using letters A-Z, excluding G, L, N, O, and P.

Macros are written just like normal programs. However, macro programs may access their arguments by using #A, #B, etc., or by using numbers: #1 for A, #2 for B, etc. (exceptions: #4-6 for I-K, #7-11 for D-H). Arguments I, J, and K can be used more than once in a macro call, with the first set of values stored as #4-6, the second as #7-9, etc., to a maximum of 10 sets.

* See example at the end of this G65 section.

A macro can use the negative of an argument by placing a minus sign before the '#'. No other arithmetic operations are supported. Macros can call other macros (up to 4 levels of depth), Macro M-functions, and subprograms. Macro M-functions and subprograms can similarly call macros.

Macros 9100 - 9999 may be embedded into a main program, using 091*xx* to designate the beginning of the macro and M99 to end it. CNC7 will read the macro and generate a file 091*xx*.CNC, but will not execute the macro. It will be executed when G65 is issued.

Example 1:

Main program:

```
G65 "TEST.CNC" A5 B3 X4
```

Macro TEST.CNC:

```
G1 X#X Y#A Z-#B
```

This call will produce

```
G1 X4 Y5 Z-3
```

Example 2:

Main program:

```
G65 "TEST2.CNC" I5 J3 K40 I-1 J2 I0 J0
```

Macro TEST2.CNC:

```
G1 X#4 Y#5 F#6
```

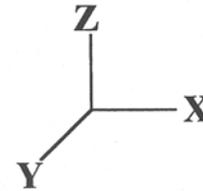
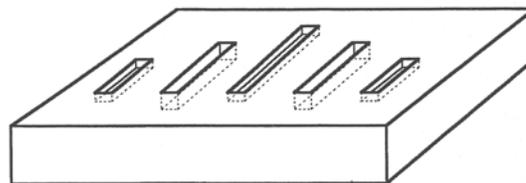
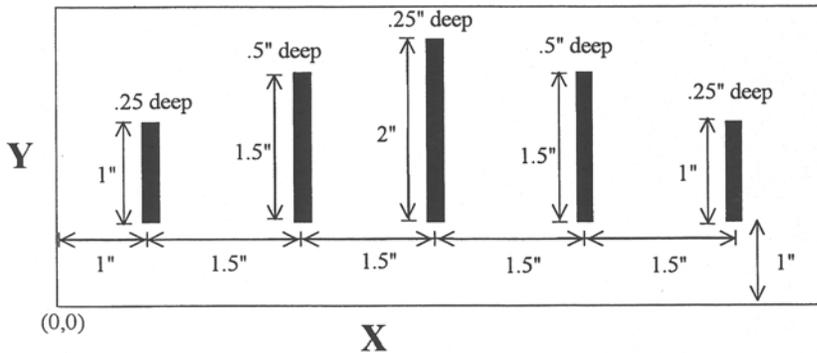
```
G1 X#7 Y#8 F#9
G1 X#10 Y#11 F#12
```

This call will produce

```
G1 X5 Y3 F40
G1 X-1 Y2
G1 X0 Y0
```

Example 3:

Suppose a piece is to have notches of different lengths and depths along the x-axis:



The macro variables would handle the length in the Y direction and depth in the Z direction:

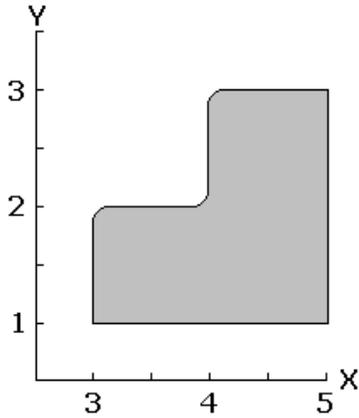
```
O0002
G90 G1 Z0 F30
Z#Z F5 ;Cut to variable depth
G91Y#Y F10 ;Cut variable length
G90 G0 Z0.1 ;Retract
```

The main program would call this macro five times, each time specifying the depth and length required.

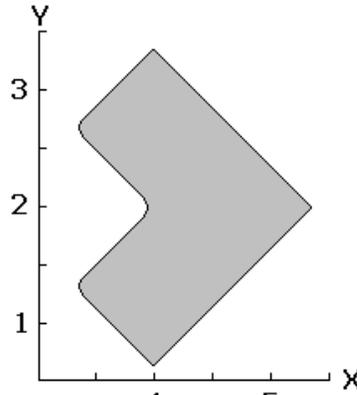
```
:Main Program
G90 G0 X1 Y1 Z0.1 ;Move to first notch
G65 P0002 L1 Y1 Z.25 ;Call macro and assign Y=1" and Z=.5"
G90 G0 X2.5 Y1
G65 P0002 L1 Y1.5 Z.5 ;Call macro and assign Y=1.5" and Z=.5"
G90 G0 X4 Y1
G65 P0002 L1 Y2 Z.25 ;Call macro again
G90 G0 X5.5 Y1
G65 P0002 L1 Y1.5 Z.5 ;Call macro again
G90 G0 X7 Y1
G65 P0002 L1 Y1 Z.25 ;Call macro again
:End program
```

G68, G69 - Coordinate Rotation on/off

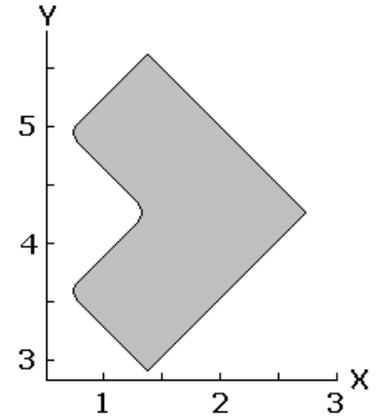
G68 rotates program G-codes a specified angle R. G68 rotates all positions, lines, and arcs until a G69 is entered. The center of rotation can be specified by X, Y and Z values (X, Y for G17 plane). If the center is not specified then a default center of rotation is used as determined by machine parameter #2 (see page 15-11 for parameter #2). The default plane of rotation is G17 (X, Y).



Original Unrotated Part



Part rotated 45° about X4, Y2



Part rotated 45° about X0, Y0

Example:

```
G0 X3.0 Y1.0           :Rapid to position
G68 R45 X4 Y2         :Rotate 45 degrees centered on X4 Y2
G1 X5.0 Y1.0 F20       :Start part profile
X5.0 Y3.0
X4.125 Y3.0
G3 X4.0 Y2.875 J-0.125
G1 X4.0 Y2.125
G2 X3.875 Y2.0 I-0.125
G1 X3.125 Y2.0
G3 X3.0 Y1.875 J-0.125
G1 X3.0 Y1.0           :End part profile
G69                     :Rotate Off
```

G73, G80, G81, G82, G83, G85, G89 - Canned Drilling Cycles

G74, G84 - Canned Tapping Cycles

G code	-Z direction (machine hole)	Operation at bottom of hole	+Z direction	Use
G73	Intermittent Feed (Set with the Q parameter)	-----	Rapid traverse	High speed peck drilling cycle
G74	Feed	Spindle CW, then Dwell (Set with the P parameter) parameter)	Feed	Counter tapping (Left-hand thread)
	-----	-----	-----	Cancels canned

G80				cycles
G81	Feed	-----	Rapid traverse	Regular and spot drilling cycles and air drill cycle
G82	Feed	Dwell (Set with the P parameter)	Rapid traverse	Regular and counter boring cycles, spot facing
G83	Intermittent Feed (Set with the Q parameter)	-----	Rapid traverse	Peck and deep hole drilling cycles
G84	Feed	Spindle CCW, then Dwell (Set with the P parameter)	Feed	Tapping (Right-hand thread)
G85	Feed	-----	Feed	Boring cycle
G89	Feed	Dwell (Set with the P parameter)	Feed	Boring cycle

Table 1. Canned drilling and tapping cycles

Canned Cycle Operation

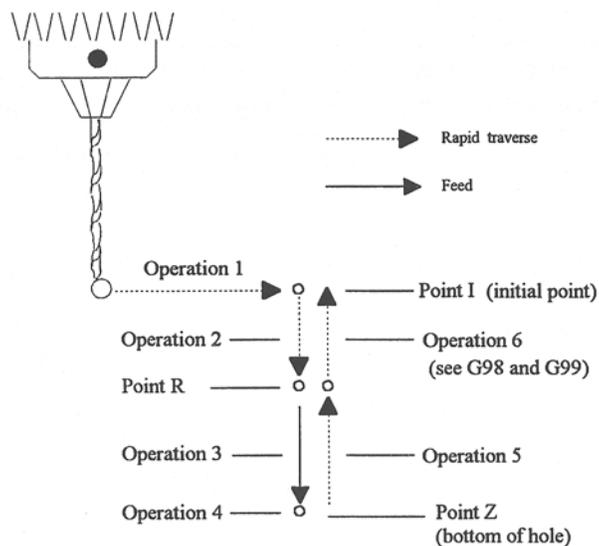
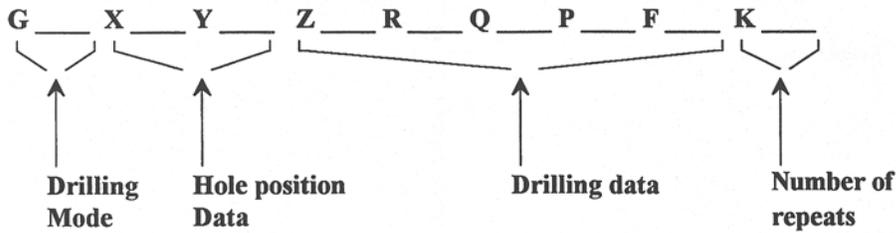


Figure 1. Drilling cycle operation

- Operation 1 Position the X, Y-axis.
- Operation 2 Rapid traverse to the position labeled R.
- Operation 3 Machine hole.
- Operation 4 Bottom hole operation.
- Operation 5 Return to point R.
- Operation 6 Rapid traverse to initial point.

Canned cycle G-code syntax



(Cycle codes do not have to be on the same line)

G ____ Canned cycle G-code from table 1.

X ____ X position of the hole to be drilled.

Y ____ Y position of the hole to be drilled.

Z ____ Specifies point Z in figure 1.
In incremental mode Z is measured from point R.
In absolute mode Z is the position of the hole bottom.

R ____ Specifies the distance to point R (figure 1) with an absolute or incremental value.

Q ____ Determines the cut-in depth for the G73 and G83 cycles.
Determines the thread lead for G74 and G84 if Rigid Tapping is enabled. (In the case of Rigid Tapping Q is not modal)

P ____ Sets the dwell time at the bottom of the holes for G74, G82, G84, and G89 cycles. The dwell time is measured in seconds (same as G04).

F ____ Sets the feed rate. Remains the feedrate even after G80 cancel canned cycles.

K ____ Sets the number of repeats for drilling cycles. Operations 1 through 6 of figure 1 will be repeated K number of times. If K is not specified K = 1. K is only useful when using incremental positioning mode (G91) and is not retained from cycle to cycle. In absolute mode, K causes the drilling of the same hole in the same position K times.

* NOTE: Canned cycles are modal and should be canceled with G80. However G00, G01, G02 and G03 will also cause the cancellation of canned cycles. All parameters are stored until canned cycles are canceled except for the hole position and K, which must be set each time the cycle is used. When G80 is issued the movement mode will be the last one issued (G0, G1, G2, G3). Canned cycles will not be performed unless X and/or Y is specified.

When performing canned cycle operations, the distances can be either incremental or absolute, depending on the current active mode (G90 = absolute, G91= incremental). Figure 2 illustrates canned cycle Z-axis distances in both modes.

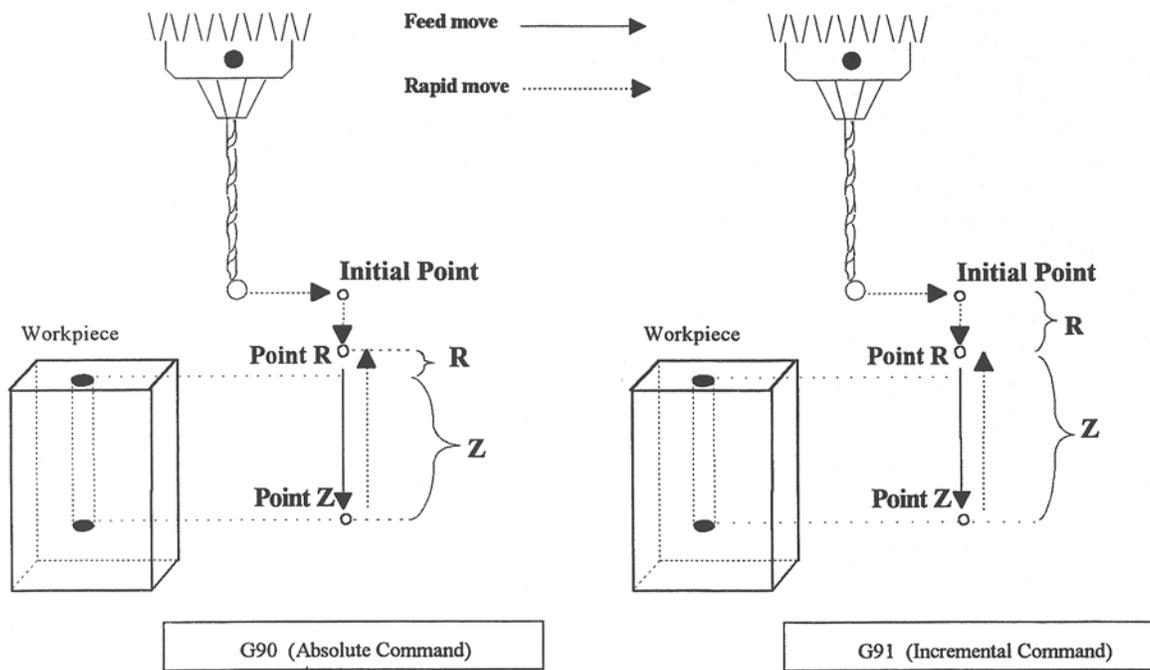


Figure 2: Canned Cycle Absolute and Incremental modes

* NOTE: In incremental mode the Z depth of the hole is measured from R, and R is measured from the initial tool position.

Example:

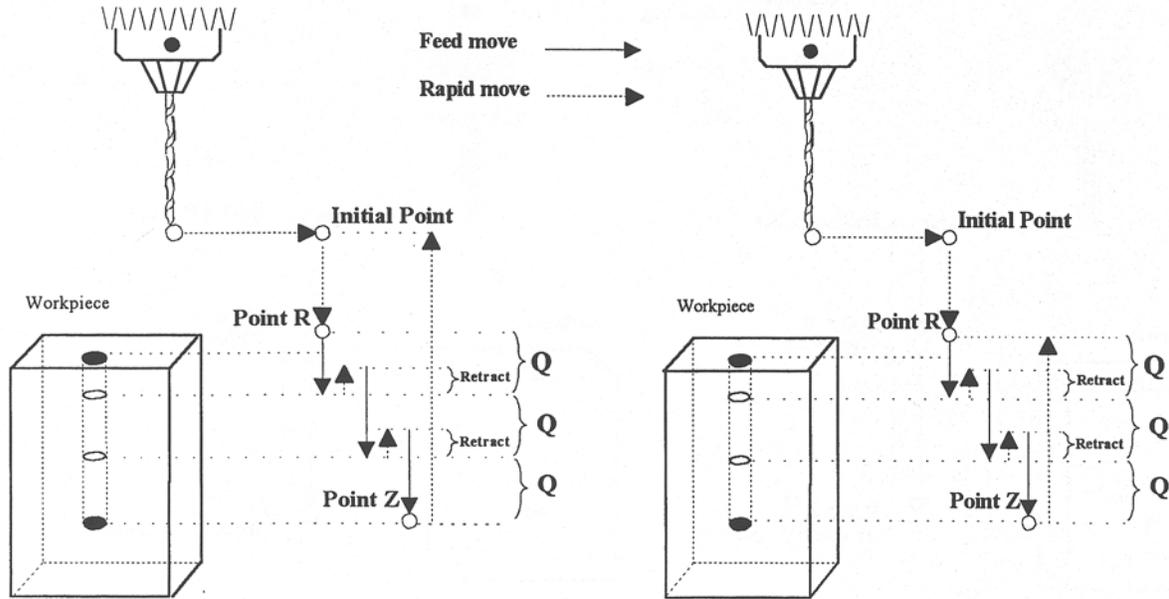
(Part surface height is Z = 0, initial tool position is X.50 Y1.0 Z.625. Drill 0.50 deep hole at X1.0 Y1.0; clearance height (R) is 0.10 above surface.)

```

; Absolute                               ; Incremental
G90                                       G91
G81 X1 Y1 R.1 Z-.5                       G81 X.5 Y0 R-.525 Z-.6
G80                                       G80

```

G73 - High Speed Peck Drilling



G73 using G98

G73 using G99

G73 is the peck drilling cycle. The hole is drilled in a series of moves: down at feedrate a distance Q, up at the rapid rate the retract distance, and then down again at feedrate. The retract amount is set with G10 as shown in the example below.

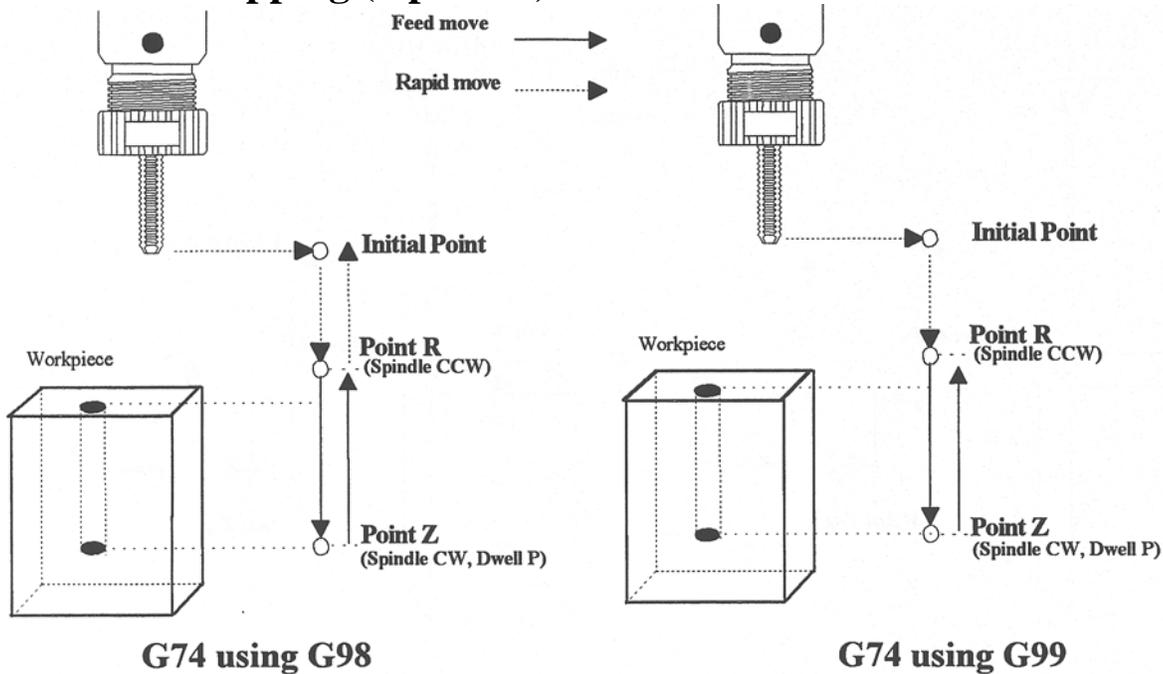
Example:

```

G90                                       ; Absolute positioning
G01 X3.00 Y1.50 Z.5                       ; G01 mode before canned cycle
G98                                       ; Set for initial point return
G10 P73 R.1                               ; Sets the retract amount to .1
G73 X3.250 Y1.75 Z-.650 R.1 Q0.325 F3    ; Peck drill at X3.25 Y1.75
X4.5 Y3.5                                 ; Peck drill at X4.5 Y3.5
G80                                       ; Cancel canned cycle, return
; to G01

```

G74 - Counter Tapping (Optional)



G74 performs left-hand tapping using a floating tap head. The spindle speed and feedrate should be set and the spindle started in the CCW direction before issuing G74. By default, G74 uses M3 to select spindle CW (at the bottom of the hole) and M4 to re-select spindle CCW (after backing out of the hole). Alternate M functions may be specified by setting parameters G74 (for CCW) and G84 (for CW).

The tap will continue to cut a short distance beyond the programmed Z height as the spindle comes to a stop before reversing. When tapping blind holes, be sure to specify a Z height slightly above the bottom of the hole to prevent the tool from reaching bottom before the spindle stops. The exact distance you must allow will depend on your machine and the diameter and pitch of the tapping tool.

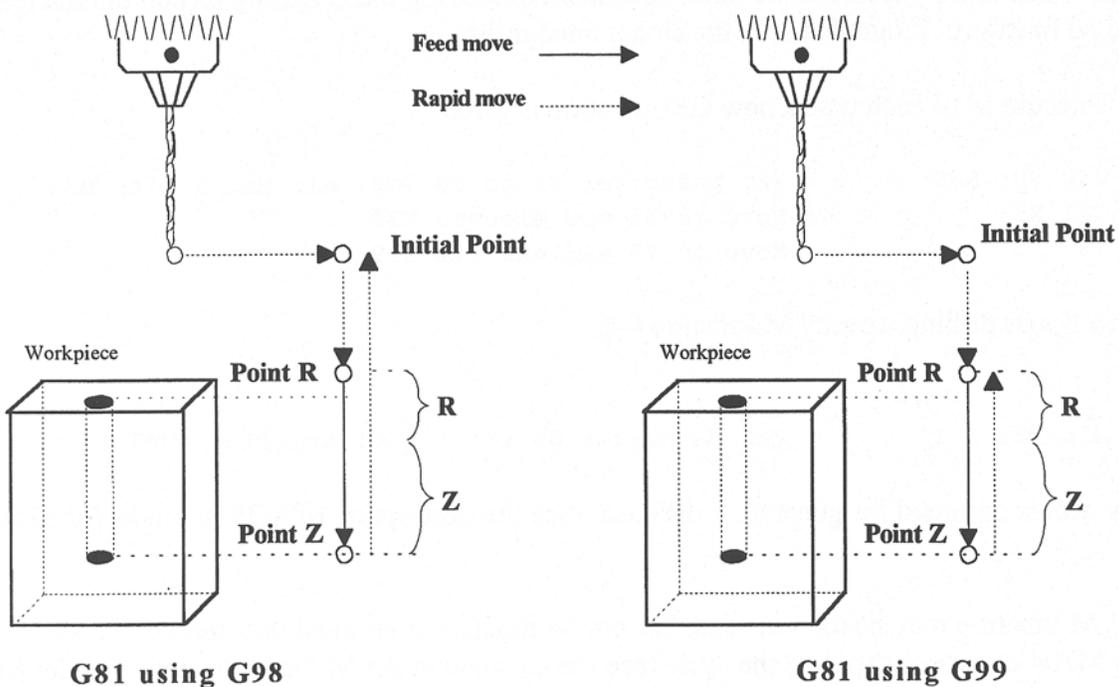
Note: If rigid tapping is enabled, a Q may be used to set the thread lead or pitch. However, because Q is not modal in the case of Rigid Tapping, you must specify Q on every line at which Rigid Tapping is to occur.

*** WARNING: Do not press FEED HOLD or CYCLE CANCEL while the tap is in the hole.**

Example:

```
M4 S500 F27.78           ; start spindle CCW, set up for 18 pitch tap
G74 X1 Y1 R.1 Z-.5      ; counter-tap a 0.5 deep hole at X1 Y1
Y1.5                    ; ... and another one at X1 Y1.5
G80                      ; cancel canned cycles
```

G81 - Drilling and Spot Drilling



G81 is a general purpose drilling cycle. The hole is drilled in a single feedrate move, and then the tool is retracted at the rapid rate.

Example:

```
G90 ; Absolute positioning
G01 X3.00 Y1.50 Z.5 ; G01 mode before canned cycle
G99 ; Set for R point return
G81 X3.250 Y1.75 Z-.650 R.1 F3 ; Drill at X3.25 Y1.75
X4.5 Y3.5 ; Drill at X4.5 Y3.5
G80 ; Cancel canned cycle, return to G1
```

G81 - Drill Cycle Transformation to G81 Air Drill Cycle

G81 may be modified to execute an M function instead of moving the Z-axis by setting parameter #81 to the desired M function. Example use is for air-actuated drills.

Example: Execute M39 each time a new G81 position is given:

```
G10 P81 R39 ; Set parameter 81 to 39 (G81 air drill with M39)
G81 X5 ; Move to X5 and execute M39
Y3 ; Move to Y3 and execute M39
```

To revert to Z-axis drilling, specify M function #-1.

Example:

```
G10 P81 R-1 ; Set parameter 81 to -1 (G81 drilling cycle)
```

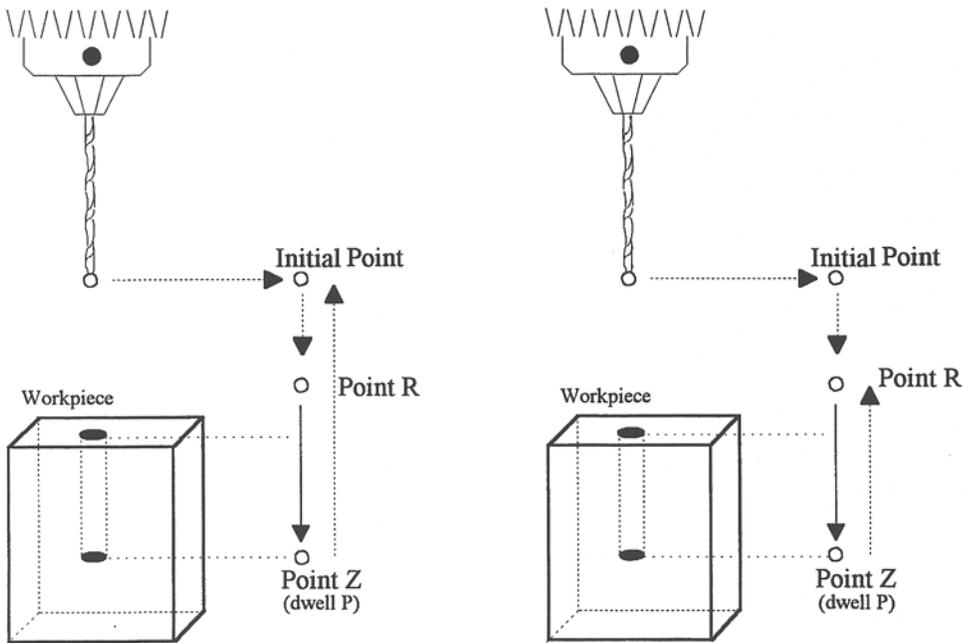
M function #39 is designed for general air drill use. See the description of M39 in the M functions section.

A different M function may be used instead, but any M function used must be a macro file that uses the M103 and M104 commands to time the cycle (see the example in the M function section under M103). If the macro file does not use M103, the control will automatically cancel the job 1/2 second after starting G81. For information on creating customized M functions, review Macro M functions in Chapter 13.

The M39 default air drill cycle has a time out of 2.0 seconds. As a result, if the cycle does not complete within 2 seconds then the cycle aborts and the output relay is turned off under PLC program control.

* NOTE: The PLC program must be involved in the execution of the cycle. The PLC program is responsible for turning on relays based on M function requests and the status of program execution. The PLC program must also stop all programmed machine functions when the program is canceled. See the M39 description (Chapter 13) for a sample of an air drill cycle M function.

G82 - Drill with dwell



G82 Using G98

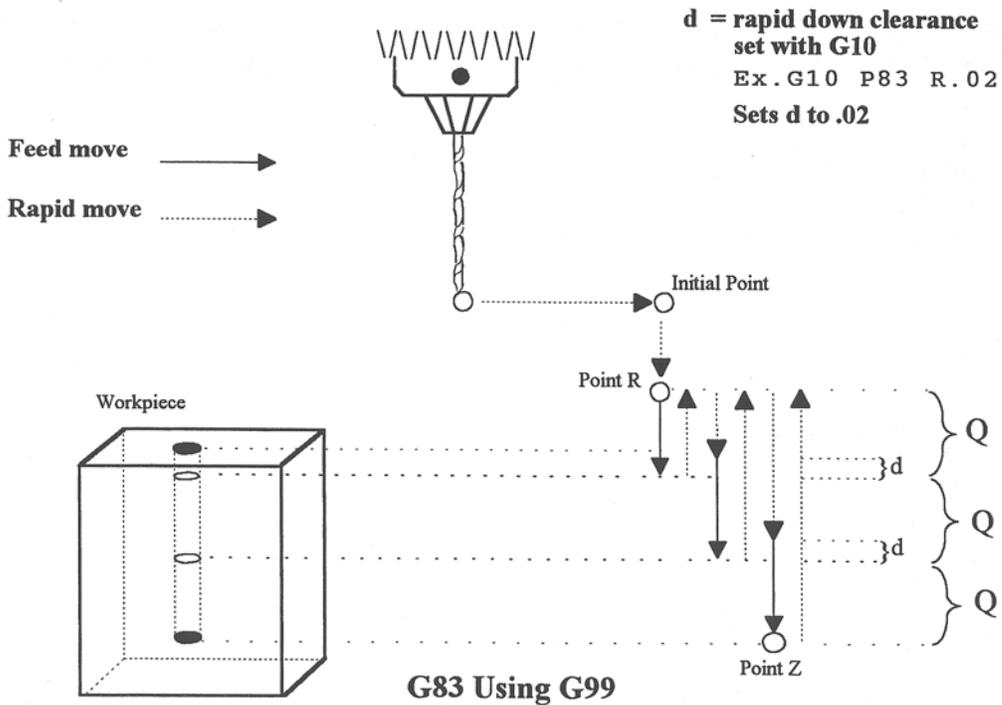
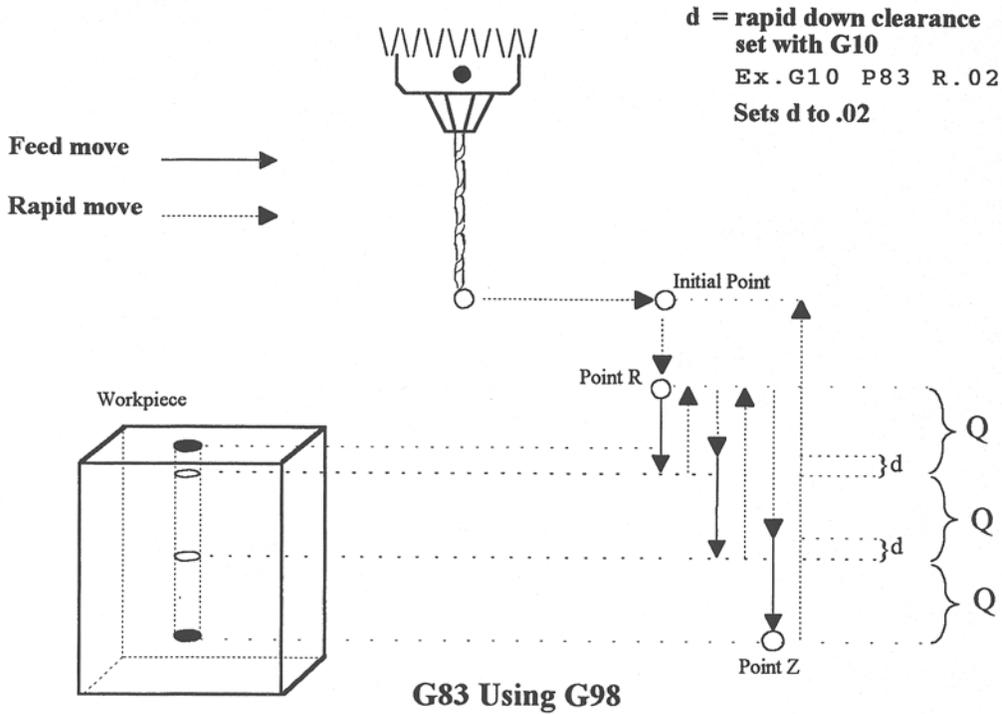
G82 Using G99

G82 is a general purpose drilling cycle similar to G81. However, G82 includes an optional dwell at the bottom of the hole before retracting the tool. This can make the depth of blind holes more accurate.

Example:

```
G82 X1 Y1 R.1 Z-.5 P.5 ; drill to Z-.5, dwell .5 seconds
```

G83 - Deep hole drilling

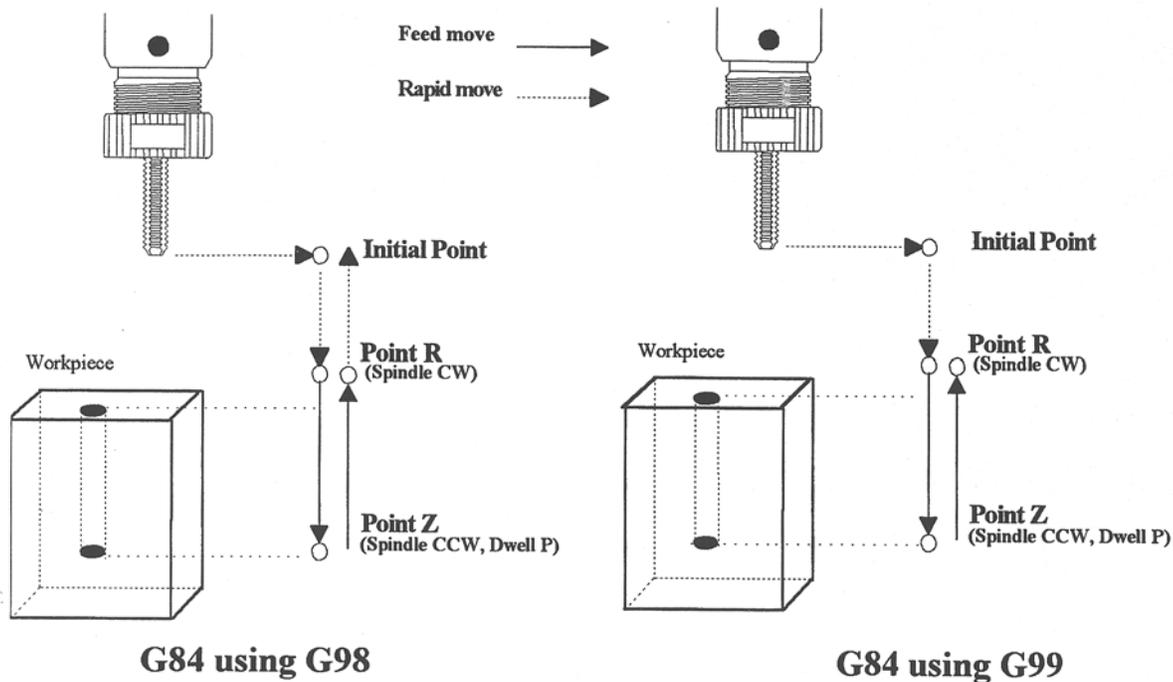


G83 is a deep hole drilling cycle. It periodically retracts the tool to the surface to clear accumulated chips, then returns to resume drilling where it left off. The retract and return are performed at the rapid rate. Because there may be chips in the bottom of the hole, the tool does not return all the way to the bottom at the rapid rate. Instead it slows to feedrate a short distance above the bottom. This clearance distance is selected by setting Parameter 83 with G10 (see example below).

Example:

```
G10 P83 R.05 ; set clearance to .05"
G83 X0 Y0 R.1 Z-2 Q.5 ; drill 2" deep hole in 0.5" steps
```

G84 - Tapping (Optional)



G84 performs right-hand tapping using a floating tap head. The spindle speed and feedrate should be set and the spindle started in the CW direction before issuing G84. By default G84 uses M4 to select spindle CCW (at the bottom of the hole) and M3 to re-select spindle CW (after backing out of the hole). Alternate M functions may be specified by setting parameters 74 (for CCW) and 84 (for CW). See G10 for examples.

The tap will continue to cut a short distance beyond the programmed Z height as the spindle comes to a stop before reversing. When tapping blind holes, be sure to specify a Z height slightly above the bottom of the hole to prevent the tool from reaching bottom before the spindle stops. The exact distance you must allow will depend on your machine and the diameter and pitch of the tapping tool.

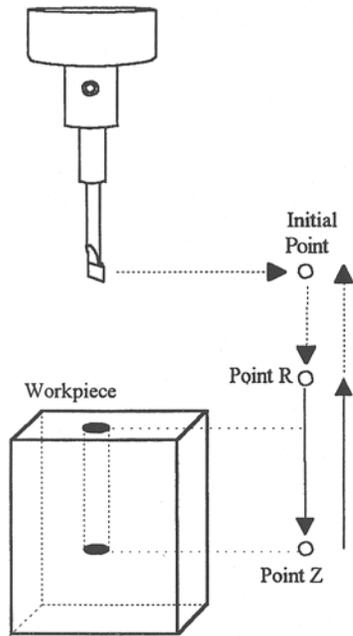
Note: If rigid tapping is enabled, a Q may be used to set the thread lead or pitch. However, because Q is not modal in the case of Rigid Tapping, you must specify Q on every line at which Rigid Tapping is to occur.

* WARNING: Do not press FEED HOLD or CYCLE CANCEL while the tap is in the hole.

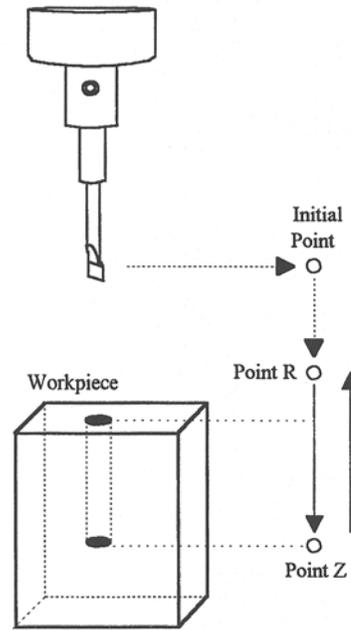
Example:

```
M3 S500 F27.78           ; start spindle CW, set up for 18 pitch tap
G84 X1 Y1 R.1 Z-.5      ; tap a 0.5 deep hole at X1 Y1
Y1.5                    ; ... and another one at X1 Y1.5
G80                      ; cancel canned cycle
```

G85 – Boring



G85 Using G98



G85 Using G99

G85 is similar to G81, except that the tool is retracted with a feedrate move instead of a rapid move. G85 may be used for tapping with reversing tap heads such as the Tapmatic NCR series.

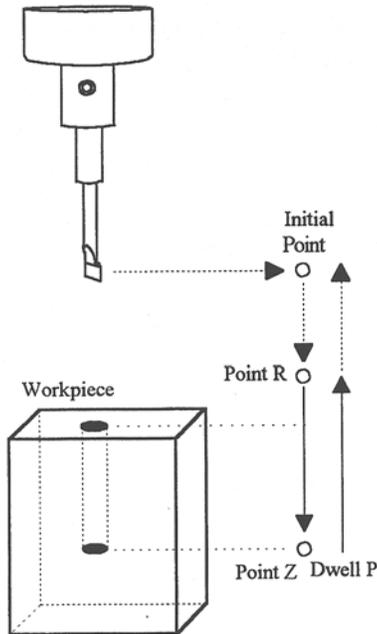
Example 1:

```
G85 X1 Y1 R.1 Z-.5 ; bore a 0.5" hole at X1 Y1  
G80 ; cancel canned cycle
```

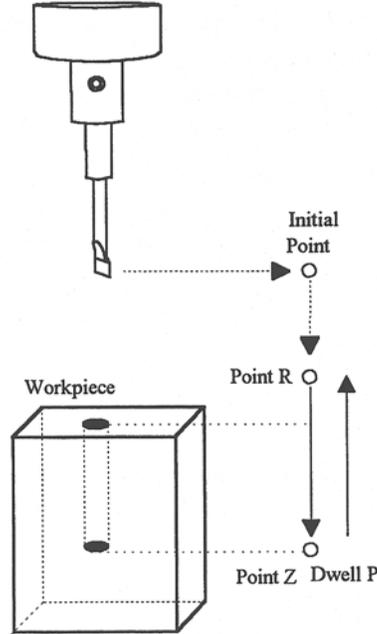
Example 2:

```
M3 S500 F27.78 ; start spindle CW, set for 18 pitch tap  
M109/1/2 ; disable feedrate and spindle overrides  
G85 X1 Y1 R.1 Z-.4 ; tap hole at X1 Y1 to a depth of 0.4"  
M108/1/2 ; reenable feedrate and spindle overrides  
G80 ; cancel canned cycle
```

G89 - Boring cycle with dwell



G89 Using G98



G89 Using G99

G89 is similar to G85, except that it includes an optional dwell at the bottom of the hole before retracting the tool.

Example:

```
G89 X1 Y1 R.1 Z-.5 P.1 ; bore 0.5" hole at X1 Y1, dwell .1 seconds
G80 ; cancel canned cycle
```

G90 & G91 - Absolute/Incremental Positioning Mode

G90 selects absolute positioning, and G91 selects incremental positioning. In absolute positioning, all coordinates are relative to the origin (0,0,0). In incremental positioning, all coordinates are distances relative to the last point.

G90 Absolute positioning

G91 Incremental coordinates

Example:

```
G90 X2 Y3 ; moves the X and Y axes from the current position to
; the new position referenced from the absolute machine ;
zero.
G91 X1 Y0 ; moves the X axis 1 inch referenced from the last X
; position, the Y axis does not move.
```

G92 - Set Absolute Position

G92 sets the current absolute position to the coordinates specified. If you are using multiple work coordinate systems, positioning in all coordinate systems will be changed by the same amount (-4 in X, -3 in Y, +2 in Z, and -4 in W in the example below).

Example:

```
G0 X5 Y3 Z-2 W5 ; Moves to the specified location
G92 X1 Y0 Z0 W1 ; Sets the current position to the absolute
; position specified.
```

G98 - Initial Point Return

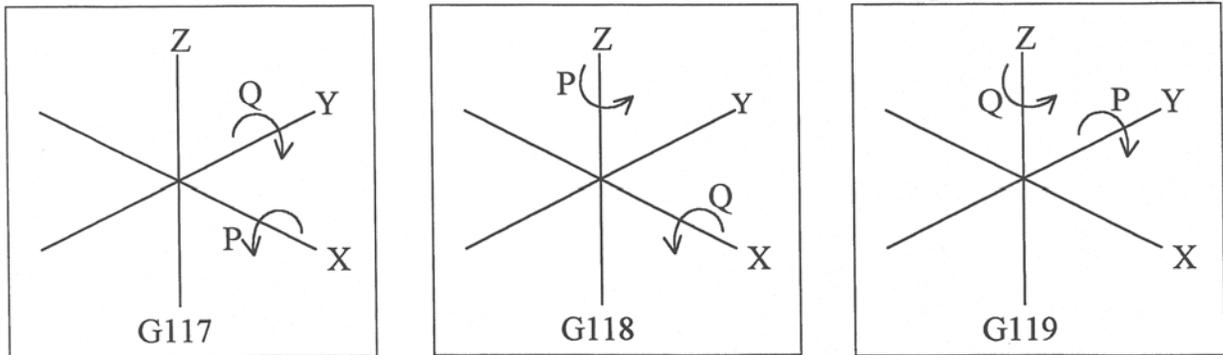
G98 sets the +Z return level to point I as pictured in Figure 1 in the Canned Cycle Section.
(G98 is the default setting)

G99 - R Point Return

G99 sets the +Z return level to point R as pictured in Figure 1 in the Canned Cycle Section.

G117, G118, G119 - Rotation of Pre-set Arc Planes

G117, G118 and G119 have the same functionality as G17, G18 and G19, respectively, except that they include 2 optional parameters P and Q to specify the arc plane rotation away from the pre-set arc plane: P specifies the arc plane angle of rotation (in degrees) around the first axis and Q specifies the arc plane angle of rotation around the second axis.



For the G117 plane, the "first axis" is X and the "second axis" is Y.

For the G118 plane, the "first axis" is Z and the "second axis" is X.

For the G119 plane, the "first axis" is Y and the "second axis" is Z. If P and/or Q is not specified, the angles are assumed to be 0 degrees. If both P and Q parameters are 0, then the plane is assumed to be an orthogonal (pre-set) arc plane. The center of the arc can be specified by the user in a 3D form both in G17-G19 and in G117-G119 (all I, J, K values are allowed at the same time with G2 and G3). Any arc center component outside the circular plane is ignored.

Example:

```
G00 X0 Y0 Z1 ; rapid move
G03 G18 X1 Y0 Z0 K-1 F20 ; arc mill
G00 X0 Y0 Z1.1 ; retract move
G01 Z1 ; move to start of contour
G03 G118 P1.000000 X0.9998 Y0.0175 Z0 K-1 ; arc mill rotated about Z
```

- NOTE: G117-G119 will not be permitted while cutter compensation is turned on. Also, scaling is not allowed while G117-G119 is specified and G117-G119 is not allowed while scaling is active.

CHAPTER 13

CNC Program Codes: M functions

M functions are used to perform specialized actions in CNC programs. Most of the M-series Control M functions have default actions, but can be customized with the use of macro files.

Macro M functions

Most M-Series CNC M functions from 0 through 90 can be fully customized. Exceptions are M2, M6, and M25 that can be customized, but will always move the 3rd (Z) axis to the home position before executing the macro M function commands. No M functions above 90 may be customized with macros. The default action listed will be performed unless that M function has been customized.

To create a macro for an M function, a file must be created in the C:\CNC7 directory. The file's name must be CNC7.Mxx where xx is the M function number used to call the macro. M functions 0-9 must use single digits in the filename (e.g. Use M3, **not** M03). The contents of the file may be any valid M and G codes.

Each time the M function is encountered in a program, the macro file will be processed line by line.

* NOTE: Nesting of macro M functions is allowed. Recursive calls are not: if a macro M function calls itself, the default action of the function will be executed.

Example: Turn on spindle with variable frequency drive and wait for "at speed" response.

Create file C:\CNC7\CNC7.M3 with contents as follows:

```
M94/1 ; request spindle start
M101/5 ; wait for up to speed signal
```

M2, M6, and M25 **always** move the Z-axis to the home position before any other motion. All other M functions are performed after the motion of the current line is complete.

The M and G codes in a macro M function are not usually displayed on the screen as they are executed, and are all treated as one operation in block mode. If you wish to see or step through macro M functions (e.g. for testing purposes), set parameter #10 as follows:

- 0 Don't display or step through macro M functions
- 1 Display macro M functions, but don't step through them
- 2 Display and step through macro M functions

* NOTE: You cannot use block mode to step through a macro M function called using the G81 transformation (see Chapter 13); the action timer will expire before you can press CYCLE START.

* NOTE: Only one M function per line is permitted.

M00 - Stop For Operator

Motion stops, and the operator is prompted to press the CYCLE START button to continue.

Default action:
M100/75

M01 - Optional Stop for Operator

M1 is an optional pause, whose action can be selected by the operator

When optional stops are turned on, M1 will pause the currently running job until CYCLE START is pressed. However, if optional stops are turned off, M1 will not pause the program.

*NOTE: If you plan to override the default action of M1 with a macro file, you may want to include a call to M1 within the macro file so that the default actions of M1 will still be effective in the overridden M1. Otherwise, if a call to M1 is not included within the macro file, the new overridden M1 will cause optional stops to be ineffective.

M02 - Restart Program

M2 moves the Z-axis to the home position, performs any movement requested, and restarts the program from the first line. The operator is prompted to press the CYCLE START button to continue.

M03 - Spindle On Clockwise

M3 requests the PLC to start the spindle in the clockwise direction.

Default action:

M95/2
M94/1

M04 - Spindle On Counterclockwise

M4 requests the PLC to start the spindle in the counterclockwise direction.

Default action:

M95/1
M94/2

M05 - Spindle Stop

M5 requests the PLC to stop the spindle.

Default action:

M95/1/2

M06 - Tool Change

M6 moves the Z-axis to the home position and stops the spindle and coolant. If an automatic tool changer is installed, it then commands the tool changer to switch to the requested tool. Otherwise, it prompts the operator to insert the tool and then press the CYCLE START button on the Operator Panel.

Default action (no tool changer):

```
M25           ;always does M25 first
M95/1/2/3/5   ; turn off spindle & coolant
M100/75       ; wait for CYCLE START button
```

Default action (tool changer installed):

```
M25           ; always does M25 first
M95/1/2/3/5   ; turn off spindle & coolant
M95/16        ; turn off tool changer strobe
M107          ; send tool number to tool changer
M94/16        ; turn on tool changer strobe
M101/32       ; wait for acknowledge from changer
```

```
M95/16      ; turn off tool changer strobe
M100/32     ; wait for acknowledge from changer
```

Manual tool changes are selected by setting Parameter 6 to 0 in the Machine Parameters table. The automatic tool changer is selected by setting Parameter 6 to 1 (see Chapter 6).

The PLC program must be involved in commanding an automatic tool changer and its associated strobe, BCD and ACKnowledge lines. See Chapter 5 of the service manual for details of how such a PLC program could be constructed.

M07 - Mist Coolant On

M7 causes the PLC to start the mist coolant system.

Default action:

```
M95/3
M94/5
```

M08 - Flood Coolant On

M8 causes the PLC to start the flood coolant system.

Default action:

```
M95/5
M94/3
```

M09 - Coolant Off

M9 causes the PLC to stop the coolant system.

Default action:

```
M95/3/5
```

M10 - Clamp On

M10 causes the PLC to activate the clamp.

Default action:

```
M94/4
```

M11 - Clamp Off

M11 causes the PLC to release the clamp.

Default action:

```
M95/4
```

M25 - Move To Z Home

M25 moves the Z-axis to the home position at the Z-axis maximum rate. The Z home position defaults to zero in machine coordinates, but may be changed by changing the Z coordinate of the first Reference Point (on the Work Coordinate System Configuration screen).

Default action:

```
G0 <Z home >
```

M26 - Set Axis Home

M26 sets the machine home position for the specified axis to the current position (after the line's movement). If no axis is specified, M26 sets the Z-axis home position.

Example:

```
M91/X ; home X axis to minus home switch
M26/X ; set machine home for X-axis there
M92/Z ; home Z-axis to plus home switch
M26   ; set machine home for Z-axis there
```

M39 - Air Drill

M39 is a default air drill activation sequence with a timeout. The sequence of operations is as follows:

```
M94/15 ; activate M function request 15
M103/2 ; start 2 second timer
M100/15 ; wait for input 15 to open
M95/15 ; deactivate M function request 15
M104   ; cancel timer
```

* NOTE: This program will be canceled by timer expiration if input 15 does not open within 2 seconds after M function request 15 is activated. The PLC program must be involved in taking away the drill output when the CNC program stops:

Example:

```
; PLC program
CNC_program_running is INP65 ;program running indicator
M15 is INP47 ;M function 15 indicator
drill_out is OUT5 ;air drill output relay
drill_out = M15 & CNC_program_running ;Drill On if M94/15 and the
;CNC program is running. Drill
;Off if M95/15 or the CNC
;program is terminated.
```

M91 - Move to Minus Home

M91 moves to the minus home switch of the axis specified at the slow jog rate for that axis. After the minus home switch is reached, the tool is moved back until the home switch untrips, and then the next encoder index pulse is reached.

Example:

```
M91/X ; move the X axis to the minus home switch.
G92 X-10 ; sets X minus home switch at -10
```

M92 - Move to Plus Home

M92 moves to the plus home switch of the axis specified at the slow jog rate for that axis. After the plus home switch is reached, the tool is moved back until the home switch untrips, and then the next encoder index pulse is reached.

Example:

```
M92/X ; moves the X-axis to the plus home switch.
G92 X+10 ; Sets X plus home switch at +10
```

M93 - Release Motor Power

M93 releases the motor power for the axis specified. If no axis is specified, then all axes are released.

Example:

M93/X ; releases the X axis.
M93 ; releases the motors on all axes.

* NOTE: Any axis freed within a CNC program should **not** be used in that program afterwards. Incorrect positioning may result.

M94/M95 - Output On/Off

There are sixteen user definable M function requests. M94 and M95 are used to request those inputs to turn on or off respectively. M function requests 1-16 are mapped to the PLC as inputs 33 - 48, as shown in the following table:

On	Off	PLC Input
M94/1	M95/1	33
M94/2	M95/2	34
M94/3	M95/3	35
M94/4	M95/4	36
M94/5	M95/5	37
M94/6	M95/6	38
M94/7	M95/7	39
M94/8	M95/8	40

On	Off	PLC Input
M94/9	M95/9	41
M94/10	M95/10	42
M94/11	M95/11	43
M94/12	M95/12	44
M94/13	M95/13	45
M94/14	M95/14	46
M94/15	M95/15	47
M94/16	M95/16	48

M Function request to PLC Input map

To use M94 and M95 to control a function external to the servo control, such as an indexer, the input request must be mapped to one of the PLC outputs in the PLC program. See M94/M95 function usage in the PLC section of the service manual.

Example:

M94/5/6 ; turns on output requests 5 and 6.

* NOTE: Requests 1, 2, 3, 4 and 5 are by default used to control the spindle CW, spindle CCW, flood coolant, clamp, and mist coolant.

* NOTE: The request number need not be (and generally is not) the same as the M function number or the PLC output number. For example, M3 turns on output request #1 (PLC Input #33), which may activate PLC output #14.

M98 - Call Subprogram (Optional)

M98 calls a user-specified subprogram. A subprogram is a separate program that can be used to perform certain operation (e.g. a drilling pattern, contour, etc.) many times throughout a main program.

Calling methods:

M98 Pxxxx Lrrrr

or

M98 "program.cnc" Lrrrr

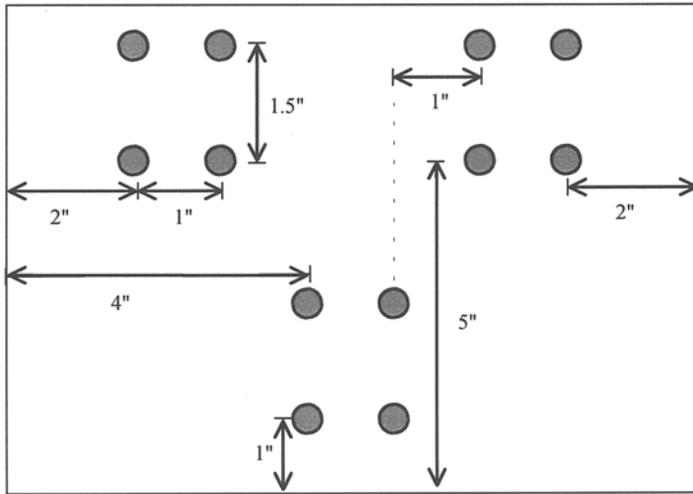
Where *xxxx* is the subprogram number (in file O*xxxx*.CNC, 0000-9999 allowed, leading 0's required in filename), *rrrr* is the repeat value, and "program.cnc" is the name of the subprogram file.

Subprograms are written just like normal programs, with one exception: an M99 should be at the end of the subprogram. M99 transfers control back to the calling program.

Subprograms can call other subprograms (up to 20 nested levels of calling may be used), Macro M-functions, and Macros. Macro M-functions and Macros can similarly call subprograms.

Subprograms 9100-9999 can also be embedded into a main program, using O9xxx to designate the beginning of the subprogram and M99 to end it. CNC7 will read the subprogram and generate a file O9xxx.cnc. CNC7 will not execute the subprogram until encounters M98 P9xxx.

NOTE: An embedded subprogram definition must be placed before any calls to the subprogram.



(0,0)

Suppose that a drilling pattern of 4 holes is needed in 3 different locations:

This subprogram would handle the drilling and incremental moves between the holes:

```
O0001           ;Program O0001.cnc
G91 F10         ;Incremental positioning
G81 X0 Y0 R -.4 Z-.6 ;Drill lower left hole
Y1.5 R -.4 Z-.6   ;Drill upper left hole
X1 R -.4 Z-.6     ;Drill upper right hole
Y-1.5 R-.4 Z-.6   ;Drill lower right hole
G80             ;Cancel canned cycles
M99             ;End of subprogram
```

The main program would call this subprogram three times:

```
:Main program
G90 G0 X2 Y5 Z0.5 ;Move to first hole pattern
M98 P0001 L1      ;Call subprogram O0001.cnc
G90 G0 X4Y1 Z0.5 ;Move to second hole pattern
M98 P0001 L1      ;Call subprogram
G90 G0 X6 Y5 Z0.5 ;Move to third hole pattern
M98 P0001 L1      ;Call subprogram
:End program
```

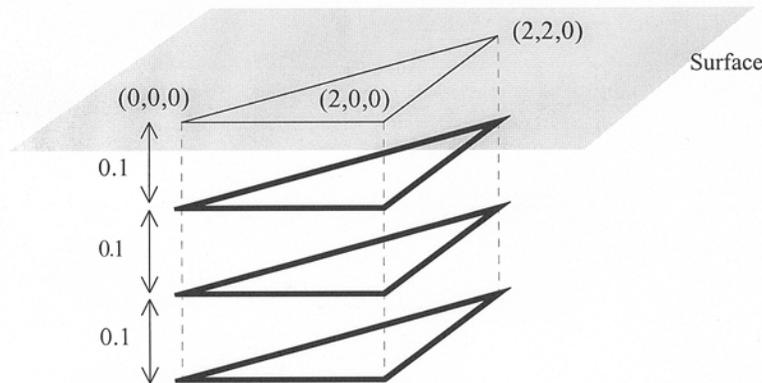
Another example is "looping" or consecutively repeating a section of code. Here, the subprogram will be part of the main program.

```
:Main program
G90 G0 X0 Y0 Z0.1
```

```

G1 Z0 F30
O9100                                ;Beginning of subprogram
G91 G1 Z-0.1 F5
G90 X2 F30
Y2
X0 Y0
M99                                    ;End of subprogram 9100
M98 P9100 L3                          ;Repeat O9100 3 times
M25 G49                                ;End main program

```



M99 - Return From Macro or Subprogram

M99 designates the end of a subprogram or macro and transfers control back to the calling program when executed. M99 may be specified on a line with other G codes. M99 will be the last action executed on a line. If M99 is not specified in a subprogram file, M99 is assumed at the end of the file:

Example:

```
G1 X3 M99 ;Move to X3 then return to calling program.
```

If M99 is encountered in the main job file, it will be interpreted as the end of the job. If M99 is encountered in an M function macro file, it will be interpreted as the end of any enclosing subprogram or macro, or as the end of the job.

M100 - Wait For Input to Open

M100 waits for the specified input to open.

Example:

```
M94/7 ; turns on output 7.
M100/1 ; waits for acknowledgment on input 1.
```

M101 - Wait For Input to Close

M101 waits for the specified input to close.

Example:

```
M95/7 ; turns off output 7.
M101/1 ; waits for acknowledge on input 1.
```

M102 - Restart Program

M102 performs any movement requested, and restarts the program from the first line. The Z-axis is **NOT** moved to the home position, and the operator is **NOT** prompted to press the CYCLE START button to continue.

M103 - Programmed Action Timer

M103 starts a timer for the operations in a program. If M104 (stop timer) is not executed before the specified time expires, the program will be canceled and the message "Programmed action timer expired" will be displayed. This function detects the failure of a device connected to the PLC and prevents further programmed action. M103 and M104 must be used for air drill cycles.

Example: Activate a device and wait for a response. If there is no response within 4.5 seconds, cancel the program.

```
M94/12 ; turn on relay 12
M103/4.5 ; start 4.5 second timer
M100/4 ; wait for input 4 to open
M104 ; input 4 opened, cancel timer
```

* NOTE: The PLC program must detect the cancellation of the program and deactivate all programmed machine functions.

Example:

```
;PLC program
CNC_program_running is INP65 ;program running indicator
M12 is INP44 ;M function 12 indicator
relay_out is OUT5 ;relay On/Off
relay_out = M12 & CNC_program_running ;Relay On if M94/12 and the
;CNC program is active. Relay
;Off if M95/12 or the CNC
;program is terminated.
```

M104 - Cancel Programmed Action Timer

M104 stops the timer started by the last M103 executed.

M105 - Move Minus to Switch

M105 moves the requested axis in the minus direction at the current feedrate until the specified switch opens.

Example:

```
M105/X P5 F30 ; move the X axis minus at 30"/min until
; switch #5 opens
G92 X10 ; Sets X position to 10
```

M106 - Move Plus to Switch

M106 moves the requested axis in the plus direction at the current feedrate until the specified switch opens.

Example:

```
M106/X P3 F30 ; move the X axis plus at 30"/min until
; switch #3 opens
G92 X10 ; Sets X position to 10
```

M107 - Output BCD Tool Number

M107 sends the current tool number to the automatic tool changer, via the PLC. The number is sent as BCD. M107 does not set the tool changer strobe or look for an acknowledgement from the changer (see M6).

Example:

```
M107 ; send request for tool to changer
M95/16 ; turn on tool changer strobe
M101/5 ; wait for acknowledge on input 5
M95/16 ; turn off strobe
M100/5 ; wait for acknowledge to be removed
```

M108 - Enable Override Controls

M108 re-enables the feedrate override and/or spindle speed override controls if they have previously been disabled with M109. A parameter of 1 indicates the feedrate override; 2 indicates the spindle speed override.

Example:

```
M109/1/2 ; disable feedrate and spindle speed overrides
M108/1 ; re-enable feedrate override
M108/2 ; re-enable spindle speed override
```

M109 - Disable Override Controls

M109 disables the feedrate override and/or spindle speed override controls. It may be used before tapping with G85 to assure that the machine runs at the programmed feedrate and spindle speed. It is not necessary to specify M109 with G74 or G84; those cycles automatically disable and re-enable the override controls. M109 cannot be used in MDI mode.

Example:

```
M3 S500 ; start spindle clockwise, 500 rpm
F27.78 ; set feedrate for 18 pitch tap
M109/1/2 ; disable feedrate and spindle speed overrides
G85 X0 Y0 R.1 Z-.5 ; tap a hole
M108/1/2 ; re-enable overrides
```

M115/M116/M125/M126 – Protected Move Probing Functions

The protected move probing functions provide the capability to program customized probing routines.

The structure for these commands is:

```
Mnnn /Axis pos Pp Ff
```

where:

nnn	is either 115, 116, 125, or 126.
Axis	is a valid axis label, i.e., X, Y, Z, etc.
pos	is an optional position
p	is a plc bit number, which can be negative.
f	is a feedrate (in units per minute.)

For M115 and M116 functions, the indicated *axis* will move to *pos* (if specified) until the corresponding plc bit *p* state is 1, unless *p* is negative, in which case movement is until the plc bit state is 0. A *p* value of 1 to 80 (or -1 to -80) specifies plc bits INP1-INP80, 81 to 160 (or -80 to -160) specifies plc bits OUT1-OUT80, and 161 to 240 (or -161 to -240) specifies plc bits MEM1-MEM80. Warnings are generated in the CNC7 message window for "*Missing P value*" and "*Invalid P value*."

If *pos* is not specified, M115 will move *axis* in the negative direction, and M116 will move *axis* in the positive direction. Note that if *pos* is specified, then it does not matter whether M115 or M116 is used.

If *pos* is not specified, the movement is bounded by the settings in the software travel limits. In the absence of software travel limits, movement is bounded by the maximum probing distance (Machine Parameter 16). In cases where *pos* is specified, it is still bounded by the software travel limits.

If the bounded position is reached before the awaited plc bit state is found, a "Probe unable to detect surface" error will be generated, which will stop any running job.

For M125 and M126 protected move functions, the behavior is identical to that of the M115 and M116 commands, except in regards to the plc bit state. M125 and M126 will generate an "Unexpected probe contact" error message if the specified plc bit state is triggered, again stopping any running job.

In summary, the M115 and M116 commands are to be used when one expects contact to be made, and M125 and M126 commands are to be used when one does not expect any contact to be made.

Example:

Finding the center of a vertical slot. In this example, it is assumed that there is a probe connected to INP15 and that the probe tip is positioned somewhere in the slot, such that movement along the X axis will cause a probe trigger.

```
G20                ; Set mode to English
M115/X P-15 F20    ; Move X minus waiting for probe trip at 20 ipm
M116/X P15 F5      ; Move X plus until no contact at 5 ipm
#100 = #5041        ; Record the point in user variable #100
M116/X P-15 F20    ; Move X plus at 20 ipm until probe trip
M115/X P15 F5      ; Move X minus at 5 ipm until probe clears
X[[#100+#5041]/2] ; Move X to center of slot
```

M120 - Open data file (overwrite existing file)

This M function will open the requested data file for writing. If no drive or directory is specified with the file name, then the file will be opened in the same directory as the CNC program. If the file cannot be successfully opened, then an error will be returned, ultimately terminating the job. If a data file is already open when M120 is called, that file will first be closed, then the new file opened.

Example:

```
M120 "probetst.dat"
```

M121 - Open data file (append to existing file)

This M function will open the requested file for writing at the end of the file. If no drive or directory is specified with the file name, then the file will be opened in the same directory as the CNC program. If the file does not already exist, it will be created. This is not an error. If the file cannot be successfully opened, then an error will be returned, ultimately terminating the job. If a data file is already open when M121 is called, that file will first be closed, then the new file opened.

Example:

```
M121 "c:\probetst.dat"
```

M122 - Record position(s) and optional comment in data file

This M function will write the current expected position value to the data file, in the usual format (i.e. axis label before number, 4 decimal places in inch mode, 3 decimal places in millimeter mode). Any comment that appeared on the line with M122 will be output after the position(s). With no axis arguments, M122 will write the positions of all installed axes. With axis arguments, it will write the positions only of the requested axes. Positions will be written in local (not machine) coordinates, in native machine units. If no data file has been opened with M120 or M121 before M122 is called, then M122 will return an error and terminate the job. The parameter L1 may be used to suppress the new line character normally outputted after the last position.

Examples (M function and sample output):

```
M122          ->   X1.2345 Y-3.2109 Z-0.5678
M122 /Z ; at 10 ipm  ->   Z-.4321 ; at 10 ipm
M122 /X/Y          ->   X-1.0000 Y0.8732
M122 /X L1         ->   X-1.5000
M122 /X           ->   X-1.5000 X-2.0000
```

M123 - Record value and/or comment in data file

This M function will write the specified parameter value (if any) to the data file, followed by any comment that appeared on the line with M123. If a P value is specified, M123 will output a ';' character followed by the numeric value (4 decimal places in inches, 3 in millimeters). If no P value is specified, then M123 outputs the comment only. If neither a P value nor a comment was specified, M123 does nothing. This is not an error. If no data file has been opened with M120 or M121 before M123 is called, then M123 will return an error and terminate the job. The parameter L1 may be used to suppress the new line character normally outputted after the last value.

Examples (M function and sample output):

```
M123 P1.2345          ->   ; 1.2345
M123 P#A ; first macro argument  ->   ; 1.2345 ; first macro argument
M123 ; Probing X+ to surface      ->   ; probing X+ to surface
M123                      ->   <nothing>
```


CHAPTER 14

Operator Panels

M-Series Jog Panel

The M-Series operator panel is a sealed membrane keyboard that enables you to control various machine operations and functions. The panel contains momentary membrane switches, which are used in combination with LED indicators to indicate the status of the machine functions.

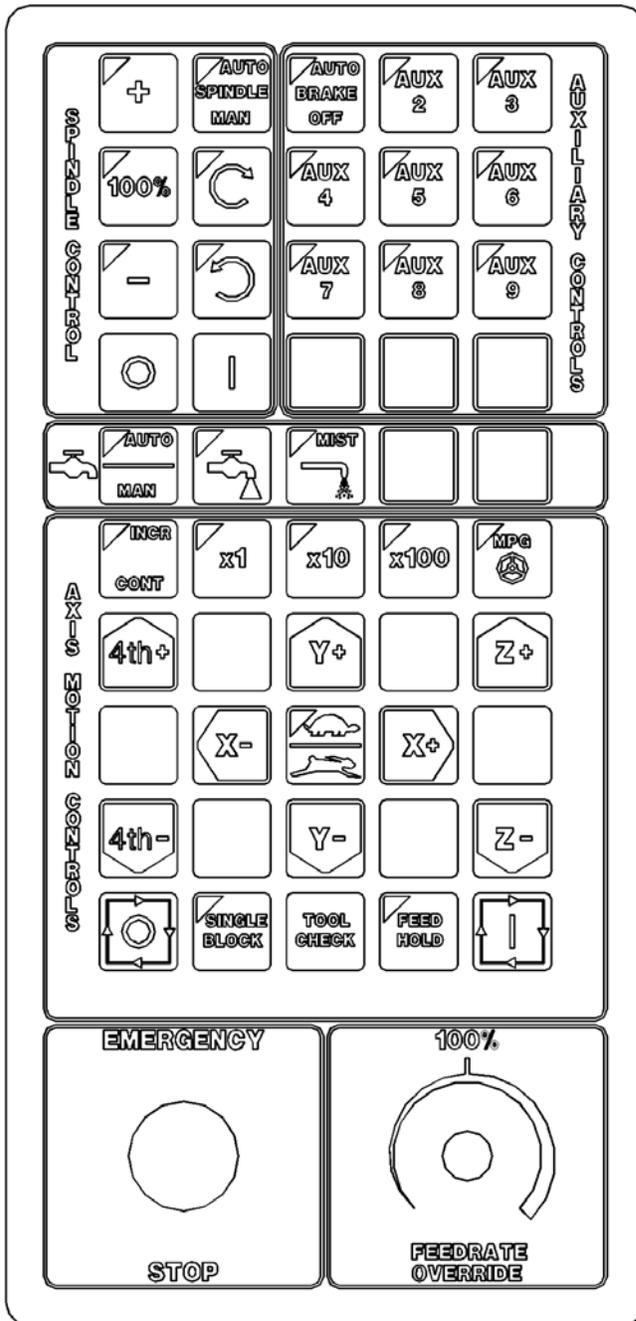


Fig 1 - M-Series Jog Panel

The M-Series operator panel is a sealed membrane keyboard that enables you to control various machine operations and functions. The panel contains momentary membrane switches. The M-Series jog panel can be customized as to the location of various keys. The jog panel displayed in the figure above is representative of a default configuration found on most M-series controls.

Axis Jog Buttons

X+ X- Y+ Y- Z+ Z- 4TH+ 4TH-

The yellow X, Y, Z, and 4TH keys are momentary switches for jogging each of the four axes of the machine. There are two buttons for each axis (+/-). Only one axis can be jogged at a time.

Slow/Fast



The slow/fast key is located in the center of the Axis Motion Controls section and is labeled with the turtle and rabbit icon shown to the right. The turtle represents slow jogging mode. When SLOW jog is selected (LED on) and a jog button is pressed, the axis moves at the slow jog rate. If FAST jog is selected, the axis will move at the fast jog rate. See Chapter 15 for information on setting the fast and slow jog rates for each axis.

Inc/Cont

INC/CONT selects between incremental and continuous jogging. Pressing the key will toggle between these two modes. The LED is lit when INC is selected. When set to INC jog and a jog button is pressed, the axis will move the current jog increment distance and stop. The jog button must be released and then pressed again before any further axis movement can occur. The LED is not lit when set to CONT. If CONT jog is selected and an axis jog button is pressed, the axis will move continuously until the button is released.

* NOTE: The jog buttons will not operate if the M-Series CNC software is **not** running, or a job (a CNC program) is running.

x1, x10, x100

Press any one of these keys to set the jog increment amount. The amount you select is the distance the control will move an axis if you make an incremental jog (x1=0.0001", x10=0.0010" and x100=0.0100"). You may select only one jog increment at a time, and the current jog increment is indicated by the key that has a lit LED. The jog increment you select is for all axes; you cannot set separate jog increments for each axis. The jog increment also selects the distance the control will move an axis for each click of the MPG handwheel.

MPG



The MPG is housed in a separate hand-held unit. Press the MPG key to set the control jog to respond to the MPG handwheel, if equipped. When selected, the LED will be on. Select the Jog Increment and desired axis and slowly turn the wheel. When the LED is not lit, the MPG is disabled and the jog panel is on.

* WARNING: Do not spin the handwheel too quickly. Damage to the machine or part may result.

Rapid Over

The RAPID OVER key controls rapid override. If the RAPID OVER LED is on, the FEEDRATE OVERRIDE knob applies to rapid (GO) moves and to jogging. If the RAPID OVER LED is off, the FEEDRATE OVERRIDE knob will have no effect on rapid moves and jogging.

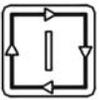
Tool Check

Press TOOL CHECK while **no** program is running to move the Z-axis to its home position. Press TOOL CHECK while a program is running to abort the currently running program. The control will stop normal program movement, pull Z to its home position, clear all M-functions, and automatically display the Resume Job Screen. From the Resume Job Screen, you can change tool settings (height offsets, diameter offsets, etc.) and resume the job with the new tool settings.

Single Block

The SINGLE BLOCK key selects between auto and single block mode. When the SINGLE BLOCK LED is on, the single block mode has been enabled. Single Block mode allows you to run a program line by line by pressing CYCLE START after each block. While in block mode you can select auto mode at any time. While in auto mode and a program is running you cannot select single block mode. Auto mode runs the loaded program after CYCLE START is pressed. Auto mode is the default (LED off).

Cycle Start



When the CYCLE START button is pressed, the M-400/M-39 Control will immediately begin processing the current program at the beginning and will prompt you to press the CYCLE START button again to begin execution of the program. After an M0, M1, M2, or M6 is encountered in the program, the message

Press CYCLE START to continue

will be displayed on the screen, and the M-400/M-39 Control will wait until you press the CYCLE START button before continuing program execution.

* NOTE: Pressing CYCLE START will cause the M-Series Control to start moving the axes immediately without further warning. Be certain that you are ready to start the program when you press this button. Pressing the FEED HOLD button or the CYCLE CANCEL button will stop any movement if CYCLE START is pressed accidentally.

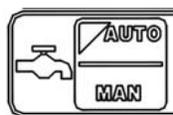
Cycle Cancel



Press CYCLE CANCEL to abort the currently running program. The control will stop movement immediately, clear all M-functions, and return to the Main Screen. It is recommended that you press FEED HOLD first before CYCLE CANCEL. If you press CYCLE CANCEL, program execution will stop; if you wish to restart the program you must rerun the entire program or use the search function. See search function operation in Chapter 2.

Coolant Control Keys

The coolant control keys are located in a single row between the Spindle Control section and Axis Motion Controls section of the jog panel.



Coolant Auto/Manual selection. This key will toggle between automatic and manual control of coolant. In automatic mode, M7 (Mist) and M8 (Flood) can be used in G-code programs to select the coolant type to be enabled. In manual mode, flood coolant and mist coolant are controlled by separate keys

When switching from automatic to manual mode, both flood and mist coolant are turned off automatically.

Coolant Flood



In manual coolant control mode, flood coolant can be toggled off and on by pressing this key. The LED will be on when flood control is selected in either automatic or manual mode.

Coolant Mist



In manual coolant control mode, mist coolant can be toggled off and on by pressing this key. The LED will be on when mist control is selected in either automatic or manual mode.

Auxiliary Function Keys (AUX1 – AUX12)

The M-Series jog panel has nine auxiliary keys, some of which may be defined by customized systems.

Brake (Auto/Off)

The BRAKE AUTO/OFF key controls the operation of the spindle brake (if present) on the mill. If the LED is on, the brake is "ON" or is in automatic mode, meaning every time the spindle is started, the brake will automatically release, and every time the spindle is stopped, the brake will be applied. If the LED is off, the brake will always be released. The default for this key is AUTO.

Spindle Controls

Spindle (CW/CCW)



The SPINDLE CLOCKWISE/COUNTERCLOCKWISE keys determine the direction the spindle will turn if it is started manually. If the spindle is started automatically, the direction keys are ignored and the spindle runs according to the program. The default direction is CW.

Spindle Override Controls



Speed increase. Pressing this key will increase the spindle speed by 10% of the commanded speed in Auto spindle mode, limited by the maximum speed or 200% of commanded speed, whichever is less. For manual spindle mode, the spindle speed is increased by 5% of the maximum spindle speed (up to the maximum speed). The LED is on if the spindle speed is set above the 100% point.



Pressing this key will set the spindle speed at the 100% point, which is defined as the commanded speed in Auto spindle mode, or ½ the maximum spindle speed in manual mode. The LED will be on when the spindle is at the 100% point.

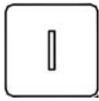


Speed decrease. Pressing this key will decrease the spindle speed by 10% of the commanded speed in Auto spindle mode, limited to 10% of commanded speed. For manual spindle mode, the spindle speed is decreased by 5% of the maximum spindle speed down to 5% of maximum. The LED is on if the spindle speed is set below the 100% point.

Spindle (Auto/Man)

This key selects whether the spindle will operate under program control (automatic) or under operator control (manual). When the LED is lit, the spindle is under automatic control. If the LED is off, the spindle is under manual control. Pressing the SPINDLE (AUTO/MAN) key will toggle it from AUTO to MAN and back again. The default is AUTO mode.

Spin Start



Press the SPIN START key when manual spindle mode is selected to cause the spindle to start rotating. Press SPIN START when automatic mode is selected to restart the spindle if it has been paused with SPIN STOP.

Spin Stop



Press the SPIN STOP key when manual spindle mode is selected to stop the spindle. Press SPIN STOP when automatic mode is selected to pause spindle rotation and can be restarted with SPIN START.

*** WARNING: SPIN STOP should only be pressed during FEED HOLD or when a program is NOT running.**

Feedrate Override

This knob controls the percentage of the programmed Feedrate that you can use during feedrate cutting moves: lines, arcs, canned cycles, etc. This percentage can be from 2% to 200%.

Spindle Override

This knob is the percentage of the Spindle Speed that will be used by variable frequency spindle drive. In AUTO Mode, the SPINDLE OVERRIDE knob is a percentage override (2% - 200%) of the programmed Spindle Speed (see Chapter 12 for how to assign a spindle speed). In Manual Mode, the knob controls Spindle Speed directly, from 0 to the maximum spindle speed.

Feed Hold

Feed Hold decelerates motion of the current movement to a stop, pausing the job that is currently running. Pressing CYCLE START will continue the movement from the stopped location.

Emergency Stop

EMERGENCY STOP releases the power to all the axes and cancels the current job immediately upon being pressed. EMERGENCY STOP also resets certain faults if the fault condition has been fixed or cleared.

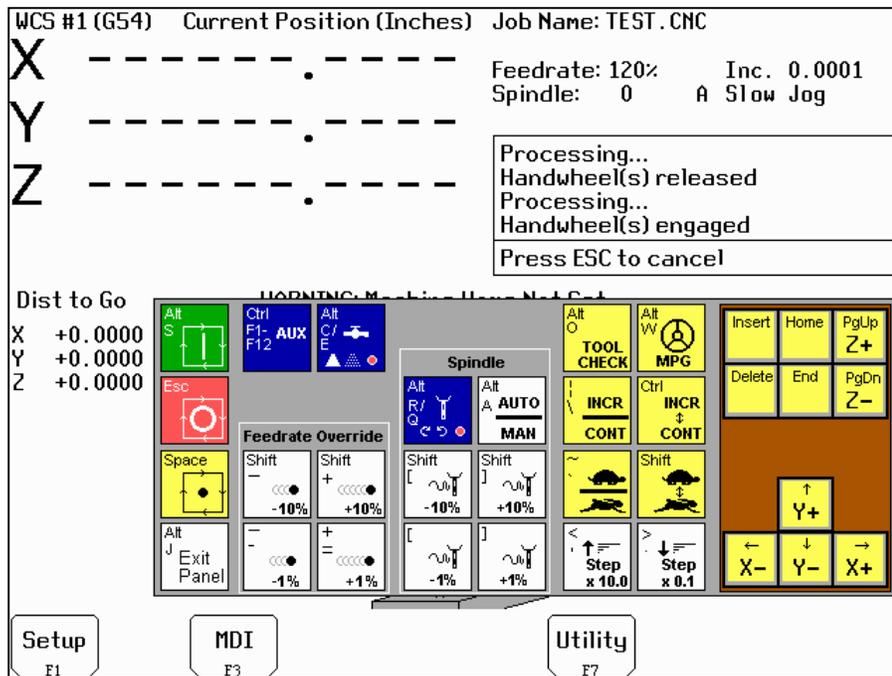
*** WARNING: On some machines, the Z or W axis will sometimes fall due to the lack of power.**

Notes about operator panels

The behavior of the control system in response to the functions listed above for the M-Series jog panel is dependant upon optional software options, the PLC program, machine parameters, and hardware wiring of the system. It is possible that the functioning explained in this chapter does not apply to a particular control system or that it may differ in some aspects.

Keyboard Jog Panel

The keyboard may be used as a jog panel. Press <Alt-J> to display and enable the keyboard jog panel. The jog panel appears as shown below:



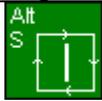
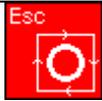
For full functionality of the keyboard jog panel, “Keyboard” must be selected as the console type in the Console Configuration menu.

The jog panel shows the mapping of keys to jogging functions. Normally, the keyboard performs menu navigation and data entry functions. The keyboard can jog the axes only when the keyboard jog panel is displayed. Ctrl and Alt functions are available, for the most part, even when the jog panel is not shown.

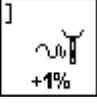
The status window in the upper right corner of the screen displays the jogging mode (continuous/incremental), incremental step size, and jog speed (fast/slow). In continuous mode, the jog keys start movement when pressed and movement stops when you release the key. In incremental mode, the axis will move the indicated incremental step amount.

As shown in the picture above, the jog keys are located in the cursor key block to the right of the main keyboard and to the left of the numeric keypad. If a jog key controls an axis, it will be overlaid with the axis symbol (“X”, “Y”, etc.) The jog keys are the arrow keys, <Insert>, <Delete>, <Home>, <End>, <Page Up>, and <Page Down>.

The remaining keys are described below:

Legend	Key(s)	Function	Description	Availability (Notes)
	<Alt S>	Cycle Start	Same as Cycle Start.	Always, with few exceptions. (1)
	<Esc>	Cycle Cancel	Same as Cycle Cancel.	During a job run; otherwise, Esc is used to exit CNC7 menus.

Legend	Key(s)	Function	Description	Availability (Notes)
	<Space> or <Alt H>	Feed Hold	Turns Feed Hold on and off	The space key may be used for editing and may not be available at all times. Alt-H is always available.
	<Alt J>	Start/Exit Panel	Invokes or exits the jog panel.	Always, with few exceptions. (1)
	<Ctrl F1> - <Ctrl F12>	Aux 1 – Aux 12	Executes the corresponding Aux function and signals the PLC. A custom PLC program is required to act upon jog panel signals.	Always, with few exceptions. (1,3)
	<Alt C> and <Alt Q>	Flood Coolant and Mist Coolant	Alt C turns flood coolant on and off. Alt E turns mist coolant on and off. Both flood and mist may be on at the same time. Either key automatically selects manual coolant mode. If requested by CNC7, Alt C and Alt E will select “Auto Coolant Mode”. Press either when prompted.	Always, with few exceptions. (1,3)
	<Shift -> or <_>	Feed Rate Override -10%	Decreases the feed rate override by 10%.	Jog panel, job run, graphing, and some other times. (2,4)
	<Shift => or <+>	Feed Rate Override +10%	Increases the feed rate override by 10%.	Jog panel, job run, graphing, and some other times. (2,4)
	<->	Feed Rate Override -1%	Decreases the feed rate override by 1%.	Jog panel, job run, graphing, and some other times. (2,4)
	<=>	Feed Rate Override +1%	Increases the feed rate override by 1%.	Jog panel, job run, graphing, and some other times. (2,4)
	<Alt R> and <Alt Q>	Spindle On/Off CW/CCW	Alt R turns the spindle on clockwise if the spindle is off; otherwise, it turns the spindle off. Alt Q is similar except counter-clockwise. Either will automatically select manual spindle operation.	Always, with few exceptions. (1,3)
	<Alt A>	Spindle Auto/ Manual	Toggles between automatic and manual spindle operation.	Always, with few exceptions. (1,3)
	<Shift [> or <{>	Spindle Override -10%	Decreases the spindle override by 10%.	Only in jog panel, and during a job. (2,4)
	<Shift] > or <}>	Spindle Override +10%	Increases the spindle override by 10%.	Only in jog panel, and during a job. (2, 4)

Legend	Key(s)	Function	Description	Availability (Notes)
	<[>	Spindle Override -1%	Decreases the spindle override by 1%.	Only in jog panel, and during a job. (2,4)
	<]>	Spindle Override +10%	Increases the spindle override by 1%.	Only in jog panel, and during a job. (2,4)
	<Alt O>	Tool Check	Performs a tool check.	Always, with few exceptions. (1)
	<Alt W>	MPG on/off	Turns MPG (handwheel) control on and off.	Available most times that jogging is available.
	< > or <>	Incremental/ Continuous Jog Selection	Selects incremental or continuous jog mode. Press again to select the opposite mode.	Only in jog panel.
	Ctrl (as modifier)	Incremental/ Continuous Jog	Fast and temporary incremental/continuous mode switch. Hold down simultaneously with a jog key. (This is like holding down the Shift key to type a capital letter instead of pressing Caps Lock.)	Only in jog panel.
	<~> or <~>	Fast/ Slow Jog Selection	Selects fast or slow jog mode. Press again to select the opposite mode.	Only in jog panel.
	Shift (as modifier)	Fast/Slow Jog Selection	Fast and temporary fast/slow mode switch. Hold down simultaneously with a jog key. (This is like holding down the Shift key to type a capital letter instead of pressing Caps Lock.)	Only in jog panel.
	<, > or <<>	Increase Jog Step 10x	Changes incremental jog step from .0001 to .001 to .01, etc. (The “1” moves to the left in the status window.) This also selects handwheel speed.	Only in jog panel.
	<. > or <>>	Decrease Jog Step 10x	Changes incremental jog step from .1 to .01 to .001, etc. (The “1” moves to the right in the status window.) This also selects handwheel speed.	Only in jog panel.
	<F1> – <F10>	F key pass-thru	Exits the jog panel and executes the corresponding F key.	Where F keys are visible.

Notes:

1. Hot key. In general, the key can be used at any time. Some CNC7 menus may prevent the use of certain keys.
2. The console type in the console configuration menu must be set to “Keyboard” to use this key.
3. The PLC program must be programmed to support this key. Keyboard only systems have this support built-in. Systems with other jog panels may not have this support.
4. Available if not in use by CNC7. For example, feed rate override can be adjusted from the main menu. If you are editing a value in a table or menu, you cannot adjust feed rate override.

MDI and the Keyboard Jog Panel

Many of the keys used by the keyboard jog panel are also possible commands to MDI. To use the keyboard jog panel functions in MDI, you must press <Alt J>. You may jog, use the handwheels or any other jog panel function. Press <Alt J> or <Esc> to return to MDI.

CHAPTER 15

Configuration

(F3 from Setup)

WCS #1 (G54)	Current Position (inches)	Job Name: _PC_TEST.CNC
X	+0.0000	Tool: T1 H0
Y	+0.0000	Feedrate: 120% Part Cnt: 0
Z	+0.0000	Spindle: 0 Part #: 30
		Feed Hold: Off
Stopped		
Press CYCLE START to start job		
Configuration		
Contrl F1	Mach. F2	Parms F3
		PID F4

General

The first five options, <F1> through <F4>, will display a set of parameters. Each option is explained in detail below. The <ESC> key will return you to the previous screen (Setup).

The configuration option provides you with a means for modifying the machine and controller configuration. The majority of information in this section should not be changed without contacting your dealer. Some of the data, if corrupt or incorrect, could cause personal injury or machine damage.

Password

When you press <F3> from the Setup Screen, you may be prompted to enter a password. This level of security is necessary so that users do not accidentally change vital parameters. The original default password is distributed in the documentation provided to the owner of the machine when the control is installed. This password is changeable via Parameter 42.

If you know the password, type it and press <ENTER>. If the password you enter is incorrect, a message will appear telling you the password was incorrect and the password prompt will reappear. Pressing <ESC> will remove the prompt.

If you don't know the password, simply press <ENTER>. You will be given access to the configuration options so that you can view the information. However, you will not be able to change any of the data.

Control Configuration

Pressing <F1> from the configuration screen will display the Control Configuration screen in the edit window. The Control Configuration screen provides you with a method of changing controller dependent data. Each of the fields is discussed in detail below.

If you wish to change a field, use the up and down arrow keys to move the cursor to the desired field. Type the new value and press <ENTER>, or press <SPACE> to toggle. When you are done editing, press <F10> to save any changes you have made. If you wish to discard your changes and restore the previous values, press <ESC>.

WCS #1 (G54)	Current Position (Inches)	Job Name: TEST .CNC
X	-3.16084	Tool: T1 H---
Y	-2.23898	Feedrate: 100%
Z	+0.00000	Spindle: 1506 M
B	+0.00000	Processing... Waiting for PLC operation Processing... Stopped
A	+0.00000	

Control Configuration		
DRO display units:	Inches	(Inches / Millimeters)
Machine units:	Inches	(Inches / Millimeters)
Max spindle (high range):	3000.0	(1.0 to 500000.0 RPM)
Min spindle (high range):	0.0	(0.0 to 500000.0 RPM)
Machine home at pwrup:	Jog	(Jog / Home Switch / Ref Mark-HS)
PLC type:	Absent	(Absent / Normal / Lite / Dual)
Console type:	Keyboard	
Jog panel required:	Yes	(No / Yes)
Screen blank delay:	20	(1 to 200 minutes)
Remote Drive & Directory:		

Press SPACE to change

Save
F10

DRO Display Units

This field controls the units of measure the DRO displays. The two options are 'Millimeters' and 'Inches.' When this field is highlighted by the cursor, "Press SPACE to change" appears at the bottom of the screen. This message is explaining that pressing the <SPACE> key will toggle the value of this field between the two options.

The DRO display units do not have to be the same as the machine units of measure (explained below). This field is provided for users of the G20 & G21 codes so that they may view the tool position in terms of job units (see Chapter 11).

Machine Units of Measure

This field controls which units of measure the machine uses for each job. The two options are 'Millimeters' and 'Inches'. Press <SPACE> to toggle the field between the two options.

This field determines the default interpretation of job dimensions and feedrates. If 'Inches' is selected, all feedrates and dimensions will be interpreted as inches as well as any unit dependent parameters.

* NOTE: This field should rarely, if ever, be changed. If you wish to run a job in units other than the default machine units, use the G20 & G21 codes.

Maximum Spindle Speed (High Range)

This field sets the high range maximum spindle speed for those machines that have a variable frequency spindle drive controller (VFD). All spindle speeds entered in a CNC program are output to the PLC as percentages of this maximum value.

If your machine is equipped with a dual range drive and VFD, the controller will not exceed the spindle speed set by this field while in high gear. See the Machine Parameters section below for information on setting the gear ratios for medium and low gear ranges. If your machine has a VFD but is not equipped with a dual range drive, this field determines the maximum spindle speed.

Minimum Spindle Speed (High Range)

This parameter is used to adjust the minimum spindle speed for the high range. This parameter allows the operator to set the minimum value for spindle speed to a value other than 0. All changes in spindle speed are made in relationship to this value, with this parameter as the minimum value. The values stored can range from 0 to 500000.0 RPM.

Machine Home At Powerup

This field controls how the machine will home at powerup. Set Machine Home at Powerup to Limit Switch if you have limit/home switches or safe hard stops for all axes, and wish to use the switches or stops for homing. Set Machine Home at Powerup to Ref Mark if you have fixed reference marks for any axis. In Ref Mark homing, axes that contain a zero (0) for the plus or minus home switch in the Machine Configuration designate that axis to have a Ref Mark home, while non-zero values specify Limit Switch homing. Set Machine Home at Powerup to Jog if you need to manually move or jog the machine to its home position. See Chapter 1 for more information about machine home.

PLC Type

This field tells the controller which PLC type is installed. The possible values are 'Absent,' 'Lite,' 'Normal,' and 'Dual.' The value should not be changed unless a different PLC type is installed. Use the <SPACE> key to select among the four options.

The standard PLC types installed are dependent on your M-series number and the options that may have been purchased. Check the information sheet on page *ix* for which type of PLC is installed on your machine, or check with your dealer for more information.

Console Type

Set for type of console installed. "T-" for lathe, "M-" for mill.

Jog Panel Required

This field tells the controller whether a Jog Panel must be installed in order to run jobs.

Screen Blank Delay

This field determines the delay used for the screen blanker function. When a value other than zero is set, the screen will blank after the specified number of minutes. The blanking function only works if no jobs are running.

The value you enter is measured in minutes. Therefore, a value of 5 would blank the screen in 5 minutes if no actions were taken. When the screen is blank, pressing any key will restore the screen.

If you do not wish to use this feature, enter a value of zero to disable it. However, if the display is kept on for long periods of time without the blanker enabled, the image of a screen may become 'burned' into the monitor. That is, you will be able to see this image of the screen on the monitor whether the monitor is on, off, or in some other screen.

Remote Drive & Directory

This field sets up the remapped default drive and directory for the <F3> key in the Load Job screen. This allows you to conveniently load files from an attached computer via LAN network (via RJ-45 Ethernet connection). The Control will usually remap the attached computer's C hard drive as drive E, depending on the way it was set up.

User Specified Paths

Operators can now specify paths for INTERCON files, posted INTERCON files, Digitize files and CAD files. These paths are specified in PATHM.INI. This file is automatically generated by CNC7 if it does not exist. The default PATHM.INI file is:

```
INTERCON_PATH=C:\INTERCON\  
ICN_POST_PATH=C:\CNC7\NCFILES\  
DIGITIZE_PATH=C:\CNC7\NCFILES\  
CAD_PATH=C:\CNC7\NCFILES\
```

Path tag	Purpose of path
INTERCON_PATH	Main directory containing *.ICN files
ICN_POST_PATH	Directory INTERCON places *.CNC files created when posting *.ICN files.
DIGITIZE_PATH	Directory digitize files are saved to. Directory used by F4 key in Load Job menu when parameter 4 is set to 2.
CAD_PATH	Directory for CAD files generated with the DIG->CAD option in the Utility menu.

Machine Configuration

Pressing <F2> from the configuration screen will display the machine configuration screen in the edit window. The machine configuration screen provides you with a method of changing machine dependent data.

If you wish to change the Jog or Motor parameters, press <F1> or <F2> to select the Jog or Motor screens, use the arrow keys to move the cursor and select the desired field. Type the new value and press <ENTER> or press <SPACE> to toggle. When you are done editing, press <F10> to save any changes you have made. If you wish to discard your changes and restore the previous values, press <ESC>. Pressing <ESC> again will return you to the previous screen (Setup).

F1 - Jog Parameters (Values should be recorded on the Control Parameters page at beginning of manual.)

This screen contains jog and feedrate information. See the figure below.

WCS #1 (G54)	Current Position (Inches)	Job Name: C_ROD.CNC
X	+0.0000	Tool: T---H1
Y	+0.0000	Feedrate: 100%
Z	+0.0000	Spindle: 0 A
W	+0.0000	FLOOD M
B	+0.0000	Waiting for PLC operation Stopped

Jog Parameters

Axis	Slow Jog (in/min)	Fast Jog (in/min)	Max Rate (in/min)	Deadstart (in/min)	Delta Umax (in/min)	Travel (-) (Inches)	Travel (+) (Inches)
1	50	170	205	4.0000	5.0000	-30.0000	30.0000
2	50	100	170	5.0000	5.0000	-30.0000	30.0000
3	36	36	36	4.0000	5.0000	-30.0000	30.0000
4	21	36	36	5.0000	5.0000	-999.0000	999.0000
5	0	0	0	0.0000	0.0000	0.0000	10.0000

Save
F10

A description of each of these parameters is listed below.

* NOTE: Some of these values are set automatically by the Autotune option (See PID Configuration below).

Slow Jog: Determines the speed of motion on an axis when slow jog is selected and a jog button is pressed. The slow jog rate cannot be set to a value greater than the maximum rate.

Fast Jog: Determines the speed of motion on an axis when fast jog is selected and a jog button is pressed. The fast jog rate cannot be set to a value greater than the maximum rate.

Max Rate: Determines the maximum feedrate of each individual axis. The feedrate on each axis can never exceed Max Rate, even if the feedrate override knob on the front panel is turned up above 100%. (See also the Machine Parameters section for the "Multi-Axis Max Feedrate" parameter that limits the feedrate along move vectors, not just each individual axis.)

* NOTE: The maximum rate may be set to a smaller value if you wish to run your machine at a slower rate.

Deadstart: Determines the speed to which an axis decelerates before stopping or reversing direction. A low setting will cause a large slowdown before reversals of direction, causing your machine to be more accurate. A high setting will cause less slowdown before reversals, but this may cause your machine to "bang" and you may lose accuracy. This parameter should not be changed.

Delta Vmax: The maximum instantaneous velocity change that will be commanded on a vector transition. This parameter should not be changed.

Travel (-): The maximum distance the axis can travel in the minus direction from the home position. Set this parameter to create a software limit that stops the axis before the fixture or tool collides with the machine.

Travel (+): The maximum distance the axis can travel in the plus direction from the home position. This parameter is especially useful when using a part or fixture larger than the table. Set this parameter to create a software limit that stops the axis before the fixture or part collides with the machine.

F2 - Motor Parameters (Values should be recorded on the Control Parameters page at beginning of manual.)

This screen contains information about the motors, ballscrews, and switches installed on your machine. See the figure below.

WCS #1 (G54)		Current Position (Inches)		Job Name: 2AXTST01.CNC				
X		+11.0863		Tool: T0 H0				
Y		-0.4519		Feedrate: 120%				
Z		+3.5334		Spindle: 0 A				
				Waiting for PLC operation Stopped				
Motor Parameters								
Axis	Label	Motor revs/in	Encoder counts/rev	Lash Comp. (Inches)	Limit - +	Home - +	Dir Rev	Screw Comp
1	X	9.99970	8000	0.00000	0 0	0 0	Y	N
2	Y	10.00120	8000	0.00000	0 0	0 0	N	N
3	Z	5.00050	8000	0.00000	12 11	12 11	N	N
4	hx M	5.00000	8000	0.00000	0 0	0 0	N	N
5	s N	3.09790	8192	0.00000	0 0	0 0	N	N
						Save F10		

A description of each of these parameters is listed below.

* **WARNING:** The Motor Parameters **should not** be changed without contacting your dealer. Corrupt or incorrect values could cause damage to the machine, personal injury, or both.

Special function indicators: These appear, if present, between the axis number and the label. 's' – axis is the spindle, 'p\$' – axis is paired with axis '\$', 'h\$' – axis is a handwheel paired with axis '\$', '*' – pairing conflict. See Machine Parameters for more information on setting up special functions.

Label: The letter you want to use to identify the axis. The first three axes should normally be X, Y, and Z. If a fourth axis is installed, it is usually named W or B. If you change a label, for example from X to A, the controller will then accept G-codes for axis A instead of X.

If fewer than four axes are present, the unused entries should be labeled N. If an axis is manually operated (it has an encoder but no motor), it should be labeled M.

For a manual Z axis, the 3rd axis label should be set to @. This setting allows for two axes posting in Intercon.

* **WARNING:** Intercon does NOT post two axis programs if the 3rd axis is labeled M.

* **NOTE:** Tool length compensation (G43-G44) and canned drilling cycles (G73-G89) always affect the third axis, regardless of its axis label. Tool diameter compensation (G41-G42) always affects the first and second axes, regardless of their axis labels.

Motor revs/unit: The number of revolutions of the motor that results in one unit of measurement of movement. That is, if the machine units of measurement are inches, then Motor revs/inch is the number of revolutions of the motor that results in one inch of movement. Handwheel note: For handwheels, this number is the number of clicks per revolution of the handwheel. If your handwheel has no detents (click positions) use “100”.

Encoder counts/rev: The counts per revolution of the encoders on your servomotors.

Lash compensation: The amount of backlash in the axis. This occurs when the table loses distance due to loose parts during direction reversals. Consult your machine manual or M-Series Service Manual for instructions on measuring backlash.

Limits: The PLC input numbers corresponding to any limit switches that you may have on your machine. Your installer should provide this information. If no limit switch is installed, this field should be set to 0.

Homes: The PLC input numbers of any Home Switches you may have. These are similar to the limit switches. If your machine does not have home switches, this field should be set to the Limit Switch value. If no home or limit switch is installed, this field should be set to 0. You may then use hard stops as homing points if you choose.

* **NOTE:** The Home Switch should never be physically located beyond the Limit Switch.

Direction reversed: Used to match the +/- reference of your machine to the control electronics. Toggle this value if you actually move in the X direction (reverse) when you jog X+.

Screw Compensation: This value indicates whether mapping ballscrew compensation is enabled. When enabled a preset ballscrew map compensates for error along the entire ballscrew. For more information, contact your dealer. It is recommended you enable ballscrew error compensation at all times.

F3 - Find Home

Press <F3> to move an axis to its plus or minus home switch.

F4 - Set Home

Press <F4> to set Machine Home for an axis at its current position. This is usually performed after Find Home. This operation should not be used to set the part zero position. To set the part zero position, use the Part Setup screen.

F5 – Manual Ballscrew Compensation

This option lets you edit the ballscrew compensation tables.

* **WARNING:** The ballscrew compensation tables **should not** be changed without contacting your dealer. Corrupt or incorrect values could adversely affect the accuracy of the positioning of your machine.

Machine Parameters

Pressing <F3> from the configuration screen will display the machine parameters screen. This screen provides you with a method of changing various parameters that are used by the control.

If you wish to change a field, use the arrow keys to move the cursor and select the desired field. Type the new value and press <ENTER>. When you are done editing the fields, press <F10> to accept any changes you have made and save them. Press <ESC> to return to the previous screen (Setup). A short description of the parameter will appear below the table. In the screen below, parameter 6 determines whether an Auto Tool Changer is installed.

Machine Parameters									
0	0.0000	20	65.5000	40	0.0008	60	0.0000	80	0.0000
1	0.0000	21	0.0200	41	0.0394	61	0.5000	81	-1.0000
2	0.0000	22	0.0200	42	54.0000	62	115.0000	82	0.0000
3	0.0000	23	0.0200	43	0.0100	63	1.5000	83	0.0500
4	0.0000	24	0.0200	44	0.0010	64	0.0000	84	3.0000
5	0.0000	25	0.6800	45	0.0000	65	1.0000	85	1.0000
6	0.0000	26	0.6800	46	0.0000	66	1.0000	86	0.0000
7	1.0000	27	0.6800	47	1.0000	67	1.0000	87	48.0000
8	2.0000	28	0.6800	48	0.1000	68	0.0000	88	48.0000
9	0.0000	29	115.0000	49	0.0000	69	1.0000	89	48.0000
10	0.0000	30	115.0000	50	1.0000	70	0.0025	90	48.0000
11	15.0000	31	0.0000	51	0.0000	71	0.0000	91	0.0000
12	10.0000	32	19200.0000	52	0.0030	72	0.0000	92	0.0000
13	0.0500	33	1.0000	53	0.0100	73	0.0500	93	0.0000
14	100.0000	34	8000.0000	54	0.0000	74	4.0000	94	0.0000
15	10.0000	35	4.0000	55	0.0000	75	0.0000	95	2.0000
16	10.0000	36	0.0000	56	32000.0000	76	0.0000	96	2.0000
17	0.0000	37	10.0000	57	32000.0000	77	0.0000	97	45.0000
18	0.0000	38	0.0000	58	32000.0000	78	0.0000	98	5.0000
19	0.0000	39	120.0000	59	32000.0000	79	70.0000	99	2.0000

Auto Tool Changer Installed

Next
Table
F3

Save
F10

<F3> Next Table will toggle the display parameters between parameters 0-99 and parameters 100-199.

* NOTE: Many machine parameters can also be set with the G10 G-code. Some of them are set by the Intercon setup menu.

The following parameters are currently defined:

Parameter	Definition	Default
1	Y jog key orientation	0
2	G-Code Interpretation Control	0
3	Modal Tool and Height Offset Control	0
4	Remote File Loading Flag	0
5	Suppress Machine Home Setup	0
6	Auto Tool Changer Installed	0
7	Display Colors	0
8	Available Coolant System(s)	2
9	Display Language	0
10	Macro M-Function Control	0
11	Touch Probe PLC Input	0
12	Touch Probe Tool Number	0
13	Probing Recovery Distance	0.05
14	Fast Probing Rate	n/a
15	Slow Probing Rate	n/a
16	Probing Search Distance	10
17	Tool Detector Reference Number	0
18	PLC Input Spindle Inhibitor	0
19	MPG modes	0
20	Ambient Temperature	72
21-24	Motor Heating Coefficients	Refer to text
25-28	Motor Cooling Coefficients	Refer to text
29	Warning Temperature	150
30	Limit Temperature	180
31	Spindle Speed Output Port	0
32	Spindle Serial Port Baud Rate	19,200
33	Spindle Motor Gear Ratio	1
34	Spindle Encoder Counts/Rev	8,000
35	Spindle Encoder Input	4
36	Rigid Tapping Enable/Disable	0
37	Spindle Deceleration Time	10
38	Multi-Axis Max Feedrate	0
39	Feedrate Override Knob Limit	120
40	Basic Jog Increment	0.0001"
41	Handwheel 100x Speed, User Jog Increment	0.25"
42	Password for Configuration Menus	0
43	Automatic tool measurement options	0
44	TT1 PLC input #	0
60	Digital Filter Size	1
61	High Power Stall Timeout	0.5
62	High Power Stall PID Limit	115
63	High Power Idle PID Multiplier	1.5
64	Fourth Axis Pairing	0
65-67	Spindle Gear Ratios	1

68	Minimum Rigid Tapping Spindle Speed	100
69	Duration For Minimum Spindle Speed	1.25
70	Offset Library Inc/Decrement Amount	.001"/.02mm
71	Part Setup Detector Height	0
72	Data M-Function Options	0
73	Peck Drill Retract Amount	0.05
74	M-Function executed at bottom of tapping cycle	4
75	Axis Summing Display Control	0
76	Manual Input Unrestricted Distance	0
77	Manual Input Movement Tolerance	0
78	Display of Spindle Speed	0
79	Auto Brake Mode PLC Bit for Uniconsole-2	70
80	Voltage Brake Applied Message Frequency	1
81	Air Drill M-Function (executed instead of Z movement in drilling)	-1
82	Spindle Drift Adjustment	108
83	Deep Hole Clearance Amount	0.05
84	M-Function executed at return to initial point of tapping cycle	3
87-90	Autotune Accel Time and ka.	48
91-94	Axis Properties	0
95-98	Autotune Move Distance	2
99	Cutter Compensation Look-ahead	2
100	Intercon comment generation	0
101	Intercon clearance amount	0.1
102	Intercon spindle coolant delay	3.0
103	Intercon corner federate override	50.0
104	Intercon modal line parameters	0
105	Intercon modal arc parameters	0
106	Intercon modal drilling cycle parameters	0
115	Intercon Help	0
120	Probe stuck clearance amount	0.10"
121	Grid digitize prediction minimum Z pullback	0.002"
122	Grid digitizing deadband move distance	0.0002"
123	Radial digitizing clearance move	0
128	Handwheel MPG mapping	0
129	Handwheel MPG display control	0
130	Z axis on/off selection	0
131	4 th axis on/off selection	0
132	5 th axis heating coefficient	Refer to text
136	5 th axis cooling coefficient	Refer to text
140	Message log priority level	1
141	Maximum message log lines	1000
142	Message log trim amount	1000
143	DRO properties (load meters, 4/5 digits, DTG)	0
144	Comparison rounding	0
145	Advanced macro properties (fast branching)	0
146	Feed hold threshold for feed rate override	0
150	Run-Time Graphics	0

152	5 th axis Autotune accel time and Ka	48
156	5 th axis Autotune move distance	2
160	Enhanced ATC	0
161	ATC Maximum Tool Bins	0
166	5 th axis properties	0
170-179	XPLC parameters	0
180-187	Inverter Parameters	0
188-199	Aux key functions	0

Parameter 1 – Y jog key orientation

This parameter is a 3 bit field where bit 0 is not used in the mill software. Bit 1 sets the direction of movement that the Y+ and Y- jog keys would cause and bit 2 will swap the X and Y jog keys. This should always be set to 0 except for very special applications.

Bit	Function Description	Parameter Value
0	Not Used	
1	Flip movement direction of Y jog keys?	Yes=2, No=0
2	Exchange X axis and Y axis jog keys?	Yes=4, No=0

Parameter 2 - G-code Interpretation Control

This parameter is a 3-bit field that controls optional interpretation of several G-codes. The following table shows the functions performed by the value entered in this parameter:

Bit	Function Description	Parameter Value
0	Arc centers I, J, K are absolute in G90 mode?	Yes = 1 No = 0
1	Allow Z being specified alone to be sufficient to trigger execution of a canned tapping or drilling cycle to be executed?	Yes = 2 No = 0
2	Interpret dwell times associated with G4, G74, G82, G84, and G89 as milliseconds rather than seconds?	Yes = 4 No = 0
3	Slaving rotary axis feedrate to non-rotary axis feedrate	Yes = 8 No = 0
4	Selects the center for scale, mirror and rotate. By default the center will be 0,0,0. Add 16 to this parameter to make the center of scale, mirror and rotate the current position.	Yes = 16 No = 0

Parameter 3 - Modal Tool and Height Offset Control

This parameter controls whether or not the last tool and height offset activated during a job run will remain active after the job is complete. This also controls the Tool status display in the Status Window.

Value	Meaning
0	Tool and Height Offset numbers will be modal and remain active between jobs. Tool Status display will remain active even when job is not running. Tool Status display will show the current T and H number.
1	Tool and Height Offset numbers will be reset to defaults upon job completion. Tool Status display will only be updated during job run.

Tool Status display will only show the current T number.

Parameter 4 - Remote File Loading Flag

This parameter controls the action of the Load Job menu when CNC job files are selected from drives letters higher than C. These drives (i.e. drives D, E, F, etc.) are presumed to be network, Interlink drives or extra hard drives.

Value	Meaning
0	Job files are not copied or cached. They are run from whichever drives they reside on.
1	Job files are copied to the C drive (C:\CNC7\NCFILES) when they are loaded. The local copy is used when the job runs.
2	Turn on file caching. Job files are temporarily cached on the C drive. The cached copy is used while the job is running. The cached copy is deleted when the next job is loaded or when Parameter 4 changes to a 0 or 1. Digitize files are cached as the machine is digitizing. When digitizing is complete, the resulting file is copied to the digitize directory specified in PATHM.INI.

File caching is useful for machines with both a flash card and a hard drive. By caching job files from the hard drive on the flash card, the hard drive is not used while the job is running. As a result, the life of the hard drive is extended and the flash card does not fill up with job files.

Parameter 5 - Suppress Machine Home Setup

This parameter controls machine homing upon startup of the control. The following table details the functions controlled by this parameter:

Bit	Function Description	Parameter Value
0	Suppress the requirement to set machine home before running jobs?	Yes = 1 No = 0
1	Display router bit map at homing screen	Yes = 2, No = 0

Bit 0 suppresses the requirement to set machine home before running. If bit 0 of Parameter 5 is 0, machine home must be set before jobs may be run. If bit 0 of Parameter 5 is 1, machine home is not requested or required.

NOTE: Parameter 5 Bit 0 is separate from the "Machine Home at Powerup" flag in the Control Configuration Screen. Parameter 5 Bit 0 determines **whether** you must home the machine; the "Machine Home at Powerup" flag determines **how** you will home the machine, if you must do so.

Parameter 6 - Automatic Tool Changer Installed

This parameter tells the control whether you have an automatic tool changer installed on your machine. This field affects the action of M6 in your CNC programs. See M6 under M functions in Chapter 13. It also affects whether the ATC key is present in the Tool Offset Setup and whether to save the last tool change number in the job files.

Value	Meaning
0	Auto Tool Changer NOT Installed
1	Auto Tool Changer Installed

Parameter 7 - Display Colors

This parameter determines what combination of colors will be used for display. If you have a color display, set this parameter to 0. If you have a monochrome display (especially a monochrome LCD panel) set this parameter to 1.

Parameter 8 - Available Coolant Systems

This parameter is used by Intercon to determine what coolant systems are available on the machine. It should be set as follows:

Value	Meaning
1	Mist Coolant (M7) only
2	Both coolant systems
3	Flood Coolant (M8) only

Parameter 9 - Display Language

This parameter determines what language will be used for menus, prompts and error messages.

Value	Meaning
0	English
1	Spanish
2	French
3	Traditional Chinese
4	Simplified Chinese

Parameter 10 - Macro M function handling

This parameter is a 4-bit field that controls various aspects of M functions. The following table shows the functions performed by the value entered in this parameter. The default value is 0.

Bit	Function Description	Parameter Value
0	Display M & G codes in M function macros?	Yes = 1, No = 0
1	Step through M function macros in Block Mode?	Yes = 2, No = 0
2	Brushless motor option: Decelerate smoothly to stop (pause) on M105 and M106? (Digitizing and Probing moves.) "Yes" means decelerate smoothly. Choosing "yes" takes more time on each probing move and is slightly less accurate. "No" means hard stop. "No" is faster and slightly more accurate but can cause excessive vibration on brushless systems.	Yes = 4, No = 0
3	Move to Z home on M6?	No = 8, Yes = 0

Parameters 11-17 - Touch Probe Parameters

These parameters control touch probe and tool detector operation. See Chapter 8 for more information.

Parameter 18 - PLC Input Inhibit Parameter (M39 and M400 only)

This parameter stores the input for the control's PLC I/O unit for the Spindle Inhibit feature. A positive value must be entered if a "normally closed" probe is to be used with the control. A negative value must be entered if a "normally open" probe is to be used with the control. The absolute value of Parameter 18 will directly reflect the PLC input the Spindle Inhibit is wired to. When this parameter is set, Digitizing and Probing cycles will not run unless a probe or touch-off block is connected. This parameter is used to prevent the tool or probe from crashing into the table. The default for this parameter is 0, which disables this feature.

Parameter 19 - MPG modes

The MPG is a hand-held device that is used as an alternate way of jogging the machine. This parameter defines the MPG's mode of operation.

Bit	Function Description	Parameter Value
0	Enable MPG when powering up control?	Yes = 1, No = 0
1	MPG speed limit	x100 = 2, x10 = 0

Parameters 20-30 - Motor Temperature Estimation

These parameters are used for motor temperature estimation. Parameters 20, 29 and 30 correspond respectively to the ambient temperature of the shop, the overheat warning temperature, and the job cancellation temperature, all in degrees Fahrenheit. Parameters 21 through 24 are the heating coefficients for each of the four axes. Parameters 25 through 28 are the cooling coefficients for each of the four axes.

The following table contains the default values for parameters 20 through 30:

Parameter Number	Axis	Values	Values	Values	Values	Values
		8A Drive, 15 in/lb motors	12A Drive, 29 in/lb motors	15A Drive, 29 in/lb motors	15A Drive, 40 in/lb motors	25A Drive, 40 in/lb motors
20	N/A	72	72	72	72	72
21	X	0.028	0.02	0.027	0.03	0.04
22	Y	0.028	0.02	0.027	0.03	0.04
23	Z	0.028	0.02	0.027	0.03	0.04
24	4TH	0.028	0.02	0.027	0.03	0.04
25	X	0.68	0.68	0.68	0.68	0.68
26	Y	0.68	0.68	0.68	0.68	0.68
27	Z	0.68	0.68	0.68	0.68	0.68
28	4TH	0.68	0.68	0.68	0.68	0.68
29	ALL	150	150	150	150	150
30	ALL	180	180	180	180	180

Parameter 31 – Spindle Speed Output Port

Parameter 31 determines the destination for the raw spindle speeds generated and output by the Control. Below are the possible values for this parameter. Note that if your machine uses a serial type spindle controller, you should not set this parameter to 0.

Value	Meaning
-1	12 bits to CPU7, then to Koyo PLC & PLCIO2 Direct
0	8 bits to CPU7 controlled spindle and RTK2
1	12 bits to COM1 serial port
2	12 bits to COM2 serial port

Parameter 32 - Spindle Vector Drive Serial Port Baud Rate

The baud rate (e.g. 9600, 19200, etc.) of the serial port at which the control should communicate with the SPIN232 board. This parameter has meaning only if Parameter 31 is set to 1 or 2, for COM1 or COM2 spindle speed output.

Parameter 33 - Spindle Motor Gear Ratio

The gear or belt ratio between the spindle motor and the chuck in high gear range. Should be greater than 1.0 if the motor turns faster than the chuck and less than 1.0 if the chuck turns faster than the motor. Note: this value applies to high range. The ratio between high range and lower ranges is established by the gear ratio parameters (65-67).

Parameter 34 - Spindle Encoder Counts/Rev

This parameter controls the counts/revolution for the spindle encoder. If the encoder counts up when running CW (M3), the value of this parameter must be positive. If the encoder counts up when running CCW (M4), the value of this parameter must be negative.

Parameter 35 – Spindle Encoder Input

This parameter specifies the axis input to which the spindle encoder is connected. Input from the spindle encoder is required for the spindle-slaved movements used in the Rigid Tapping cycles. So, if Rigid Tapping is used, this parameter must be set to the correct value. Otherwise, this value is generally ignored. A value of 2 means the third encoder input; a value of 3 means the fourth encoder input; a value of 4 means the fifth encoder input.

Parameter 36 - Rigid Tapping Enable/Disable

This parameter is a 3-bit field that enables or disables Rigid Tapping and its options. Bit 1 and 2 have no meaning unless bit 0 is turned on.

Bit	Function Description	Parameter Value
0	Enable Rigid Tapping?	Yes = 1, No = 0
1	Suppress sending "Wait for Index Pulse" during Rigid Tapping?	Yes = 2, No = 0
2	Allow Spindle Override during Rigid Tapping?	Yes = 4, No = 0

Parameter 37 - Spindle Deceleration Time

This parameter is used in conjunction with parameter 36 when rigid tapping is enabled. This sets the amount of time required for the spindle to decelerate before it switches direction during a rigid tapping operation.

Parameter 38 - Multi-Axis Max Feedrate

This parameter is used to limit the feedrate along all commanded move vectors. This parameter can be used to limit the speed of multi-axis moves on machines that may have enough power to move a single axis rapidly, but starve out of power on 2 or 3 axis rapid moves. A zero in this parameter will disable this feature.

Parameter 39 - Feedrate Override Percentage Limit

This parameter is used for limiting the upper end of the Feedrate Override Knob percentage to a value from 100% to 200%. This parameter can be used to restrict the Feedrate Override Knob effect on machines with maximum rates over 200 in/min. The Feedrate Override Knob percentage is normally allowed to go to 200%. However, on machines with high cutting speeds, if the knob is turned up to 200%, it creates overshoots on corners. If this parameter is set to something like 110, it will stop the Feedrate Override Knob from exceeding 110% and thus causes the overshoots to disappear.

Parameter 40 - Basic Jog Increment

This parameter holds the basic jog increment (0.0001" or 0.002 mm by default). This value is used by the x1, x10, and x100 jog keys (0.0001, 0.001 and 0.01 on older consoles). It also specifies the distance per click for handwheels (MPG).

Parameter 41 - Handwheel 100x Speed, User Jog Increment

On newer consoles, this parameter holds the actual handwheel speed in 100x mode. For normal 100x operation it should be 100. On some systems 100x is way too fast and this value is set to a more reasonable value such as 20 or 30.

On older consoles, this parameter holds the user jog increment (0.250" or 1.0 mm by default). The 0.250 jog key on older consoles uses this value.

Parameter 42 – Password for Configuration Menus

This parameter determines the password that the user must enter in order to gain supervisor access to the configuration menus.

Value	Meaning
54.0	No password required for supervisor access; the user is not prompted for a password
ABCD.ABCD	Password is 4 digits represented by "ABCD"
Any other number	Password is "137"

Parameter 43 – Automatic tool measurement options

If this parameter is set to 1, the height of the tool detector (parameter 71) will be subtracted from the measured height of the tool.

Parameter 44 – TT1 PLC input number

PLC input number that the TT1 is wired into on the PLC. If a shared PLC input is used for the TT1 and the DP4 probe, then the value can be left at zero or set to the same value as parameter #11.

Warning: If using a different PLC input for the TT1 and DP4, when setting the Z reference in the tool library with the DP4 make sure not to use a ruby probe tip. The TT1 is continuity based and the ruby tip is not conductive!

Parameter 60 - Digital Filter Size

This parameter defines the PID output filter size for the motor outputs. This parameter is meant to provide a software filter where no hardware filter exists in order to slow down the PID output frequency (normally 4000 times/sec.), or to supplement a hardware filter that appears to be inadequate. It is the number of samples to average the PID output over. For example, a value of 2 says to average the PID output over 2 samples, which would reduce the PID output frequency to 2000 (4000/2) times/sec. The default value of this parameter is 1 (no averaging).

Parameters 61-62 - Stall Detection Parameters

The M-Series control will detect and report several stall conditions. The low power stall occurs if the control has been applying a specified minimum current for a specified time, and no encoder motion has been detected. This may indicate a loose or severed encoder cable. A high power stall occurs if the control has been applying at least 90% current for a specified time, and no motion greater than 0.0005" has been detected. This may indicate a physical obstruction.

Parameter 61 is the time limit, in seconds, for a high power stall. The default is 0.5 seconds.

Parameter 62 is the PID output threshold for a high power stall. The default is 115.

Parameter 63 - High Power Idle PID Multiplier

This parameter holds the value of a constant used for motor temperature estimation when an axis is not moving and no job is running, but there is power going into the motor to maintain its position. The default value is 1.5. This temperature estimation is intended to detect early if an axis is stopped against some abnormal resistance, such that it will probably overheat later.

Parameter 64 - Fourth Axis Pairing

This feature enables the 4th axis motor to be run in a paired fashion with any of the other 3 axes. This parameter is set to 1, 2, or 3 to indicate that the X, Y, or Z-axes are paired with the 4th axis. This is intended to drive 2 screws on opposite sides of a table (probably a router table or gantry system). Set this parameter to 0 (default) to indicate that no other axis is paired with the 4th axis.

Value	Meaning
0	No Pairing (Default)
1	Pair with X Axis
2	Pair with Y Axis
3	Pair with Z Axis

Parameters 65-67 - Spindle Gear Ratios

These parameters tell the control the gear ratios for a multi-range spindle drive. Up to four speed ranges are supported; high range is the default. Parameters 65-67 specify the gear ratio for each lower range, relative to high range. For example, if the machine is a mill with a dual range spindle, and the spindle in low range turns 1/10 the speed it turns in high range, then parameter 65 should be set to 0.1.

Parameter 65 is the low range gear ratio.

Parameter 66 is the medium-low range gear ratio.

Parameter 67 is the medium-high range gear ratio.

These parameters work in conjunction with the PLC program, which uses the states of INP63 and INP64 to signal to the CNC7 software which range is in effect, according to the table below.

PLC INPUT	Spindle Range			
	High Range	Medium High Range	Medium Low Range	Low Range
INP63	0	1	1	0
INP64	0	0	1	1

Parameter 68 – Minimum Rigid Tapping Spindle Speed

This parameter holds the value that the spindle slows down to from the programmed spindle speed towards the end of the tapping cycle. The lower the value, the more accurately the Z axis will land on target, but at the expense of possibly stalling the spindle motor which in turn will cause Z to fall short. If this value is too large, the off-target error increases. The suggested starting value is 100 RPM.

Parameter 69 – Duration for Minimum Spindle Speed Mode

This is the duration of time, in seconds, that the control will stay at minimum spindle speed. If the number is too small, overshoot will occur. If the number is too large, the user waits longer for the hole to be tapped at the slow speed specified by parameter 68. The suggested starting value is 1.25 seconds.

Parameter 70 - Offset Library Inc/Decrement Amount

Sets the increment and decrement amount used in the offset library.

Parameter 71 – Part Setup Detector Height

If this Parameter is set to a non-zero value, it indicates that the F3/Auto feature in part setup should be available using the tool detector (TT1) instead of the probe. The value in this parameter is the height of the detector. A value of 0 disables this feature.

When this feature is enabled:

- Probe detection (Parameter 18) is not checked
- The tool number and/or edge finder diameter entered by the operator are used; Parameter 12 is ignored.
- The value from Parameter 71 is added to (or subtracted from, depending on approach direction) the part position.

Parameter 72 – Data M Function Options

The setting of this parameter affects the operation of the data M functions M122 and M123.

Bit	Function Description	Parameter Value
0	Suppress output of axis labels by M122?	Yes = 1, No = 0
1	Insert commas between positions/values with M122 and M123?	Yes = 2, No = 0
2	Suppress spaces between positions/values outputted by M122 and M123?	Yes = 4, No = 0

Parameters 73, 74 - Canned Cycle Parameters

P74 specifies the number of the M-function that is executed at the bottom of the G74 or G84 tapping cycle.
P73 specifies the retract amount used during a G73 peck drilling cycle.

Parameter 75 – Summing Display Control

This parameter indicates which axes are to be summed and how the results are to be displayed on the DRO. The parameter can contain up to four digits. The position and value of each digit has special significance as indicated in the tables below:

Parameter Digit Position	Axis Display Controlled	Digit Value	Meaning
		0	Summing off
1's Column	Axis 1	1 – 4	Axis to Sum
10's Column	Axis 2	5	(reserved)
100's Column	Axis 3	6	Disable display
1,000's Column	Axis 4	7	Display if moved
		8	Display if other moves
		9	(reserved)

Here are some examples using the axis summing display parameter:

Desired Display	Parameter
Sum Z axis (3) with M (4), display sum in Z DRO position	400
Sum Z axis (3) with M (4), display sum in M DRO position.	3000
Sum Z axis (3) with M (4), display sum in Z DRO position, and suppress M display.	6400
Sum Z axis (3) with M (4), display sum in M DRO position, and suppress Z display.	3600
Sum Z axis (3) with M (4), display sum in Z DRO position, and show M only if M moves.	6300
Sum Z axis (3) with M (4), display sum in Z DRO position, show M if either Z moves.	7400

The DRO will display both labels when displaying a summed axis. For example, "ZM" or "MZ" depending on where the sum is displayed.

Parameter 76 – Manual Input Unrestricted Distance

This parameter is intended to be used with Z axis summing. It defines the maximum distance from the summed axis start of travel in which manual movements can occur without causing a fault. Use a negative value to specify a distance from the minus travel limit, a positive value for a distance from the plus travel limit.

When used with manual drilling, for example, setting this parameter will allow the operator to keep a hand on the quill at all times and even begin pulling on the quill in anticipation of a programmed stop.

Setting this value to zero will cause a fault if there is any manual movement.

To completely disable manual movement restrictions, set this parameter to a value exceeding the total travel of the summed axis.

Minimum = -99999.9999, maximum = 9999.9999, default = 0, typical = +/- 1.0 inch or +/- 20.0 mm

Parameter 77 – Manual Input Movement Tolerance

This parameter specifies the manual movement tolerance while a job is running. It is intended for use with a quill locking mechanism. It allows the lock to distort and/or slip a small amount when under stress. If the quill moves more than the given tolerance, the job will stop with a fault. A typical setting for Parameter 77 is 0.005 inches.

Parameter 78 – Display of Spindle Speed

This parameter specifies how the spindle speed is determined and displayed in the CNC7 status window. When set to 1.0, the spindle speed is determined from reading the encoder feedback from the axis specified according to parameter 35, which has the number of encoder counts/revolution specified in parameter 34. When set to 0.0, the displayed speed is not measured- the speed is calculated based upon the set speed, spindle override adjustment, and gear range.

Parameter 79 – Auto Brake Mode PLC Bit for Uniconsole-2

This parameter specifies which PLC bit signals the state of automatic brake mode when using the Uniconsole-2 console type. For other console types, it has no effect. This parameter can be changed to allow the Auto Brake mode key to be located in different positions on the Uniconsole-2 jog panel. The PLC program must be updated to reflect any change in this parameter.

Parameter 80 – Voltage Brake Message Frequency

This parameter specifies the number of time the “450 Voltage brake applied message has to occur before we show it in the message window and message log. A value of 0 or 1 will display the message for every instance that it occurs.

Parameter 81 – Canned Cycle Parameter

P81 (when not equal to -1.0) specifies the M-function to be called in place of Z axis movement during a G81 drilling cycle.

Parameter 82 – Spindle Drift Adjustment

This value is the number of degrees that the spindle will take to coast to a stop if it is cut off while it is spinning at the spindle speed specified by parameter 68.

Parameter 83 – Canned Cycle Parameter

P83 specifies the clearance amount used during a G82 deep hole drilling cycle.

Parameter 84 – Canned Cycle Parameter

P84 specifies the number of the M-function that is executed after the return to the initial point of a G74 or G84 tapping cycle.

Parameters 87-90 - Autotune Accel Time and Ka

These parameters are used by autotune. Increasing the value will lengthen acceleration time and reduce the ka value given by autotune. Lowering the value will decrease the acceleration time and increase Ka. First set the parameters and then run autotune. The default value is 48. The maximum value is 64 and the minimum value is 1.

Parameters 91-94 – Axis Properties

These parameters may be used to set various axis properties. These parameters correspond to X, Y, Z and the fourth axis, respectively.

Bit	Function Description	Parameter Value
0	Rotary/Linear Axis Selection	Rotary Axis= 1, Linear Axis= 0
1	Rotary Display Mode	Wrap Around = 2, Show Rotations = 0
2	Suppress direction check when doing Tool Check?	Don't Check = 4, Check= 0
3	Suppress park function?	Don't Park = 8, Park = 0
4	NOT USED ON MILL	
5	Linear Display of Rotary Axis	Linear Display = 32, Default Rotary = 0

Notes on Bit 0: Turning this bit on will cause the DRO display for the affected axis to be displayed in degrees. Also this information is used by Intercon to make rotary axis support available (by setting parameter 94 to 1, indicating that the fourth axis is rotary). This bit is also used when performing inch/mm conversions: values for a rotary axis will not be converted since they are assumed to be in degrees regardless of the system of linear units.

Notes on Bit 1: This bit has no effect unless Bit 0 (mentioned above) is turned on. When this bit is turned on, a “Wrap Around” display is shown on the DRO. A “Wrap Around” Rotary Display is a display in degrees without the number of rotations shown. If this bit is turned off, the number of rotations away from 0 degrees will be shown alongside the degree display.

Notes on Bit 2: This bit will only affect the Z axis. It controls whether or not a direction check will be performed when the Tool Check button is pressed. If this bit is turned on, direction checking is turned off, and thus, there is a possibility for the Z axis to move downward unexpectedly, depending on the Z value of Return Point #1 (G28). Therefore, it is best in most cases to leave this bit turned off to allow direction checking to be turned on (value = 0).

Notes on Bit 5: This setting overrides only the DRO display options for an axis that has bit 0 set (including the Rotary Display Mode – bit 1) so that the display does not reflect a degree symbol or any indication of the number of rotations, but appears as a linear axis.

Parameters 95-98 - Autotune Move Distance

These parameters hold the maximum distance that the control will move each axis in either direction from the starting point when Autotune is executed. The default value for these parameters is 2.0 inches.

Parameter 99 – Cutter Compensation Look-ahead

This parameter sets the default number of line or arc events for the G-code interpreter to scan ahead when Cutter Compensation (G41 or G42) is active. Values of 1 to 10 are allowed for this parameter.

Parameters 100–115 - Intercon parameters

These parameters are some of the Intercon setup parameters. See Chapter 10 for more information about these parameters. Changing values will change Intercon settings and may effect the output of the G-code program if it is re-posted.

Parameter 120 – Probe Stuck Clearance Amount

This parameter specifies the distance that digitizing or probing functions will move to try to clear a stuck probe condition. A stuck probe condition exists when the probe detects a point and then moves away but the probe input has not changed. It is recommended that this parameter is not changed from its default value without consulting a qualified technician.

Parameter 121 – Grid digitize prediction minimum Z pullback

This parameter specifies the minimum distance the Z axis will move upward when pulling back from a surface. The digitizing function attempts to predict the slope of a part surface because time is saved when the Z axis does not have to travel upward to the starting Z depth for every digitized point. When probe contact is made traversing in the XY plane, this parameter specifies the minimum distance the Z axis moves upward before attempting another XY plane move. Smaller values are better when the surface being digitized has smooth curves. Larger values are better for surfaces that have steep walls. It is recommended that this parameter is not changed from its default value without consulting a qualified technician.

Parameter 122 – Grid digitizing deadband move distance

This parameter specifies a deadband distance used for internal calculations when doing a clearance move. It is recommended that this parameter is not changed from its default value.

Parameter 123 – Radial Clearance Move

This parameter only applies to radial digitizing and determines what type of positioning move the digitizing probe will make should it encounter an unexpected probe contact with the surface of the part during Radial Digitizing. Unexpected probe contact is defined as probe contact occurring while the probe is traversing towards the user defined center point.

With Parameter 123 set to 0: When the probe encounters an unexpected probe contact, the digitizing program stops data collection. The control then prompts the operator to jog the probe to a clear position. This can be any place inside the digitizing radius and above the part, that the probe stylus has a clear path to the defined center position. To restart data collection press “Cycle Start”. The probe moves in the XY plane from the position the operator placed it to the center position defined in the radial setup menu. After reaching the center position, the probe will feed down to the Z axis position it was at when the data collection was interrupted. The digitizing run will resume with the probe approaching from the defined center position.

With Parameter 123 set to 1: When the probe encounters an unexpected probe contact, it will automatically move (with probe detection turned off) to the maximum Z height, then moves the X and Y-axis to the defined center position. The probe will then move to the Z position it was at when the unexpected contact occurred. It will Then move from the defined center position, towards the measurement position it was trying to approach when the unexpected probe contact occurred and continue digitizing.

With Parameter 123 set to 2: When the probe encounters an unexpected probe contact, it will automatically move back to the defined center position (with probe detection turned off), at its present Z height. It will then move from the defined center position, towards the measurement position it was trying to approach when the unexpected probe contact occurred and continue digitizing.

Settings 1 and 2 should only be used with extreme caution because probe detection during some positioning moves is turned off, and damage to the probe or work piece could occur!

Parameter 128 – Handwheel (MPG) Mapping

This parameter selects how the axes are paired for handwheel operation. Each digit in the displayed number represents an axis. The first axis is at the far right. The value of each digit represents the companion axis, 1 to 5. A zero digit means no pairing. The table below shows how the digits are mapped to axes:

Axis:	5	.	4	3	2	1
Parameter value	0	.	0	0	0	0

Example Value	Axis/Companion					Comments
	5	4	3	2	1	
0.0000						No pairing.
0.1000		1			4	Axes 1 & 4 paired.
0.0043				4	3	Axes 1 & 3, 2 & 4 paired.
0.2100		2	1			Axes 1 & 3, 2 & 4 paired.
0.0021				2	1	Invalid – does nothing. Axes are paired with themselves.

Only manual axes that are paired with powered axes will produce a valid configuration. Manual axes specified by Parameter 128 must be properly configured as handwheel axes in the Motor Parameters screen of the Machine Configuration. See the Machine Configuration section earlier in this chapter.

Parameter 129 – Handwheel (MPG) Display

By default, manual axes paired by Parameter 128 are not displayed in the DRO. This parameter can force display of the manual axis in the DRO, if desired. The parameter has the same axis mapping for each digit as shown in Parameter 128. To display an otherwise hidden manual axis, set the digit corresponding to the axis number to a “1”. For example, “0.1000” would display axis 4, if it is a manual axis that is paired with some other powered axis.

Parameter 130 – Z axis on/off selection

Parameter 131 – 4th axis on/off selection (only uses 1’s and 10’s digit)

These parameters control the display of the 3rd and 4th axes, respectively. The tens digit of the parameter value specifies the label of the affected axis when it is enabled, with values 1-9 corresponding to axis labels ABCUVWXYZ. The ones digit specifies the label of the axis when it is disabled, with 0.0 meaning the axis is not switchable, 1.0 meaning it turns off (N), a 2.0 meaning manual (M), and a 3.0 meaning 2-axis Z (@). P130 also supports additional modes depending upon the value of the hundreds digit. For example, a value of 92 will toggle between a 3rd axis Z and a 3rd axis M and power off just the Z axis. A value of 192 will toggle the 3rd axis between Z and M and power off all axes. A value of 392 will toggle the 3rd axis label between Z and M and power off all axes and receive its positions from the 4th axis encoder input. When P130/P131 is configured for axis switching, the Setup menu displays function keys <F5>/<F6> to switch the axes.

Parameter 130 Options:

Hundred's Digit	Function Description	Parameter Value (add
Bit1	Axis motor power when switching to two axis mode.	1 = power all axes off, 0 = power 3rd off only
Bit2	Use 4 th encoder input for scale input	2 = Use 4 th encoder input, 0 = no manual input
Bit3	Use 5 th encoder input for scale input	4 = Use 5 th encoder input, 0 = no manual input

Enabled Axis:	A	B	C	U	V	W	X	Y	Z
Ten's Digit	1	2	3	4	4	6	7	8	9

Disabled Axis:	N	M	@
One's Digit	1	2	3

Parameters 132 – 5th Axis Heating Coefficient

This parameter sets the heating coefficient for the 5th axis. See parameters 20-30 for more information.

Parameters 136 – 5th Axis Cooling Coefficient

This parameter sets the cooling coefficient for the 5th axis. See parameters 20-30 for more information.

Parameter 140 – Message log priority level

This parameter controls the messages that are written to the message log, which can be accessed through the <F9> Logs function in the Utilities menu. With the Log Level set to 1, CNC7 logs numbered error messages and most other messages except "Moving...", "Jogging...", "Stopped", etc. At Log Level = 9, all messages are logged including user prompts. Message logging can be disabled by setting this parameter to -1.

Parameter 141 – Maximum message log lines

This parameter is the number of lines that will be kept in the message log. If this parameter is set to 10,000, for example, the newest 10,000 messages will be retained. CNC7 will delete the oldest messages, trimming the log file to the given number of lines at startup and periodically while CNC7 is in an idle state. Parameter 142 controls the frequency of the log cleanup.

Parameter 142 – Message log trim amount

This parameter is the number of additional lines above the minimum that can be added to the log before it is reduced to the minimum size. Setting this parameter to a lower value will cause the log file to be trimmed to its minimum size more often. The higher the value, the less often the log will be trimmed. The speed of the disk drive and total size of the log file at the time it is trimmed will determine how long the log cleanup takes. Under most circumstances, using 10,000 and 1,000 for parameters 141 and 142 will provide a reasonable and useful log size with no noticeable effects on performance. If parameters 141 and 142 are set to excessively high values, the message "Trimming excess lines from log file" will be presented. This message will appear at startup and very infrequently when CNC7 is idle. Normal operation can proceed after the message disappears. If the delay is unacceptable, reduce the values of parameters 141 and 142.

Parameter 143 – DRO Properties (load meters, 4/5 digits, DTG)

This parameter controls the display of the axis load meters and 4/5 digit DRO precision.

Bit	Function Description	Parameter Value
0	Enable Load Meters	Enable = 1, Disable = 0
1	Load Meter Outline	Enable = 2, Disable = 0
2	DRO 4/5 Digit Precision	4 digits = 4, 5 digits = 0
3	Mini DRO (Distance to Go)	Enable = 1, Disable = 0

Use a value of 3 to display load meters with outlines. The axis load meters will be colored green for values that are up to 70% of maximum power output, yellow for values between 70% and 90%, and red for values between 90% and 100%. The axis load meters appear below the DRO for each axis (see Chapter 1).

Parameter 144 – Comparison Rounding

This parameter determines the built in rounding for the comparison operators ('EQ', 'NE', 'LT', 'GT', etc.) in expressions. Rounding of comparison arguments is necessary due to extremely small errors that are part of every floating point calculation. The result of such errors is that two floating point values are rarely exactly equal. The value of parameter 144 represents the precision of comparison in places after the decimal point. If the parameter is set to 9.0, for example, then comparison operators will declare two numbers that differ in value by less than 0.0000000005 as being equal. The value 0.0 is a special value that turns comparison rounding off. When comparison rounding is off, it is up to the G code programmer to build the precision into conditional statements, for example "IF ABS[#A - #B] LT 0.00005 THEN GOTO 100". When comparison rounding is off, the "EQ" usually returns "false". If parameter 144 is set to 9, the programmer can shorten the previous example to "IF #A EQ #B THEN GOTO 100".

Parameter 145 – Advanced Macro Properties (Fast Branching)

This parameter turns fast branching on (1) and off (0). The other bits of this parameter are reserved for future use.

If fast branching is disabled, CNC7 searches forward in the program for the first matching block number and resumes searching, if necessary, from the top of the program. For this reason, backward branches take longer than forward branches and backward branch times depend on the total program size. If the program is sufficiently large, use of the GOTO statement could introduce temporary pauses.

When fast branching is enabled, CNC7 remembers the locations of block numbers as it finds them during program execution. Backward branches always take place immediately. The first forward branch to a block not yet encountered will take additional time as CNC7 searches forward for the block number; however, subsequent forward branches to that block number will take place immediately. The trade-off for using fast branching is that all line numbers at a given level of program or subprogram must be unique and programs will use more memory (approximately 16kilobytes of memory for every 1000 block numbers in the program.)

Parameter 146 – Feed Hold Threshold for Feed Rate Override

This parameter sets the lowest value permitted as the feed rate override percentage before feed hold is engaged. Feed hold will be released when the override percentage is greater than this value.

Parameter 150 – Run-Time Graphics

This parameter controls the default value of the Run-Time Graphics option in the Run Menu. If this parameter is set to 0.0, the RTG option in the Run Menu defaults to OFF when CNC7 is started. If the parameter is set to 1.0, the RTG option defaults to ON when CNC7 is started.

Parameters 152 – 5th Axis Autotune Accel Time and Ka

This parameter sets the autotune accel time and Ka for the 5th axis. See parameters 87-90 for more information.

Parameters 156 – 5th Axis Autotune Move Distance

This parameter sets the autotune move distance for the 5th axis. See parameters 95 – 98 for more information.

Parameter 160 – Enhanced ATC

This parameter controls enhanced automatic tool changer (ATC) options. A value of 1 indicates a nonrandom type of ATC and a value of 2 indicates a random type ATC. A value of 0 disables enhanced ATC features. A warning is displayed when attempting to enable enhanced ATC features as these features work in conjunction with specific PLC programs. The enhanced ATC option has the following characteristics:

- (1) The beginning of an M6 call, whether it be a customized CNC7.M6 routine or not, flags the job file, setting the ATC error flag field to 1.
- (2) The end of an M6, whether customized or not, performs the following:
 - (a) The atc error flag is set to zero.
 - (b) The tool number displayed on the screen is updated and this value is saved in the CNC7.JOB file.
 - (c) The tool library bin fields are updated in this manner:
 - If there was a valid tool in the spindle at the start of the M6, then the tool library bin field for this tool will be updated with either the “putback” field for that tool (if nonrandom type) or the current ATC carousel position (for random type). For both random and nonrandom types, the “putback” field is set to 0. The “putback” field is an internal field for each tool in the tool library. It can be displayed by using the CNC7CONV utility with the –dt option to display the tool library.
 - For nonrandom types, the new tool now in the spindle will have it’s “putback” field updated to the current ATC carousel position.
 - For both random and nonrandom types, the new tool now in the spindle has the bin field set to 0.
- (3) The current ATC carousel position is constantly monitored. When there is a change, the ATC bin field in the CNC7.JOB file is updated and the file is saved. The ATC carousel position is read from PLC bits OUT41-OUT48, which should be written by the PLC program in a binary format (not BCD).
- (4) At the start of running a job, to include MDI mode, the ATC error field is checked. If this field is 1, then a warning message is displayed with a prompt to either clear the fault by entering a ‘Y’ or canceling the job by pressing some other key.
- (5) A tool change is not performed if the requested tool is already in the spindle.
- (6) An M107 command sends the bin number for the specified tool number, not the tool number.

- (7) For random types, tool changes in Intercon are posted as a tool change (Tnn M6) followed by a pre-fetch command for the next tool in the program (Tn2 M107). This allows the PLC program to rotate the tool carousel to the next tool while a job continues with the current tool.
- (8) For random types, a job search for a tool number will look for lines of the form Tnn M6, i.e., the search bypasses lines of the form Tnn M107, which are just pre-fetch commands.
- (9) The tool library allows editing of the bin fields to specify which carousel bin number the tools are stored in.

Parameter 161 –ATC Maximum Tool Bins

This parameter sets the number of tool changer bins (carousel positions) used with the enhanced ATC option described above. PLC programs are responsible for reading this value. The tool library interface uses this parameter to validate bin fields and perform initialization of the bin fields.

Parameters 166 – 5th Axis Properties

This parameter sets the axis properties for the 5th axis. See parameters 91-94 for more information.

Parameters 170-179 – XPLC Parameters

These parameters are accessed by the XPLC through LP0 - LP9 commands. Please see the Service and Installation manual for more information regarding these parameters.

Parameters 180-187 – Inverter Parameters

These parameters describe various properties of the inverter.

Parameters 188-199 – Aux Key Functions

These parameters are used to assign a function to aux keys 1-12. The following is the list of possible functions that can be executed when an aux key is pressed.

Function	Parameter Value
No Function	0
Input X Axis Position	1
Input Y Axis Position	2
Input Z Axis Position	3
Set Absolute Zero	4
Set Incremental Zero	5
One Shot - Drill	6
One Shot - Circular Pocket	7
One Shot - Rectangular Frame	8
One Shot - Frame	9
One Shot - Face	10
Execute M Code file	<i>m11*</i>
Free Axes	14
Power Axes	15

Function	Parameter Value
XYZ Set Absolute Zero	16
One Shot - Drill Bolt Hole Circle	17
One Shot - Drill Array	18
Jog Axis 1 (+)	21
Jog Axis 2 (+)	22
Jog Axis 3 (+)	23
Jog Axis 4 (+)	24
Jog Axis 5 (+)	25
Jog Axis 1 (-)	31
Jog Axis 2 (-)	32
Jog Axis 3 (-)	33
Jog Axis 4 (-)	34
Jog Axis 5 (-)	35

The Input Axis Position functions must be used with the Set ABS/INC Zero functions. After entering the desired value at the input field provided by the Input Axis Position function, press an aux key assigned either the function Set ABS Zero or Set INC Zero.

* *m* is the number of the M code to execute. For example, if the parameter value is set to 7211, the file CNC7.M72 will be executed when the Aux key was pressed,

All remaining parameters are reserved for further expansion.

PID Configuration

Pressing <F4> from the Configuration screen will bring up the PID Configuration screen. The PID Configuration screen provides qualified technicians with a method of changing the PID dependent data to test and configure your machine. The PID Parameters **should not** be changed without contacting your dealer. Corrupt or incorrect values could cause damage to the machine, personal injury, or both.

WCS #1 (G54)	Current Position (Inches)	Job Name: TEST . CNC							
X	-3.16089	Tool: T1 H---							
Y	-2.23898	Feedrate: 105%							
Z	+0.00000	Spindle: 1506 M							
B	+0.00000								
<div style="border: 1px solid black; padding: 5px;"> Processing... Stopped Processing... Stopped Press CYCLE START to start job </div>									
PID Configuration									
Axis	Kp	Ki	Kd	Limit	Kg	Kv1	Ka	Accel.	Max Rate
X	1.000	0.00391	15.000	32000	0	0	0	0.500	300.0
Y	1.000	0.00391	15.000	32000	0	0	0	0.500	300.0
Z	1.000	0.00391	15.000	32000	0	0	0	0.500	300.0
B	1.000	0.00391	15.000	32000	0	0	0	0.500	300.0
N	1.000	0.00391	15.000	32000	0	0	0	0.500	300.0
Axis	Error	Sum	Delta	PID Out	Abs Pos	Line	PID Collection Program		
X	0	0	0	OFF	-2	1			
Y	0	0	0	OFF	0	2			
Z	0	0	0	OFF	0	3			
B	0	0	0	OFF	0	4			
N	0	0	0	OFF	0	5			
PID Collection Axis: X		Density: 1		Type (0-4): 0		File:			
PID	Prog.	Collect	Tune		Drag	Laser	Plot		
F1	F2	F3	F5		F6	F7	F9		

F1 - PID Parameters

(These parameter values should be recorded on the Control Parameters page at beginning of manual.)

This option is for qualified technicians **only**. Altering these values will cause **DRAMATIC** changes in the way the servo system operates, leading to possible machine damage. **DO NOT** attempt to change these parameters without contacting your dealer.

* NOTE: Some of these values are set automatically by the Autotune option. (See F5 – Autotune)

The parameters Kp, Ki, Kd, Limit, Kg, Kv1, and Ka at the top of the edit window are values used by the PID control algorithm. These parameters should not be changed at any time. The remaining two PID parameters are acceleration time and maximum rate. These parameters are described below.

Accel: (Acceleration Time) the time required for an axis to accelerate to its maximum rate. Although each axis has its own acceleration time, the actual acceleration time used during a job will be the slowest time of all the axes.

DO NOT change this field unless you have a thorough understanding of its operation.

Max Rate: See section Machine Configuration: Jog Parameters above.

WARNING: Improper PID values can ruin the machine, cause personal injury, and/or destroy the motor drives!!!

F2 - PID Collection Program

This option allows qualified technicians to test the PID parameters by entering up to 5 lines of G-codes to be executed with the Collect Data command below.

F3 - Collect Data

This option allows qualified technicians to collect data on the movement of one of the motors. It uses the values located in the axis and density fields at the bottom of the screen and the PID collection program to collect the data. When this option is selected, the controller executes the PID collection program and collects data on the selected axis. The data is saved using the file name entered at the file prompt at the lower right hand side of the screen. The information in the lower left hand side of the edit window provides information to qualified technicians about the selected axis.

F5 - Autotune

This option is used by qualified technicians to automatically determine values for Max Rate, Accel/decel time, and Deadstart (See section Motor Configuration: Jog Parameters) as well as the PID parameters for each installed axis. The Autotune procedure will make a series of moves on each axis, traveling up to 2" (see parameters 95-98) from the initial position in all directions to determine the friction and gravity of each axis. The initial high-speed move will use half of this distance. This will allow Autotune to work on axes with less than 4" of travel, on rotary axes that need more than 1 degree to get up to speed, and on very fast/slow accelerating machines that need more than 1 inch to get up to speed. (In order to use less than 4", or more than 4 degrees, you must change the corresponding parameter.)

* NOTE: Do not run Autotune unless requested to do so by a qualified technician.

F6 - Drag

This option is used by qualified technicians to determine whether your machine is binding anywhere along the axis travel. Press <F6> to begin the drag test. Press <F1> to select the axis you wish to check. Hit the CYCLE START button. A text file DRAG_X.OUT, DRAG_Y.OUT, DRAG_Z.OUT, or DRAG_W.OUT file is generated and stored in the C:\CNC7 directory. If significant drag occurs, a message will be displayed on-screen. Contact your dealer to correct the problem as soon as possible.

F7 - Laser

This option is used by qualified technicians to take automated laser measurements and create or adjust the ballscrew compensation tables using accordingly. Do not attempt to run automatic laser compensation without first contacting your dealer for details.

F9 - Plot

This option is used by qualified technicians to plot data.

Handwheel Configuration

If you are using a manual input as a handwheel (MPG) input, be sure to configure all handwheel/MPG parameters. This list serves as a guide to configuration of the handwheels. Motor Parameters do not apply to MPG's that use the special MPG input. You may configure any unused encoder input as a handwheel input.

Screen	Parameter	Value	Comments
Jog Parameters	Travel (-), Travel (+) for an axis controlled by a handwheel.	Actual travel limits of the powered axis.	Axis controlled by a handwheel must have travel limits set.
Motor Parameters	Label	M	Handwheel input must be a manual axis.
Motor Parameters	Motor Revs/Unit	Number of "clicks" per rev.	If the wheel has no detents, use 100.
Motor Parameters	Encoder Counts/Rev	Actual number of counts generated per rotation of the handwheel.	Use higher resolution encoders for smoother operation.
Motor Parameters	Lash, Limits, Homes	0, 0, 0	Do not apply to handwheels.
Motor Parameters	Direction reversed, Screw Compensation	N, N	Do not apply to handwheels.
Machine Parameters	Parameter 19 – MPG Modes	As desired to select MPG on at power-up and MPG speed limit.	Be sure to enable or disable 100x operation here. See Machine Parameters for more information.
Machine Parameters	Parameter 40 – Basic Jog Increment	0.0001 in. or 0.002 mm by default.	This specifies the distance per "click" in x1 mode. Note: Also used for jogging.
Machine Parameters	Parameter 41 – Handwheel 100x Speed, User Jog Increment	Set to 100 for 100x movement. If this is too fast, choose a smaller value.	This speed will be used in 100x mode.
Machine Parameters	Parameter 128 – Handwheel Mapping	As needed to achieve the desired mapping.	See Machine Parameters for more information.
Machine Parameters	Parameter 129 – Handwheel Display	0 will work fine. Handwheel display will be suppressed.	See Machine Parameters for more information.
Machine Parameters	Parameters 130, 131 – Z and 4 th axis on/off	0 will work fine. See Machine Parameters for more information.	Be aware that use of this feature may cause the handwheel to be turned on and off when the axis is switched.

The distance per turn of the handwheel in 1x mode is determined by the following equation:

$$\text{Distance/Turn} = \text{Distance/Click} * \text{Clicks/Turn}$$

Parameter 40 is the distance/click. Motor parameter Revs/Unit holds the Clicks/Turn value. You may adjust the Clicks/Turn value to achieve a different distance per turn. For example, if Parameter 40 is 0.0001 inches and Clicks/Turn is 100, the distance per turn is 0.01 inches. To get 0.05 inches per turn, use 500 clicks per turn. (This assumes that the encoder counts per rev is accurate.)

Be aware that Axis Summing parameter (75) may conflict with handwheel configuration. If you wish to use both handwheels and axis summing, be sure that the manual input for axis summing is the first manual axis. Axis summing cannot use a manual input that is used as a handwheel.

CHAPTER 16 - CNC7 Messages

CNC7 Startup errors

101. Message: Error initializing graphics... cannot continue (text mode).
Cause: Missing GFT files or no VGA adapter found.
Effect: Exit CNC7 with return code 63. 'Fix files and try again' prompt from CNC7M4.BAT
Removed: By CNC7M4.BAT.
102. Message: Error initializing CPU7... cannot continue (text mode).
Cause: Error while sending CNC7.HEX. Other messages with more detail of error appear on screen before this message.
Effect: Exit CNC7 with return code 63. 'Fix files and try again' prompt from CNC7M4.BAT
Removed: By CNC7M4.BAT.
103. Message: **Error sending setup** (windowed message).
Cause: ESC key pressed while sending setup.
Effect: No setup command sent to CPU7. CPU7 probably not responding.
Removed: Timed message.
104. Message: **Error sending PID setup** (windowed message).
Cause: ESC key pressed while sending PID setup.
Effect: No PID setup command sent to CPU7. CPU7 probably not responding.
Removed: Timed message.
105. Message: CNC7.PLC file read error... cannot continue (text mode).
Cause: Missing or error in CNC7.PLC.
Effect: Exit CNC7 with return code 63. 'Fix files and try again' prompt from CNC7M4.BAT
Removed: By CNC7M4.BAT.
106. Message: The PC clock appears to be wrong.

- Cause: The time on the PC internal clock is earlier than the time recorded in a previously stored file.
Effect: None.
Removed: Start of new job.

Messages issued upon exit from CNC7

201. Message: **Return code 60** (text mode).
Cause: Utility button pressed (CNC7M4.BAT probably not running).
Effect: Exit CNC7 with return code 60.
Removed: By CNC7M4.BAT.
202. Message: **Return code 61** (text mode).
Cause: Edit button pressed (CNC7M4.BAT probably not running).
Effect: Exit CNC7 with return code 61.
Removed: By CNC7M4.BAT.
203. Message: **Return code 62** (text mode).
Cause: CAM button pressed (CNC7M4.BAT probably not running).
Effect: Exit CNC7 with return code 62.
Removed: By CNC7M4.BAT.
204. Message: **Return code 63** (text mode).
Cause: CPU7 not responding, or CNC7.HEX, CNC7.PLC, or font file is missing or damaged.
Effect: Exit CNC7 with return code 63.
Removed: By CNC7M4.BAT.
205. Message: **Return code 64** (text mode).
Cause: A floating-point math error occurred.
Effect: Exit CNC7 with return code 64.
Removed: By CNC7M4.BAT.
206. Message: **Return code 65** (text mode).
Cause: CNC7.CFG file is missing or damaged.

Effect: Exit CNC7 with return code 65.
 Removed: By CNC7M4.BAT.

Cause: M0, M1, M100/75, or Block Mode.
 Effect: 'Block Mode' message displayed if CNC program running in block mode.
 Removed: After CYCLE START pressed.

Messages and Prompts in the Operator Status Window

Status messages

301. Message: **Stopped.**
 Cause: No operations in progress.
 Effect: None.

302. Message: **Moving...**
 Cause: Motors are moving while a CNC program is running.
 Effect: None.

303. Message: **Paused...**
 Cause: Motion is paused while a CNC program is running (FEED HOLD).
 Effect: None.

304. Message: **MDI...**
 Cause: CPU7 running in MDI mode.
 Effect: None.

305. Message: **Processing...**
 Cause: CPU7 running in a mode other than MDI.
 Effect: None.

306. Message: **Job finished.**
 Cause: Normal end of CNC program.
 Effect: None.

307. Message: Operator abort: job canceled.
 Cause: ESC or CYCLE CANCEL pressed.
 Effect: Job canceled.
 Removed: Start of new job.

308. Prompt: Waiting for input #NN.
 Cause: M100 or M101 executing.
 Effect: 'Mnn' or 'M6: Insert Tool # NNN Tool library description' message displayed if M function macro executing.
 Removed: After input is received.

309. Prompt: Waiting for CYCLE START button.

310. Prompt: Waiting for output #NN.
 Cause: M100 or M101 executing.
 Effect: 'Mnn' or 'M6: Insert Tool # NNN Tool library description' message displayed if M function macro executing.
 Removed: After output is in correct state.

311. Prompt: Waiting for memory #NN.
 Cause: M100 or M101 executing.
 Effect: 'Mnn' or 'M6: Insert Tool # NNN Tool library description' message displayed if M function macro executing.
 Removed: After memory is in correct state.

312. Prompt: Waiting for PLC operation (Mnn).
 Cause: PLC program not clearing PLC operation in progress.
 Effect: 'Mnn' or 'M6: Insert Tool # NNN Tool library description' message displayed if M function macro executing.
 Removed: After PLC program completes operation in progress.

313. Prompt: Waiting for dwell time.
 Cause: G4 executing.
 Effect: 'Mnn' or 'M6: Insert Tool # NNN Tool library description' message displayed if M function macro executing.
 Removed: After specified time has elapsed.

314. Message: Input search data.
 Cause: Run/search key pressed.
 Effect: None.
 Removed: After search data input.

315. Message: **Searching...**
 Cause: Run/search in progress.
 Effect: None.

316. Message: Search complete. Processing...
 Cause: Run/search mode. Search successful. Preprocessing job.

Effect: None.

317. Message: Waiting for automatic tool change.

Cause: M6 executing with automatic tool changer.

Effect: None.

Removed: After changer signals that tool change is complete.

Abnormal stops (faults)

Abnormal stops are detected in the following order:PLC, servo drive, spindle drive, lube, ESTOP. This means that if both the servo drive and the spindle drive have faulted, the servo drive fault message would appear.

401. Message: PLC failure detected.

Cause: CPU7 stopped with PLC failure bit set.

Effect: Job canceled.

Removed: When PLC failure bit removed.

Typical implementation: correct PLC then press and release EMERGENCY STOP.

402. Message: **PLC Online.**

Cause: PLC has returned on line.

Effect: None.

404. Message: Spindle drive fault detected.

Cause: CPU7 stopped with spindle drive fault bit set.

Effect: Job canceled.

Removed: when spindle drive fault removed by PLC. Typical implementation: Check inverter for fault or reset spindle contactor OCR, then press and release EMERGENCY STOP.

405. Message: Lubricant level low.

Cause: CPU7 stopped with low lube fault bit set.

Effect: Current job continues to run, but a new job cannot be started.

Removed: when low lube fault bit removed by PLC. Typical implementation: add lube then press and release EMERGENCY STOP.

406. Message: Emergency Stop detected.

Cause: CPU7 stopped with no fault bits set.

Effect: 'Fault: Job canceled.' prompt.

Removed: when Emergency Stop released.

407. Message: X+ limit (#1) tripped.

Cause: CPU7 stopped with limit switch status.

Effect: Job canceled.

Removed: Start of new job, when limit cleared.

408. Message: Programmed action timer expired.

Cause: M103 time expired before M104 encountered.

Effect: Job canceled.

Removed: Start of new job.

409. Message: **_ axis lag.**

Cause: Lag Distance (Allowable Following Error) is detected on any axis for more than 1.5 seconds. Where:
Lag Distance=
Feedrate inch/min

+ .0005 inch/int
240,000 ints/min
(Allowable Following Error)
All axis motion is stopped and the CNC program is aborted. The probable causes of this error are:
1.The machine is doing a very heavy cut.
2.The maximum rates or the acceleration values for the motors are set too high.
3.The motors are undersized for the application.

Effect: Job canceled

Removed: 1.If the problem is occasional heavy cuts, slowing down the cutting feedrate can solve the problem.
2.If the problem only occurs on high speed moves then either the maximum speed or the acceleration is set too high. Lower the values in the Motor

Setup screen or rerun Autotune again to determine new values.
3.If the problem is persistent lag errors in normal operations it indicates that the motors are too weak to handle the required loads. Increase the gear ratios or get more powerful motors.

Note: If the Lag Distance (Allowable Following Error) is exceeded for more than .025 seconds, then no acceleration will occur on any axis. However, no error message is generated at this point because no fatal error exists.

410. Message: _ axis position error.
Cause: A position error > .25 inches is detected on any axis. All axis motion is stopped, power to the motors is released (all servo drive commands cease) and the CNC program is aborted.

The probable causes of this error are:

- 1.The motor is wired up backwards.
- 2.Noise is getting into the system via the motor cables (the line integrity has been violated).
- 3.An encoder error occurred.

Effect: Job canceled

Removed: Try a slow jog on the motor and watch the DRO position. If the position on the DRO goes opposite the direction indicated on the jog button, then the motor is wired up backwards. Change the motor wiring.

Check the motor cabling paying particular attention to the ground connections. Replace the cable if it is damaged or repair the motor connections.

Jog the motor awhile, at the maximum rate, using the fast jog buttons. (Check the fast jog rate

in the motor jog parameters screen to make sure it is set equal to the maximum motor rate.) If the motor seems to jump around rather than accelerate and decelerate smoothly then you are probably fighting an encoder error.

Swap the motor with one from another axis and see if the error follows the motor. If it stays with the axis, replace the CPU. If it follows the motor, replace the motor cable. If the problem still persists, replace the motor and encoder.

Start of new job.

411. Message: _ axis full power without motion.
Cause: 90% Power (PID Output > 115) is applied to any axis and no motion >.0005 inches is detected, for more than the time specified in parameter 61 (usually .5 seconds). All axis motion is stopped and the CNC program is aborted.

The probable causes of this error are:

One of the axes is against a physical stop.
The servo drive has shutdown due to a limit switch input.
The Z home switch is the same as the Z + limit switch.

Effect:

If the axis has obviously run into a physical stop, use the slow jog mode to move the axis away from the end. Change the CNC program to remove moves that are out of bounds or rezero to a point that permits the required CNC moves to be made.

If the axis is not on a physical stop, check the limit switch to

	see if it is tripped. If it is then the software is commanding a move into the switch but the hardware is shutting the move down. Go to the motor setup screen and enter the limit switch if this is applicable.	Effect:	Power down, then power up the system. The error should disappear.
		Removed:	Never - system must be powered down.
	Make sure the switch input is not unstable or noisy. If it is then replace the switch. If the problem persists it may be necessary to create separate home and limit switch inputs.		
	Use slow jog to move opposite the direction causing the error and clear all limit switches. Jog toward the direction causing the error, if no motion occurs then a servo drive failure is indicated.		
	Removed:		Start of new job.
412.	Message: _ axis encoder connection is bad. Cause: Axis is enabled but a differential encoder signal is not detected. May indicate a loose or severed encoder cable or a bad encoder. Effect: All axis motion is stopped and the CNC program is aborted. Removed: Reconnect encoder or repair encoder and/or encoder cable.	416. Prompt:	Motion fault #XX detected.
		Cause:	Invalid motion status from CPU7.
		Effect:	Power down, then power up the system. The error should disappear.
		Removed:	never - system must be powered down.
		417. Prompt:	Abnormal end of job.
		Cause:	Job ended without reason.
		Effect:	Job canceled.
		Removed:	Start of new job.
		418. Message:	Search data not found.
		Cause:	Requested search input data not found in loaded CNC file. Removed: Jogging, start of new job, other error.
		Effect:	Job canceled.
		Removed:	Start of new job.
		419. Message:	Search line in embedded subprogram.
		Cause:	Requested search line is part of an embedded subprogram; Search can only be used to start in the main program.
		Effect:	Job canceled.
		Removed:	Start of new job.
413.	Prompt: CPU Failure #01: power down. Cause: CPU7 has experienced a problem with the PC reset line. Effect: Power down, then power up the system. The error should disappear. Removed: Never - system must be powered down.	420. Message:	_ axis motor overheating.
		Cause:	CNC7 estimates that a motor has reached the warning temperature (set in Parameter 29).
		Effect:	No effect on a job which is currently running. However, a new job cannot be started until the motor has cooled below the warning temperature.
		Removed:	When next message appears.
414.	Prompt: CPU Failure #02: power down. Cause: CPU7 detected CPU failure. Effect: Power down, then power up the system. The error should disappear. Removed: Never - system must be powered down.	421. Message:	Motor(s) too hot: job canceled.
		Cause:	CNC7 estimates that one or more motors have reached the limit temperature (set in Parameter 30).
415.	Prompt: CPU fault #XX detected. Cause: invalid stop reason from CPU7.		

Effect:	The current job is canceled and power is released.	430. Message:	CPU7 PIC Offline.
Removed:	Start of new job (after motors have cooled below warning temperature).	Cause:	Power supply or hardware problem.
		Effect:	Power down, then power up the system. The error should disappear.
422. Message:	Jog Panel Offline.	431. Message:	CPU7 PIC Online.
Cause:	Jog panel failure or loose cable.	Cause:	CPU7 is back on line.
Effect:	All buttons on jog panel are inoperative.	Effect:	None.
Removed:	By reconnecting jog panel cable and appearance of next message.	432. Message:	External PLC Offline.
		Cause:	Koyo PLC Direct failure or loose cable.
423. Message:	Jog Panel Online.	Effect:	None.
Cause:	Loose jog panel cable has been reconnected.	Removed:	When PLC failure removed or cable reconnected.
Effect:	All buttons on jog panel are operative.		
424. Message:	Feedrate Override Offline.	433. Message:	External PLC Online.
Cause:	Jog panel failure or loose cable.	Cause:	PLC failure corrected.
Effect:	Feedrate knob and some jog panel keys are inoperative.	Effect:	None.
Removed:	By appearance of next message.	434. Message:	_ idling too high: Releasing power.
		Cause:	Axis is not moving and no job is running but axis has stopped against some abnormal resistance.
425. Message:	Feedrate Override Online.	Effect:	Power to motors is released.
Cause:	Loose jog panel cable has been reconnected.	Removed:	Start of new job.
Effect:	Feedrate Override knob is operative.		
426. Message:	Spindle Override Offline.	435. Message:	_ axis runaway: Check motor wiring.
Cause:	Jog panel failure or loose cable.	Cause:	Axis was moving more than 120 in/min while power was supposed to be off. Motor may be wired backwards or may be a shorted servo drive.
Effect:	Spindle knob and some jog panel keys are inoperative.	Effect:	Power to motors is released.
Removed:	By appearance of next message.	Removed:	Start of new job (after wiring or servo drive failure has been removed).
427. Message:	Spindle Override Online.		
Cause:	Loose jog panel cable has been reconnected.	436. Message:	Servo drive shutdown.
Effect:	Spindle Override knob is operative.	Cause:	This error message is produced by hardware detection of a physical error.
428. Message:	MPG Offline.		
Cause:	MPG failure or loose cable.		
Effect:	MPG is inoperative.		
Removed:	By reconnecting MPG cable and appearance of next message.		
429. Message:	MPG Online.		
Cause:	Loose MPG cable has been reconnected.		
Effect:	MPG is operative.		

The servo drive hardware originates this error message if it detects either an overcurrent or overvoltage condition. The particular hardware condition is reflected on the servo

drive LEDs. Once the servo drive detects this error condition it stops all motion and removes power to the motors. The hardware indicates the presence of this condition to the CNC7 software via the servo drive fault input to the PLC.

Effect: The particular condition can be resolved by observing the servo drive LEDs.
 Removed: This message is removed and the condition is reset only if the ESTOP is pressed and released. The PLC program that is responsible for latching this condition is also responsible for clearing this condition.

439. Message: _ axis servo drive processor failure.
 Cause: Logic power failure or processor failure.
 Effect: No motor power.
 Removed: Power complete unit down and check connections to drive.

440. Message: _ axis overvoltage
 Cause: Drive input power greater than 350 vdc.
 Effect: No motor power.
 Removed: Check input voltage and cycle start.

442. Message: _ axis undervoltage
 Cause: Drive input power less than 80 vdc.
 Effect: No motor power.
 Removed: Check input voltage and cycle start.

443. Message: _ axis commutation encoder bad.
 Cause: Bad connection from encoder.
 Effect: No motor power.
 Removed: Check encoder cable and cycle input power.

444. Message: _ axis overtemperature detected.
 Cause: Drive overtemp sensor tripped.
 Effect: No motor power.
 Removed: Wait for drive to cool and cycle start.

445. Message: _ axis overcurrent detected.
 Cause: Overcurrent detected on axis.
 Effect: No motor power.
 Removed: Check motor power connection and cycle start.

446. Message: _ axis synchronization failure.
 Cause: Communication CheckSum error.
 Effect: No motor power.
 Removed: Check fiber optic cable.

CNC syntax errors

501. Message: Invalid character on line NNNNN.
 Cause: Invalid character on CNC line.
 Effect: Job canceled.
 Removed: Start of new job.

502. Message: Invalid G code on line NNNNN.
 Cause: Invalid G code encountered on CNC line.
 Effect: Job canceled.
 Removed: Start of new job.

503. Message: Invalid M function on line NNNNN.
 Cause: Invalid M function encountered on CNC line.
 Effect: Job canceled.
 Removed: Start of new job.

504. Message: Invalid parameter on line NNNNN.
 Cause: Invalid or missing number after letter.
 Effect: Job canceled.
 Removed: Start of new job.

505. Message: Invalid value on line NNNNN.
 Cause: Value out of range (T, H, D).
 Effect: Job canceled.
 Removed: Start of new job.

506. Message: Only 1 M code per line.
 Cause: More than one M code appears on the line.
 Effect: Job canceled.
 Removed: Start of new job.

507. Message: No closing quote.
 Cause: The closing quotation mark (") in a quoted string is missing.
 Effect: Job canceled.

	Removed:	Start of new job.		Cause:	Brackets or parentheses are nested too deeply.
508.	Message:	Macro nesting too deep.		Effect:	Job canceled.
	Cause:	Macro nesting limit exceeded on attempt to invoke a subroutine.		Removed:	Start of new job.
	Effect:	Job canceled.	518.	Message:	Undefined variable.
	Removed:	Start of new job.		Cause:	The variable name does not exist.
509.	Message:	Option not available.		Effect:	Job canceled.
	Cause:	Attempt to access a locked software option.		Removed:	Start of new job.
	Effect:	Job canceled.	519.	Message:	Too many variables.
	Removed:	Start of new job.		Cause:	The space allotted for user-defined variables has been exceeded.
510.	Message:	Too many macro arg's.		Effect:	Job canceled.
	Cause:	Too many arguments were given in a G65 macro call.		Removed:	Start of new job.
	Effect:	Job canceled.	520.	Message:	Invalid variable name.
	Removed:	Start of new job.		Cause:	The variable name contains an illegal character.
511.	Message:	Missing parameter.		Effect:	Job canceled.
	Cause:	A parameter is required or expected but not found.		Removed:	Start of new job.
	Effect:	Job canceled.	521.	Message:	Divide by zero.
	Removed:	Start of new job.		Cause:	Attempt to divide by zero.
513.	Message:	Expected “=”.		Effect:	Job canceled.
	Cause:	Error in expression to left of “=”, missing “=”, or orphaned parameter.		Removed:	Start of new job.
	Effect:	Job canceled.	522.	Message:	Domain error.
	Removed:	Start of new job.		Cause:	Imaginary number would result (square root of a negative number).
514.	Message:	Empty expression.		Effect:	Job canceled.
	Cause:	The expression contains no operands.		Removed:	Start of new job.
	Effect:	Job canceled.	523.	Message:	Invalid value in assignment.
	Removed:	Start of new job.		Cause:	Attempt to assign an illegal value to a system variable.
515.	Message:	Syntax error in expression.		Effect:	Job canceled.
	Cause:	Illegal character in number, variable or function.		Removed:	Start of new job.
	Effect:	Job canceled.	524.	Message:	Variable is read-only.
	Removed:	Start of new job.		Cause:	Attempt to assign a value to a read-only system variable.
516.	Message:	Unmatched bracket (parenthesis).		Effect:	Job canceled.
	Cause:	Brackets or parentheses are paired improperly or misplaced.		Removed:	Start of new job.
	Effect:	Job canceled.			
	Removed:	Start of new job.			
517.	Message:	Evaluation stack overflow.			

Cutter compensation errors

601.	Message:	Error: no compensation in MDI.
	Cause:	G41 or G42 entered in MDI.
	Effect:	MDI is not canceled, but cutter compensation does NOT go into

- effect. Remainder of line processed.
602. Message: Arc as first comp. move on line NNNNN.
Cause: Cutter compensation started with arc as first move.
Effect: Job canceled.
Removed: Start of new job.
603. Message: Arc as first uncomp. move on line NNNNN.
Cause: Arc specified as first move after end of compensation (G40).
Effect: Job canceled.
Removed: Start of new job.
604. Message: Plane must be XY on line NNNNN.
Cause: Cutter compensation started with YZ or ZX plane selected.
Effect: Job canceled.
Removed: Start of new job.
605. Message: Canned cycle not allowed on line NNNNN.
Cause: Canned cycle attempted during compensation.
Effect: Job canceled.
Removed: Start of new job.
606. Message: G53 not allowed on line NNNNN.
Cause: G53 attempted during compensation.
Effect: Job canceled.
Removed: Start of new job.
607. Message: Set home not allowed on line NNNNN.
Cause: M26 attempted during compensation.
Effect: Job canceled.
Removed: Start of new job.
608. Message: Ref. point move not allowed on line NNNNN.
Cause: G28, G29, or G30 attempted during compensation.
Effect: Job canceled.
Removed: Start of new job.
609. Message: File read error on look ahead.

- Cause: Error reading file used for cutter comp look ahead.
Effect: Job canceled.
Removed: Start of new job.

Parameter setting errors

701. Message: G10 error: no R-value on line NNNNN.
Cause: G10 used with no R-value.
Effect: Job canceled.
Removed: Start of new job.
702. Message: G10 error: invalid D on line NNNNN.
Cause: G10 D0 Rxx specified.
Effect: Job canceled (D0 cannot be set; it is always zero).
Removed: Start of new job.
703. Message: G10 error: invalid H on line NNNNN.
Cause: G10 H0 Rxx specified.
Effect: Job canceled (H0 cannot be set; it is always zero).
Removed: Start of new job.
704. Message: G10 error: invalid P on line NNNNN.
Cause: G10 used with unknown P value.
Effect: Job canceled.
Removed: Start of new job.
705. Message: G10 error: No D, H, or P on line NNNNN.
Cause: G10 used without D, H, or P to assign value.
Effect: Job canceled.
Removed: Start of new job.

Canned cycle errors

801. Message: Error: No R point on line NNNNN.
Cause: No R-value specified.
Effect: Job canceled.
Removed: Start of new job.
802. Message: Error: Q = 0 on line NNNNN.
Cause: Q value of 0 specified (Q used for G73 and G83 only).
Effect: Job canceled.
Removed: Start of new job.

803.	Message:	Error: No Z point on line NNNNN.	Cause:	I, J, or K specified with wrong plane (e.g. K with G17, or I with G19).
	Cause:	No Z value specified for canned cycle.	Effect:	Job canceled.
	Effect:	Job canceled.	Removed:	Start of new job.
	Removed:	Start of new job.		
804.	Message:	Error: Ggg invalid on line NNNNN (gg = 76, 86, 87, 88).	905. Message:	Warning: 0 radius arc on line NNNNN.
	Cause:	Unimplemented canned cycle requested.	Cause:	Arc move was specified with zero radius.
	Effect:	Job canceled.	Effect:	Move is done as a linear move; job continues.
	Removed:	Start of new job.	Removed:	When next message appears.
805.	Message:	Error: No Q value on line NNNNN.	906. Message:	Warning: unknown arc on line NNNNN.
	Cause:	Q value not specified for G73 or G83.	Cause:	Position of arc move could not be determined from parameters (e.g. G91 G2 X0 Y0 R1).
	Effect:	Job canceled.	Effect:	Move is done as a linear move; job continues.
	Removed:	Start of new job.	Removed:	When next message appears.
806.	Message:	Error: No P value on line NNNNN.	907. Message:	_ axis travel exceeded on line NNNNN.
	Cause:	P value (dwell time) not specified for G82 or G89.	Cause:	Software travel limit would be exceeded by the requested move.
	Effect:	Job canceled.	Effect:	Job canceled.
	Removed:	Start of new job.	Removed:	Start of new job.

Miscellaneous errors

901.	Message:	Ref. point invalid on line NNNNN.	908. Message:	Option not available on line NNNNN.
	Cause:	G30 with invalid P value (must be 1 or 2).	Cause:	A code for an extra-cost option was specified, but the option has not been licensed.
	Effect:	Job canceled.	Effect:	Job canceled.
	Removed:	Start of new job.	Removed:	Start of new job.
902.	Message:	No prior G28 or G30 on line NNNNN.	909. Message:	Program too long: job canceled.
	Cause:	G29 with no preceding G28 or G30.	Cause:	Attempt to run a job over 640K in length, without the unlimited program size option.
	Effect:	Job canceled.	Effect:	Job canceled.
	Removed:	Start of new job.	Removed:	Start of new job.
903.	Message:	Warning: No coordinates for G92 on line NNNNN.	910. Message:	No subroutines in MDI.
	Cause:	G92 with no axis coordinates to set.	Cause:	Specified O9100 - O9999 in MDI, which would begin an embedded subprogram.
	Effect:	Remainder of line processed; job continues.	Effect:	MDI canceled.
	Removed:	When next message appears.	Removed:	Start of new job.
904.	Message:	Invalid plane for arc on line NNNNN.	911. Message:	Illegal recursion.

	Cause:	Attempt to execute a subprogram or macro that calls itself, either directly or indirectly.	Cause:	Tried to rescale while scaling is turned on.	
	Effect:	Job canceled.	Effect:	Job canceled.	
	Removed:	Start of new job.	Removed:	Start of new job.	
912.	Message:	Too many subprogram calls.	1005.	Message:	Cannot scale arcs with different scale factors.
	Cause:	Attempt to run a job with 20 or more levels of subprogram nesting.		Cause:	Scaling factors of the arc axes are different.
	Effect:	Job canceled.		Effect:	Job cancelled.
	Removed:	Start of new job.		Removed:	Start of new job.
913.	Message:	Could not open file <i>filename.ext</i> .	1100 – 1199.		Custom messages defined in CNC7XMSG.TXT. Please contact your dealer if you have any questions regarding a particular message.
	Cause:	Attempt to call a subprogram or macro, but the subprogram file does not exist.			
	Effect:	Job canceled.			
	Removed:	Start of new job.			
914.	Message:	Tool library invalid for <i>Tnn</i>			
	Cause:	Enhanced ATC is enabled and the tool library does not have a valid bin number assigned.			
	Effect:	Job canceled.			
	Removed:	Start of new job.			

Scaling/Mirroring errors

1001.	Message:	Invalid scaling parameter on line NNNNN.
	Cause:	Invalid parameter specified (I, J, K, P).
	Effect:	Job canceled.
	Removed:	Start of new job.
1002.	Message:	Invalid scaling center on line NNNNN.
	Cause:	Invalid parameter specified (X, Y, Z).
	Effect:	Job canceled.
	Removed:	Start of new job.
1003.	Message:	G code not allowed when scaling on line NNNNN.
	Cause:	G28/G29/G30/G92 is not allowed when scaling or mirroring is turned on.
	Effect:	Job canceled.
	Removed:	Start of new job.
1004.	Message:	Turn scaling off before rescaling.

