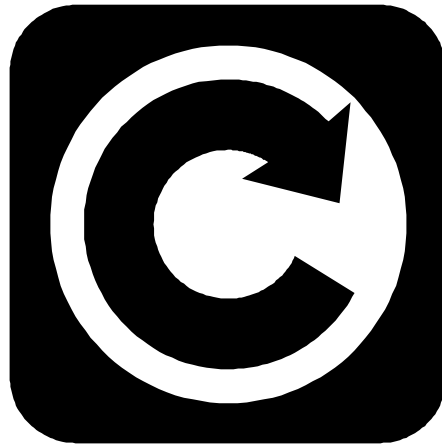


CENTROID™



M-SERIES Operator's Manual

Version 7.17
Rev. 010325

© 2002 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: _____

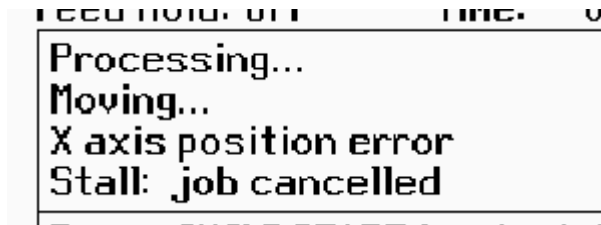
Dealer: _____ Machine Brand: _____

Machine type (check below)

- | | |
|-----------|------------------|
| Knee Mill | Machining Center |
| Bed Mill | Lathe |
| Router | 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					

Motor Parameters

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

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									

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	Param	Value	Param	Value	Param	Value	Param	Value	Param	Value

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

TABLE OF CONTENTS

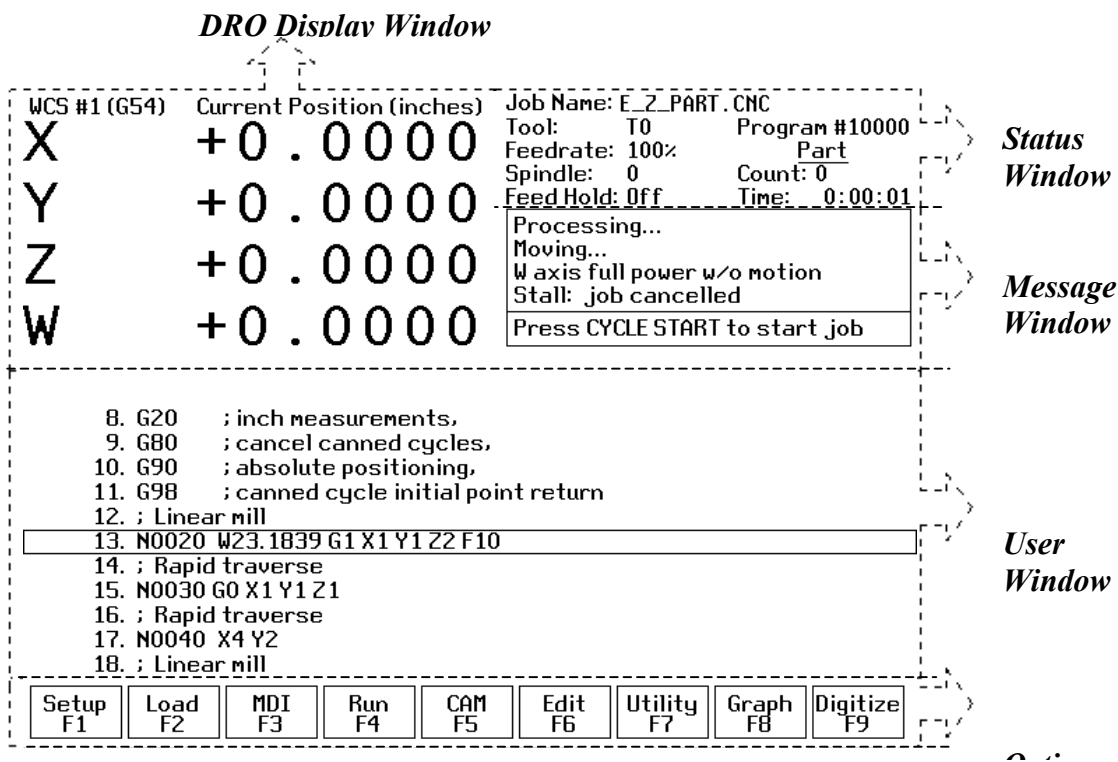
<u>CHAPTER 1 - Introduction</u>	<u>1-1</u>	Probe Parameters	6
Window Description	1		
Conventions	3		
Machine Home	4		
<u>CHAPTER 2 - CNC7 Main Screen</u>	<u>2-1</u>	<u>CHAPTER 9 - Engraving</u>	<u>9-1</u>
Option Descriptions	1	Quick Start Engraving Software Tutorial	1
Canceling and Resuming Jobs	5		
Canceling a Job in Progress	5	<u>CHAPTER 10 - Intercon Software</u>	<u>10-1</u>
Resuming a Canceled Job	5	Intercon Main Screen	1
M-Series CNC G-Code Editor Description	6	Edit Part Program	4
		Insert Operation	6
<u>CHAPTER 3 - Part Setup (F1 from Setup)</u>	<u>3-1</u>	Teach Mode	35
Operation Description	2	Graphics	36
Part Setup Examples	4	Math Help	39
Work Coordinate Systems Configuration	6	Intercon Tutorial #1	51
		Intercon Tutorial #2	61
<u>CHAPTER 4 - Tool Setup (F2 from Setup)</u>	<u>4-1</u>	<u>CHAPTER 11 - Mastercam Hints</u>	<u>11-1</u>
Offset Library	1		
Automatic Tool Measurement	3	<u>CHAPTER 12 - CNC Program Codes</u>	<u>12-1</u>
Setting up Tool Height Offsets	4	Miscellaneous CNC Program Symbols	1
Tool Library	5	<u>CHAPTER 13 - G-codes</u>	<u>13-1</u>
<u>CHAPTER 5 - Power Feed (F4 from Setup)</u>	<u>5-1</u>	<u>CHAPTER 14 - M functions</u>	<u>14-1</u>
		Macro M functions	1
<u>CHAPTER 6 - The Utility Menu</u>	<u>6-1</u>	<u>CHAPTER 15 - Operator Panels</u>	<u>15-1</u>
F1 - Format	1	M-400 Operator Panel / M-39 Jog Pendant	1
F2 – Update, F3 - Backup	2	M-15 Operator Panel	6
F4 - Restore	3	<u>CHAPTER 16 – Configuration</u>	<u>16-1</u>
F5 - File Ops, F6 - PLC Diag	4	Password	1
F7 – Report, F8 - Options	4	Control Configuration	2
		Machine Configuration	5
<u>CHAPTER 7 – Digitize (F9 from Main Menu)</u>	<u>7-1</u>	Machine Parameters	8
Grid Digitize	2	PID Configuration	16
Radial Digitize	6	<u>CHAPTER 17 - CNC7 Messages</u>	<u>17-1</u>
<u>CHAPTER 8 - Probing</u>	<u>8-1</u>		
Part Setup with Probing	1		
Probing Cycles	2		
Calibrating the Probe Tip Diameter	6		

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.



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 16).

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. 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 15 for descriptions of the Feed Hold Button, Feedrate Override Knob, and Spindle controls. For a description of the Program Number see G65 in Chapter 13 or M98 in Chapter 14.

The Part Count 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 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.

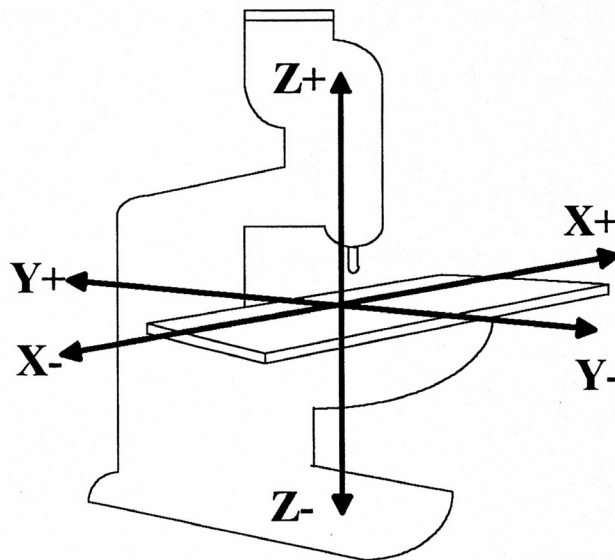
Conventions

*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>.

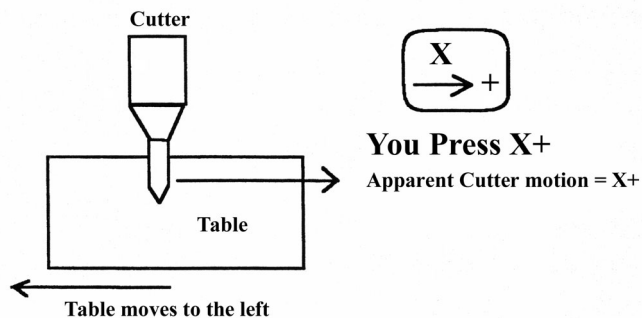
*All data entry screens in the M-Series Control use <F10> to save changes.

*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.



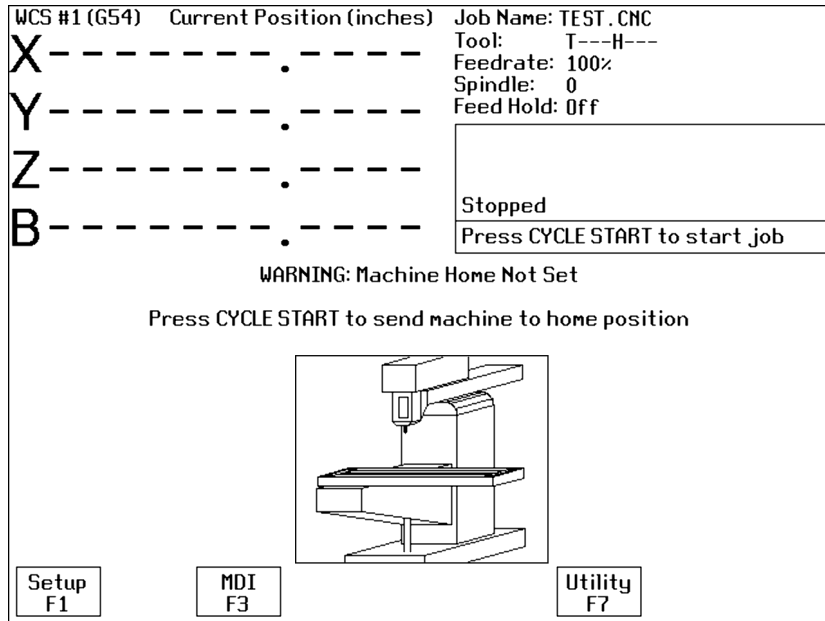
*The direction of motion is defined by the CUTTER motion, not the TABLE motion.



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

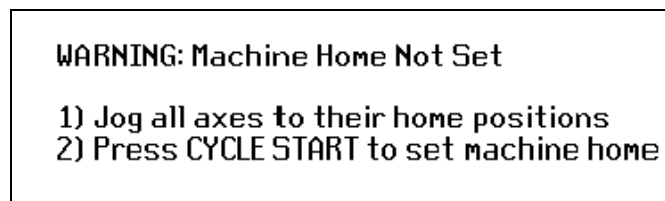
Machine Home

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 or safe hard stops, the control can automatically home itself. 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 to its plus limit, then home X to its minus limit and Y to its plus limit.

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.

CHAPTER 2

CNC7 Main Screen

When the M-Series control is started, the first screen to appear is the CNC7 Main Screen.

Option Descriptions

F1 – Setup

This will bring up the Setup Screen. The options displayed in the options window are Part, Tool, Config, and Feed. These options are explained in detail in Chapters 3, 4, 5 and 16.

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>.

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. You can select an alternate starting point, turn single block mode on or off, and turn optional stops on or off. The options on the Run screen are:

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.

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.

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.

F8 - Graph

Graphs the part. For more information, see the "<F8> - Graph" as described later. 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.

F5 - CAM

Choose <F5> from the Main Menu to load an installed CAM software package. Currently, the default CAM system is Intercon (**Interactive Conversational**) 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

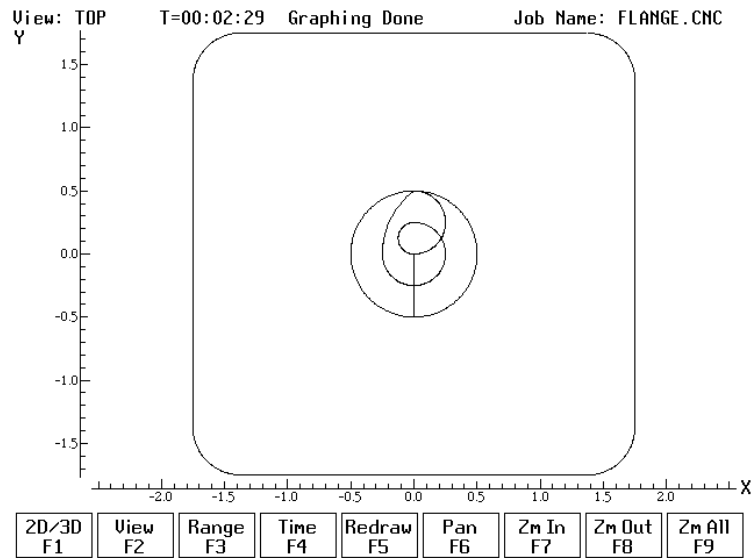
The edit function from the Main Screen loads a text editor so that you may edit CNC files. Press <F6> to load the current job file. When you exit the text editor, you will return to the CNC7 Main Screen.

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 grey. Rapid traverse movements are shown in red. Feedrate movements are shown in yellow. Press <F1> 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. The screen shows the following keys:

<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.

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.

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

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.

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.

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:

```
C:\CNC7\NCFILES>cnc7edt           Invoking editor from
command line
C:\CNC7\NCFILES>cnc7edt cnc40.nc   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	See user dialog table below. Save current file to disk.
Exit	F10	See user dialog table below. Quit using editor. See user dialog table below.
Search forward	F3	Specify string to search for; this is a case-sensitive search
Search forward again	F4	Replace all occurrences of one text string with another string.
Replace	F5	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.	"Do you want to replace the original file <current file name>? Y/N"

	You answer N to the "replace" question. You choose a file name that already exists.	"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.

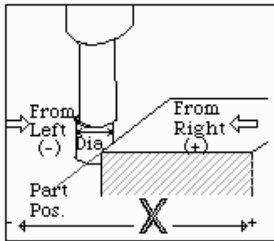
Table 2 - User Dialogs

CHAPTER 3

Part Setup

(F1 from Setup)

General

WCS #1 (G54)	Current Position (inches)	Job Name: 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
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
X	0.0000	0.0000
		Approach From Right(+)
Nxt Axis F1	Prv WCS F2	Nxt WCS F3
Auto F4	Probe F5	WCS F9
		Set F10
		Cancel ESC

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.

<F2> will select the previous work piece coordinate system. The position that will be set only affects the currently selected coordinate system.

<F3> will select the next work piece coordinate system. The position that will be set only affects the currently selected coordinate system.

<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.

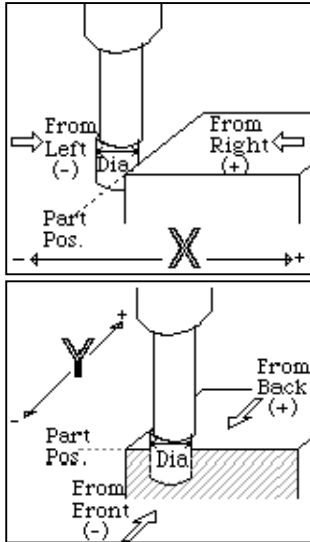
Two additional Probing features may be available on <F4> and <F5>. See Chapter 8.

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

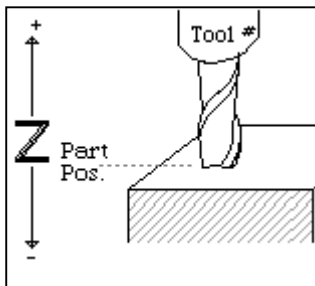
Axis	Part Position	Edge Finder Diameter	Approach from
X	0.0000	0.0000	Left (-)

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: enter the direction the edge finder is approaching the part from. The X-Axis and Y-Axis default to zero.

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):

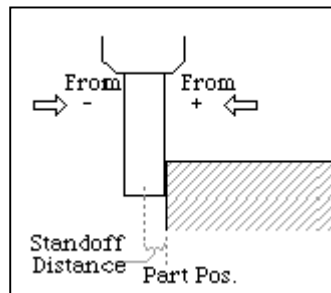
Set Tool Number to the number this tool is assigned in the tool library.

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 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.

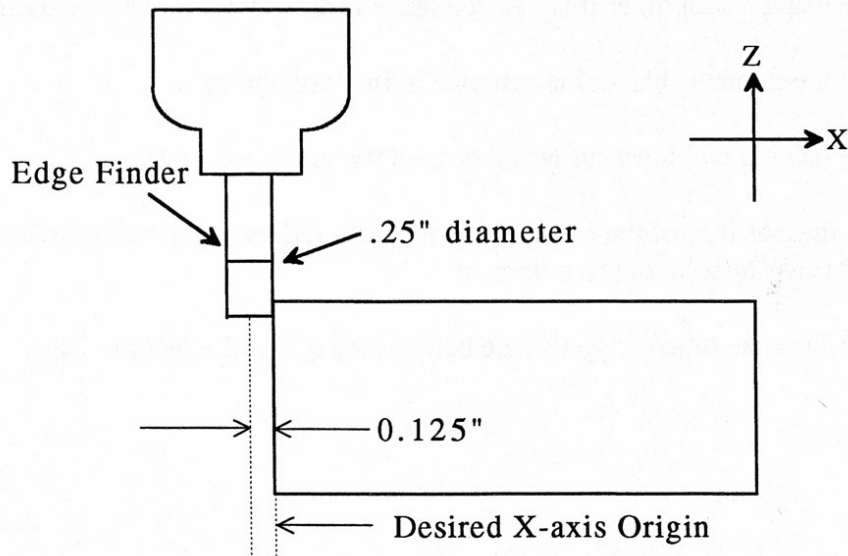
Approach from: enter the direction the edge finder is approaching the part from. Enter the correct direction given the nature of your 4th-Axis.

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

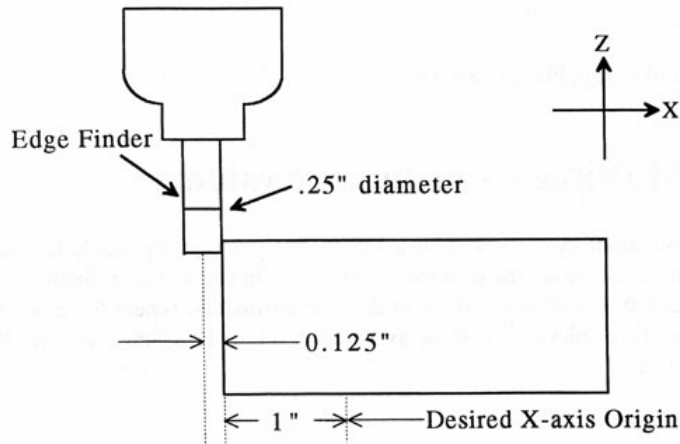
Axis	Position	Edge Finder Diameter	Approach From
X	0	0.25	Left (-)

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 13). 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 14).

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.

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.

CHAPTER 4

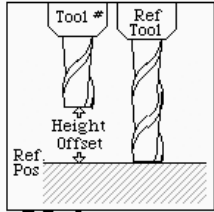
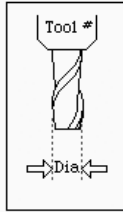
Tool Setup

(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

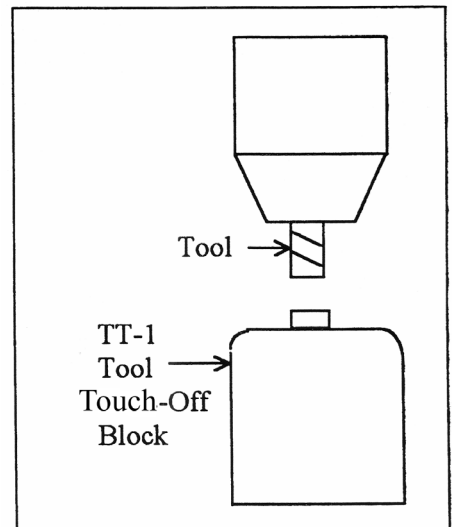
Z-minus single-surface probing, using the TT-1 tool touch-off post, is available in the Tool Offset Library.

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.



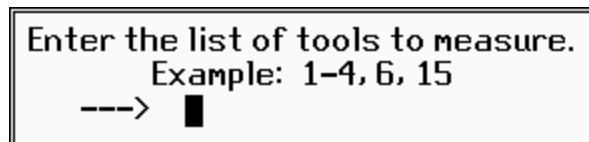
Setting the Tool Height Offsets:

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>.

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:



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: FLANGE . CNC				
X	+ 0 . 0000	Feedrate: 100%				
Y	+ 0 . 0000	Spindle: 0				
Z	+ 0 . 0000	Feed Hold: Off				
		Stopped				
Tool Library						
Tool	Height	Diameter	Coolant	Spindle	Speed	Description
T001	H001	D001	FLOOD	OFF	3000	.325 end mill
T002	H002	D002	FLOOD	OFF	4200	.5 end mill
T003	H003	D003	MIST	OFF	4000	.25 ball mill
T004	H004	D004	MIST	OFF	4200	.125 end mill
T005	H005	D005	FLOOD	OFF	3500	.375 end mill
T006	H006	D006	OFF	OFF	0	
T007	H007	D007	OFF	OFF	0	
T008	H008	D008	OFF	OFF	0	
T009	H009	D009	OFF	OFF	0	
T010	H010	D010	OFF	OFF	0	
						<input type="button" value="Save F10"/> <input type="button" value="Cancel ESC"/>

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.

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.

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.

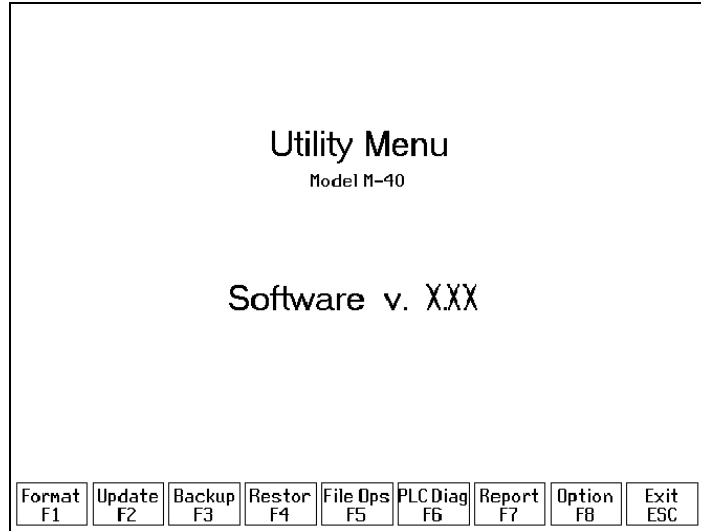
F4 - Power XY

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

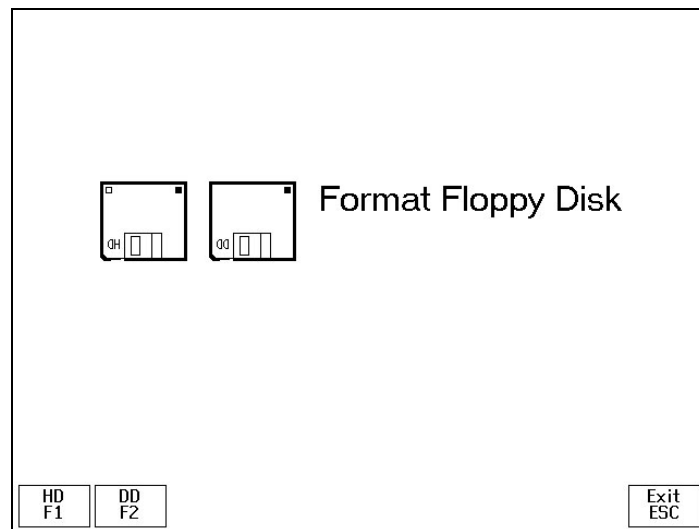
The Utility Menu

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

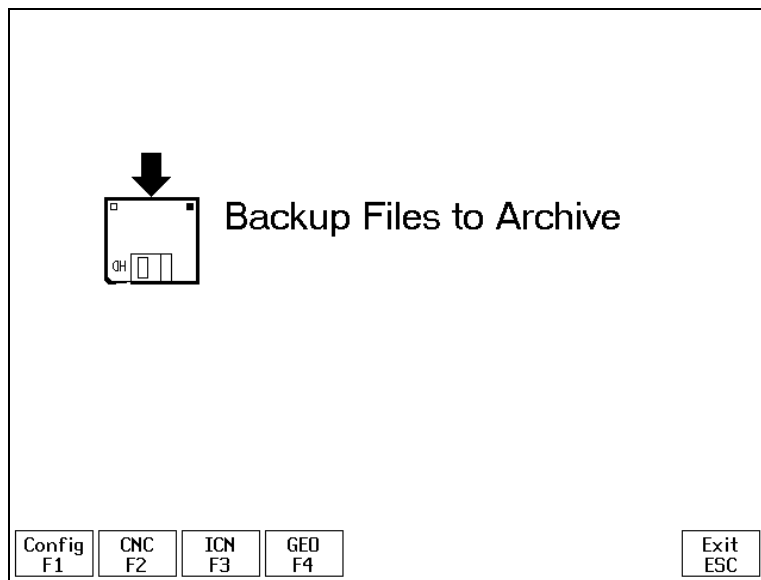
Pressing <F2> will perform the same function as above, except that the floppy disk will be formatted at double density (720K).

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

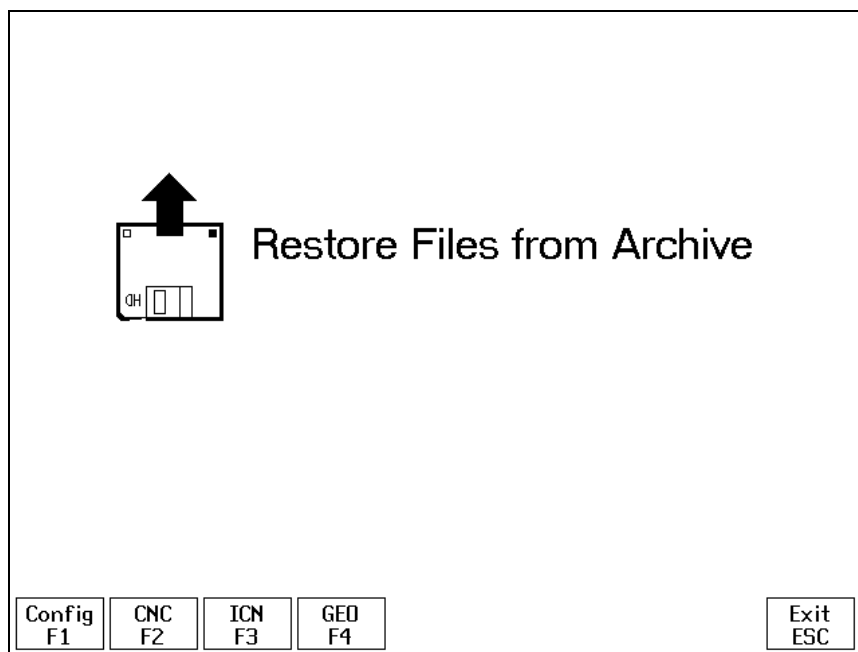
Pressing <F3> will do the same function as above, except that this will back up the .ICN files that are stored on the Controller (in the directory C:\ICN).

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 C:\ICN 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 13). 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 C:\NC directory. 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.

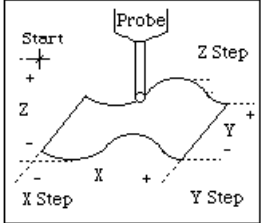
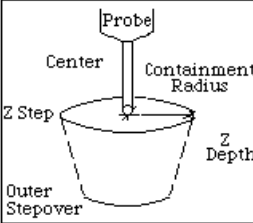
F8 - Options

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

CHAPTER 7

Digitize

(F9 from Main Menu)

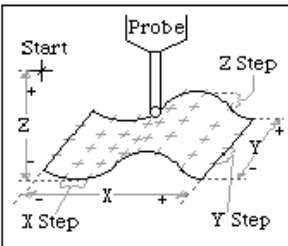
WCS #1 (G54)	Current Position (inches)	Job Name: FLANGE . CNC	
X	+ 3 . 5 0 0 0	Feedrate: 100%	Stopped
Y	+ 2 . 7 4 3 0	Spindle: 2992	
Z	- 0 . 2 5 5 0	Feed Hold: Off	
			
Grid		Radial	
Grid F1	Radial F2	Probe F4	Cancel ESC

The Digitize feature of CNC7 is used to digitize a rectangular surface area or an inside circular bore area and record the resulting data points in a disk file. The file is in the form of CNC codes. If you choose the digitize probe tip to match your milling cutter size, you can then load and run this data file directly to produce an exact copy of the digitized part. If you wish to view, edit, or refine the data and make a new part program, you can translate the data into Mastercam[®] Level 2's spline format using the Utility Screen (Chapter 12), then load the data into Mastercam[®] Level 2.

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). 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.

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
		
		<input type="button" value="Save F10"/> <input type="button" value="Cancel 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.

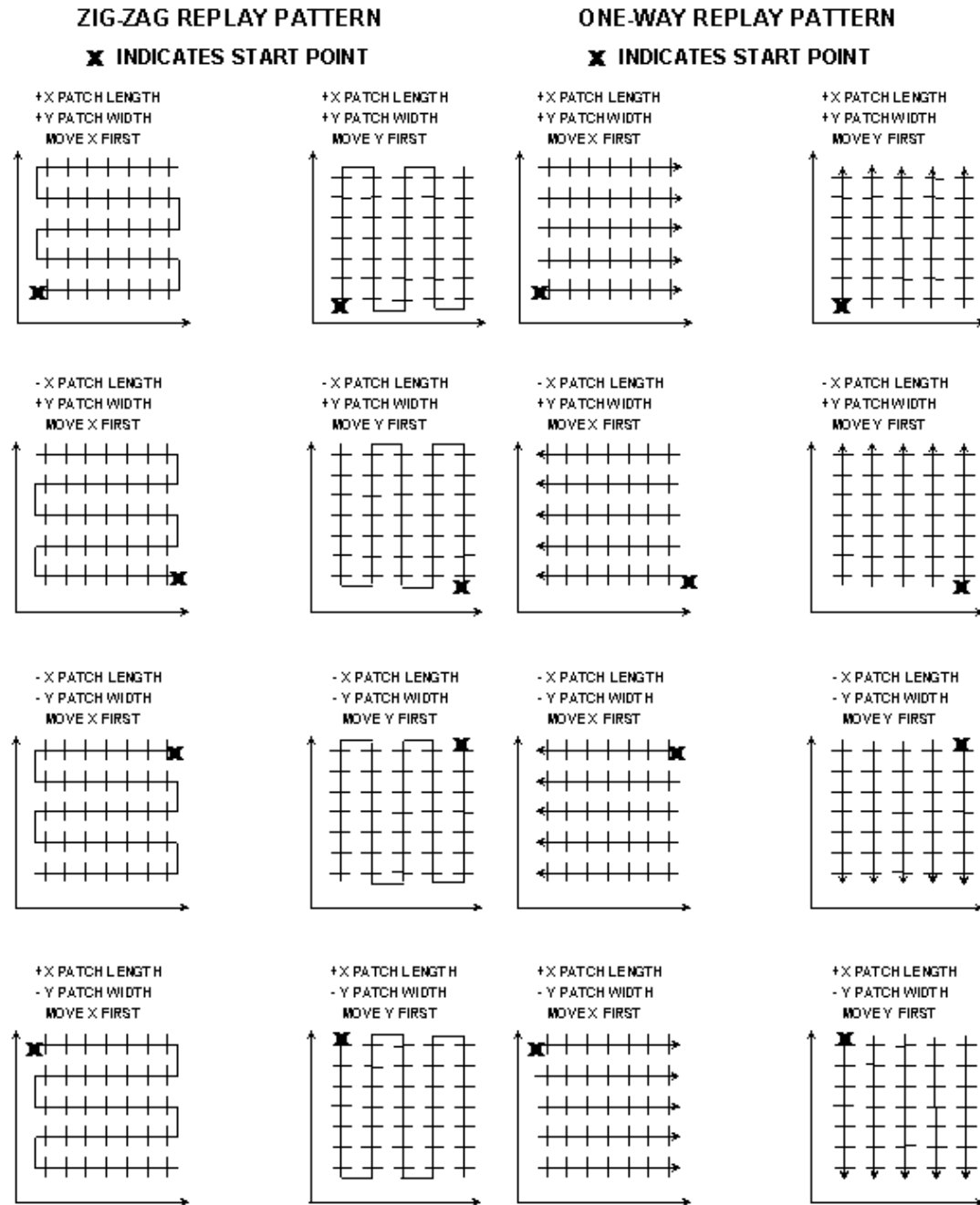
Replay Feedrate: The feedrate to include with the G1 command on the first line of the data file. If the data file is run as a CNC program, this is the feedrate at which the machine will retrace the digitized surface.

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.

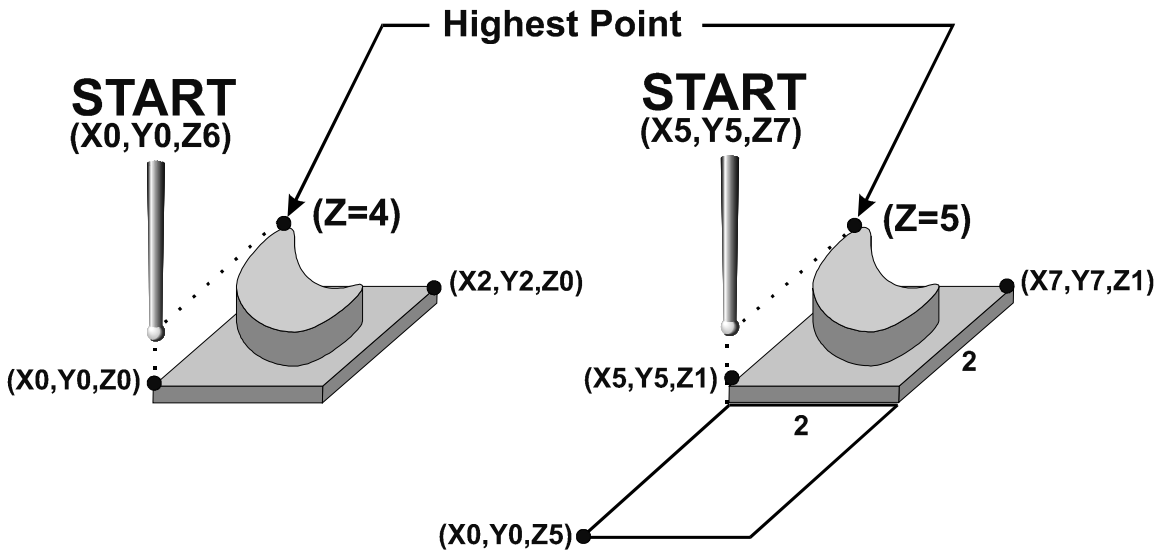
Replay Pattern: Indicates the replay movement pattern. If ZIG ZAG is selected, the replay pattern will alternate between positive and negative directions on each successive pass. If ONE WAY is selected, the replay pattern will maintain a constant "one-way" direction throughout the playback.

Grid Digitize Notes

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

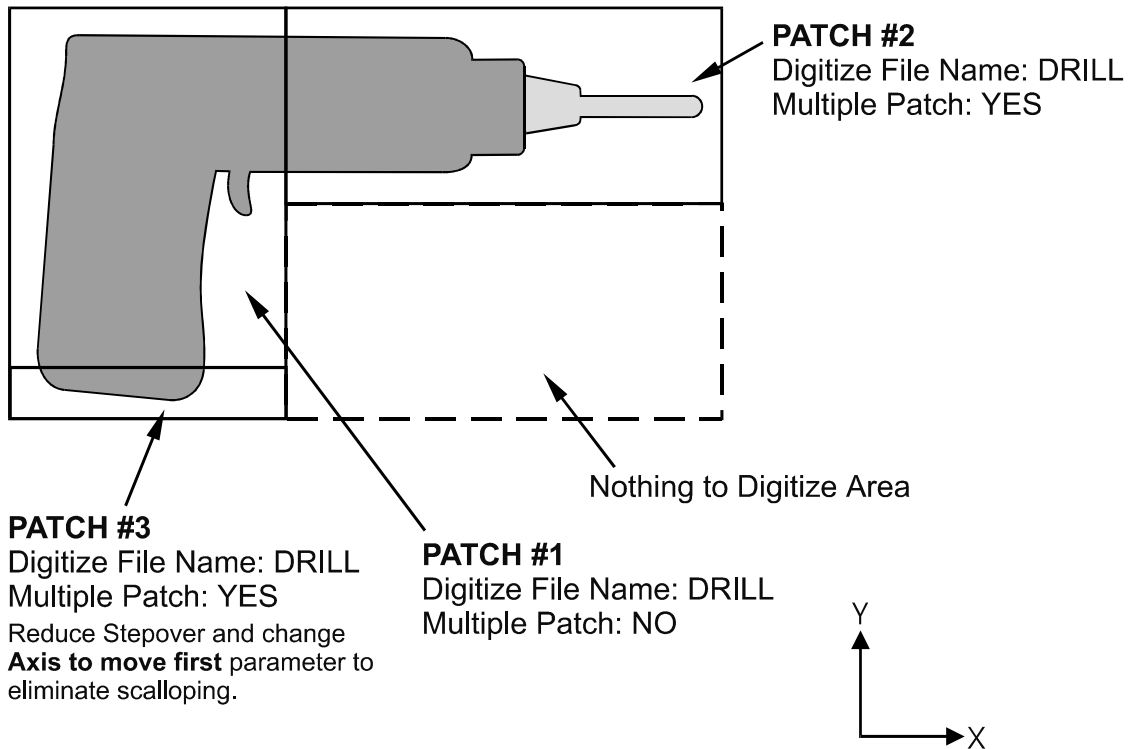


2. A digitizing patch can be located anywhere in the coordinate system. The digitizing starting point is referenced from the part zero. For example, setting up digitizing, as shown in the figure on the right below, will record the first point at (X5, Y5, Z1) and the last point at (X7, Y7, Z1). If the 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.

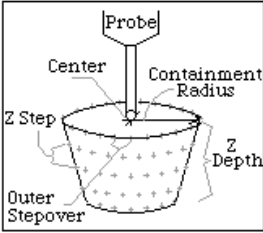
4. Multiple patches are useful in the following situations: completing a canceled digitize run, digitizing parts with large areas that contain nothing to be digitized (shown below), and patching vertical walls to eliminate scallops caused by the cutting tool.



The drill shown on the previous page is L-shaped. Therefore, it can be digitized faster using two 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 2) Jog Probe Tip to Center of Well 3) Edit Radial Digitize Parameters 4) Press CYCLE START to Begin		Containment Radius: 1.0000 Z Patch Depth: 1.0000 Z Step: 0.0100 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	<div style="display: inline-block; border: 1px solid black; padding: 2px 5px;">Save F10</div> <div style="display: inline-block; border: 1px solid black; padding: 2px 5px; margin-left: 10px;">Cancel ESC</div>

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, and an extension of .DOC when translated for import into Mastercam[®] Level 2.

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 angles section below).

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.

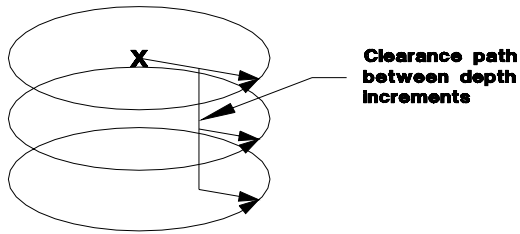
Move Between Levels: This field is enabled only if Partial and CCW or CW option is selected. It indicates the move between Z levels on replay of a partial sector radial digitize file. This move may now be done in three different ways: Clearance, which goes to the clearance height as in previous versions,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

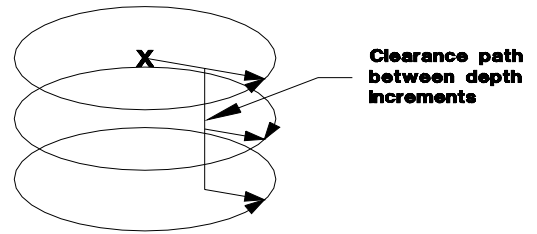
Radial Digitize Notes

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

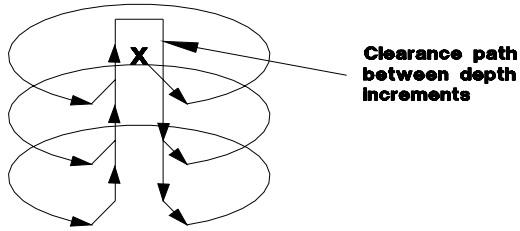
**FULL CONTAINMENT ANGLE
ONE WAY REPLAY PATTERN**



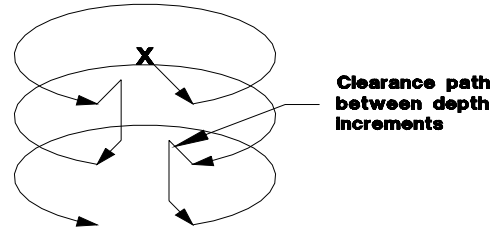
**FULL CONTAINMENT ANGLE
ZIG ZAG REPLAY PATTERN**



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

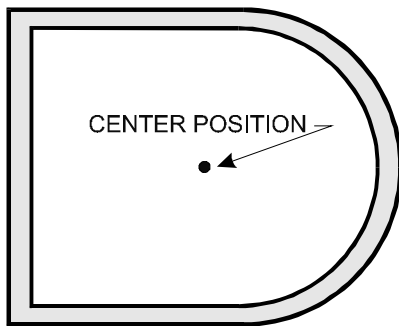


**PARTIAL CONTAINMENT ANGLE
ZIG ZAG DIRECT REPLAY PATTERN**

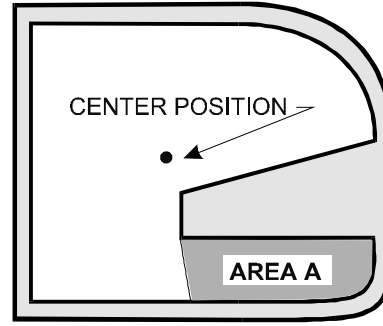


2. When radial digitizing, make sure the probe can fully retract to the center position without obstructions. Observe the two parts below. The cross section on the left has no obstructions that could keep the probe from full retraction to the center position. The cross section on the right does not allow 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

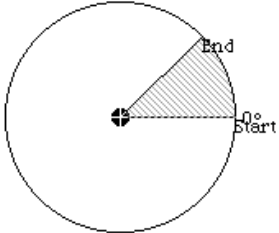


PROBLEM PART



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
Partial Digitizing Sector		
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.		
2) Press F10 to Save changes or ESC to exit.		
	Start Angle:	0.00
	End Angle:	45.00
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 teach in 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.

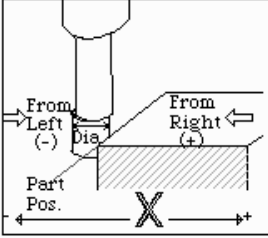
CHAPTER 8

Probing

NOTE: Refer to the Probe Parameters section 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.

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
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
X	0.0000	0.0000
		Approach From Right(+)
Nxt Axs F1	Prv WCS F2	Nxt WCS F3
Auto F4	Probe F5	WCS F9
		Set F10
		Cancel ESC

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 <F2>. Then select the axis you want to probe by pressing <F1>. Both <F1> and <F2> are described in Chapter 4 (Part Setup).

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.

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.

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:

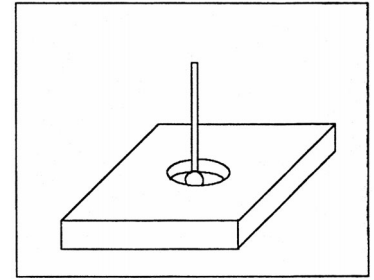
WCS #1 (G54)	Current Position (inches)	Job Name: TEST . CNC
X	+0.0000	Tool: T---H---
Y	+0.0000	Feedrate: 100%
Z	+0.0000	Spindle: 0
B	+0.0000	Feed Hold: Off
		Stopped
Probing Cycles		
Probe diameter (Tool #101):		0.2000
Bore F1	Boss F2	Slot F3
Web F4	In Cnr F5	Out Cnr F6
1 Axis F7	Exit ESC	

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 for more information.

F1 – Bore

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

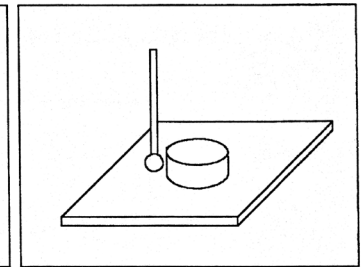
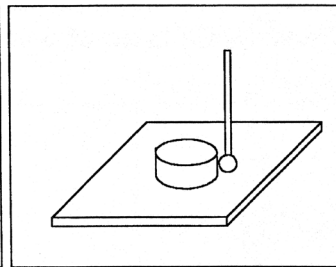
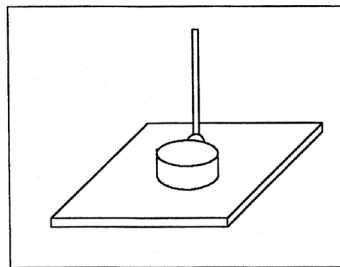
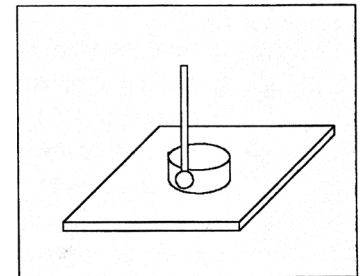
1. Make sure the probe is clear of any obstacles.
2. Manually jog the probe inside the hole. The probe tip should be just below the top edge of the surface.
3. Press <F10> to start the probing.



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.



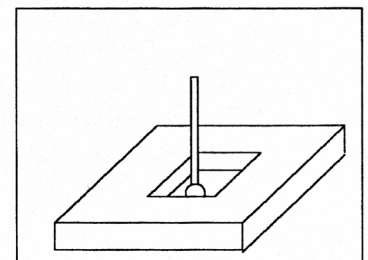
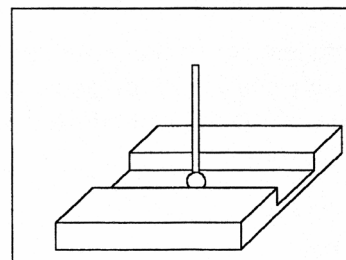
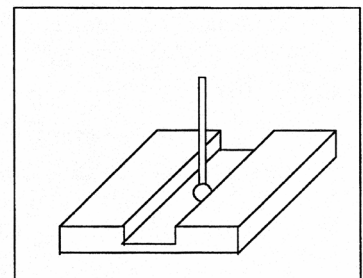
If the Z clearance you entered is too small, the probe will stop and show an error message. Correct the problem by repeating the previous steps. If the approximate diameter you entered is too small, the probe will bounce its way across the top surface until it finds the correct edge.

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.
3. Press <F10> to begin the probing cycle.

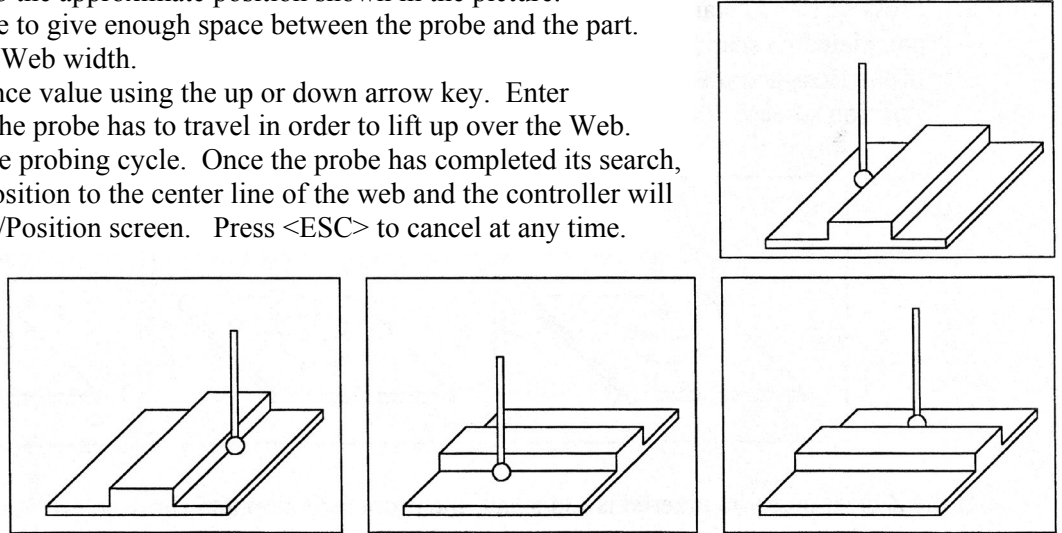
Once the cycle is finished, the probe will be located at the center of the slot.



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:

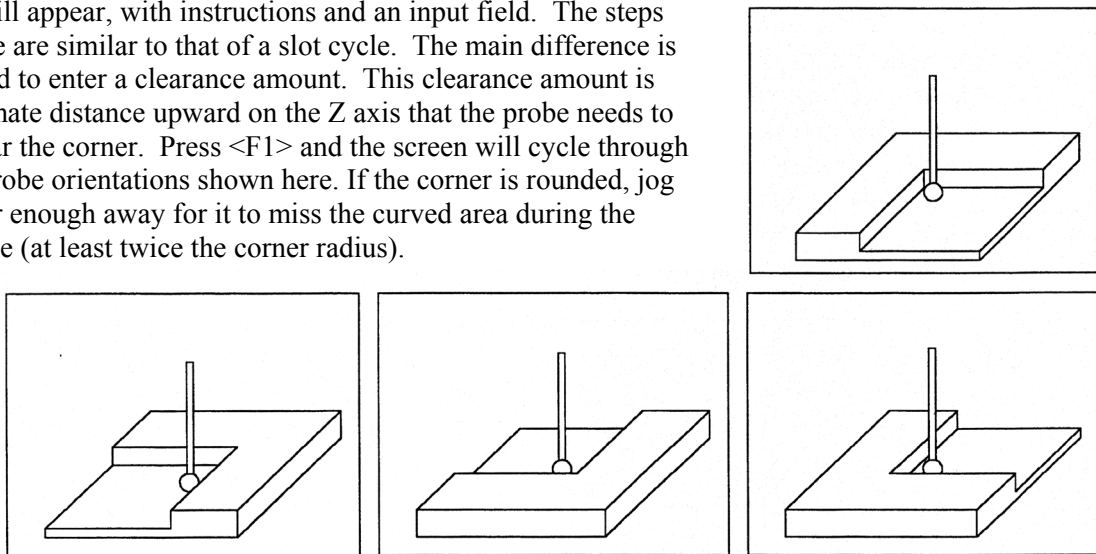
1. Press <F1> to select the orientation of the probe with respect to the Web. You will see one of the screens shown below.
2. Slowly jog the probe to the approximate position shown in the picture. During this jog, be sure to give enough space between the probe and the part.
3. Enter the approximate Web 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.
5. 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 and the controller will return to the Set Part 0/Position screen. Press <ESC> to cancel at any time.



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 its way across the top surface until it finds the correct edge.

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).

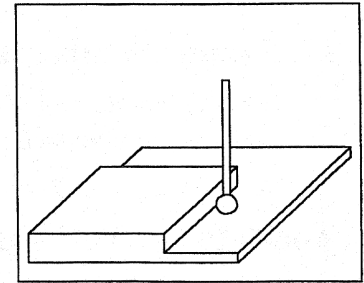
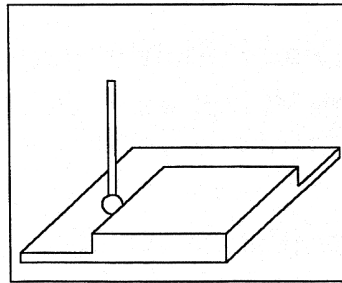
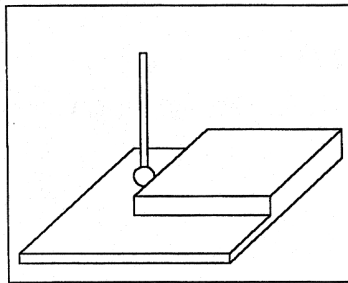
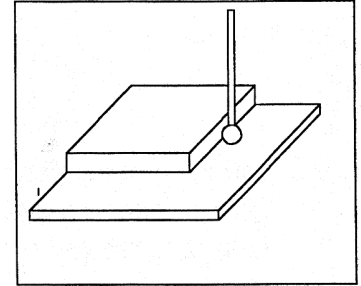


F6 - Out Cnr (Outside Corner)

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.

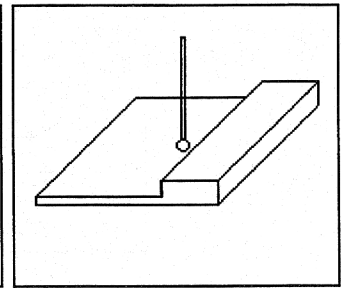
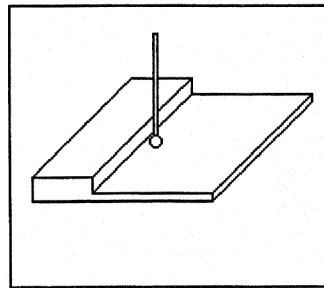
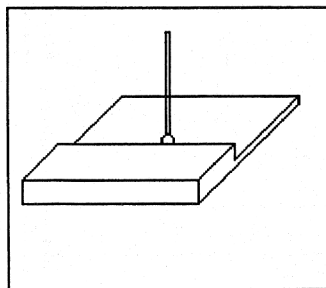
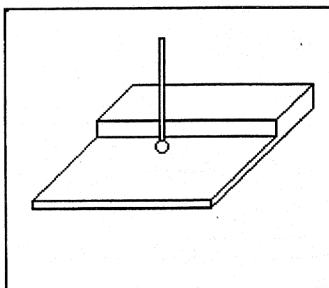
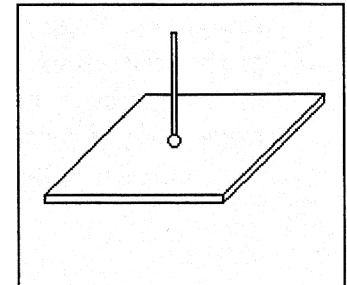


Once the probe has completed its search, it will automatically move above the corner at the Z clearance you specified.

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 probe has completed its search, it will automatically move to the corner and the controller will return to the Set Part 0 / Position screen. Press <ESC> to cancel at any time.

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.

Probe Parameters

Various probing parameters can be set on the Machine Parameters screen (see Chapter 16). 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 in parameter 11. A positive number indicates Closed on contact; a negative number will indicate Open on contact.

Touch Probe Tool Number (Parameter 12)

0 through 200 in parameter 12. This is used to look up the length offset and tip diameter of the probe in the Offset Library.

Recovery Distance (Parameter 13)

Distance in parameter 13 is 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)

The fast probing rate is used for positioning moves and initial surface detection. It 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)

The slow probing rate is used for final measuring moves. It is determined by a speed/accuracy tradeoff. The default setting is 3.5 inches/min.

Maximum Probing Distance (Parameter 16)

Distance in parameter 16 is 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)

Return point number in 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

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.

An offline version of Intercon is available 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 (<F5> CAM option from the CNC7 Main screen) the following screen will be displayed. This is the Intercon Main Screen.

Current Part: FLANGE							Main Screen			
Operation #	Type	X	Y	Z						
0010	Header	0.000	0.000	Home	Intercon Version XXX					
0020	Tool#1	0.000	0.000	Home						
0030	Circ Pkt	0.000	0.000	0.500						
0040	Drill BHC	0.884	-0.884	0.100						
0050	Frame	1.250	1.500	0.500						
0060	End Prog	1.250	1.500	Home						
New F1	Load F2	Save F3	Edit F4	Post F5	SaveAs F6	Delete F7	Graph F8	Quit F9	Setup F10	Cancel ESC

The available operations appear in abbreviated form along the bottom of the screen. If a part program has been loaded, it will be displayed on the left side of the screen.

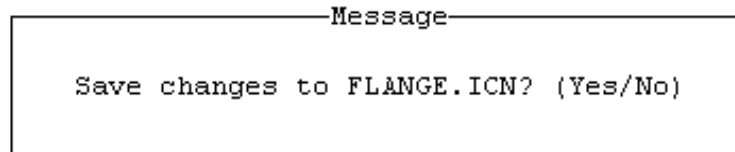
Intercon stores your part designs in files identified with the extension **.ICN**. For example, if you specify the name of a given part design as E_Z_CAM, this part design will be saved on disk in a file called E_Z_CAM.ICN. The ICN files are only readable by Intercon.

F1 - Program New Part

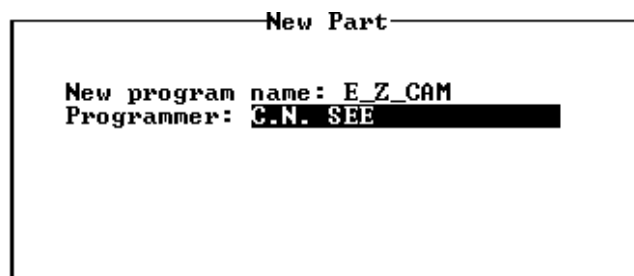
You can start a new part program by pressing <F1>. You will be prompted to specify the name of the new program and the name of the programmer. This information, along with the time and date of the program's creation, will be stored in the header of the program.

F2 - Load Part Program from Disk

When you press <F2>, Intercon will prompt you for the filename of the part program you wish to load. Type in the name of the file without the .ICN extension. For example, if you wish to load a program called E_Z_CAM.ICN, type E_Z_CAM in the space for the filename and press <ENTER>. If you already have an un-saved program in memory named FLANGE and you press <F2> to load a new program, Intercon will ask you if you wish to save the old program before loading the new one:



If you type <N>, Intercon will prompt you for the name of the new file you wish to load. If you type <Y>, Intercon will prompt you for a filename to save the program under, and then prompt you for the name of the new file you wish to load:



Type the name of the program, or use the arrow keys to highlight the program you wish to load. The programs in the current directory are displayed below the program name prompt. You can change disk drives or directories by typing a drive letter followed by a colon, or a directory name. For example, to load a part program from the floppy drive, type "A:\", then type the name of the program.

After the program has been loaded, Intercon briefly displays the message: "Program Loaded", and then returns to the Main Screen.

F3 - Save Current Part

Press <F3> to save the current part program. The current program will be saved under the specified name; the program will not prompt you for a new file name.

F4 - Edit Part

Press <F4> to make changes or additions to the current part. See the section in this chapter titled "Edit Part Program" for a detailed explanation of the many options.

F5 - Post Part Program to G-Codes

When you press <F5> Intercon generates CNC codes for the current part program (on-line version only), then returns to the Main Screen where you can press CYCLE START to run the job. If you have not yet saved your changes to the part program, Intercon will ask you if you wish to save the program before returning to CNC7.

F6 - Save As

When you press <F6> Intercon prompts you to save the current program under a different file name as the current one selected. The name of the file can be any combination of letters or numbers up to eight characters long. Symbols (-+=\[]{}':"/;>?) may not be used in a filename; however, underscore '_' is valid.

When you type in the filename and press <ENTER>, Intercon saves the current part design onto the disk drive with the specified name plus the .ICN extension. For example, if you called your part TESTPART, it would be saved on disk as TESTPART.ICN.

You can save the part program on another disk drive or in another directory by typing the drive letter or directory name. For example, to save the program on a floppy disk, type "A:\" followed by the name of the program.

After the program has been saved, Intercon automatically generates CNC codes for the part (on-line version only).

F7 - Delete Part Program

When you press <F7>, Intercon prompts you for the name of a part program to delete. Delete unwanted part programs to save space on your disk and to simplify the list of files that is displayed.

F8 - Graphics

Pressing <F8> provides access to the graphical display operations of Intercon. For more information, refer to the "Edit Part Program" section.

F9 - Quit

Pressing <F9> exits Intercon and returns you to the CNC7 Main Screen. Pressing <ESC> from this screen will also exit Intercon and return you to the CNC7 Main Screen.

F10 - Setup

Before starting to use Intercon, you must first set up some computer and machine parameters. From the Intercon Main Menu, press <F10> to enter the Setup Menu:

```
-----Intercon Setup-----
Comment Generation      : Enabled
Clearance Amount       : 0.500
Spindle/Coolant Delay  : 0.10
Corner Feedrate Override : 50.00 %
Modal Linear           : Yes
Modal Arc              : No
Modal Drill/Bore/Tap   : Yes
```

Use the up and down arrow keys to move from field to field. Clearance Amount and Spindle/Coolant delay require you to type in a value; the other fields have fixed values which you may toggle 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 let the operator decide whether he wants to insert one of these operations after another once the first one is accepted. If one of these operations is enabled, the operator must press <ESC> to exit after accepting the operation.

Alt-M - Display Free Memory

When you press <Alt-M>, Intercon briefly displays the amount of free memory. If you are working with a very large part program, this will tell you how close you are to running out of memory.

Edit Part Program

The Edit Part option of Intercon allows you to program parts to be machined. A part is described by a series of machining operations such as lines, arcs, positioning moves, and canned cycles. Math assistance is provided to help you fill in the dimensions for the operations, and a graphic view of the operations programmed is provided for verification. When you press <F4> from the Main Screen, the Edit Part screen appears:

Current Part: E_Z_PART						Edit Part						
Operation #	Type	X	Y	Z	End Position	Status						
0010	Header	0.0000	0.0000	Home		Tool #:0	Diameter: 0.0000	Length: 0.0000	Cutter Compensation:None	Feedrate: 0.0000	Coolant: Off	Spindle Speed:Off
0020	End Prog	0.0000	0.0000	Home								
Modify F1	Insert F2	Delete F3	Undel F4	Copy F5	Move F6		Graph F8				Cancel ESC	

The name of the program is shown above the left side of the screen.

The left window shows the block number, type, and ending position for each operation in the program. The currently selected operation is highlighted.

The lower right side of the screen shows the status box, which displays information about the size of the current tool, the current cutter compensation, the current feedrate, and the current status of the spindle and coolant.

F1 - Modify Operation

Press <F1> to make changes to the current operation. You will see the operation editing screen for the current operation. See the Insert Operation section below for a description of each type of operation.

F2 - Insert Operation

Press <F2> to insert new operations in the program. The new operations will be inserted before the current operation. To add operations at the end of the program, press <END> to highlight the End of Program operation, and then press <F2>. When you press <F2> Intercon will display the Insert screen. See the Insert Operation section for details.

F3 - Delete Operation

Press <F3> to delete the current operation, removing it from the program. The next operation will be highlighted as the new current operation.

F4 - Undelete Operation

Press <F4> to restore the last operation which was deleted. The restored operation will be inserted before the current operation, not necessarily in the same place it was in before. Thus <F3> and <F4> may be used to "cut and paste", moving an operation from one location to another.

F5 - Copy Operation(s)

Press <F5> to make copies of one or more operations at a specified location in your part program. You will see the following screen:

Current Part: E_Z_CAM						Copy Operation(s)				
Operation #	Type	X	Y	Z	End Position					
0010	Header	0.0000	0.0000	Home		Start Block	:	0040		
0020	Rapid	1.0000	1.0000	1.0000		End Block	:	0050		
0030	Rapid	4.0000	2.0000	1.0000		Destination Block	:	0080		
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	Drill BHC	8.5562	6.2881	0.1000						
0080	Bore BHC	1.3365	-0.6810	0.1000						
0090	End Prog	1.3365	-0.6810	Home						
									Accept F10	Cancel ESC

Start Block: Selects the first operation in the block of operations to copy.

End Block: Selects the last operation in the block of operations to copy. This operation must not precede the start block.

Destination Block: Selects the location where the copy of the block of operations will begin. All operations placed after this line will be moved down.

F6 - Move Operation(s)

Press <F6> to move one or more operations at a specified location in your part program. You will see the following screen:

Current Part: E_Z_CAM						Move Operation(s)				
Operation #	Type	X	Y	Z	End Position					
0010	Header	0.0000	0.0000	Home		Start Block	:	0030		
0020	Rapid	1.0000	1.0000	1.0000		End Block	:	0060		
0030	Rapid	4.0000	2.0000	1.0000		Destination Block	:	0080		
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	Drill BHC	8.5562	6.2881	0.1000						
0080	Bore BHC	1.3365	-0.6810	0.1000						
0090	End Prog	1.3365	-0.6810	Home						
									Accept F10	Cancel ESC

The fields in this feature work just like the fields in the Copy feature, except that after accepting this operation the original operations are removed:

Start Block: Selects the first operation in the block of operations to move.

End Block: Selects the last operation in the block of operations to move. This operation must not precede the start block.

Destination Block: Selects the location where the copy of the block of operations will begin. All operations placed after this line will be moved down.

F8 - Graphics

Press <F8> to display a graphic preview of the part. See the section titled "Graphics" for details.

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:

Current Part: E_Z_PART						Insert									
Operation #	Type	X	Y	Z	End Position										
0010	Header	0.0000	0.0000	Home											
0020															
0030	End Prog	0.0000	0.0000	Home											
						Rapid F1	Line F2	Arc F3	Tool F4	Cycles F5	Other F6	Comp F7	Graph F8	Subpgm F9	Cancel ESC

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:

Current Part: PRETZEL						Edit Operation					
Operation #	Type	X	Y	Z	End Position						
0010	Header	0.0000	0.0000	Home							
0020	Comment	0.0000	0.0000	Home							
0030	Rapid	0.0000	-5.0000	0.1000							
0040	Line	0.0000	-5.0000	0.0000							
0050	Arc CW	0.0025	-5.2500	-0.2500							
0060	Line	0.0075	-5.7500	-0.2500							
0070	Arc CW	0.0100	-6.0000	0.0000							
0080	Arc CCW	0.0125	-5.7500	-0.2500							
0090	Line	0.0175	-5.2500	-0.2500							
0100	Arc CCW	0.0200	-5.0000	0.0000							
0110	Rotate	5.0000	0.0200	0.0000							
0120	Comment	5.0000	0.0200	0.0000							
0130	Line	5.0000	0.0000	0.0000							
0140	Arc CCW	5.2500	0.0025	-0.2500							
0150	Line	5.7500	0.0075	-0.2500							
0160	Arc CCW	6.0000	0.0100	0.0000							
0170	Arc CW	5.7500	0.0125	-0.2500							
0180	Line	5.2500	0.0200	-0.2500							
						End:	X:	0.0000			
							Y:	-5.0000			
							Z:	0.1000			
						Angle	:	270.00°			
						Length	:	5.0000			
Abs/Inc F1						Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC

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.

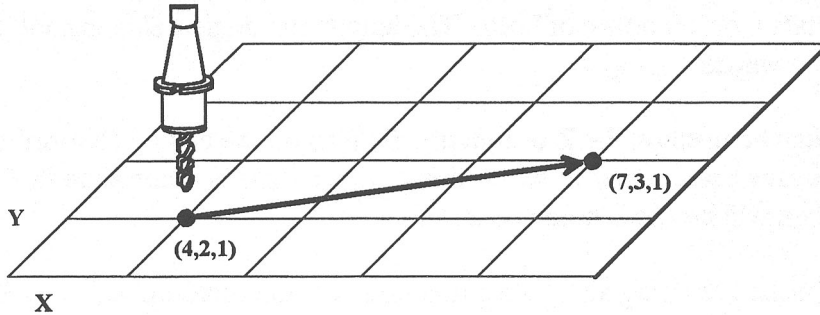
NOTE: The Rapid traverse operation can have rotary fields, if you have a rotary fourth axis. The rotary field descriptions are the same as that of the Linear Mill operation. See "F2 - Linear Mill" below.

F2 - Linear Mill

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

Current Part: E_Z_PART					Edit Operation					
Operation #	Type	X	End Position Y	Z	#0040 Linear mill					
0010	Header	0.0000	0.0000	Home	End:	X:	7.0000			
0020	Rapid	1.0000	1.0000	1.0000		Y:	3.0000			
0030	Rapid	4.0000	2.0000	1.0000		Z:	1.0000			
0040					Angle	:	18.43°			
0050	End Prog	4.0000	2.0000	Home	Length	:	3.1623			
					Connect Radius	:	0.0000			
					Feedrate	:	20.0000			
					Rotary axis B	:				
					Degrees	:	0°	INC		
					Minutes	:	0'	INC		
					Seconds	:	0"	INC		
					Decimal degrees	:	0.0000°	INC		
Abs/Inc F1					Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC

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.

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 below:

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.

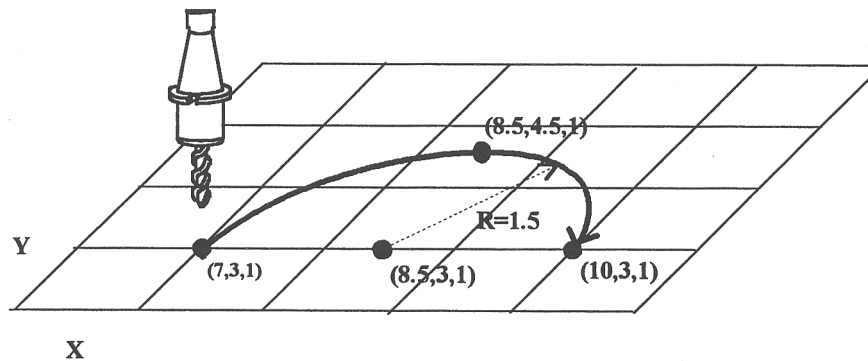
Rotary movement defaults to zero degrees, incremental. To enter an absolute (rather than incremental) rotary position, you must press F1 (Abs/Inc) to toggle to absolute.

F3 - Arc Mill

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

Current Part: E_Z_PART					Edit Operation						
Operation #	Type	X	Y	Z	#0050 Arc mill end						
0010	Header	0.0000	0.0000	Home	Arc Type	:	EP&R				
0020	Rapid	1.0000	1.0000	1.0000	Mid:	X:	8.5000				
0030	Rapid	4.0000	2.0000	1.0000		Y:	4.5000				
0040	Line	7.0000	3.0000	1.0000		Z:	1.0000				
0050					End:	X:	10.0000				
0060	End Prog	7.0000	3.0000	Home		Y:	3.0000				
						Z:	1.0000				
					Center:	X:	8.5000				
						Y:	3.0000				
						Z:	1.0000				
					Angle	:	180.00 °				
					Radius	:	1.5000				
					Plane	:	XY				
					Direction	:	CW				
					Connect Radius	:	0.0000				
					Feedrate	:	20.0000				
					Angle <= 180°	:	Yes				
					Math	Solutn	Graph	Teach	Accept	Cancel	
					F6	F7	F8	F9	F10	ESC	

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.

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:

Current Part: E_Z_CAM					Edit Operation					
Operation #	Type	X	Y	Z	#0060 Tool change					
0010	Header	0.0000	0.0000	Home	Tool Number	:	1			
0020	Rapid	1.0000	1.0000	1.0000	Description:	:	.325 end mill			
0030	Rapid	4.0000	2.0000	1.0000	Position	:	X:	0.0000		
0040	Line	7.0000	3.0000	1.0000		:	Y:	0.0000		
0050	Arc CCW	10.0000	3.0000	1.0000	Tool H Offset	:	1			
0060					<Tool Height	:	0.0000			
0070	End Prog	10.0000	3.0000	Home	Tool D Offset	:	1			
					Tool Diameter	:	0.3250			
					Spindle Speed	:	3000			
					Spindle Direction	:	CW	<M3>		
					Coolant Type	:	Flood	<M8>		
					Actual Tool Change:	:	Yes			

		Toggle F3			Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC
--	--	-----------	--	--	---------	-----------	----------	----------	------------	------------

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:

Operation		End Position			#0070
#	Type	X	Y	Z	
0010	Header	0.0000	0.0000	Home	
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					
0080	End Prog	0.0000	0.0000	Home	

Drill F1	Bore F2	Tap F3	Face F4	R Pckt F5	C Pckt F6	Frame F7				Cancel ESC
-------------	------------	-----------	------------	--------------	--------------	-------------	--	--	--	---------------

Canned Cycle Introduction #1: Using Bolt Hole Circle Pattern (Drilling, boring, tapping)

If you elect to use the Bolt Hole Circle pattern with a canned cycle, you will see a screen similar to this one (it will vary depending on the type of canned cycle):

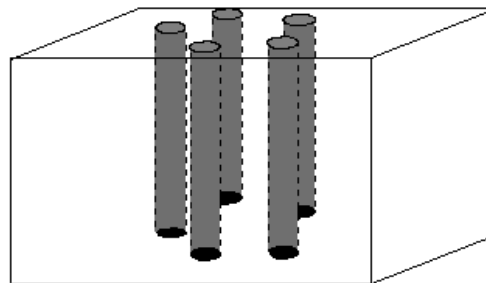
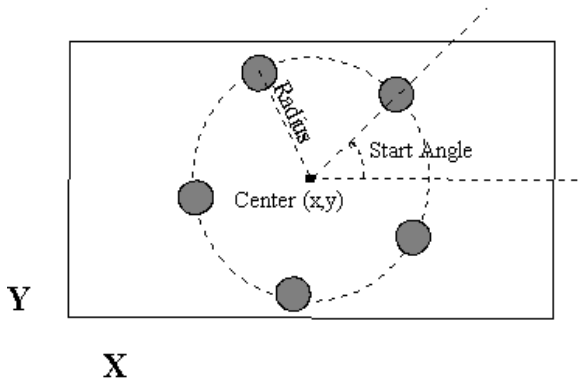
Current Part: E_Z_CAM					Edit Operation						
Operation #	Type	X	Y	Z	#0070 Drill bolt holes						
0010	Header	0.0000	0.0000	Home	Cycle Type	:	Drilling				
0020	Rapid	1.0000	1.0000	1.0000	Center:	X:	4.0000	Y:	-8.0000		
0030	Rapid	4.0000	2.0000	1.0000	Surface Height	:	0.1000				
0040	Line	7.0000	3.0000	1.0000	Clearance Height	:	0.1000	INC			
0050	Arc CCW	10.0000	3.0000	1.0000	'Rapid To' Depth	:	0.1000	INC			
0060	Tool#1	0.0000	0.0000	Home	Depth:	Total:	0.5000	INC			
0070					Plunge Rate	:	20.0000				
0080	End Prog	0.0000	0.0000	Home	Dwell Time	:	1.0000				
					Number of Holes	:	5				
					Radius	:	1.5000				
					Start Angle	:	45.00	°			
					Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC	

All canned cycle operations using the Bolt Hole Circle pattern are identical to their equivalent using the Single Hole pattern. The use of the Bolt Hole Circle pattern, however, offers the option to drill more than one hole in a pattern dictated by the three new fields in the menu: number of holes, radius, and start angle.

This pattern is explained graphically in the following figure:

BOLT HOLE CIRCLE PATTERN

Number of holes: 5

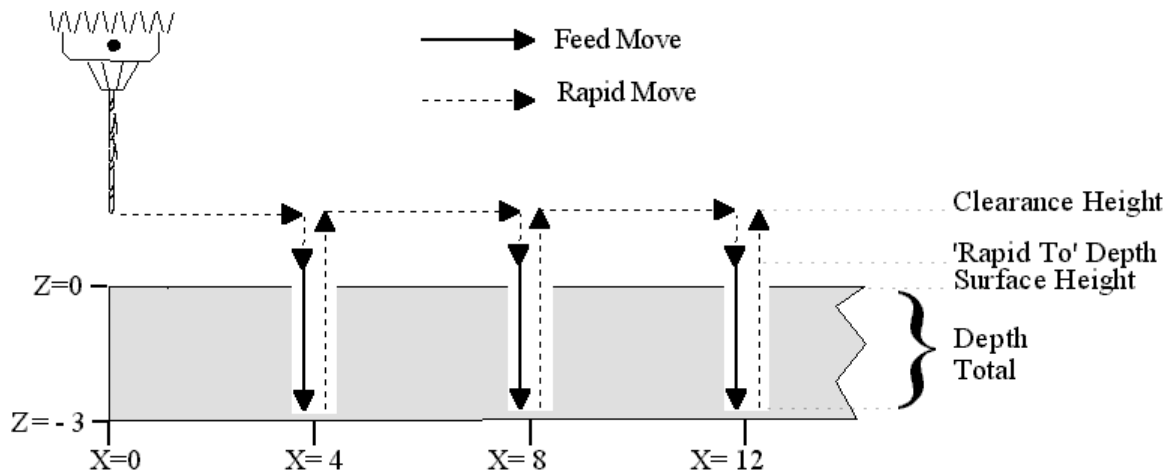


Canned Cycle Introduction #2: Linear Repetition Of Operations (Drilling, Boring, Tapping)

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:

Current Part: E_Z_CAM					Edit Operation					
Operation #	Type	X	Y	Z	#0070 Drill					
0010	Header	0.0000	0.0000	Home	Cycle Type	:	Drilling			
0020	Rapid	1.0000	1.0000	1.0000	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 CCW	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					Depth:	Total:	0.5000	INC		
0080	End Prog	0.0000	0.0000	Home	Plunge Rate	:	20.0000			
					Dwell Time	:	1.0000			
					Number of Holes	:		5		
					Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC

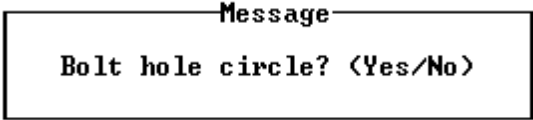
The numbers in the fields on the screen correspond to the following example, shown here graphically:



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.

After pressing <F1> (Drill) from the Canned Cycle Menu, the message will be displayed:

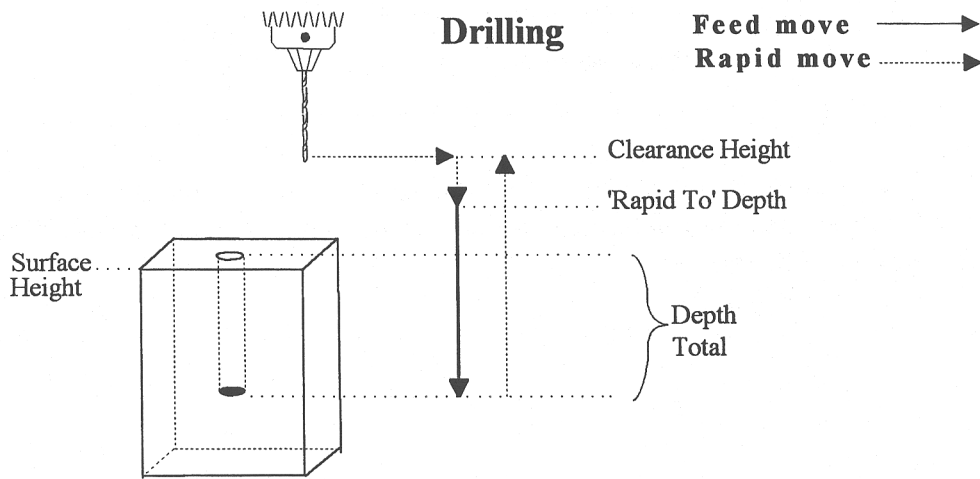


Answer YES if you want to perform the drilling of a series of holes using the Bolt Hole Circle pattern. Answer NO otherwise. For details about drilling using the Bolt Hole Circle pattern refer to the Canned Cycle Introduction #1.

If you answer NO to the question above, you will see the following screen:

Current Part: E_Z_CAM					Edit Operation					
Operation #	Type	X	Y	Z	#0070 Drill					
0010	Header	0.0000	0.0000	Home	Cycle Type	:	Drilling			
0020	Rapid	1.0000	1.0000	1.0000	Position	X:	4.0000			
0030	Rapid	4.0000	2.0000	1.0000		Y:	-8.0000			
0040	Line	7.0000	3.0000	1.0000	Surface Height	:	0.1000			
0050	Arc CCW	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					Depth:	Total:	0.5000	INC		
0080	End Prog	0.0000	0.0000	Home	Plunge Rate	:	20.0000			
					Dwell Time	:	1.0000			
					Math	Solutn	Graph	Teach	Accept	Cancel
					F6	F7	F8	F9	F10	ESC

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.

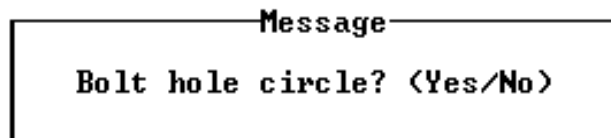
Dwell Time: Delay at bottom of hole before starting ascent.

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.

After pressing F1 (Drilling) from the Canned Cycle Menu, the following message will be displayed:

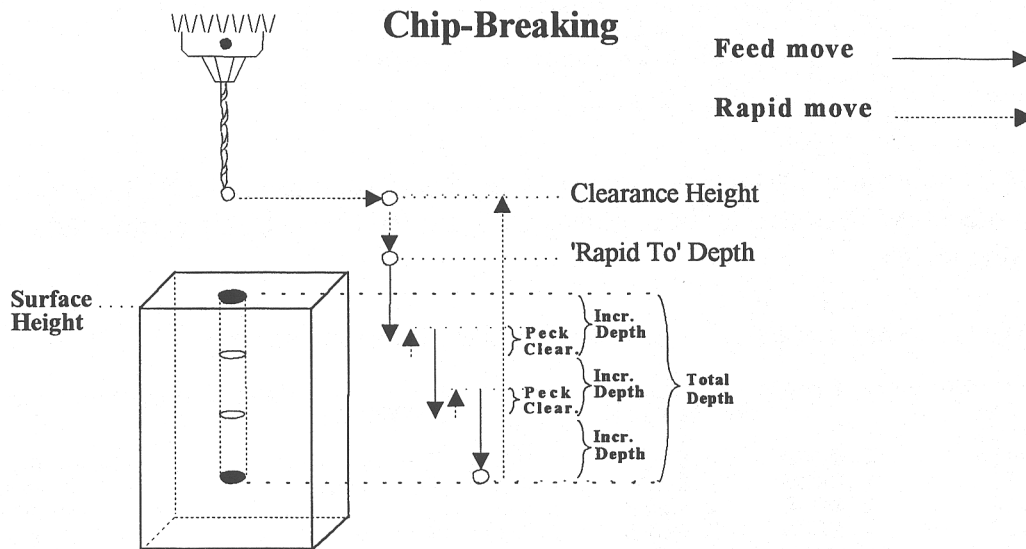


Answer Yes if you want to perform Chip Breaking on a series of holes using the Bolt Hole Circle pattern. Answer No otherwise. For details about Chip Breaking using the Bolt Hole Circle pattern, refer to Canned Cycle Introduction #1.

If you answer No to the question above, you will see the following screen:

Current Part: E_Z_CAM					Edit Operation																																																																								
<table border="1"> <thead> <tr> <th>Operation #</th> <th>Type</th> <th>X</th> <th>Y</th> <th>Z</th> </tr> </thead> <tbody> <tr> <td>0010</td> <td>Header</td> <td>0.0000</td> <td>0.0000</td> <td>Home</td> </tr> <tr> <td>0020</td> <td>Rapid</td> <td>1.0000</td> <td>1.0000</td> <td>1.0000</td> </tr> <tr> <td>0030</td> <td>Rapid</td> <td>4.0000</td> <td>2.0000</td> <td>1.0000</td> </tr> <tr> <td>0040</td> <td>Line</td> <td>7.0000</td> <td>3.0000</td> <td>1.0000</td> </tr> <tr> <td>0050</td> <td>Arc CCW</td> <td>10.0000</td> <td>3.0000</td> <td>1.0000</td> </tr> <tr> <td>0060</td> <td>Tool#1</td> <td>0.0000</td> <td>0.0000</td> <td>Home</td> </tr> <tr> <td>0070</td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>0080</td> <td>End Prog</td> <td>0.0000</td> <td>0.0000</td> <td>Home</td> </tr> </tbody> </table>					Operation #	Type	X	Y	Z	0010	Header	0.0000	0.0000	Home	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 CCW	10.0000	3.0000	1.0000	0060	Tool#1	0.0000	0.0000	Home	0070					0080	End Prog	0.0000	0.0000	Home	<table border="1"> <thead> <tr> <th colspan="2">#0070 Drill</th> </tr> </thead> <tbody> <tr> <td>Cycle Type</td> <td>: Chip Breaking</td> </tr> <tr> <td>Position</td> <td>X: 4.0000</td> </tr> <tr> <td></td> <td>Y: -8.0000</td> </tr> <tr> <td>Surface Height</td> <td>: 0.1000</td> </tr> <tr> <td>Clearance Height</td> <td>: 0.1000 INC</td> </tr> <tr> <td>'Rapid To' Depth</td> <td>: 0.1000 INC</td> </tr> <tr> <td>Depth: Total</td> <td>: 1.0000 INC</td> </tr> <tr> <td></td> <td>Increment: 0.1000</td> </tr> <tr> <td>Peck Clearance</td> <td>: 0.0500</td> </tr> <tr> <td>Plunge Rate</td> <td>: 20.0000</td> </tr> </tbody> </table>						#0070 Drill		Cycle Type	: Chip Breaking	Position	X: 4.0000		Y: -8.0000	Surface Height	: 0.1000	Clearance Height	: 0.1000 INC	'Rapid To' Depth	: 0.1000 INC	Depth: Total	: 1.0000 INC		Increment: 0.1000	Peck Clearance	: 0.0500	Plunge Rate	: 20.0000
Operation #	Type	X	Y	Z																																																																									
0010	Header	0.0000	0.0000	Home																																																																									
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 CCW	10.0000	3.0000	1.0000																																																																									
0060	Tool#1	0.0000	0.0000	Home																																																																									
0070																																																																													
0080	End Prog	0.0000	0.0000	Home																																																																									
#0070 Drill																																																																													
Cycle Type	: Chip Breaking																																																																												
Position	X: 4.0000																																																																												
	Y: -8.0000																																																																												
Surface Height	: 0.1000																																																																												
Clearance Height	: 0.1000 INC																																																																												
'Rapid To' Depth	: 0.1000 INC																																																																												
Depth: Total	: 1.0000 INC																																																																												
	Increment: 0.1000																																																																												
Peck Clearance	: 0.0500																																																																												
Plunge Rate	: 20.0000																																																																												
					Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC																																																																			

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.

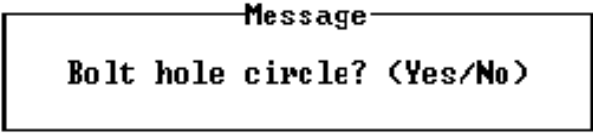
Peck Clearance: Distance the tool retracts before drilling the next peck.

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

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.

After pressing F1 (Drilling) from the Canned Cycle Menu, the following message will be displayed:

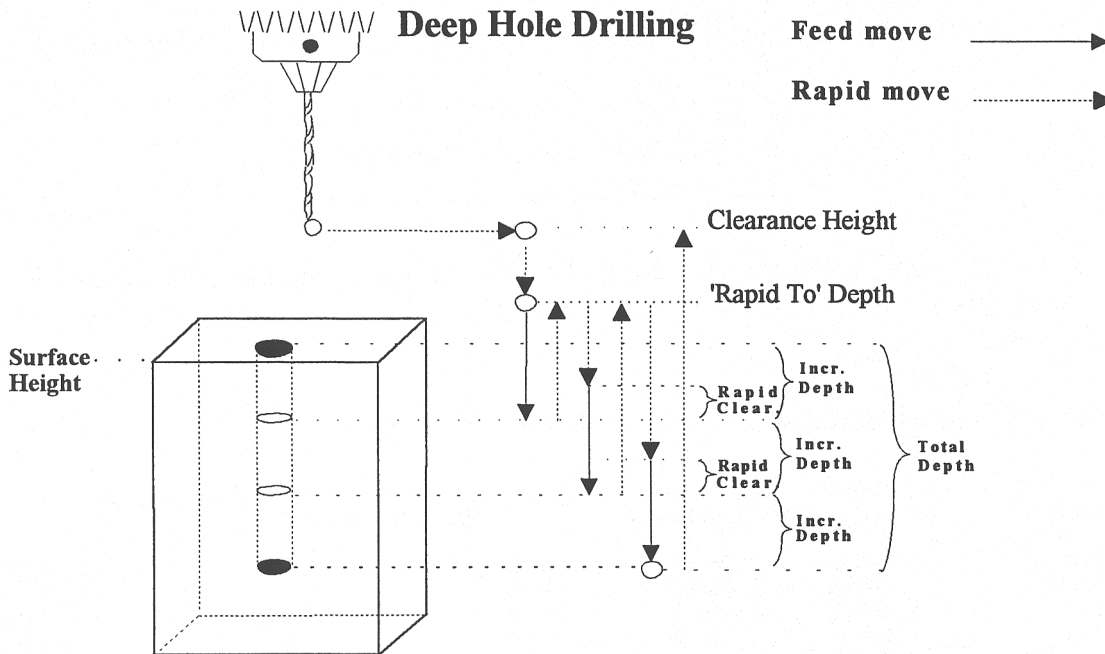


Answer Yes if you want to perform the drilling of a series of holes using the Bolt Hole Circle pattern. Answer No otherwise. For details about drilling using the Bolt Hole Circle pattern refer to Canned Cycle Introduction #1.

If you answer No to the question above, you will see the following screen:

Current Part: <u>E_Z_CAM</u>				Edit Operation					
0010	Header	0.0000	0.0000	Home	#0070 Drill Cycle Type : Deep Hole Position X: 4.0000 Y: -8.0000 Surface Height : 0.1000 Clearance Height : 0.1000 INC 'Rapid To' Depth : 0.1000 INC Depth: Total: 2.0000 INC Increment: 0.2000 Rapid Clearance : 0.0500 Plunge Rate : <u>-20.0000</u>				
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 CCW	10.0000	3.0000	1.0000					
0060	Tool#1	0.0000	0.0000	Home					
0070	0070								
0080	End Prog	0.0000	0.0000	Home					
				Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC

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: How close the cutter approaches the workpiece between rapid plunges.

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

Boring (F2 in the Canned Cycle Menu)

If you press <F2> (Boring) from the Canned Cycle Selection Menu, the following message appears:

<p>Message</p> <p>Bolt hole circle? <Yes/No></p>
--

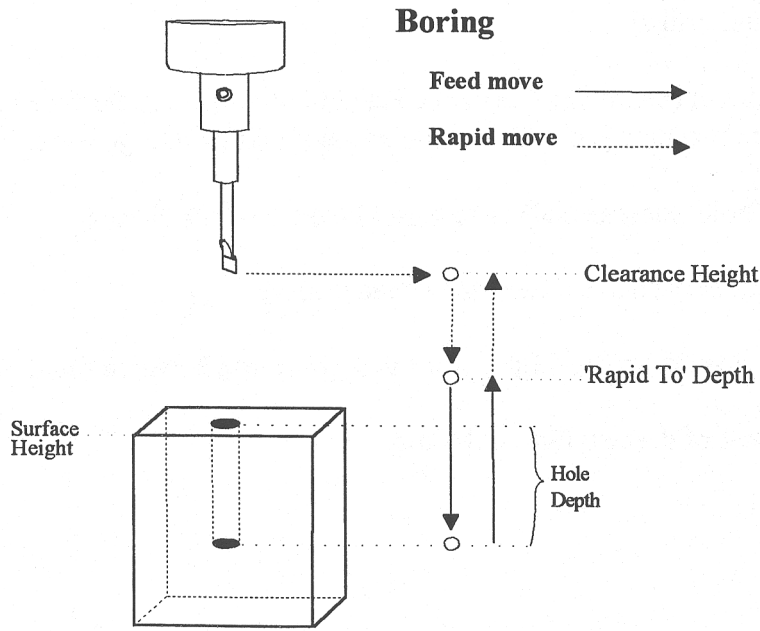
Answer Yes if you want to perform the boring of a series of holes using the Bolt Hole Circle pattern. Answer No otherwise. For details about boring using the Bolt Hole Circle pattern refer to Canned Cycle Introduction #1.

If you answer No to the question above, you will see the following screen:

Current Part: E_Z_CAM				Edit Operation	
Operation #	Type	X	Y	Z	#0080 Bore
0010	Header	0.0000	0.0000	Home	Position X: 4.0000
0020	Rapid	1.0000	1.0000	1.0000	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 CCW	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	Drill BHC	5.3365	-8.6810	0.2000	Plunge Rate : 20.0000
0080					Dwell Time : 1.0000
0090	End Prog	5.3365	-8.6810	Home	

					Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC
--	--	--	--	--	---------	-----------	----------	----------	------------	------------

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 rapid moves before beginning to drill the hole at the specified Plunge Rate.

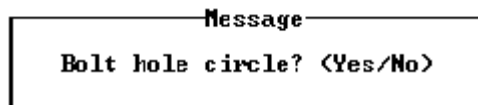
Hole Depth: Depth of hole (incremental) as measured from Surface Height.

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

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

Tapping (F3 in the Canned Cycle Menu)

If you press <F3> (Tapping) at the Canned Cycle Selection Menu, the following message is displayed:



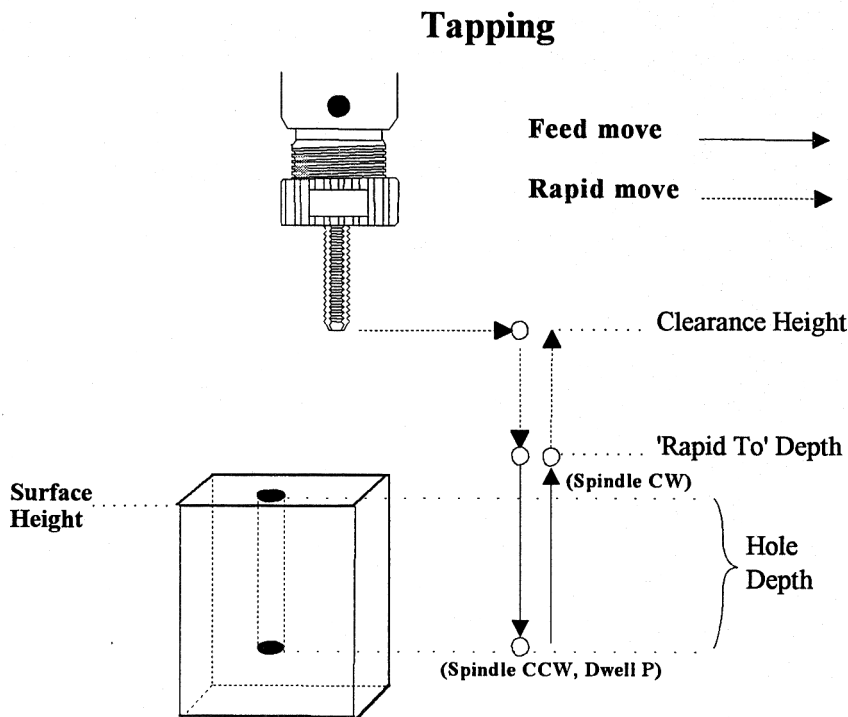
Answer Yes if you want to perform the tapping of a series of holes using the Bolt Hole Circle pattern. Answer No otherwise. For details about tapping using the Bolt Hole Circle pattern refer to Canned Cycle Introduction #1.

If you answer No to the question above, you will see the following screen:

Current Part: E_Z_CAM					Edit Operation					
Operation					#0090 Tap					
#	Type	X	Y	Z						
0010	Header	0.0000	0.0000	Home	Tap Head Type	:	Rigid			
0020	Rapid	1.0000	1.0000	1.0000	<Spindle Direction:	Off>				
0030	Rapid	4.0000	2.0000	1.0000	Position	X:	0.0000			
0040	Line	7.0000	3.0000	1.0000		Y:	0.0000			
0050	Arc CW	10.0000	3.0000	1.0000	Surface Height	:	0.0000			
0060	Tool#1	0.0000	0.0000	Home	Clearance Height	:	0.0000	INC		
0070	Drill BHC	11.5321	1.7144	0.2000	'Rapid To' Depth	:	0.0000	INC		
0080	Bore	4.0000	-8.0000	0.2000	Depth:	Total:	0.0000	INC		
0090						Increment:	0.0000	INC		
0100	End Prog	4.0000	-8.0000	Home	Threads / Inch	:	0.0000			
					<Spindle Speed	:	0			

		Toggle			Math	Solutn	Graph	Teach	Accept
--	--	--------	--	--	------	--------	-------	-------	--------

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 (Plunge Rate = Spindle Speed / 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.

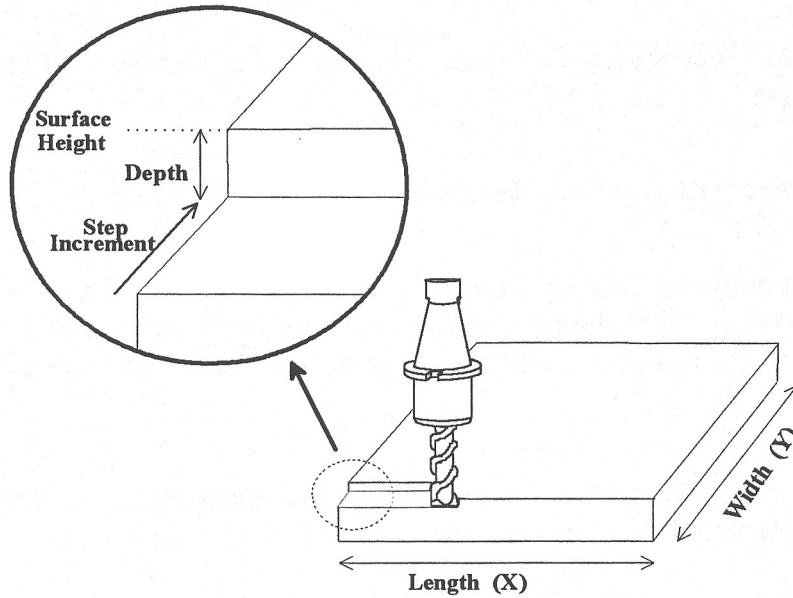
* **NOTE**: When using low gear for tapping, the spindle may turn opposite the direction specified. The operator is responsible for setting the correct spindle speed and direction.

Facing (F4 in the Canned Cycle Menu)

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

Current Part: E_Z_CAM					Edit Operation					
Operation #	Type	X	Y	Z	#0100 Face					
0010	Header	0.0000	0.0000	Home	Start:	X:	3.0000			
0020	Rapid	1.0000	1.0000	1.0000		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 CCW	10.0000	3.0000	1.0000	Width	<Y>:	3.0000	INC		
0060	Tool#1	0.0000	0.0000	Home	Depth	:	0.5000	INC		
0070	Drill BHC	5.3365	-8.6810	0.2000	Step Increment	:	0.1000			
0080	Bore	4.0000	-8.0000	0.2000	Feedrate	:	30.0000			
0090	Tap	4.0000	-8.0000	0.2000	Plunge Rate	:	10.0000			
0100										
0110	End Prog	4.0000	-8.0000	Home						
					Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC

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.

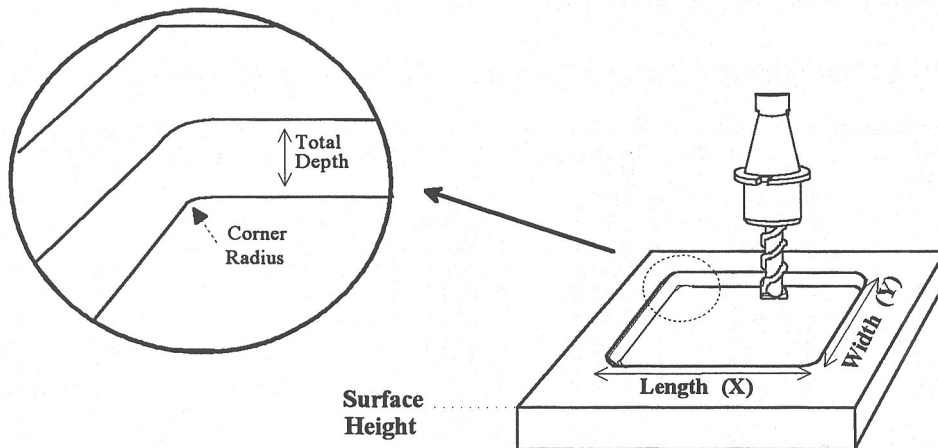
Feedrate: Speed of the cutter during facing.

Rectangular Pocket (F5 in the Canned Cycle Menu)

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

Current Part: E_Z_CAM				Edit Operation						
Operation				#0110 Rectangular pocket						
#	Type	X	Y	Z	Center:	X:	4.0000			
0010	Header	0.0000	0.0000	Home		Y:	-8.0000			
0020	Rapid	1.0000	1.0000	1.0000	Surface Height	:	0.0000			
0030	Rapid	4.0000	2.0000	1.0000	Length <X>	:	5.0000 INC			
0040	Line	7.0000	3.0000	1.0000	Width <Y>	:	3.0000 INC			
0050	Arc CW	10.0000	3.0000	1.0000	Corner Radius	:	0.2500			
0060	Tool#1	0.0000	0.0000	Home	Depth: Total:	:	0.5000 INC			
0070	Drill BHC	5.3365	-8.6810	0.2000	Per Pass:	:	0.1000			
0080	Bore	4.0000	8.0000	0.2000	Plunge Rate:	:	10.0000			
0090	Tap	3.0000	6.0000	0.1000	Plunge Type:	:	Ramped			
0100	Face	3.0000	6.0000	0.1000	Plunge Angle:	:	0.0000			
0110					Rough Cuts	:	Conventional			
0120	Circ Pkt	4.0000	-8.0000	0.2000	Stepover:	:	0.1950			
0130	Frame	1.7500	-6.6250	0.2000	Feedrate:	:	30.0000			
0140	End Prog	1.7500	-6.6250	Home	Finish Pass	:	Climb			
					Amount:	:	0.0200			
					Feedrate:	:	30.0000			
Abs/Inc F1					Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC

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: Stepover: 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.

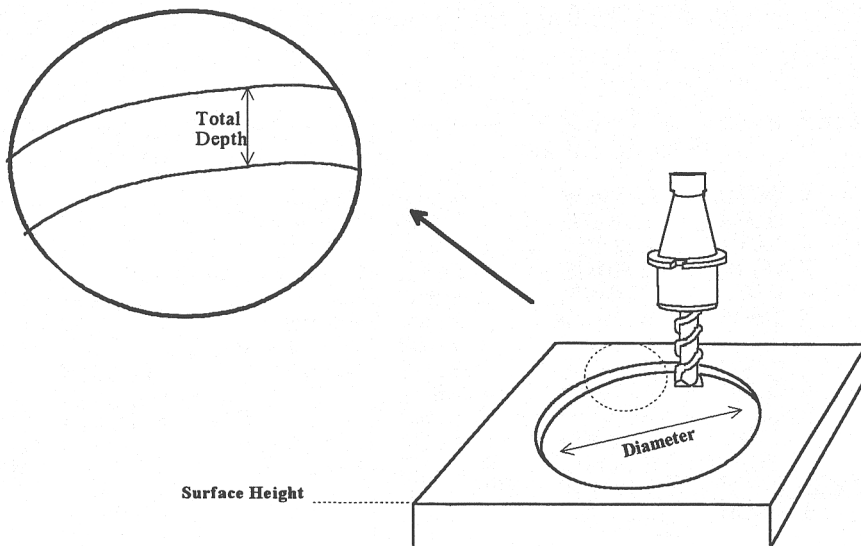
Circular Pocket (F6 in the Canned Cycle Menu)

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

Current Part: E_Z_CAM					Edit Operation							
Operation #	Type	X	Y	Z	#0120 Circular pocket							
0010	Header	0.0000	0.0000	Home	Center:	X:	4.0000	Y:	-8.0000	Surface Height	:	0.0000
0020	Rapid	1.0000	1.0000	1.0000	Diameter	:	2.0000	Cleanout	:	Yes	Depth:	Total:
0030	Rapid	4.0000	2.0000	1.0000								0.5000 INC
0040	Line	7.0000	3.0000	1.0000							Per Pass:	0.1000
0050	Arc CW	10.0000	3.0000	1.0000							Plunge Rate:	10.0000
0060	Tool#1	0.0000	0.0000	Home							Plunge Type:	Straight
0070	Drill BHC	5.3365	-8.6810	0.2000							Plunge Angle:	0.0000
0080	Bore	4.0000	8.0000	0.2000							Rough Cuts	: Conventional
0090	Tap	3.0000	6.0000	0.1000							Stepover:	0.1950
0100	Face	3.0000	6.0000	0.1000							Feedrate:	30.0000
0110	Rect Pkt	4.0000	-8.0000	0.2000							Finish Pass	: Climb
0120											Amount:	0.0200
0130	Frame	1.7500	-6.6250	0.2000							Feedrate:	30.0000
0140	End Prog	1.7500	-6.6250	Home								

Abs/Inc					Math	Solutn	Graph	Teach	Accept	Cancel
F1					F6	F7	F8	F9	F10	ESC

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: 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.

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

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

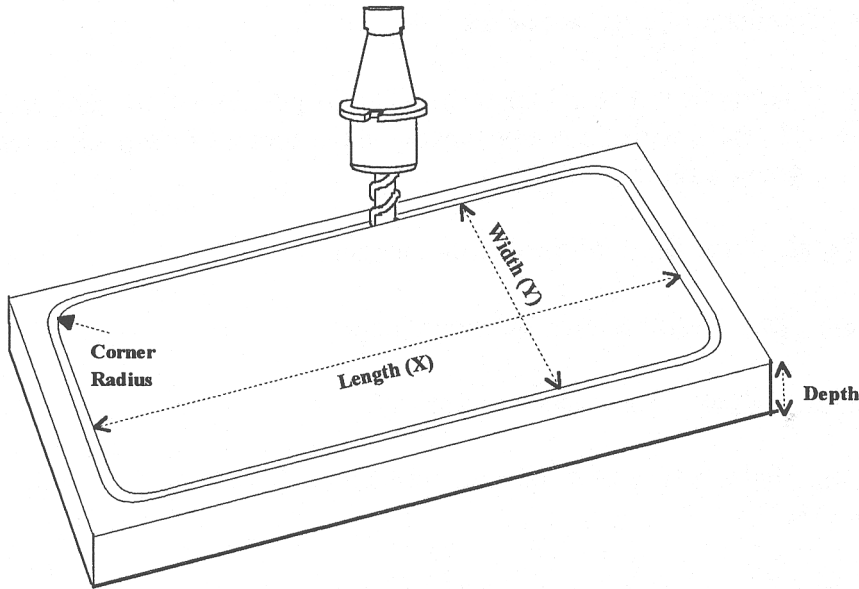
Frame Milling (F7 in the Canned Cycle Menu)

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

Current Part: E_Z_CAM					Edit Operation					
Operation #	Type	X	Y	Z	#0130 Frame mill					
0010	Header	0.0000	0.0000	Home	Frame Type	:	Inside			
0020	Rapid	1.0000	1.0000	1.0000	Center:	X:	4.0000			
0030	Rapid	4.0000	2.0000	1.0000		Y:	-8.0000			
0040	Line	7.0000	3.0000	1.0000	Surface Height	:	0.0000			
0050	Arc CW	10.0000	3.0000	1.0000	Length	<X>:	5.0000	INC		
0060	Tool#1	0.0000	0.0000	Home	Width	<Y>:	3.0000	INC		
0070	Drill BHC	5.3365	-8.6810	0.2000	Corner Radius	:	0.2500			
0080	Bore	4.0000	8.0000	0.2000	Depth:	Total:	0.5000	INC		
0090	Tap	3.0000	6.0000	0.1000		Per Pass:	0.1000			
0100	Face	3.0000	6.0000	0.1000		Plunge Rate:	10.0000			
0110	Rect Pkt	4.0000	-8.0000	0.2000		Plunge Type:	Ramped			
0120	Circ Pkt	4.0000	-8.0000	0.2000		Plunge Angle:	3.0000			
0130					Cut Type	:	Conventional			
0140	End Prog	1.7500	-6.6250	Home	Feedrate	:	30.0000			

		Toggle F3			Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC
--	--	--------------	--	--	------------	--------------	-------------	-------------	---------------	---------------

The parameters on the screen correspond to the following dimensions:



Where:

Frame Type: Selects either Inside or Outside frame type. 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.

Width: Y-axis dimension of the frame mill.

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

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.

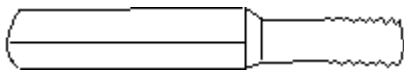
NOTE: To make a circular frame mill of radius R, specify R as the Corner Radius and set the Length and Width parameters equal to 2 x R.

Thread Milling (F8 in the Canned Cycle Menu)

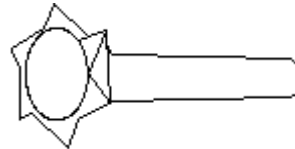
When you press <F8> (Thread Milling) from the canned cycle menu, the following screen is displayed:

Current Part: E_Z_CAM					Edit Operation					
Operation					#0140 Thread milling					
#	Type	X	Y	Z	Center:	X:	4.0000	Y:	-8.0000	
0010	Header	0.0000	0.0000	Home	Diameter	:	0.0000	Threads / Inch	:	20.0000
0020	Rapid	1.0000	1.0000	1.0000	Thread Pitch	:	(0.0500)	Thread Type	:	Right-Hand
0030	Rapid	4.0000	2.0000	1.0000	Thread Direction	:	Top-to-Bottom	Tool Type	:	Single Point
0040	Line	7.0000	3.0000	1.0000	Thread Approach	:	Internal	Clearance Amount	:	0.1000
0050	Arc CW	10.0000	3.0000	1.0000	Clearance Angle	:	0.00	Feedrate	:	30.0000
0060	Tool#1	0.0000	0.0000	Home	Surface Height	:	-0.3000	Clearance Height	:	0.1000 INC
0070	Drill BHC	5.3365	-8.6810	0.2000	'Rapid To' Depth	:	0.1000 INC	Depth: Total:	:	0.5000
0080	Bore	4.0000	-8.0000	0.2000	Plunge Rate	:	10.0000	Number Of Passes	:	1
0090	Tap	4.0000	-8.0000	0.2000						
0100	Face	3.0000	6.0000	0.1000						
0110	Rect Pkt	4.0000	-8.0000	0.1000						
0120	Circ Pkt	4.0000	-8.0000	0.1000						
0130	Frame	1.7500	-6.6625	0.1000						
0140										
0150	End Prog	1.7500	-6.6625	Home						
Abs/Inc					Math	Solutn	Graph	Teach	Accept	Cancel
F1					F6	F7	F8	F9	F10	ESC

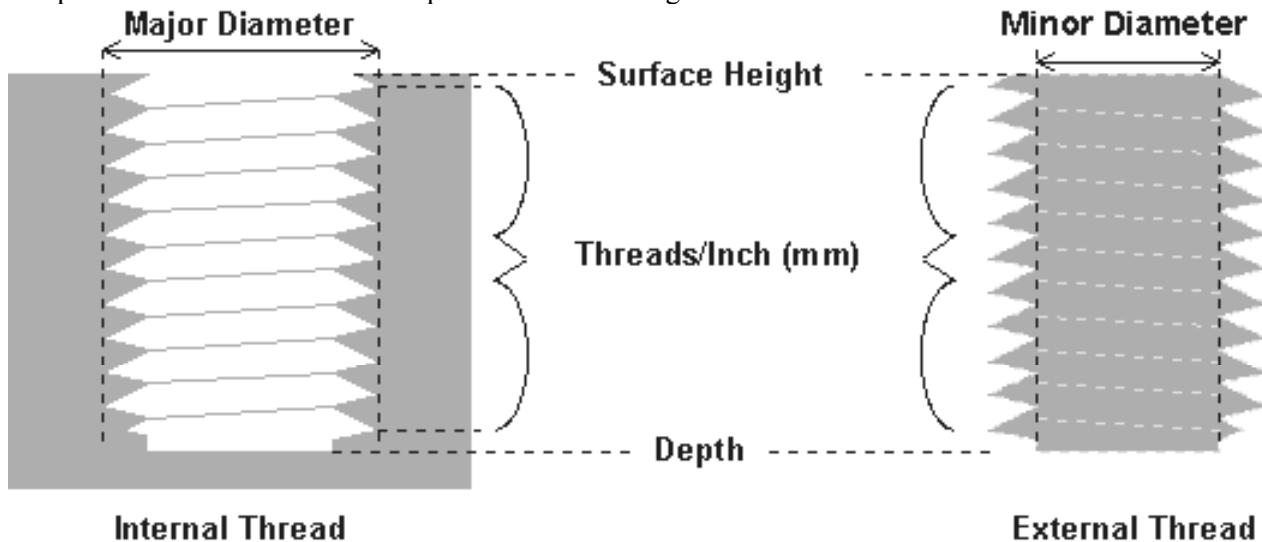
Multiple Thread Mill



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.

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:

Current Part: E_Z_PART					Insert Other				
Operation #	Type	X	Y	Z	#0140				
0010	Header	0.0000	0.0000	Home					
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	Drill BHC	5.3365	-8.6810	0.2000					
0080	Bore	4.0000	-8.0000	0.2000					
0090	Tap	4.0000	-8.0000	0.2000					
0100	Face	3.0000	6.0000	0.1000					
0110	Rect Pkt	4.0000	-8.0000	0.1000					
0120	Circ Pkt	4.0000	-8.0000	0.1000					
0130	Frame	1.7500	-6.6625	0.1000					
0140									
0150	End Prog	1.7500	-6.6625	Home					

Commnt F1	Spindl F2	Coolnt F3	Clamp F4	Z Home F5	Code F6	Rotary F7				Cancel ESC
--------------	--------------	--------------	-------------	--------------	------------	--------------	--	--	--	---------------

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:

Current Part: E_Z_CAM					Edit Operation		
Operation #	Type	X	Y	Z	#0140 M & G codes		
0010	Header	0.000	0.000	Home	M41; SELECT LOW GEAR RANGE		
0020	Rapid	1.000	1.000	1.000			
0030	Rapid	4.000	2.000	1.000	Warning!		
0040	Line	7.000	3.000	1.000	Any M & G codes entered here will be unrecognizable by Intercon		
0050	Arc CW	10.000	3.000	1.000	Careful consideration must be taken before using this function.		
0060	Tool#1	0.000	0.000	Home			
0070	Drill BHC	5.337	-8.681	0.200			
0080	Bore	4.000	-8.000	0.200			
0090	Tap	4.000	-8.000	0.200			
0100	Face	3.000	6.000	0.100			
0110	Rect Pkt	4.000	-8.000	0.100			
0120	Circ Pkt	4.000	-8.000	0.100			
0130	Frame	1.750	-6.662	0.100			
0140							
0150	End Prog	1.750	-6.662	Home			

						Graph F8		Accept F10	Cancel ESC
--	--	--	--	--	--	----------	--	------------	------------

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.

Current Part: E_Z_PART					Edit Operation			
Operation #	Type	X	Y	Z	#0140 Move rotary axis			
0010	Header	0.0000	0.0000	Home	Degrees	:	25°	INC
0020	Rapid	1.0000	1.0000	1.0000	Minutes	:	10'	INC
0030	Rapid	4.0000	2.0000	1.0000	Seconds	:	10"	INC
0040	Line	7.0000	3.0000	1.0000	Decimal degrees	:	25.1694°	INC
0050	Arc CW	10.0000	3.0000	1.0000				
0060	Tool#1	0.0000	0.0000	Home				
0070	Drill BHC	5.3365	-8.6810	0.2000				
0080	Bore	4.0000	-8.0000	0.2000				
0090	Tap	4.0000	-8.0000	0.2000				
0100	Face	3.0000	6.0000	0.1000				
0110	Rect Pkt	4.0000	-8.0000	0.1000				
0120	Circ Pkt	4.0000	-8.0000	0.1000				
0130	Frame	1.7500	-6.6625	0.1000				
0140								
0150	End Prog	1.7500	-6.6625	Home				

Abs/Inc F1					Math F6	Solutn F7	Graph F8		Accept F10	Cancel ESC
------------	--	--	--	--	---------	-----------	----------	--	------------	------------

F7 - Cutter Compensation

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

Current Part: E_Z_PART					Edit Operation				
Operation #	Type	X	Y	Z	#0150 Cutter comp				
0010	Header	0.0000	0.0000	Home	Cutter Comp : Left				
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	Drll BHC	5.3365	-8.6810	0.2000					
0080	Bore	4.0000	-8.0000	0.2000					
0090	Tap	4.0000	-8.0000	0.2000					
0100	Face	3.0000	6.0000	0.1000					
0110	Rect Pkt	4.0000	-8.0000	0.1000					
0120	Circ Pkt	4.0000	-8.0000	0.1000					
0130	Frame	1.7500	-6.6625	0.1000					
0140	M&G Code	1.7500	-6.6625	0.1000					
0150	End Prog	1.7500	-6.6625	Home					

	Toggle F3			Graph F8		Accept F10	Cancel ESC
--	--------------	--	--	-------------	--	---------------	---------------

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:

Current Part: E_Z_PART					Insert Subprogram				
Operation #	Type	X	Y	Z	#0160				
0010	Header	0.0000	0.0000	Home					
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	Drll BHC	5.3365	-8.6810	0.2000					
0080	Bore	4.0000	-8.0000	0.2000					
0090	Tap	4.0000	-8.0000	0.2000					
0100	Face	3.0000	6.0000	0.1000					
0110	Rect Pkt	4.0000	-8.0000	0.1000					
0120	Circ Pkt	4.0000	-8.0000	0.1000					
0130	Frame	1.7500	-6.6625	0.1000					
0140	M&G Code	1.7500	-6.6625	0.1000					
0150	Comp Lft	1.7500	-6.6625	0.1000					
0160	End Prog	1.7500	-6.6625	Home					

DpthRpt F1	Repeat F2	Mirror F3	Rotate F4					Cancel ESC
---------------	--------------	--------------	--------------	--	--	--	--	---------------

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

Current Part: E_Z_PART					Edit Operation					
Operation #	Type	X	End Position Y	Z	#0160 Repeat					
0010	Header	0.0000	0.0000	Home	Start Block	:	0020			
0020	Rapid	1.0000	1.0000	1.0000	End Block	:	0080			
0030	Rapid	4.0000	2.0000	1.0000	Increment	X:	2.0000	Y:	1.0000	
0040	Line	7.0000	3.0000	1.0000	Clearance Height	:	0.1000			
0050	Arc CW	10.0000	3.0000	1.0000	Plunge Rate	:	20.0000			
0060	Tool#1	0.0000	0.0000	Home	Number of Copies	:	10			
0070	Drill BHC	5.3365	-8.6810	0.2000	Skip Copy	:	3.5.7			
0080	Bore	4.0000	-8.0000	0.2000	Skip List:					
0090	Tap	4.0000	-8.0000	0.2000	003	005	007	---	---	---
0100	Face	3.0000	6.0000	0.1000	---	---	---	---	---	---
0110	Rect Pkt	4.0000	-8.0000	0.1000	---	---	---	---	---	---
0120	Circ Pkt	4.0000	-8.0000	0.1000	---	---	---	---	---	---
0130	Frame	1.7500	-6.6625	0.1000	---	---	---	---	---	---
0140	M&G Code	1.7500	-6.6625	0.1000	---	---	---	---	---	---
0150	Comp Lft	1.7500	-6.6625	0.1000	---	---	---	---	---	---
0160										
0170	End Prog	1.7500	-6.6625	Home						
					Math F6	Solutn F7	Graph F8		Accept F10	Cancel ESC

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)

Current Part: E_Z_PART					Edit Operation					
Operation #	Type	X	End Position Y	Z	#0160 Repeat to depth					
0010	Header	0.0000	0.0000	Home	Start Block	:	0020			
0020	Rapid	1.0000	1.0000	1.0000	End Block	:	0080			
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	:	20.0000			
0070	Drill BHC	5.3365	-8.6810	0.2000	Warning!					
0080	Bore	4.0000	-8.0000	0.2000	Operator must program first pass					
0090	Tap	4.0000	-8.0000	0.2000	at depth increment. Plunge must					
0100	Face	3.0000	6.0000	0.1000	be programmed into this pass.					
0110	Rect Pkt	4.0000	-8.0000	0.1000						
0120	Circ Pkt	4.0000	-8.0000	0.1000						
0130	Frame	1.7500	-6.6625	0.1000						
0140	M&G Code	1.7500	-6.6625	0.1000						
0150	Comp Lft	1.7500	-6.6625	0.1000						
0160										
0170	End Prog	1.7500	-6.6625	Home						
					Math F6	Solutn F7	Graph F8		Accept F10	Cancel ESC

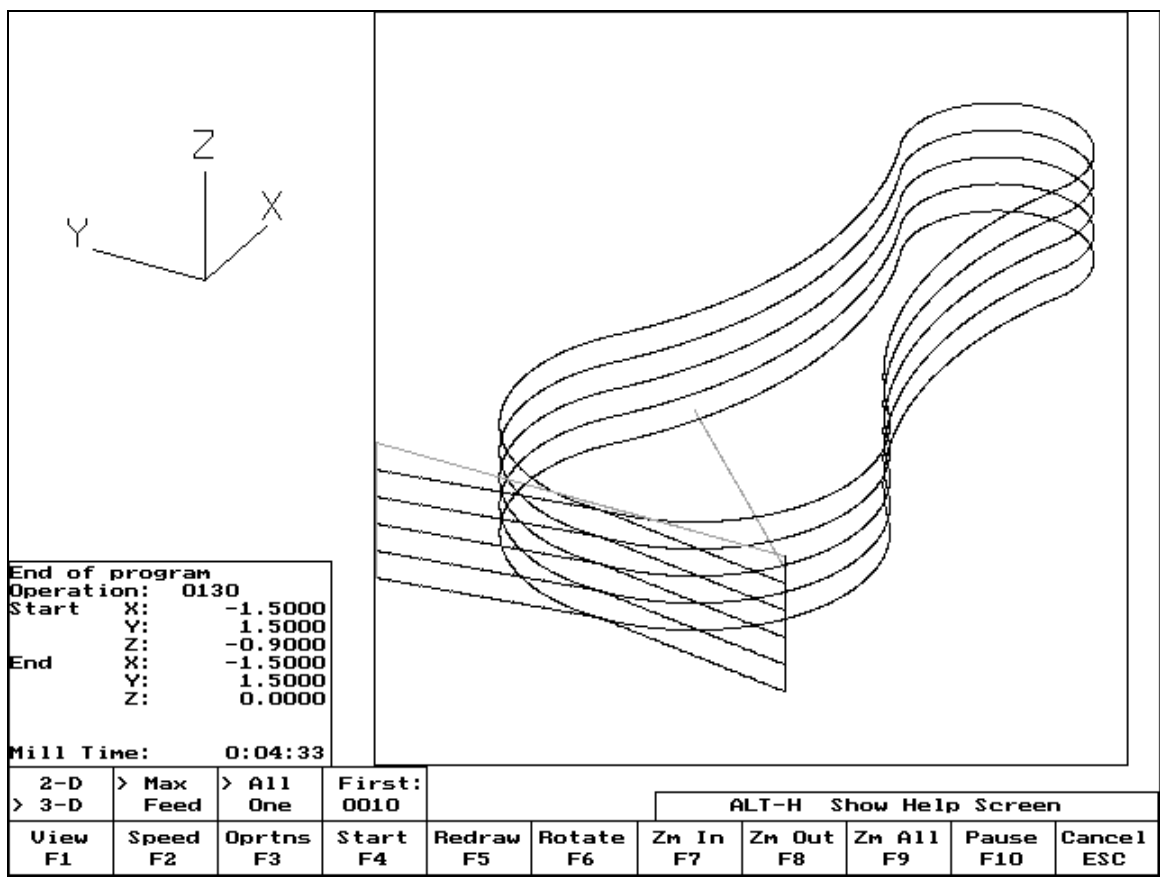
The Repeat to Depth feature is useful for repeating a part contour when the material being machined is too thick to cut in just one pass. The contour formed by these operations may either be a closed contour or an open one. If a non-vertical plunge to the start of the contour is desired, it must be programmed 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>).

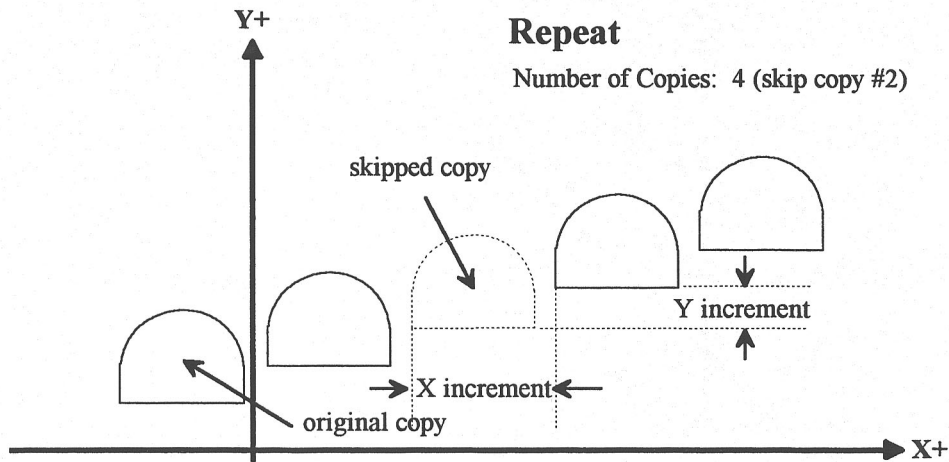
The contour shown on the sample screen above would appear as follows:



* 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.

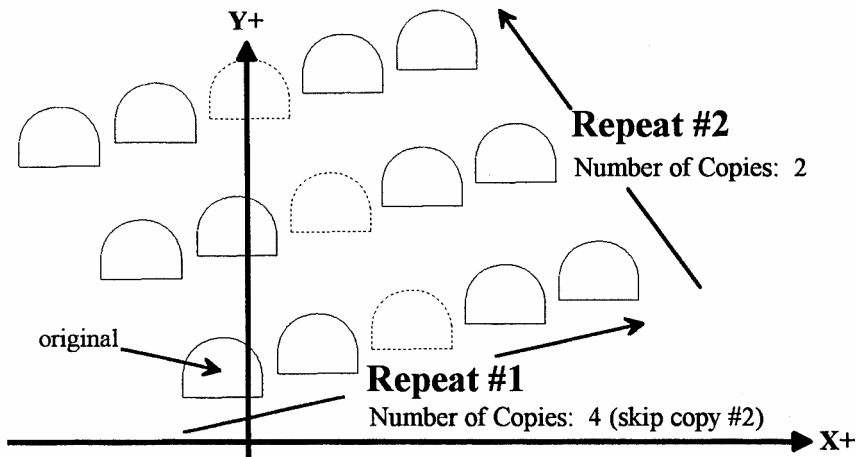


Increment: Specifies the X and Y distances between the start points of each copy of the contour.

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.

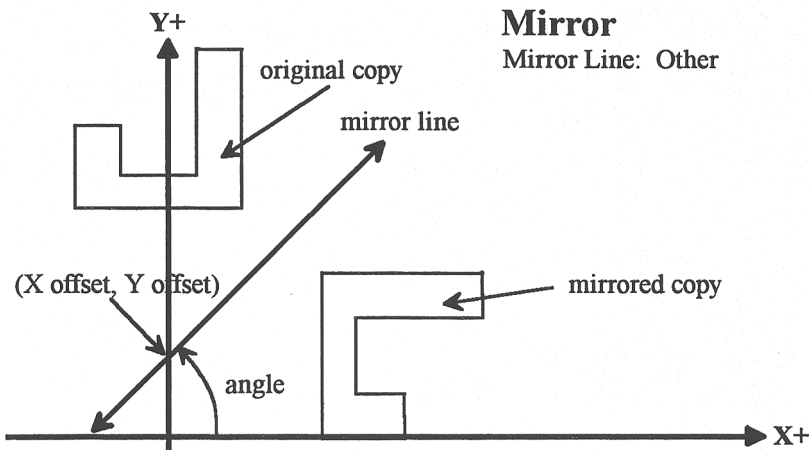
Skip List: List of skipped copies currently selected.



* 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).

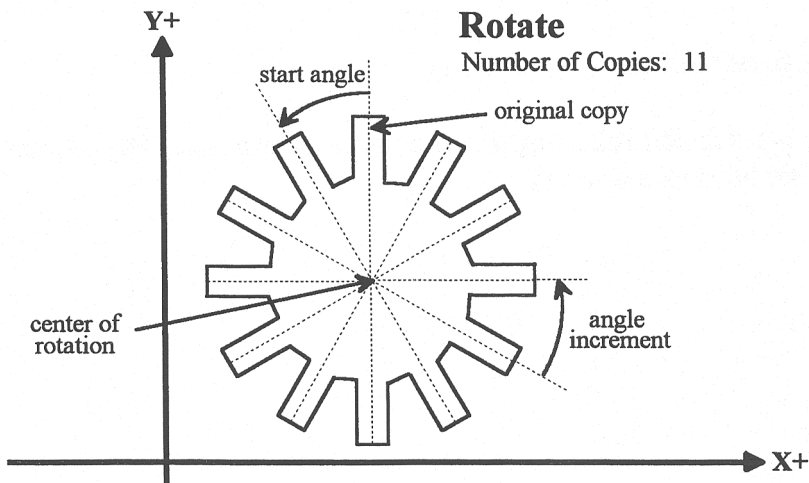
X Offset: Specifies the X coordinate of a point on the Mirror Line. This field will not be visible if a horizontal mirror line is being defined.

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.

ALT-H - Help

Press <ALT-H> to see a brief explanation of all the options available on the Insert Operation screen.

Teach Mode

Teach mode allows X/Y part coordinates to be entered into a program through machine table positioning. In any program operation being edited, while editing an X or Y coordinate, the <F9> Teach option will be displayed as shown:



Abs/Inc F1					Math F6	Solutn F7	Graph F8	Teach F9	Accept F10	Cancel ESC
---------------	--	--	--	--	------------	--------------	-------------	-------------	---------------	---------------

If you press <F9> for Teach, you will see a screen similar to the following:

Set Teach Zeroes					Edit Operation					
<p>To set part zeroes or reference:</p> <p>1) Move the table to the reference or zero position.</p> <p>2) Enter reference value if non-zero.</p> <p>3) Then press <F1> and/or <F2> to set.</p>					<p>#0050 Linear mill</p> <p>End: X: 7.0000 Y: 3.0000 Z: 1.0000</p> <p>Angle : 0.00°</p> <p>Length : 0.0000</p> <p>Connect Radius : 0.0000</p> <p>Feedrate : 20.0000</p> <p style="text-align: center;">Rotary axis B</p> <p>Degrees : 0° INC</p> <p>Minutes : 0' INC</p> <p>Seconds : 0" INC</p> <p>Decimal degrees : 0.0000°INC</p>					
<p>Current Position X: 2.3500</p> <p>Y: 2.6100</p>										
Set X F1	Set Y F2									Done ESC

The current machine position and motor status will be displayed on the left portion of the screen. The machine can be positioned manually, if the motors are not powered, by jogging using the jog panel or by using a MPG hand wheel (if available).

F1 - Zeroes

Pressing <F1> Zeroes displays the screen below and allows you to set a new reference for teach mode.

Move the table to the desired zero or reference position, and then enter the value for the new reference if other than zero. Press <F1> to set an X and/or <F2> to set a Y reference position. Press <ESC> to return to the Teach screen.

* NOTE: The new zeroes only affect Teach Mode and will **not** be saved upon exiting Intercon.

F5 - X Power

Pressing <F5> is a toggle switch to apply power to or remove power from the X motor. The X motor status is display next to the current X position.

F6 - Y Power

Pressing <F6> is a toggle switch to apply power to or remove power from the Y motor. The Y motor status is display next to the current Y position.

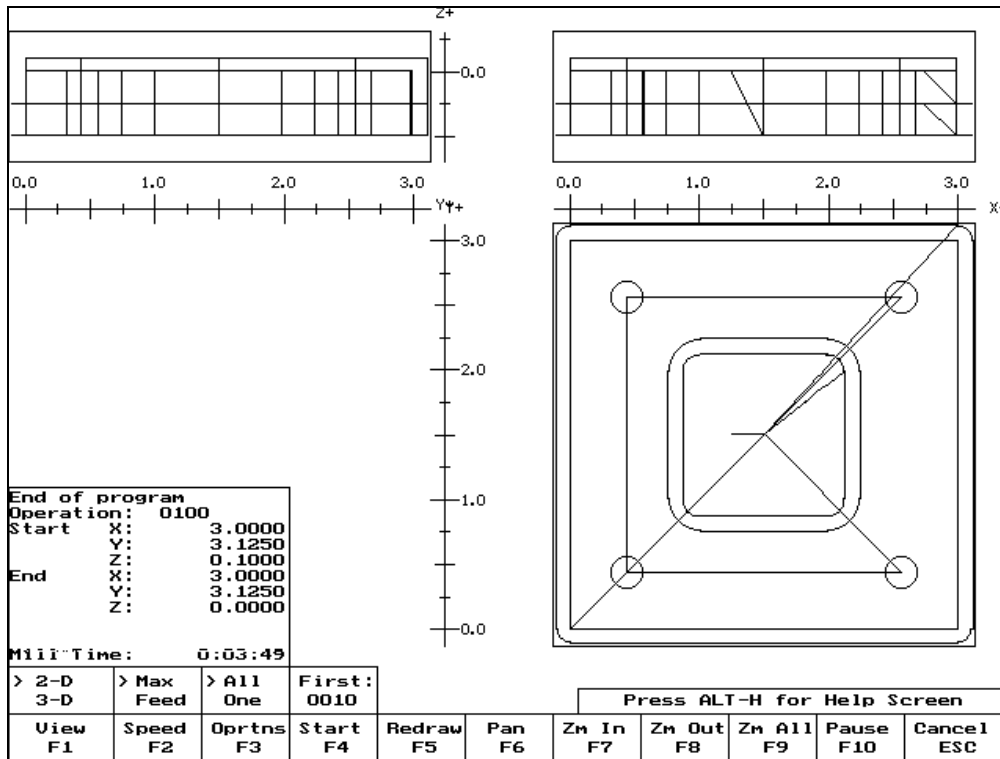
F10 - Accept

Pressing <F10> will transfer the X/Y machine position to the X/Y position fields in the current edit screen and returns to the edit screen.

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.

Pressing <ALT-H> will display a complete list of all available functions.

Graphics Help

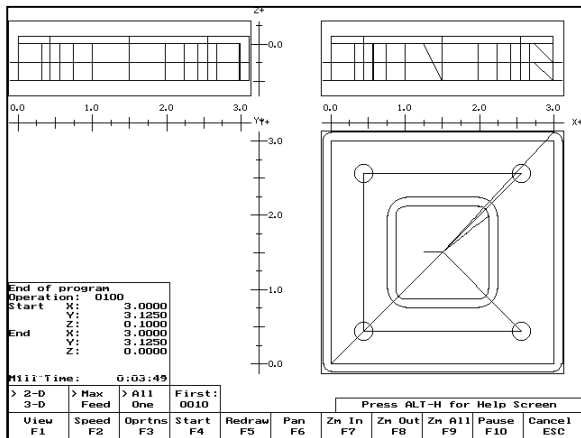
```

F1      2-D/3-D Graphics Display
F2      Maximum/Feedrate Display Speed
F3      Show All/Show One Operation(s)
F4      Change First Graphics Operation
F5      Redraw Project
F6      Pan Screen Center (2-D Display)
        Rotate Project (3-D Display)
F7      Zoom In (Enlarge Project Size)
F8      Zoom Out (Reduce Project Size)
F9      Zoom All (Redraw at Original Size)
F10     Pause/Resume Preview
ESC     Quit Graphics
        Cancel Pan/Rotate/Zoom In/Zoom Out
ALT-F   Show/Hide Feedrate Display (Max Speed)
ALT-H   Graphics Help Screen
ALT-M   Show/Hide Milling Time Estimates
+       Increase Feedrate Percent (Feed Speed)
-       Decrease Feedrate Percent (Feed Speed)
Arrows  Move Crosshairs (Pan/Zoom In/Zoom Out)
        Rotate Project (Rotate)
Enter   Get Last/Next Operation (First Operation)
        Select Screen Center (Pan/Zoom Out)
        Select Zoom Box Corner (Zoom In)
        Show Next Operation (Show One)
        Select First Operation (First Operation)
    
```

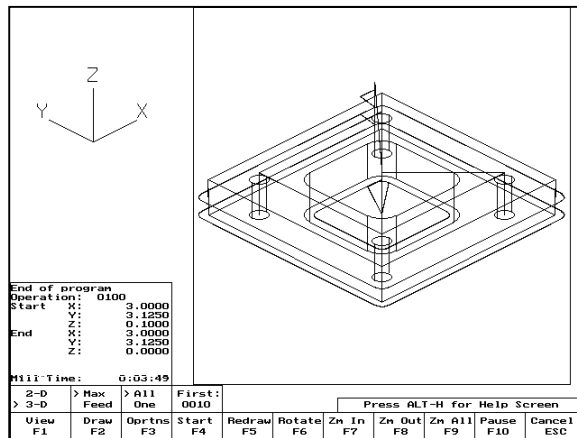
Press any key to continue

F1 - View

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).



Three-plane



Isometric

F2 - Speed

According to your preference, you may cause the simulation to proceed at its maximum speed (Max), or at the feedrate specified in the operation (Feed). Pressing <F2> toggles this option.

F3 - Operations

The part may be viewed in its entirety (Show All), or you may view the operations one at a time (One Step). Press <F3> to switch between the modes. If stepping through the operations one at a time, you will be required to press <ENTER> to allow the simulation to proceed to the next operation.

F4 - Start

Pressing the <F4> key will cause the simulation to start again, allowing you to specify the starting position.

F5 - Redraw

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

F6 - Pan or Rotate

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 or rotate mode, simply press the <F6> key. A set of crosshairs will appear in the XY plane of the three-plane display, or a second set of orientation axes (with broken lines) will appear. Table 1 below describes what keys affect the orientation of the project display during use of the pan and rotate feature.

Key	Three-Plane Display	Isometric Display
<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	Cancel Rotate

Table 1 - Pan/Rotate Mode Key Functions

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

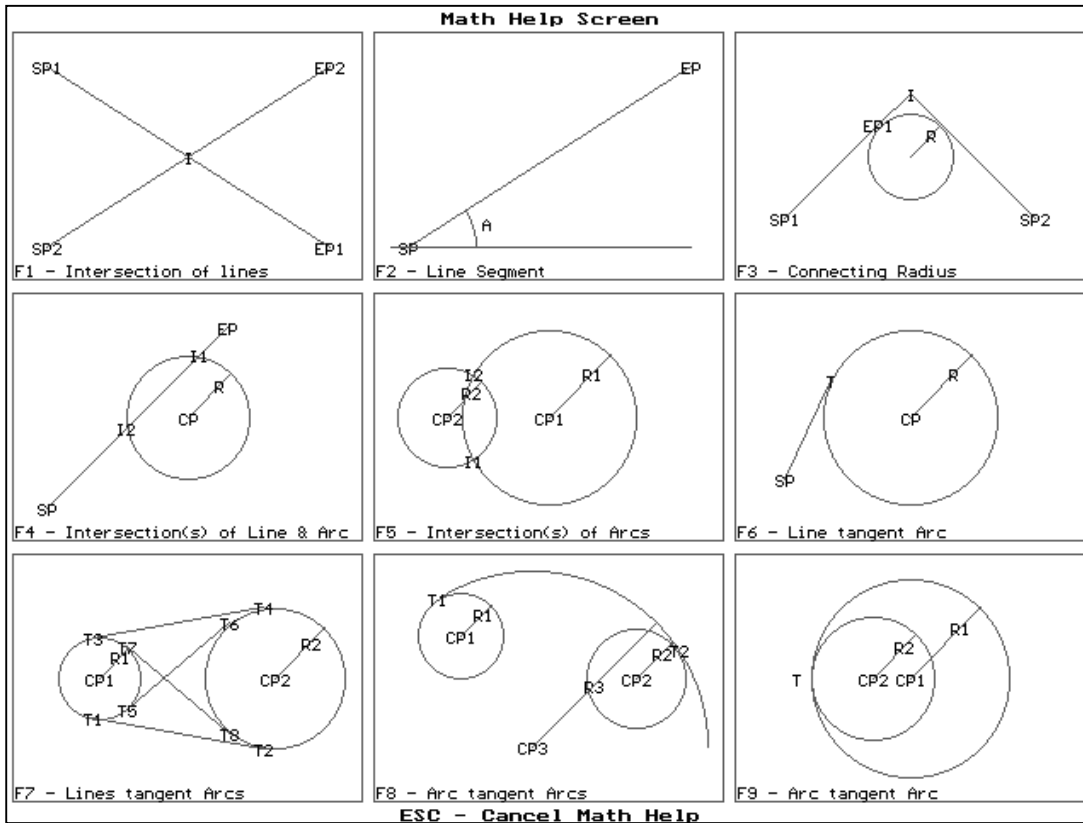
The project can also be viewed in an enlarged or reduced state by pressing the <F7> or <F8> keys to activate Zoom In and Zoom Out respectively. Pressing <F9> redraws the project at its original size. When zooming in or out, a set of crosshairs will appear on the screen with which a section of the project can be selected for magnification. Table 2 below describes what keys affect the functioning of the zoom feature.

Key	Function
<Left Arrow>, <Right Arrow>, <Down Arrow>, <Up Arrow>	Move crosshairs around on screen
<ENTER>	Select zoom box corner (Zoom In) Select screen Center (Zoom Out)
<ESC>	Cancel zoom box selection or cancel zoom

Table 2 - Zoom Mode Key Functions

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 or from the Edit Part screen. The following screen appears:



The figures show problem scenarios. Each scenario shows sample figures drawn from known information and the intersection point(s) of the figures. The labels on the figures show the required information to solve the problem (LP, CP, R, or T) and the data that will be found (I, T, CP, or R). A discussion of each scenario is presented later in this chapter.

Choose the scenario that matches the known and unknown information for your drawing and press the corresponding function key.

After you have selected the scenario to solve, you will be prompted for the parameters. The parameter entry screen is similar to an Edit Operation screen. Use the arrow keys to move from field to field, press <ENTER> to accept a value and go to the next field, press <F9> to accept all values and solve, or press <ESC> to cancel.

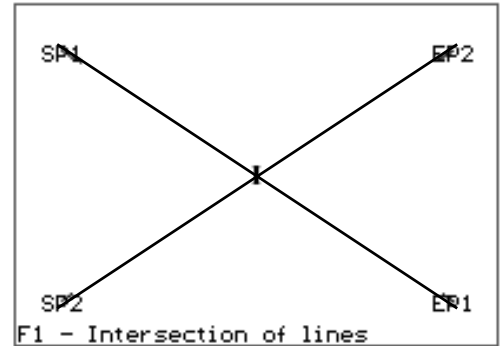
* NOTE: If there is no solution with the parameters you have entered, Intercon displays a message. You can then change the parameters or press <ESC> to cancel.

A discussion of each parameter entry screen follows.

F1 - Intersection of Lines

Given 2 lines, each defined by one point (L1P1 and L2P1) and either a second point (L1P2 or L2P2) or the angle from horizontal, find the intersection point (I).

Press <F1> to choose this scenario.



Intersection of Lines						Edit Operation				
Line 1:						#0020 Linear mill				
	SP X:	<input type="text" value="0.0000"/>				End:	X:	<input type="text" value="0.0000"/>		
	Y:	<input type="text" value="0.0000"/>					Y:	<input type="text" value="0.0000"/>		
	EP X:	<input type="text" value="0.0000"/>	Unknown				Z:	<input type="text" value="1.0000"/>		
	Y:	<input type="text" value="0.0000"/>	Unknown			Angle	:	<input type="text" value="0.00°"/>		
	Angle:	<input type="text" value="0.0000"/>	Unknown			Length	:	<input type="text" value="0.0000"/>		
Line 2:						Connect Radius	:	<input type="text" value="0.2500"/>		
	SP X:	<input type="text" value="0.0000"/>				Feedrate	:	<input type="text" value="20.0000"/>		
	Y:	<input type="text" value="0.0000"/>								
	EP X:	<input type="text" value="0.0000"/>	Unknown							
	Y:	<input type="text" value="0.0000"/>	Unknown							
	Angle:	<input type="text" value="0.0000"/>	Unknown							
Known F1									Solve F10	Cancel ESC

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

Press <F10> when you are done entering parameters. Intercon will find the intersection point if one exists, and the graphic solution will be displayed (sample on page 10-57).

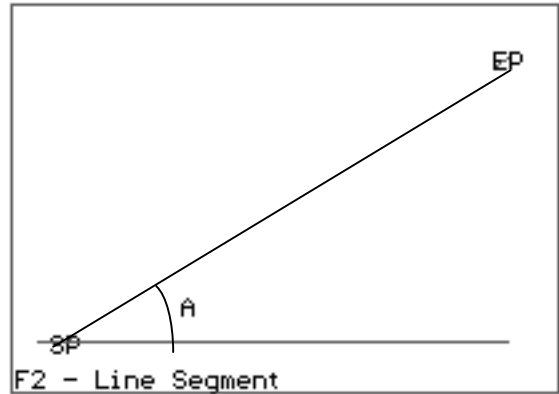
Notes:

1. When you enter a value in a field that was "UNKNOWN", it changes to "KNOWN". Press <F1> to toggle back to "UNKNOWN".
2. If you wish to enter "0" for an UNKNOWN value, you MUST press <F1> to change it to "KNOWN".

F2 - Line Segment

Given a line segment defined by the start point (LP1), one coordinate from the end point (LP2 X or Y), and the angle from horizontal, find the missing coordinate of the end point (LP2 X or Y).

Press <F2> to choose this scenario.



Line Segment					Edit Operation						
Line	Point1:	X:	<input type="text" value="0.0000"/>		#0020 Linear mill						
		Y:	<input type="text" value="0.0000"/>		End:	X:	<input type="text" value="0.0000"/>	Y:	<input type="text" value="0.0000"/>	Z:	<input type="text" value="1.0000"/>
Line	Point2:	X:	<input type="text" value="0.0000"/>	Unknown	Angle	:	<input type="text" value="0.00°"/>	Length	:	<input type="text" value="0.0000"/>	
		Y:	<input type="text" value="0.0000"/>	Unknown	Connect Radius	:	<input type="text" value="0.2500"/>	Feedrate	:	<input type="text" value="20.0000"/>	
	Angle:		<input type="text" value="0.0000"/>	Unknown							
Known F1									Solve F10	Cancel ESC	

You must enter the X and Y coordinates of the first point of the line segment and one of the following:

- * the X coordinate and the angle from horizontal
- * the Y coordinate and the angle from horizontal

When you are done, press <F9> and Intercon will find the missing coordinate, and the graphic solution will be displayed (sample on page 10-57).

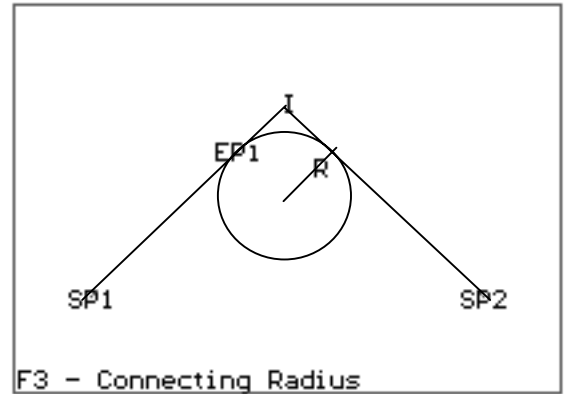
Notes:

1. When you enter a value in a field that was "UNKNOWN", it changes to "KNOWN". Press <F1> to toggle back to "UNKNOWN".
2. If you wish to enter "0" for an UNKNOWN value, you MUST press <F1> to change it to "KNOWN".

F3 - Connecting Radius

Given two points on a line (L1P1 and L1P2, where L1P2 is a tangent point to the connecting radius), a point on a second line (L2P1), and the connecting radius (R), find the intersection point (I) and the center of the arc.

Press <F3> to choose this scenario.



Connecting Radius					Edit Operation				
Circle:	Radius:	<input style="width: 100%;" type="text" value="0.0000"/>			#0020 Linear mill				
Line 1:	SP X:	<input style="width: 100%;" type="text" value="0.0000"/>	Y:	<input style="width: 100%;" type="text" value="0.0000"/>	End:	X:	<input style="width: 100%;" type="text" value="0.0000"/>	Y:	<input style="width: 100%;" type="text" value="0.0000"/>
	EP X:	<input style="width: 100%;" type="text" value="0.0000"/>	Y:	<input style="width: 100%;" type="text" value="0.0000"/>		Z:	<input style="width: 100%;" type="text" value="1.0000"/>	Angle	<input style="width: 100%;" type="text" value="0.00°"/>
Line 2:	EP X:	<input style="width: 100%;" type="text" value="0.0000"/>	Y:	<input style="width: 100%;" type="text" value="0.0000"/>	Length	<input style="width: 100%;" type="text" value="0.0000"/>	Connect Radius	<input style="width: 100%;" type="text" value="0.2500"/>	Feedrate
								Solve F10	Cancel ESC

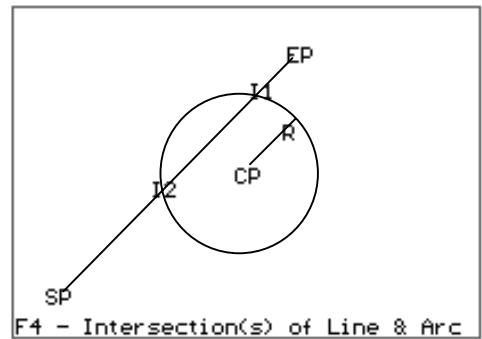
You must enter the radius of the arc which will connect line 1 and line 2, the X and Y coordinates for the start point of line 1, the X and Y coordinates for the second (tangent) point of line 1, and the X and Y coordinates for a point on line 2.

When you have finished entering parameters, press <F9> and Intercon will find the intersection point and the graphic solution will be displayed (sample on page 10-57).

F4 - Intersection(s) of Line and Arc

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).

Press <F4> to choose this scenario.



Intersections of Line and Arc						Edit Operation					
Circle:						#0020 Linear mill					
	CP X:		0.0000			End:	X:		0.0000		
	Y:		0.0000				Y:		0.0000		
	Radius:		0.0000				Z:		1.0000		
Line:						Angle	:		0.00°		
	SP X:		0.0000			Length	:		0.0000		
	Y:		0.0000			Connect Radius	:		0.2500		
	EP X:		0.0000	Unknown		Feedrate	:		20.0000		
	Y:		0.0000	Unknown							
	Angle:		0.0000	Unknown							
Known										Solve	Cancel
F1										F10	ESC

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

When you are done, press <F9> and Intercon will find the intersection(s), and the graphic solution will be displayed (sample on page 10-57).

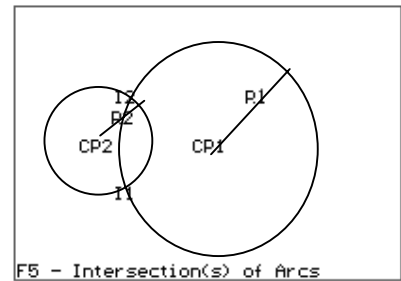
Notes:

1. When you enter a value in a field that was "UNKNOWN", it changes to "KNOWN". Press <F1> to toggle back to "UNKNOWN".
2. If you wish to enter "0" for an UNKNOWN value, you MUST press <F1> to change it to "KNOWN".

F5 - Intersection(s) of Arcs

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.

Press <F5> to choose this scenario.



Intersection(s) of Arcs						Edit Operation				
Circle 1:						#0020 Arc mill end				
	CP X:					Arc Type	:	EP&R		
	Y:					Mid:	X:	_____		
	Radius:						Y:	_____		
Circle 2:							Z:	_____		
	CP X:					End:	X:	2.0000		
	Y:						Y:	3.0000		
	Radius:						Z:	1.0000		
						Center:	X:	_____		
							Y:	_____		
							Z:	_____		
						Angle	:	_____ °		
						Radius	:	1.5000		
						Plane	:	XY		
						Direction	:	CW		
						Connect Radius	:	0.0000		
						Feedrate	:	20.0000		
						Angle <= 180°	:	Yes		
								Solve	Cancel	
								F10	ESC	

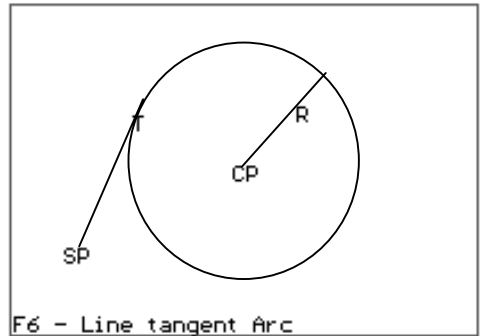
You must enter the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

When you are done, press <F9> and Intercon will find the intersection(s), and the graphic solution will be displayed (sample on page 10-57).

F6 - Lines Tangent to Arc

Given the center (CP) and radius (R) of an arc and 1 point (LP1) on a line, find the lines tangent to the arc (defined by the tangent point (T)).

Press <F6> to choose this scenario.



Line tangent to Arc					Edit Operation				
Circle:					#0020 Arc mill end				
CP X:		0.0000		Arc Type	:	EP&R			
Y:		0.0000		Mid:	X:				
Radius:		0.0000			Y:				
Line:					End:	X:	2.0000		
SP X:		0.0000			Y:	3.0000			
Y:		0.0000		Center:	Z:	1.0000			
					X:				
				Angle	:				°
				Radius	:	1.5000			
				Plane	:	XY			
				Direction	:	CW			
				Connect Radius	:	0.0000			
				Feedrate	:	20.0000			
				Angle <= 180°	:	Yes			
							Solve F10	Cancel ESC	

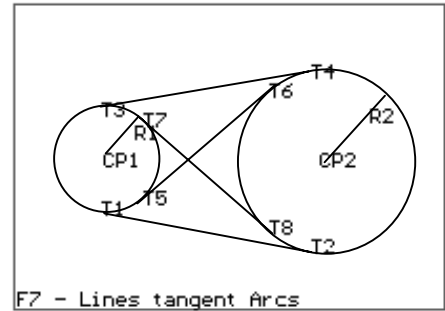
You must enter the X and Y coordinates for the circle's center point, the circle's radius, and the X and Y coordinates for a point on the line.

When you are done, press <F9> and Intercon will find the tangent points, and the graphic solution will be displayed (sample on page 10-57).

F7 - Lines Tangent to 2 Arcs

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.

Press <F7> to choose this scenario.



Lines tangent to 2 Arcs					Edit Operation				
Circle 1:					#0020 Arc mill end				
	CP X:				Arc Type	:	EP&R		
	Y:				Mid:	X:			
	Radius:					Y:			
Circle 2:					End:	X:	2.0000		
	CP X:					Y:	3.0000		
	Y:				Center:	Z:	1.0000		
	Radius:					X:			
						Y:			
					Angle	:		°	
					Radius	:	1.5000		
					Plane	:	XY		
					Direction	:	CW		
					Connect Radius	:	0.0000		
					Feedrate	:	20.0000		
					Angle <= 180°	:	Yes		
								Solve	Cancel
								F10	ESC

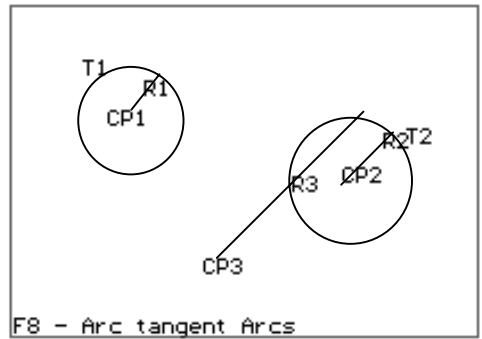
You must enter the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

When you are done, press <F9> and Intercon will find the tangent lines, and the graphic solution will be displayed (sample on page 10-57).

F8 - Arc Tangent to 2 Arcs

Given the center points (CP1 and CP2) and radii (R1 and R2) 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).

Press <F8> to choose this scenario.



Arc tangent to 2 Arcs						Edit Operation	
Tangent Arc:						#0020 Arc mill end	
Arc 1: Radius: 0.0000						Arc Type : EP&R	
CP X: 0.0000						Mid: X: _____	
Y: 0.0000						Y: _____	
Radius: 0.0000						Z: _____	
Arc 2: CP X: 0.0000						End: X: 2.0000	
Y: 0.0000						Y: 3.0000	
Radius: 0.0000						Z: 1.0000	
						Center: X: _____	
						Y: _____	
						Z: _____	
						Angle : _____ °	
						Radius : 1.5000	
						Plane : XY	
						Direction : CW	
						Connect Radius : 0.0000	
						Feedrate : 20.0000	
						Angle <= 180° : Yes	
						Solve F10	
						Cancel ESC	

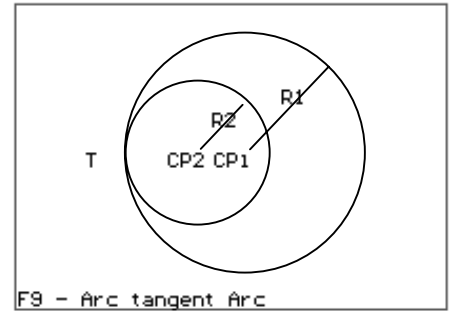
You must enter the radius of the tangent arc, the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

When you are done, press <F9> and Intercon will find the tangent arc's center and the tangent points, and the graphic solution will be displayed (sample on page 10-57).

F9 - Arc Tangent to 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.

Press <F9> to choose this scenario.



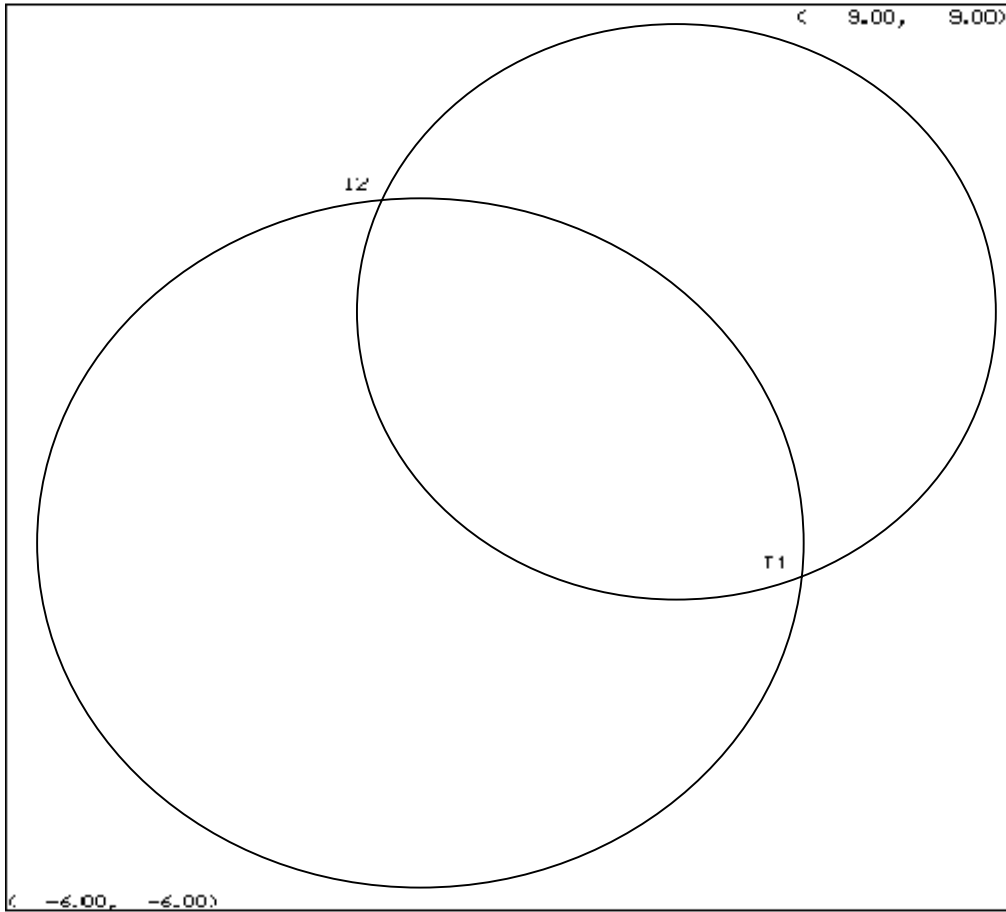
Arc tangent to Arc						Edit Operation					
Outside Circle: CP X: <input type="text" value="0.0000"/> Y: <input type="text" value="0.0000"/> Radius: <input type="text" value="0.0000"/> Inside Circle: CP X: <input type="text" value="0.0000"/> Y: <input type="text" value="0.0000"/> Radius: <input type="text" value="0.0000"/>						#0020 Arc mill end Arc Type : <input type="text" value="EP&R"/> Mid: X: _____ Y: _____ Z: _____ End: X: <input type="text" value="2.0000"/> Y: <input type="text" value="3.0000"/> Z: <input type="text" value="1.0000"/> Center: X: _____ Y: _____ Z: _____ Angle : _____ ° Radius : <input type="text" value="1.5000"/> Plane : <input type="text" value="XY"/> Direction : <input type="text" value="CW"/> Connect Radius : <input type="text" value="0.0000"/> Feedrate : <input type="text" value="20.0000"/> Angle <= 180° : <input type="text" value="Yes"/>					
<input type="text"/>	<input type="text"/>	<input type="text"/>	<input type="text"/>	<input type="text"/>	<input type="text"/>	<input type="text"/>	<input type="text"/>	<input type="text"/>	<input type="text"/>	Solve F10	Cancel ESC

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.

When you are done, press <F9> and Intercon will find the tangent point, and the graphic solution will be displayed (sample on page 10-57).

Graphic Solution Example

The Graphic Solution #1 (out of 1 solutions)



F1 - Next Sc
ESC - Cancel

The scenario is drawn to scale using the coordinates you specified, and the solution point(s) are indicated by a T position. The solution number and the total number of solutions found are displayed at the top of the screen.

Press <F1> to select another solution if multiple solutions exist. When you have the correct solution displayed, press <ESC> to return to the numeric entry screen to show the numeric solution clipboard and take the values into your program.

You may also press <ESC> to cancel Math Help.

Using the Numeric Solution Clipboard

When you press <ESC> from the Graphic Solution screen, or press <F7> (Last Solution) from an Operation Entry screen, the Last Solution Menu appears on the left side of the screen, and the operation you are editing is on the right side of the screen:

Last Solution					Edit Operation				
Circle 1:					#0020 Arc mill end				
	CP X:	2.0000			Arc Type	:	EP&R		
	Y:	3.0000			Mid:	X:	_____		
	Radius:	5.0000				Y:	_____		
Circle 2:						Z:	_____		
	CP X:	4.0000			End:	X:	2.0000		
	Y:	-4.0000				Y:	3.0000		
	Radius:	10.0000				Z:	1.0000		
Intersection Point 11:					Center:	X:	_____		
	X:	-2.9979				Y:	_____		
	Y:	3.1434				Z:	_____		
Intersection Point 12:					Angle	:	_____°		
	X:	6.1678			Radius	:	1.5000		
	Y:	5.7622			Plane	:	XY		
					Direction	:	CW		
					Connect Radius	:	0.0000		
					Feedrate	:	20.0000		
					Angle <= 180°	:	Yes		
↓ F1	↑ F2	Copy → F3			Gr Sol F8			Cancel ESC	

One field will be highlighted on the Last Solution clipboard, and one field will be highlighted on the operation parameters window. Use <F1> and <F2> to select the previous or next field of the clipboard. Use the arrow keys to select the next operation parameter. Press <F3> to copy the number from the clipboard field to the operation parameter. Continue copying values until you are finished, and then press <ESC> to remove the clipboard and return to editing the operation parameters.

You may review the graphic solution (and also choose another solution if one exists) by pressing <F8>.

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.

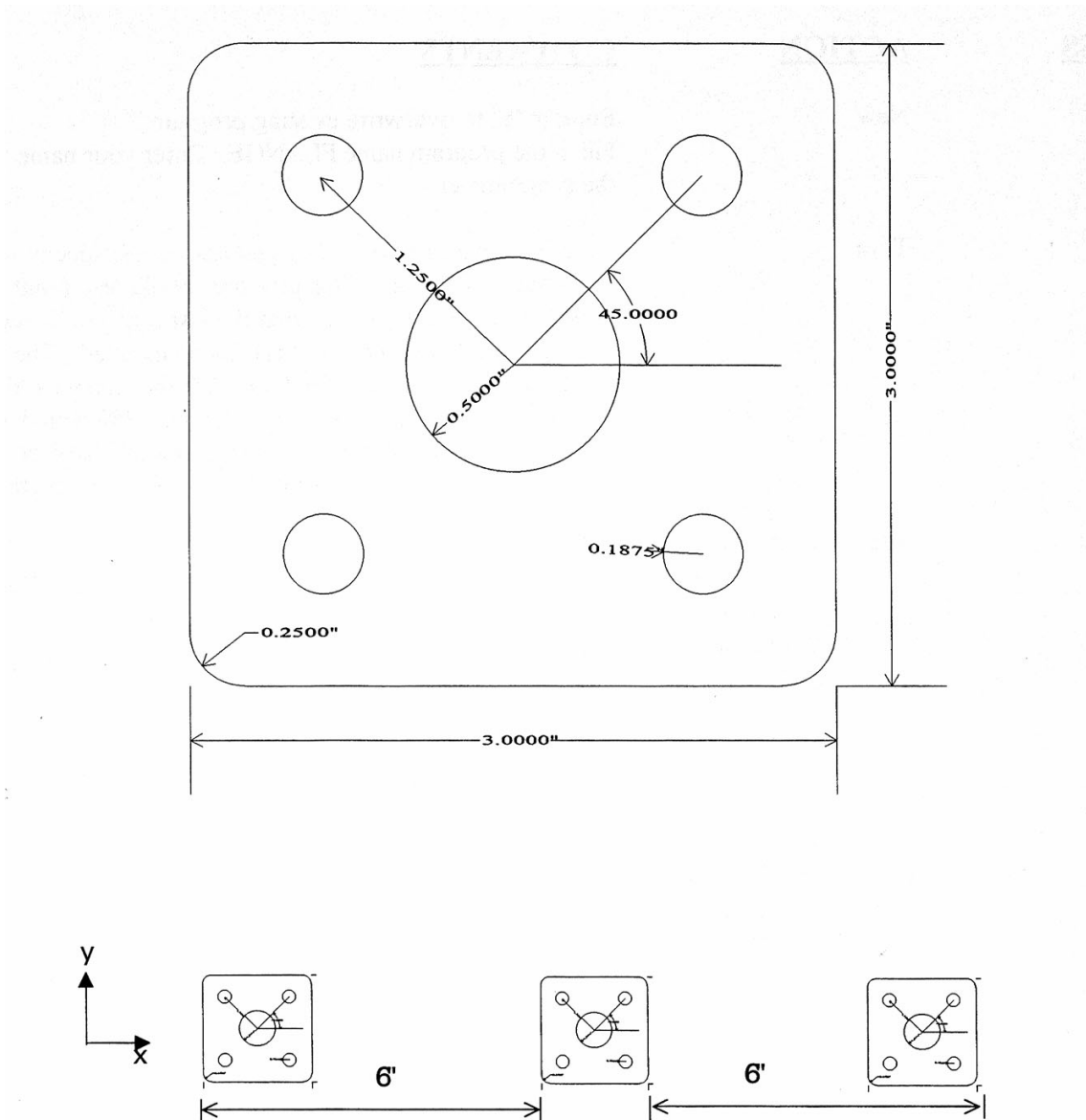


FIG. 1 Blueprint of flange part and the 3 fixtures.

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 Main Screen, 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	Enter in 'Y' to overwrite existing program. Fill in the program name FLANGE. Enter your name in as the programmer.
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).

#0020 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. "Overwrite Tool Library (Y/N)?" Type in 'Y'.
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:

#0030 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
Rough Cuts	:	Conventional
	Stepover :	0.2250
	Feedrate :	20.0000
Finish Pass	:	Climb
	Amount :	0.0020
	Feedrate :	10.0000

- F10** Accept Keep selected values.
- F5** Cycles Access the list of available Canned Cycles.
- F1** Drill "Bolt Hole Circle (Y/N)?" Type in 'Y'.
This question determines whether or not a circular pattern of bolt holes is desired. See more on drilling cycles earlier in this manual.

#0040 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.

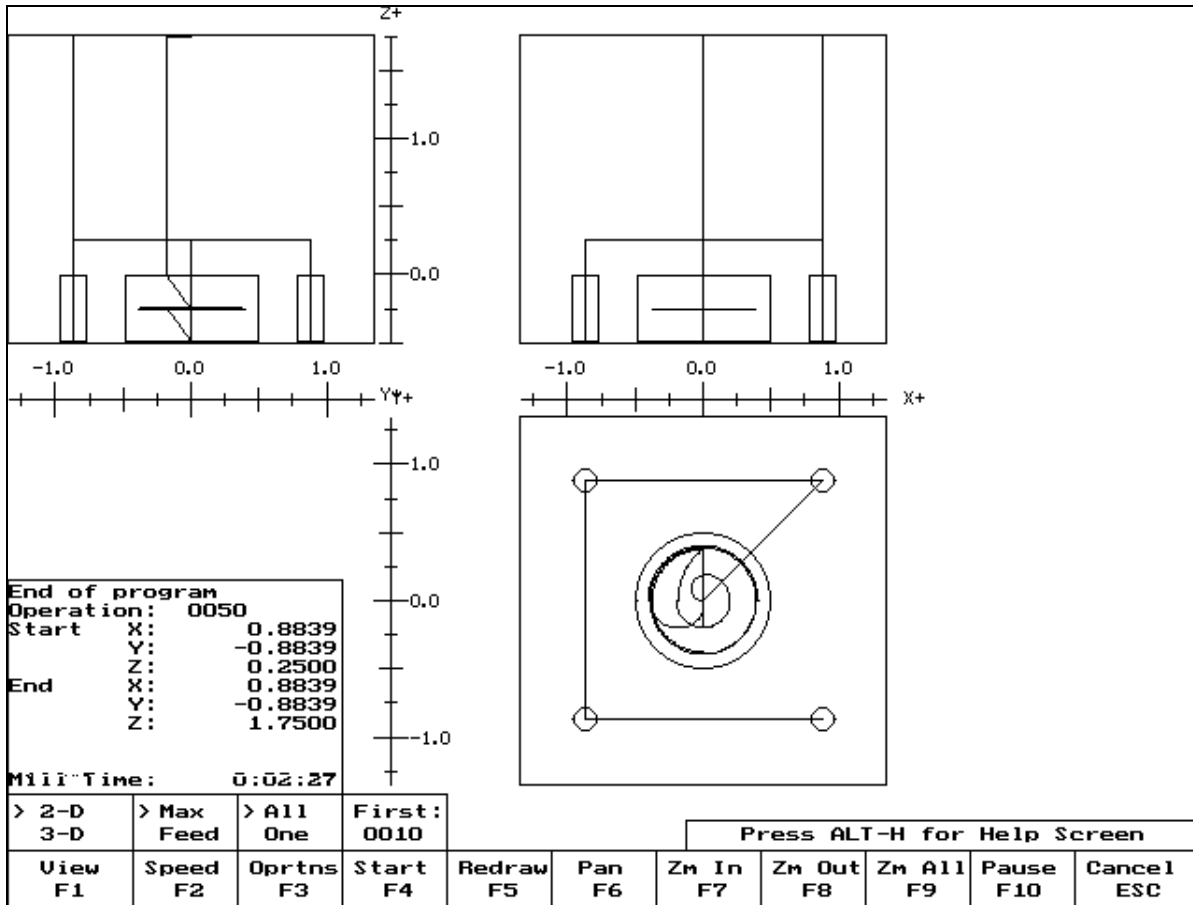


FIG. 2 - Graphics screen showing bolt holes and circular pocket

- 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.

#0050 Frame mill

Frame	:	Outside
Center:	X :	0.0000
	Y :	0.0000
Surface Height	:	0.0000
Length	(X) :	3.0000
Width	(Y) :	3.0000
Corner Radius	:	0.2500
Depth:	Total :	0.5000
	Per Pass :	0.2500
	Plunge Rate :	2.0000
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.

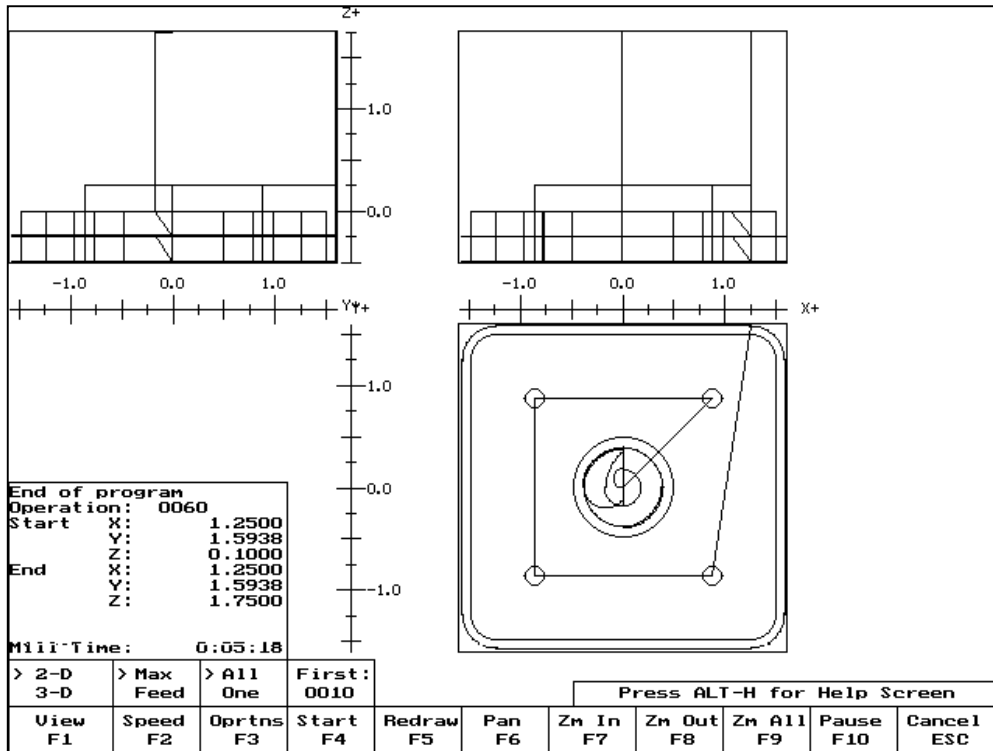


FIG. 3 - Graphics screen showing part with bolt holes and outer frame

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.

#0060 Repeat

Start Block	:	0030
End Block	:	0050
Increment	X :	6.0000
	Y :	0.0000
Clearance Height	:	Home Z
Plunge Rate	:	2.0000
Number of copies	:	2
Skip copy	:	
Skip List:		

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.
ESC/CANCEL	Cancel	Return to Main screen.
F6	Save As	Enter the name of the file in which to save this program, or press F10 and accept the name supplied. Overwrite existing program, if it exists, by pressing 'Y'.

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

F5

Post

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 Axs	Select the Y-axis next.
F10	Set	Set your Y zero position at current tool location.
F1	Next Axs	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 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!

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 program (CNC7) screen enter 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.

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.

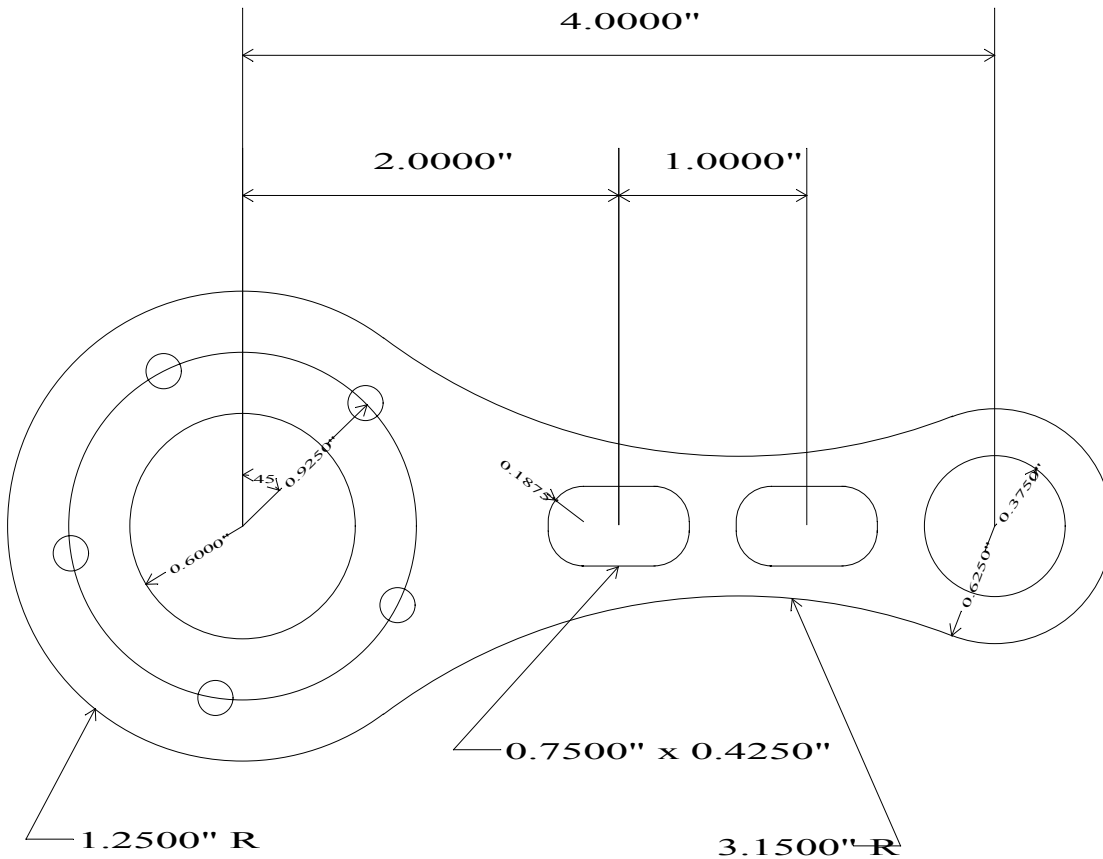


FIG. 1 - Part to be machined.

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 Main Screen, the following series of keystrokes 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 <F10> to accept each name after you fill it in.
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.

#0020 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. Overwrite Tool Library (Y/N)? Type in 'Y'.
F5	Cycles	Access the list of available Canned Cycles.
F1	Drill	Bolt Hole Circle (Y/N)? Type in 'Y'. This question determines whether or not a circular pattern of bolt holes is desired. See Figure 2 on the next page for an illustration of a bolt hole circle.

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

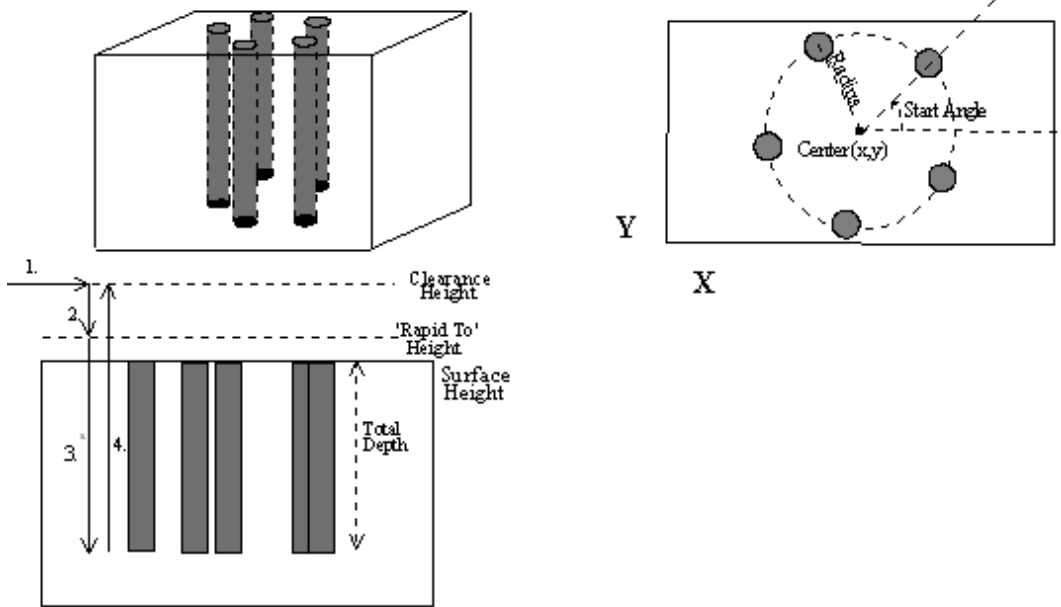


FIG. 2 - Bolt Hole Circle

#0030 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
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 #0020 above) has a height of 0.0000. Again, do not be alarmed if the Tool Height is not -1 for operation #0040. If this tool does not have the desired spindle (CW) and coolant (Flood) settings, you should also enter values specific to your machine setup.

#0040 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. Overwrite Tool Library (Y/N)? Type in 'Y'.

F5

Cycle

Access the list of available canned cycles.

F6

C Pckt

Start with 1.2000 inch diameter Pocket.

#0050 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
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.
- F6** C Pckt Repeat above pocket cycle. The center X value 4.0000 and the diameter is 0.7500 inches.

#0060 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
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.

#0070 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
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.

#0080 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
Rough Cuts	:	Conventional
	Stepover :	0.1000
	Feedrate :	2.0000
Finish Pass	:	None
	Amount :	0.0000
	Feedrate :	2.0000

- F10** Accept Keep selected values.
- F7** 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.

#0090 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.

#0100 Rapid traverse

End:	X :	5.0000
	Y :	0.5000
	Z :	0.1000
Angle	:	14.04°
Length	:	2.0616

- F10** Accept Keep selected values.

F2

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.

#0110 Arc mill

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
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 shown on the following page. 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.

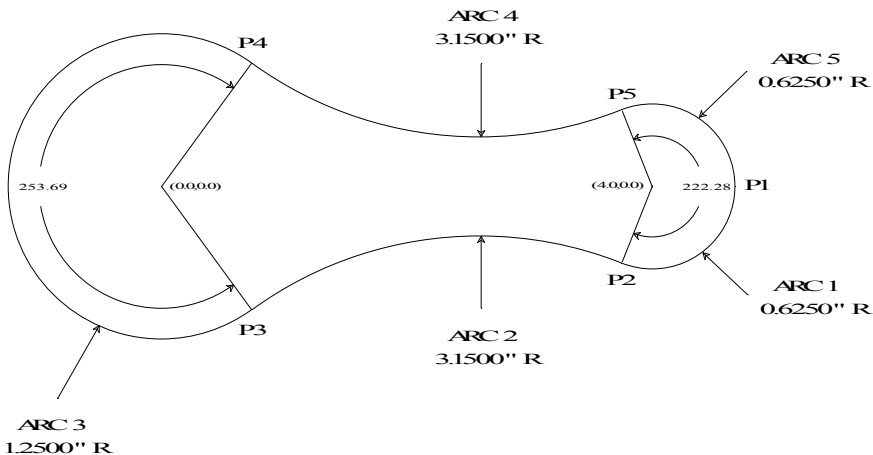


FIG 3. Tangent point and arc reference.

#0120 Arc mill

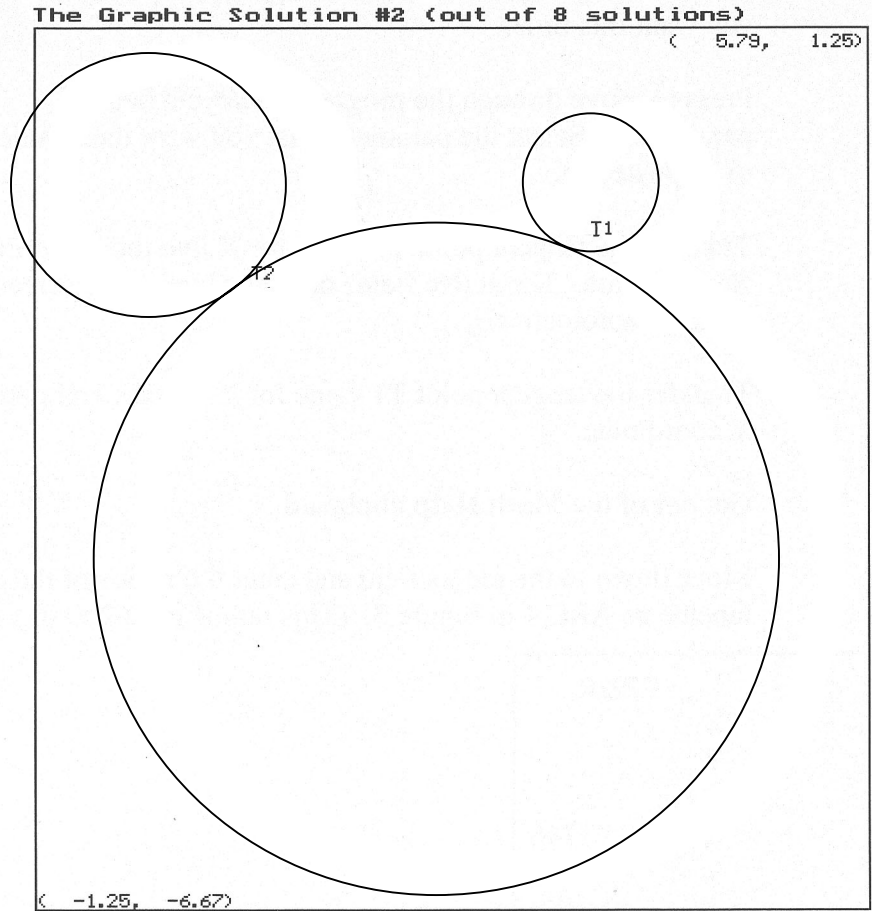
Arc type	:	EP&R
Mid	X :	
	Y :	
	Z :	
End:	X :	4.6250
	Y :	0.0000
	Z :	-0.0500
Center:	X :	
	Y :	
	Z :	
Angle	:	
Radius	:	.625
Plane	:	XY
Direction	:	CW
Feedrate	:	10.0000
Angle <= 180°	:	Yes

- F6 Math We are trying to find end points for the arcs that make up the outside edge of the part.
- F8 Arc Tangent Arcs This scenario will generate tangent points P2 - P5 of Figure 3. The following screen will appear:

Tangent Arc:		
	Radius :	3.1500
Arc 1:	:	
	CP X :	4.0000
	Y :	0.0000
	Radius :	0.6250
Arc 2:	:	
	CP X :	0.0000
	Y :	0.0000
	Radius :	1.2500

- F10 Solve Ask Intercon to calculate the missing values for this scenario. If the given data is insufficient to compute any solution, a message saying '**No solutions**' will appear.
- F1 Next Sol. Find scenario that corresponds to the actual arcs being milled. Observe Figure 4. Point T1 is the one need.

FIG. 4 - Screen showing Math Help Arc Tangent Arc solutions



F1 - Next Sol.
ESC - Cancel

ESCCancel Uses the given scenario for needed information and returns you to the arc mill input screen.

Last Solution (for arcs 1 and 2 of Figure 3)

Tangent Arc:		
	Radius :	3.1500
Arc 1:	CP X :	4.0000
	Y :	0.0000
	Radius :	0.6250
Arc 2:	CP X :	0.0000
	Y :	0.0000
	Radius :	1.2500
Tangent Arc	CP X :	2.6387
	Y :	-3.5210
Tangent Pnt T1	X :	3.7746
	Y :	-0.5829
Tangent Pnt T2	X :	0.7496
	Y :	-1.0003

F1, F2	Math Help Field	Press to highlight the needed tangent point X coordinate from the Last Solution screen. Tangent point T1 is the one you want this time.
ARROWS	Cycle Field	Press to move through the program line/cycle field of parameters. Select the parameter that you want the solution to be posted to.
F3	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.
F3	Copy	Transfer the tangent point T1 value for Y into the end point Y coordinate.
ESC/CANCEL	Cancel	Get out of the Math Help clipboard.
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

```

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.
-----------	-------	---

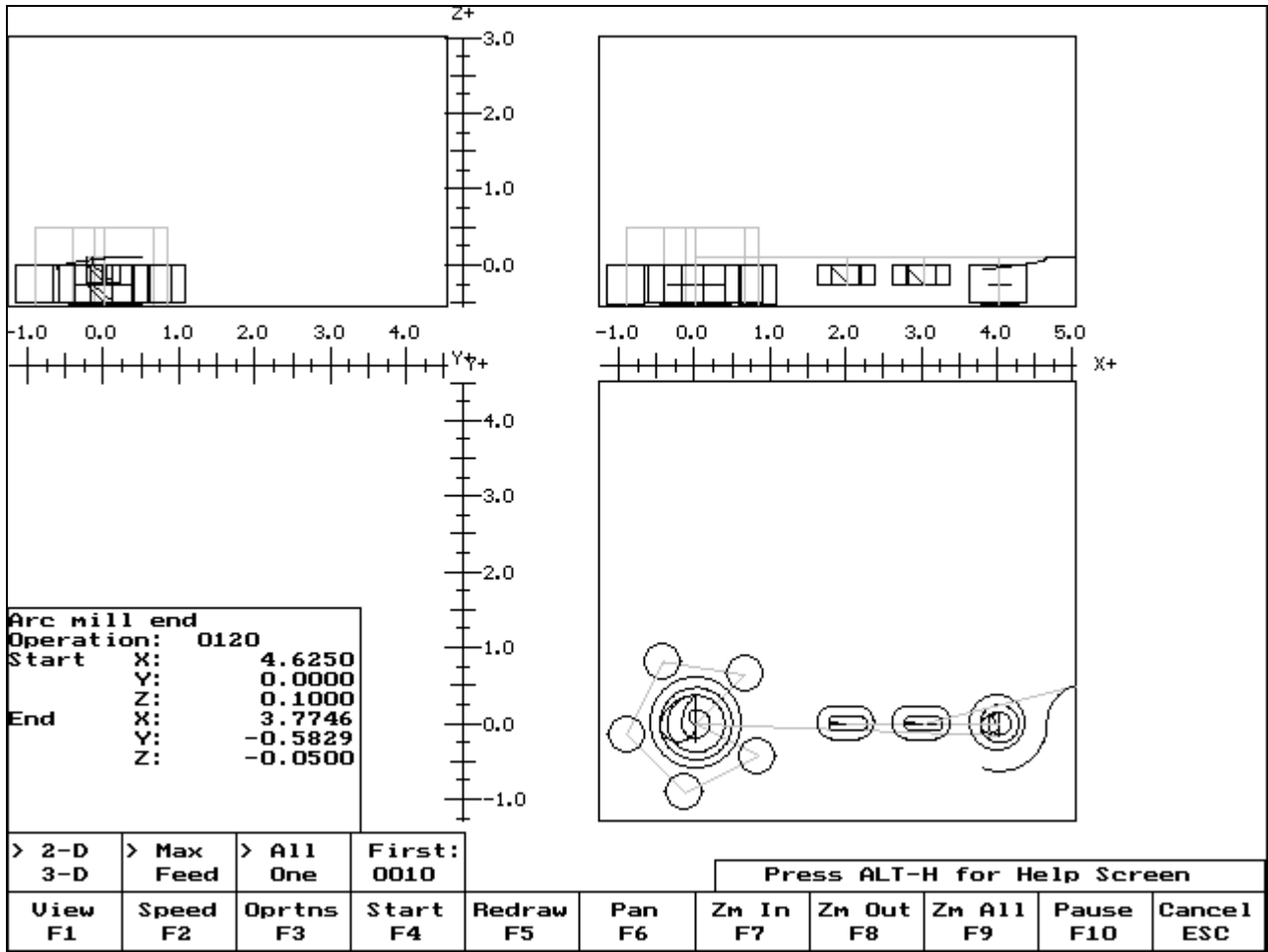


FIG. 5 - Draw screen showing Bolt Holes, Pockets and first arc of part

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.
	#0130 Arc mill	
F7	Solutn.	Redisplay the Math Help values calculated for the last arc.

Last Solution (for arcs 1 and 2 of Figure 3)

Tangent Arc	
Radius	: 3.1500
Arc 1:	CP X : 4.0000
	Y : 0.0000
Radius	: 0.6250
Arc 2:	CP X : 0.0000
	Y : 0.0000
Radius	: 1.2500
Tangent Arc CP	X : 2.6387
	Y : -3.5210
Tangent Pnt T1	X : 3.7746
	Y : -0.5829
Tangent Pnt T2	X : 0.7496
	Y : -1.0003

F1, F2	Math Help Field	Press to highlight the needed tangent point X coordinate from the Last Solution screen. Tangent point T2 is the one you want this time.
ARROWS	Cycle Field	Press to move through the program line/cycle field of parameters. Select the parameter that you want the solution to be posted to.
F3	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.
F3	Copy	Transfer the tangent point T2 value for Y into the end point Y coordinate.
ESC/CANCEL	Cancel	Get out of the Math Help clipboard.
ARROWS	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).

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**.

#0140 Arc mill

F7 Solutn. Redisplay the **Math Help** values calculated for the last arcs.

Last Solution (for arcs 1 and 2 of Figure 3)

Tangent Arc Radius:		3.1500
Arc 1:	CP :	
	X :	4.0000
	Y :	0.0000
	Radius :	0.6250
Arc 2:	CP :	
	X :	0.0000
	Y :	0.0000
	Radius :	1.2500
Tangent Arc CP	X :	2.6387
	Y :	-3.5210
Tangent Pnt T1	X :	3.7746
	Y :	-0.5829
Tangent Pnt T2	X :	0.7496
	Y :	-1.0003

F8	Gr. Sol.	Redisplays the diagram of the scenario selected to calculate arcs 1 and 2 on Figure 3.
F1	Next Sol.	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)

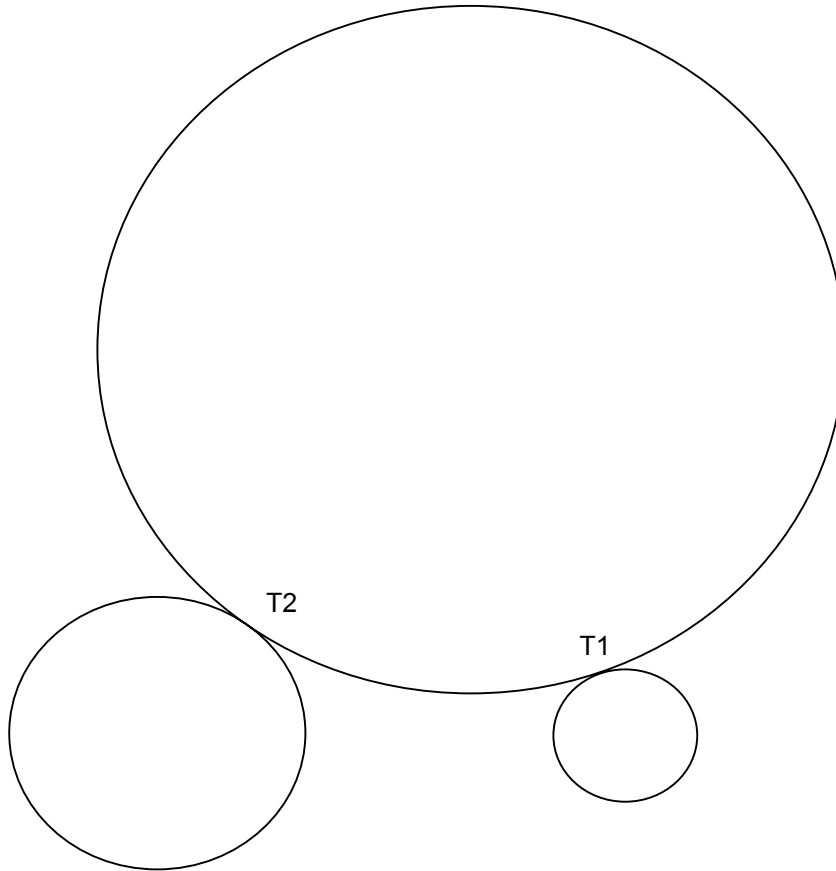


FIG. 6 - Math Help solution for arcs 3 and 4

ESC	Cancel.	Solves the given scenario for needed information and returns you to the Arc Mill input screen.
------------	---------	--

Last Solution (for arcs 3 and 4 of Figure 3)

Tangent Arc:		
	Radius :	3.1500
Arc 1:	CP X :	4.0000
	Y :	0.0000
	Radius :	0.6250
Arc 2:	CP X :	0.0000
	Y :	0.0000
	Radius :	1.2500
Tangent Arc CP	X :	2.6387
	Y :	3.5210
Tangent Pnt T1	X :	3.7746
	Y :	0.5829
Tangent Pnt T2	X :	0.7496
	Y :	1.0003

F1, F2	Math Help Field	Press to highlight the needed tangent point X coordinate from the Last Solution screen. Tangent point T2 is the one you want this time.
ARROWS	Cycle Field	Press to move through the program line/cycle field of parameters. Select the parameter that you want the solution to be posted to.
F3	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.
F3	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**.

#0150 Arc mill

F7 Solutn. Re-display the **Math Help** values calculated for the last arc.

Last Solution (for arcs 3 and 4 of Figure 3)

Tangent Arc Radius:		3.1500
Arc 1:	CP :	
	X :	4.0000
	Y :	0.0000
Radius	:	0.6250
Arc 2:	CP :	
	X :	0.0000
	Y :	0.0000
Radius	:	1.2500
Tangent Arc CP	X :	2.6387
	Y :	3.5210
Tangent Pnt T1	X :	3.7746
	Y :	0.5829
Tangent Pnt T2	X :	0.7496
	Y :	1.0003

F1, F2	Math Help Field	Press to highlight the needed tangent point X coordinate from the Last Solution screen. Tangent point T1 is the one you want this time.
ARROWS	Cycle Field	Press to move through the program line/cycle field of parameters. Select the parameter that you want the solution to be posted to.
F3	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.
F3	Copy	Transfer the tangent point T1 value for Y into the end point Y coordinate.
ESC/CANCEL	Cancel	Get out of the Math Help clipboard.
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). Also notice that this is a CCW arc.

```

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.

#0160 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 (G1) Move tool away from the edge of the part after the last arc.

#0170 Arc mill

Arc type	:	EP&R
Mid:	X :	
	Y :	
	Z :	
End:	X :	5.000
	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 #0110 and ends with the Linear Mill in operation #0170.

#0180 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.

#0190 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.

#0200 Rapid traverse

End:	X :	0.0000
	Y :	0.0000
	Z :	3.0000
Angle	:	135.00°
Length	:	7.0711

F10 Accept Keep selected values.

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).

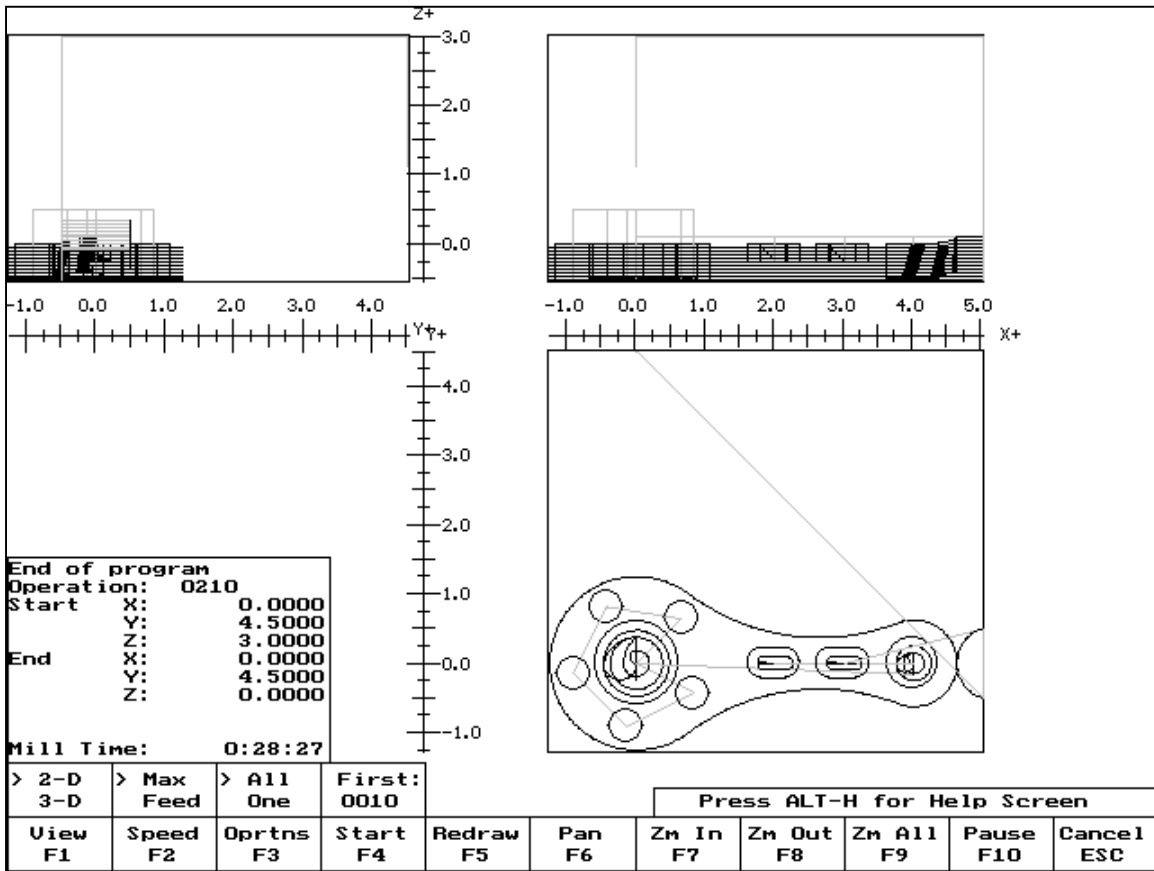


FIG. 7 - Draw screen showing complete part

ESC/CANCEL	Cancel	Return to the editing screen.
ESC/CANCEL	Cancel	Creation of the part is complete. Intercon programs automatically turn the spindle and coolant off at the end.
ESC/CANCEL	Cancel	Return to Main screen.
F3	Save	Press <F3> and accept the name supplied. Or you can save the program under a different name by pressing <F6>. Type the name you wish to save the program under, or use the ARROW keys and highlight the program name you wish to overwrite.

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.

F5

Post

Post your program to G codes. 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!

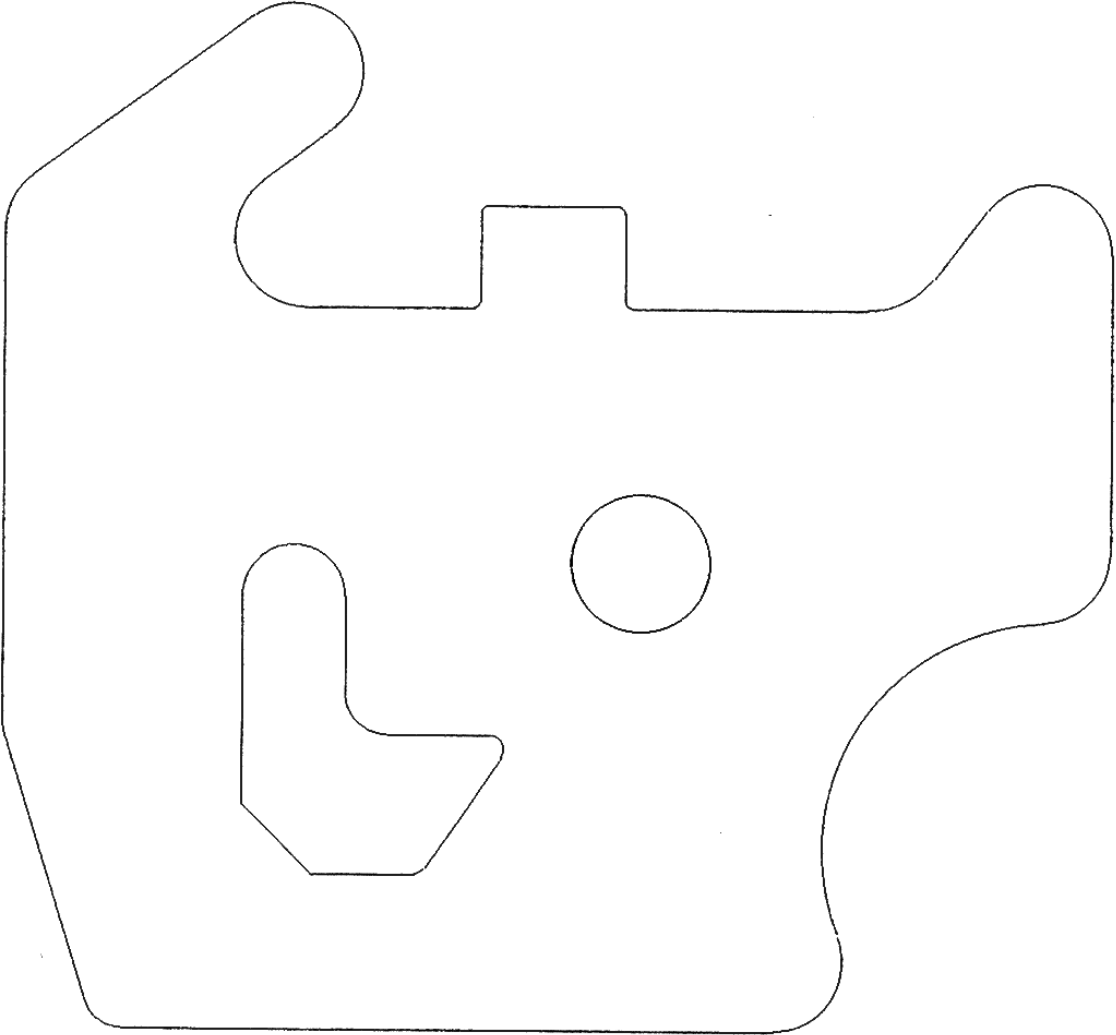
Measuring Tool Heights

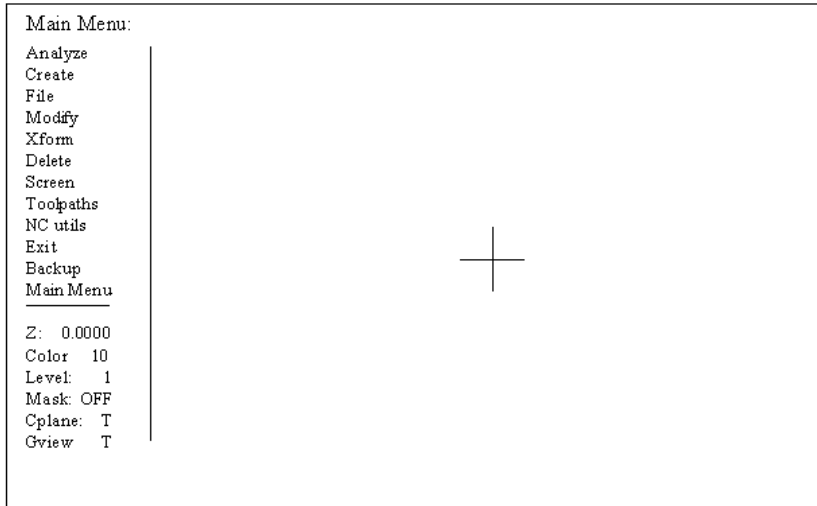
The following is a brief description of the method used to measure tool height values (offsets). You will need to insert a reference tool into the quill before beginning.

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

Mastercam Hints





When Mastercam is first started, this is the screen that should appear. The items below the main menu should have these settings. Items from the Main Menu, as well as any sub-menu can be selected using the mouse or by pressing the capital letter in the name of the menu item to be selected. Pressing <Alt-F10> will display the help keys in Mastercam.

Help Keys

F1	Zoom
Alt-F1	Fit
F2	Unzoom
F3	Redraw Screen
F4	Analyze
Alt-F4	Analyze Cursor Position
F5	Delete
Alt-F5	Delete Window
F6	File
Alt-F6	Edit
F9	Display Info
Alt-F9	Draw Axis
F10	Help
Alt-V	Version Number

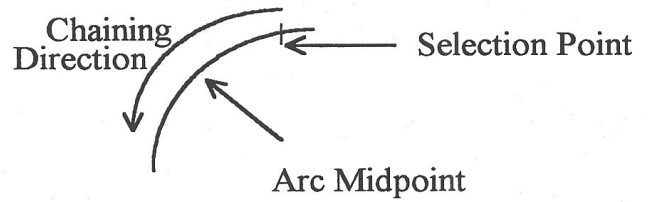
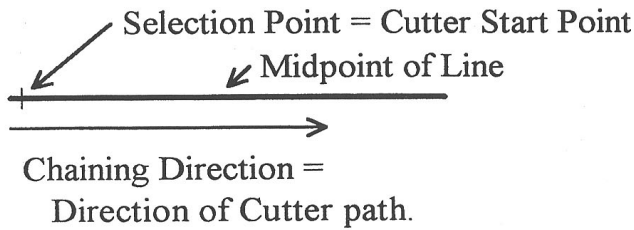
Frequently asked questions

What is an Entity?

An Entity is any object that is part of your geometry. Entities, in Mastercam are defined as lines, points, arcs and splines.

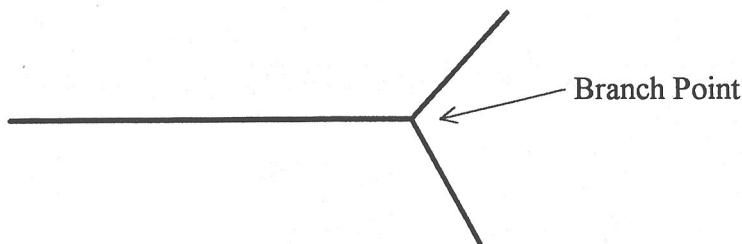
What is meant by chaining?

Chaining is the process of selecting perimeters entities for contouring and pocketing. Chaining is also used to define the cutter path direction and where cutting is to start. The first entity selected in the chain will determine where the path starts. The place that is selected on the entity will determine the direction of travel.

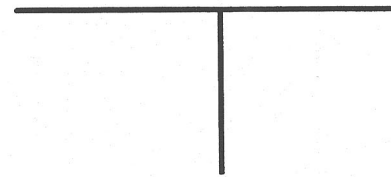


What is a Branch Point?

Branch points are where the endpoints of three or more entities meet. At these points when you are chaining a choice will have to be made as to which direction to go. Each entity must have an endpoint at the intersection to be able to branch.



No branch point because, the horizontal line is one entity and does not have an endpoint at the intersection of the vertical line



How do I put in dimensions?

Mastercam does not use dimensions and they are not needed. To check the dimensions of an entity in your part geometry, use the analyze feature from the Main Menu.

Why can't I contour around this circle?

Mastercam must have at least two entities to create a toolpath, so complete circles must be broken into at least two arcs. This can be accomplished by using Mastercam's break feature under the modify menu

Why didn't Mastercam produce G-codes for my part?

After defining the toolpath for your part, you **must** select End Program. After selecting end program you will be prompted to run the post processor. Select yes and your G-codes will be produced and written to C:\CNC7\NCFILES.

CAD Drawing Pitfalls

Problem - When laying a toolpath the message "Must have endpoint on the diamond"

Possible Pitfall - Multiple lines under lines:

This can happen when a 3D CAD drawing is imported into Mastercam in the 2D plane. The result can be noticed when laying down a toolpath and you cannot close a contour around the perimeter of a part's geometry.

Solution:

In the DELETE menu select <delete duplicate> and try again. If this does not work then go to the SCREENS menu and select endpoints. This will show the endpoints of all entities. If you see an endpoint in the middle of an entity that is not supposed to be there, delete the extra entity and try again.

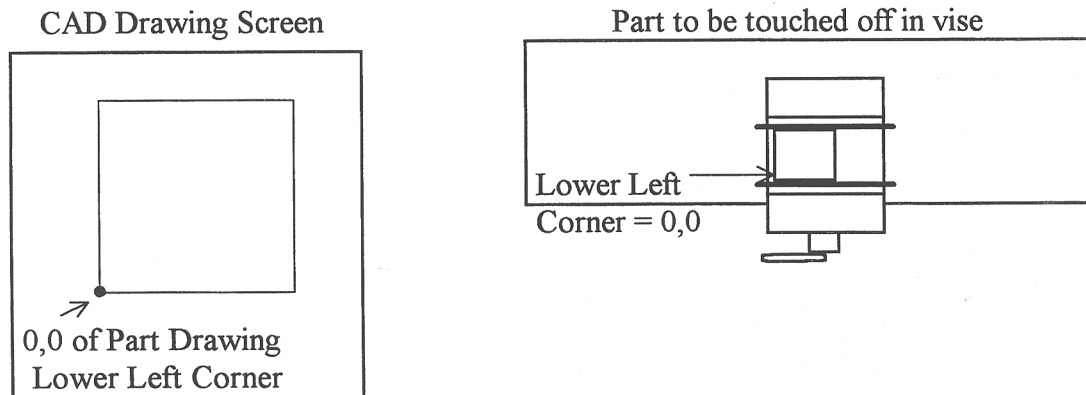
Possible Pitfall - Two or more of the entities do not have common endpoints

Solution: To see this, zoom in on the area when the toolpath stopped. If a gap between the two entities appears, those entities must be joined together. Even a gap of .0005 is too large.

Problem - After a toolpath was created and the machine zeros were set, it tried to cut my part some place other than where I set it.

Possible Pitfall - When the part was created in CAD, it was not drawn with the same **ZERO** reference.

Solution: When drawing in CAD, make sure that the zero in the drawing is the same as the zero on the machine where you touch off the part. This is for all three axes.



Mastercam Geometry Conversion (read in translators)

Mastercam allows a user to import other CAD Software files to create toolpaths. There are a few rules that must be used to ensure correct import. They are as follows:

- 1.The file must be in one of the following formats (DXF, IGES, CADL, NFL, ASCII or GEO)
- 2.Mastercam **only** imports **LINES, ARCS, POINTS and SPLINES**.
- 3.The file must be only geometry. This means **no** Dimension or Title Blocks. They must be completely removed from the drawing not just turned off levels or layers. Tell your CAD draftsman to keep all geometry on a separate layer and convert only that layer to DXF. This is just normal good practice.
- 4.For best results all geometry to be imported should be on the same cad drawing layer.
- 5.Entities cannot be blocked or grouped they must be exploded before import.
- 6.For **CadKey** users when taking a 3D (XYZ) drawing and converting to 2D (XY) use projections.

Steps for file conversion to Mastercam geometry

- 1.From the Mastercam main menu, select FILE.
- 2.From the FILE menu, select CONVERT.
- 3.Select the type of file to convert. (DXF, IGES, CADL, NFL, GEO)
- 4.Select READ to read the file.
- 5.Enter the file name to convert with full path. A:\MYPART.DXF would convert a DXF file from the A: drive to Mastercam geometry.
- 6.Save the file in the Mastercam geometry format.

When importing from newer versions of Mastercam versions. 4 and 5 you can simply save the geometry as 286 format (ver. 3.21):

- 1.In Mastercam, select File, Convert, 286, Write

2. Indicate path for file to be written
3. Ex. A:\My part.GE3

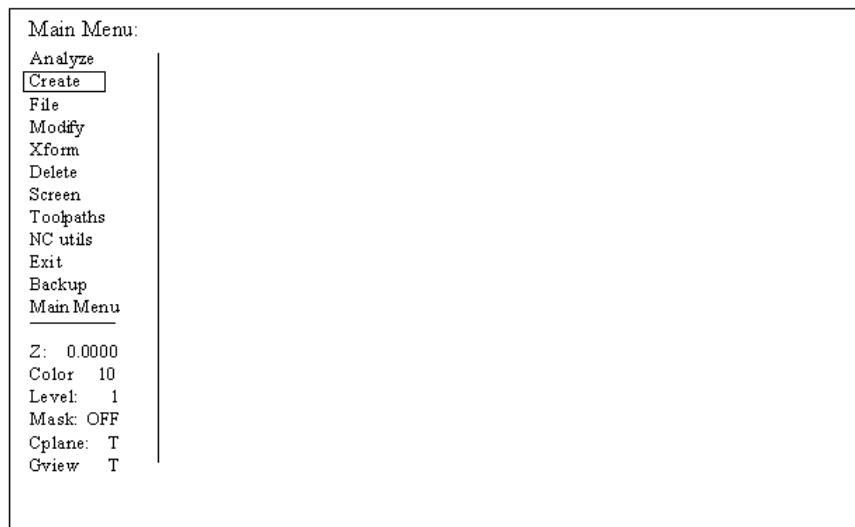
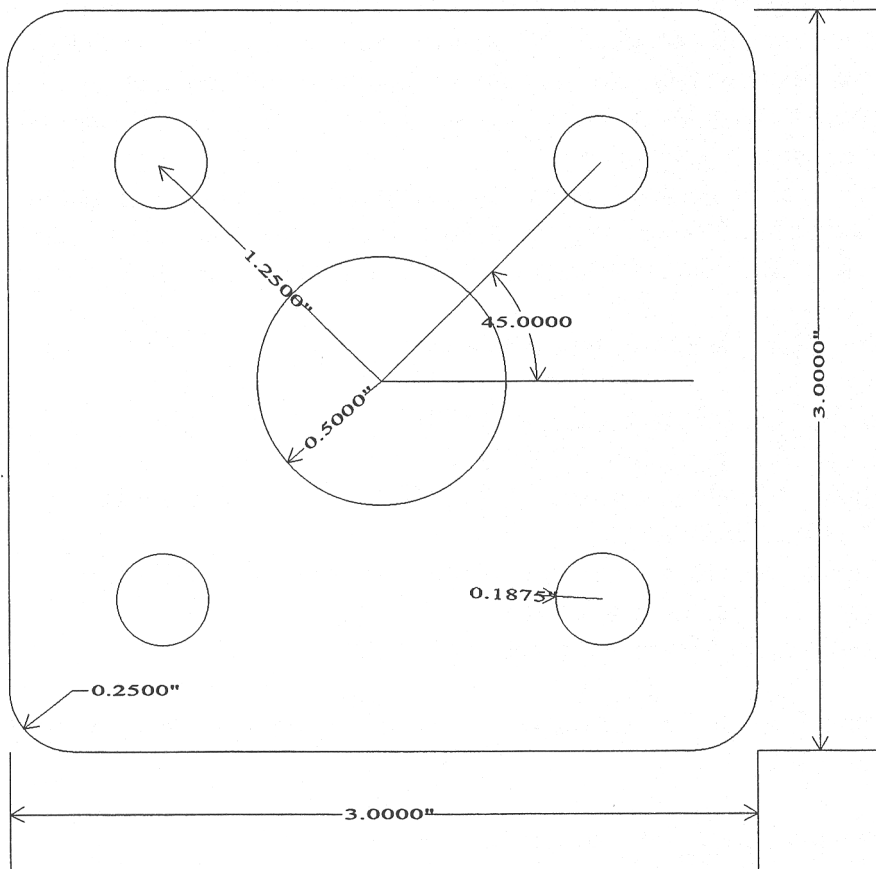
Using Corel Draw Files

Corel draw is a popular drawing program for Microsoft's windows that includes over a hundred fonts and a wide variety of clipart images and symbols. Additionally, Corel Draw can accept scanned images and will convert them to vector format. When used with Mastercam, Corel Draw becomes a powerful front end for complex engraving applications.

1. From Corel Draw, select File, Export
2. In the dialog box, enter the file name and path and select DXF or AutoCAD file format Corel Draw will automatically add the DXF extension.
3. In Mastercam, follow instructions for DXF import (above)
4. Select the Contour toolpath option to create the engraving toolpath

Mastercam Quick Start Tutorial

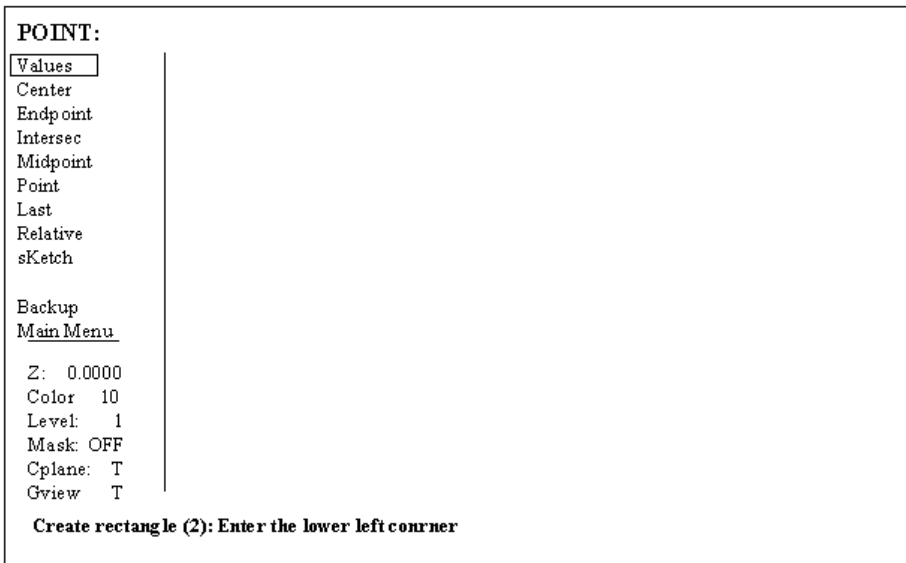
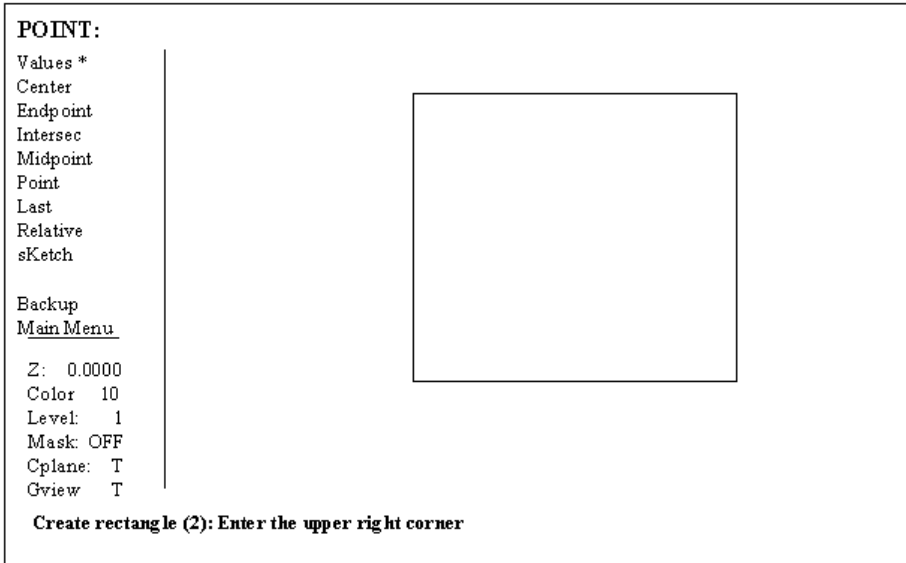
This tutorial is meant to familiarize you with use of the three basic toolpath operations contouring, pocketing and drilling. The part to be created is a simple motor flange shown below.



The first step will be to create the part geometry needed to lay a toolpath on. Starting from the Main Menu, select the following menu and sub-menu choices in the order given.

1. Create
2. Rectangle
3. 2 points

After selecting 2 *points*, The screen below will be displayed and in the lower corner of the screen, you will be ask to enter the lower left corner of the rectangle. Select **VALUES** and type <0,0>.



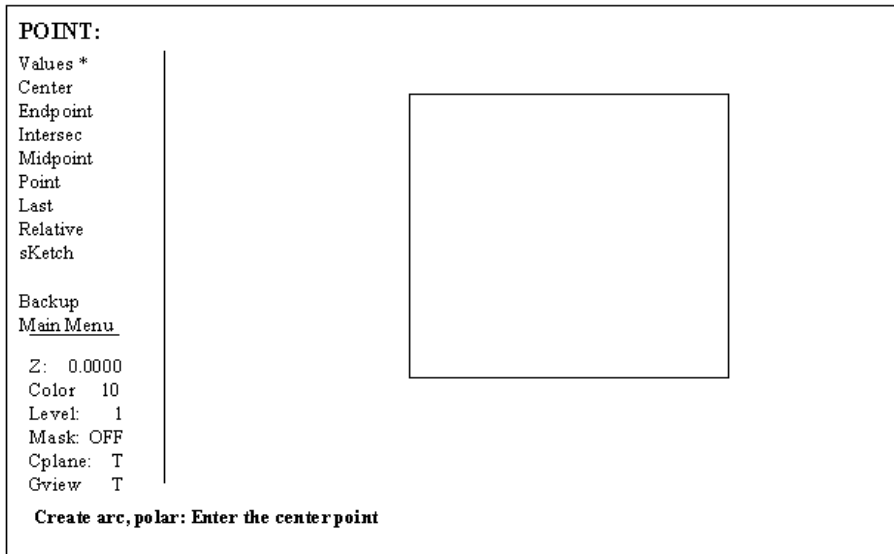
A cross will appear at 0,0 and you will be ask for the upper right corner of the rectangle. Type <3,3> and the above screen should appear with a square in the drawing area. If the square does not appear, press <Alt-F1> to fit the picture to the center of the drawing area.

Next, a circle will be created for the pocket in the center. From the **CREATE** menu select

Arc

Polar

Ctr point



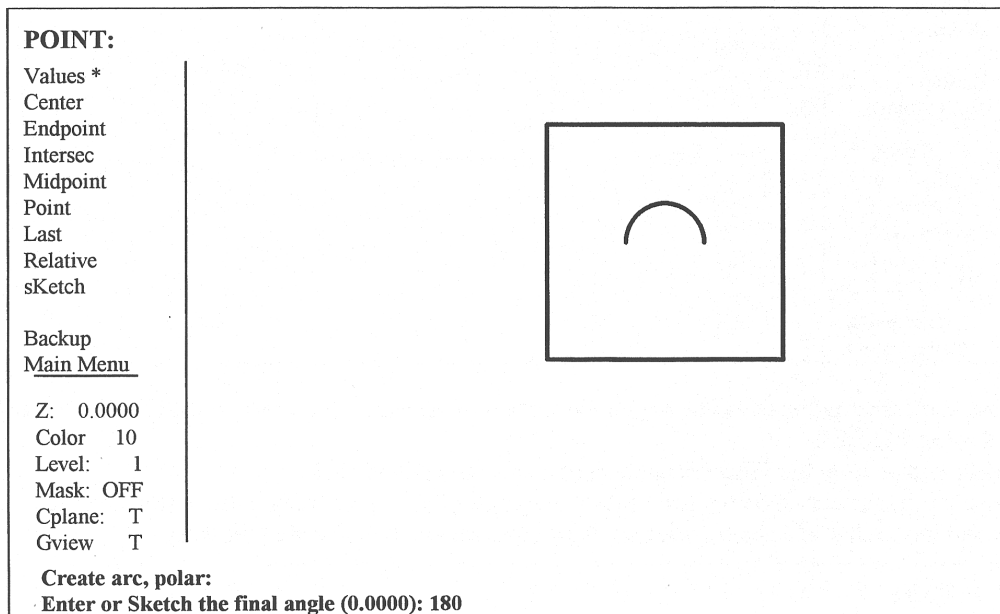
Select **VALUES** and type <1.5,1.5>. A cross will appear at 1.5,1.5 and you will be prompted for the Radius = 0.5000"

Start angle = 0

Final angle = 180

The screen should look like the following. To finish the circle repeat the above procedure, but **swap** the Start (start = 180) and Final angles (final = 0).

*NOTE: When drawing arcs the angles always go counter clockwise from the 3 o'clock position.



Points must now be placed in the four positions for drilling. From the **Create** menu select **POINT**. From the Create Point menu select the following

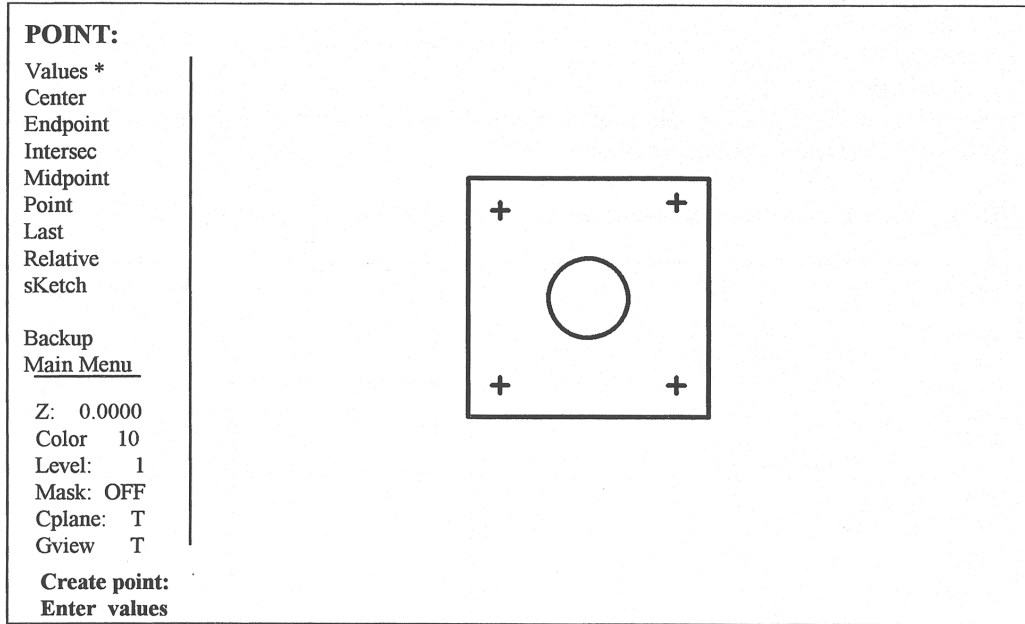
1. Position
2. Values

Type in the values for the four holes.

- POSITIONS
1. .50, .50
 2. 2.50, .50

- 3. .50,2.50
- 4. 2.50,2.50

After all four sets of values are input, screen should look like the following picture:



Next, the corner radiuses will be put on the corners of the plate. So return to the **CREATE MENU** and select **FILLET**.

Select Radius from the **Fillet: change** menu and enter a radius of .25.

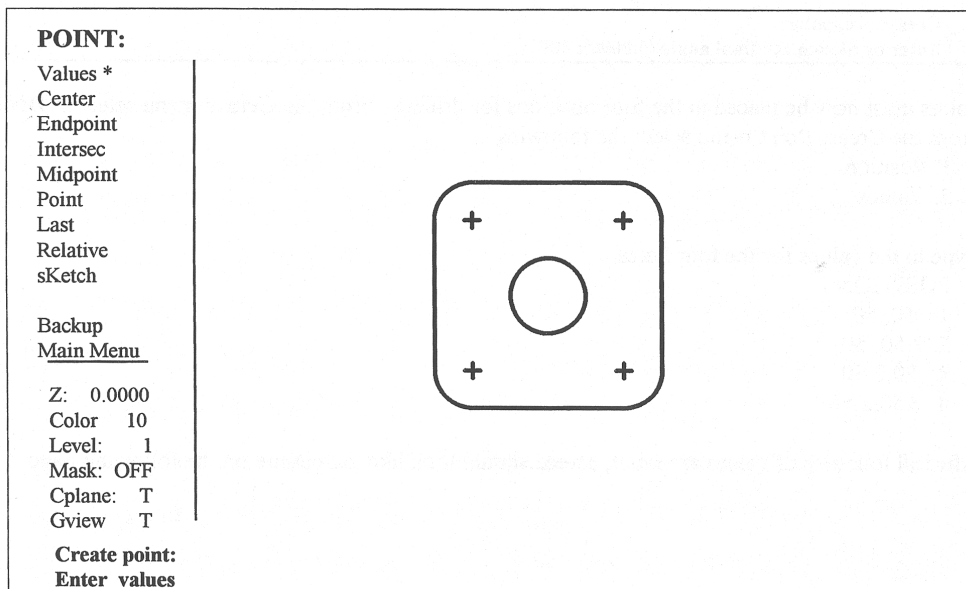
Place the mouse cursor on the first of two lines to be filleted. Press the mouse button and then select the second line. After all lines are selected, the picture should look like the following:

Save this file the drawing is complete!

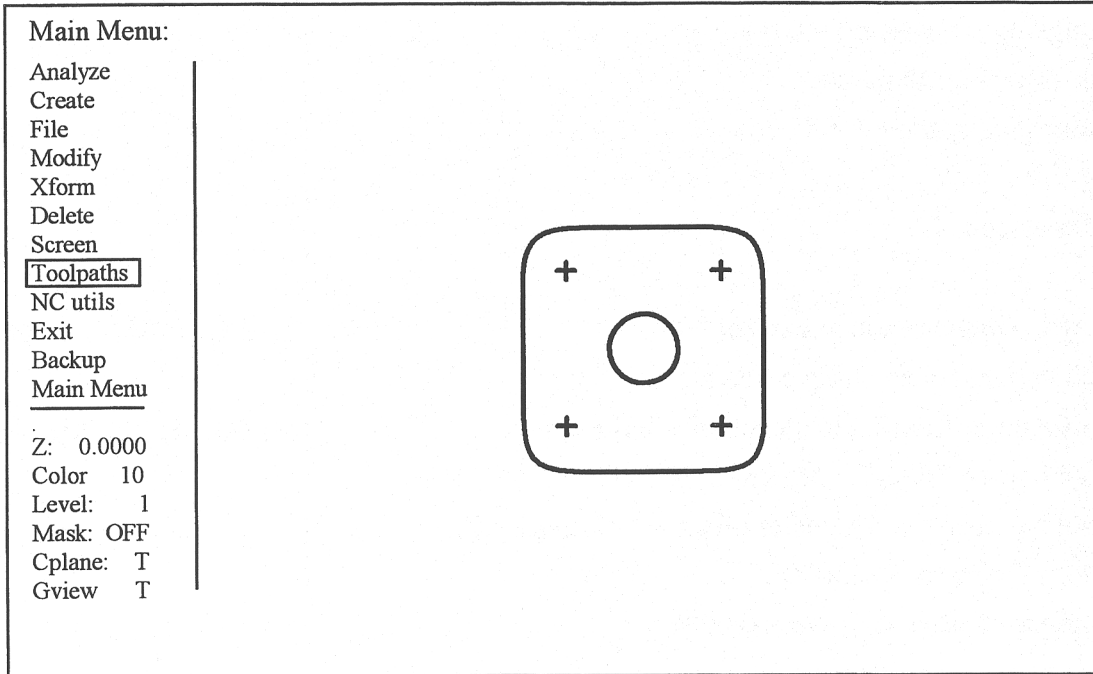
Now that the drawing is complete, we must lay down a toolpath to cut the part. From the main menu, select **Toolpath**. Three different types of toolpaths will be done.

1. Contour around the outside of the part.
2. Drill for the four holes
3. Pocket for the center hole.

*NOTE: This is not the only way to do this part.

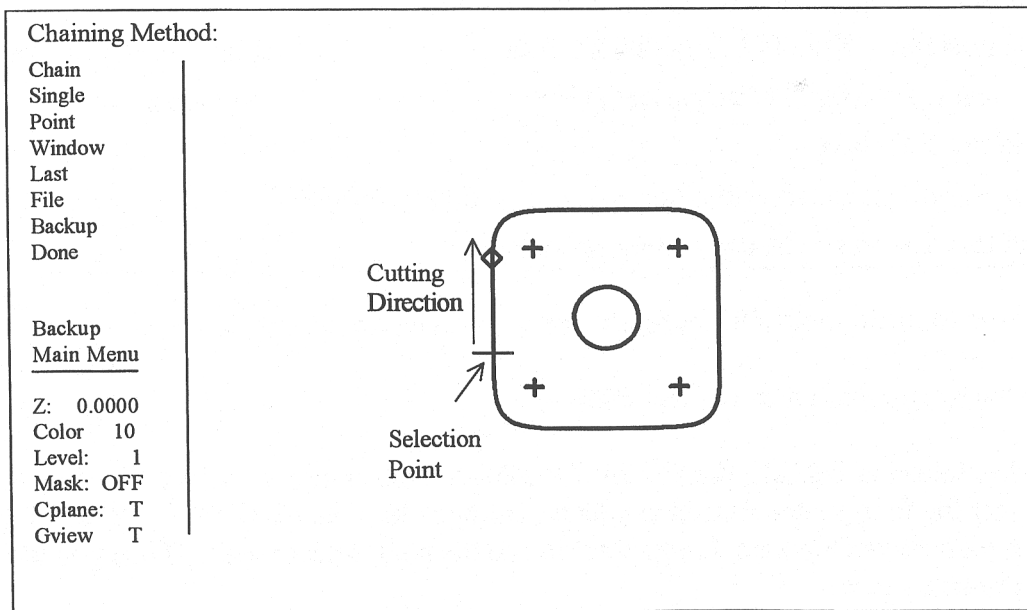


After selecting Toolpaths, the different types of paths are displayed. We will start with contour.



Select

1. Contour
2. Chain
3. With the mouse, select an entity in the proper location for direction and on the perimeter where you want the cutting to begin.
4. Close
5. End here



Select **DONE** and then **Parameters** and you will be taken to the parameters screen.

CONTOUR PARAMETERS

<p>Entry/exit line length = 0.5000 Entry/exit arc: Radius = 0.0000 Angle = 90.000 Number of roughing cuts = 0 Roughing cut spacing = 0.0000 Number of finish passes = 1 Finish pass spacing = 0.0000</p> <p>Next screen</p> <p>Cutter compensation in control = off Roll cutter around sharp corners Cutter compensation in computer = left center Tool library: NONE Material: NONE Tool Number = 1 Diameter offset = 1 Length offset = 1 Cutter Diameter = 0.5000 Amount of stock to leave = 0.0000 Feedrate = 10.0 Plunge rate = 10.0 Spindle speed = 2000 Coolant = on Rapid depth = 0.1000 Contour depth = -.2500 Starting sequence number = 10 Inc = 10 Program n.9890 No rotary axis Linear array: Nx, Ny = 1,1 Dx, Dy = 0.0000,0.0000 Depth cuts: Rough: 1 cuts at 0.0000 Finish: 0 cuts at 0.0000 Home position = X-5.0000 Y0.0000 Z0.1000 Misc. real [1] = 0.0000 Misc. integer [1] = 1 Mill in the XY plane Display Tool (animated, endpoints, run, delay=0.0000) Toolpath Select this line when through setting parameters</p>	<p>Uses G41, G42 Let Mastercam Comp</p> <p>No G40, 41, 42 - Mastercam will comp</p> <p>Contour depths in Mastercam MINUS Puts block numbers in G code program</p> <p>Machine return after program end Not used</p>
--	--

If the toolpath looks OK, accept it and start on he the next path.

From the toolpath menu, select **DRILL**.

After selecting DRILL, the SELECT POINTS menu appears. Select **MANUAL**. With the mouse, select the four points created for the mounting holes then hit <ESC>. You will be prompted to draw the path between points. If you want to see the path, select <yes>. Then you will be taken to the parameters screen.

Drill Parameters

Cycle = drill / counterbore	Select to change type drilling cycle
Dwell = 0.0000	
Initial height = 0.1000	
Reference height = 0.1000	
Depth = Incremental	
Depth = -0.3000	Depth should be negative
Next screen	
Tool library: NONE Material: NONE	
Tool Number = 1 Diameter offset = 1 Length offset = 1	
Cutter Diameter = 0.2500	

Feedrate = 10.0 Plunge rate = 10.0 Spindle speed = 2000	
Coolant = on	
Starting sequence number = 10 Inc = 10 Program n.9890	Puts block numbers in G code program
No rotary axis	
Linear array: Nx, Ny = 1,1 Dx, Dy = 0.0000,0.0000	
Home position = X-5.0000 Y0.0000 Z0.1000	Machine return after program end
Misc. real [1] = 0.0000 Misc. integer [1] = 1	Not used
Mill in the XY plane	
Select this line when through setting parameters	

The toolpath will be displayed and if it looks OK, accept it, and start on the next path. After accepting the toolpath, you will be prompted to repeat with the same holes. This allows you to do another operation on the same holes without redefining the toolpath.

From the toolpath menu, select **POCKET**. Chaining the pocket is done the same way contouring was done before. After the toolpath is created you will be prompted to ENTER AT USER DEFINED POINT. This is so you can tell Mastercam where to start if there is a clamp or other obstacle in the way. Most of the time, it is best to let Mastercam decide where to start by selecting <NO>. The parameter screens follow this selection.

Pocket Parameter

Cutting method = Zigzag	
Pocket depth = -0.2500	Pocket depth must be minus
Roughing angle = 0.0000	
Roughing cut spacing = 0.1000	
Entry/exit arc: Radius = 0.0000 Angle = 90.000	
Number of finish passes = 1	
Finish pass spacing = 0.0100	
Move between finish passes: feedrate	
Move between finish passes at: final depth	
Next screen	
Roll cutter around sharp corners	
Tool library: NONE Material: NONE	
Tool Number = 1 Diameter offset = 1 Length offset = 1	
Cutter Diameter = 0.5000	
Amount of stock to leave = 0.0000	
Feedrate = 10.0 Plunge rate = 10.0 Spindle speed = 2000	
Coolant = on	
Rapid depth = 0.1000	
Starting sequence number = 10 Inc = 10 Program n.9890	Puts block numbers in G code program
No rotary axis	
Linear array: Nx, Ny = 1,1 Dx, Dy = 0.0000,0.0000	
Depth cuts: Rough: 1 cuts at 0.0000 Finish: 0 cuts at 0.0000	
Home position = X-5.0000 Y0.0000 Z0.1000	Machine return after program end
Misc. real [1] = 0.0000 Misc. integer [1] = 1	Not used
Mill in the XY plane	

Display Tool (animated, endpoints, run, delay=0.0000) Toolpath	
Select this line when through setting parameters	

If the toolpath looks ok, accept it. Now that all three paths are done, select End Program from the toolpath menu. You will be prompted to Post Process. You must select <YES> to create a G code program to run on the mill.

Chapter 12

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 13.

N - Block Number

Block numbers are used to identify CNC program lines. Block numbers are optional, but can be used with 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
```

CHAPTER 13

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)
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.

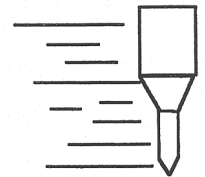
*If several G-codes from one group are used in the same line, only the G-code specified last will remain active.

*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.

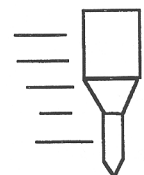
Example:

```
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

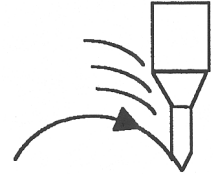
G1 moves to the specified position at the programmed feedrate. The coordinates may be either absolute positions (G90) or incremental positions (G91). The movement will be along a straight line. G1 is modal and remains in effect until another positioning mode (G0, G2, G3 etc.) is commanded.



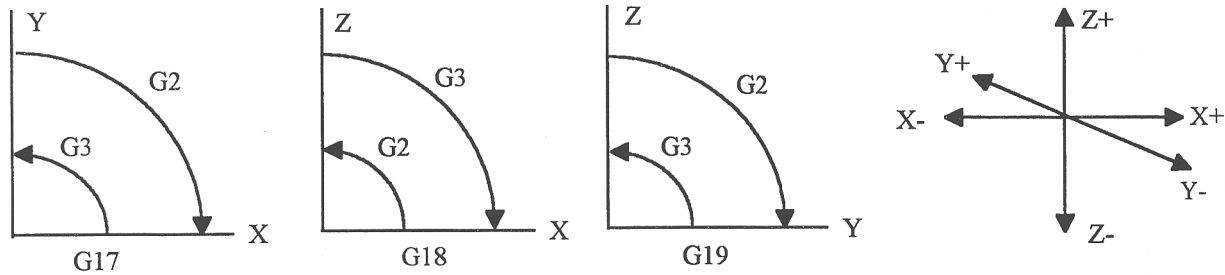
Example:

```
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:

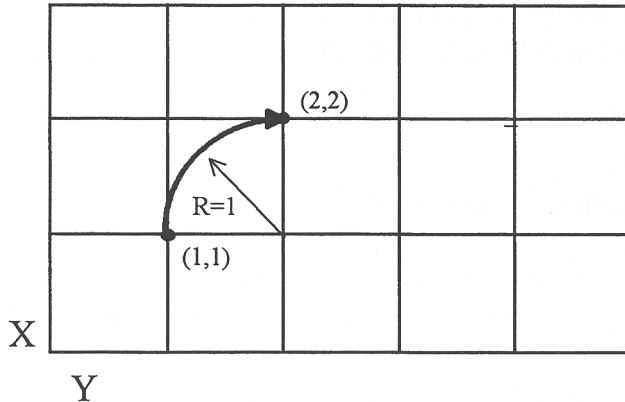
```
G2 Xa Yb Zc Rd
G3 Xa Yb Zc Rd
```

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.

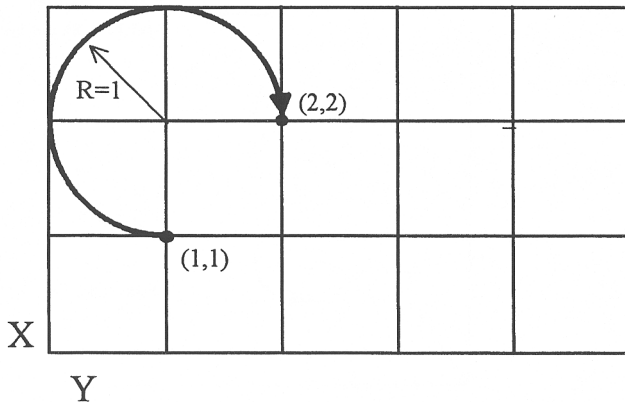
Example 1 (small arc solution: positive radius):

```
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)

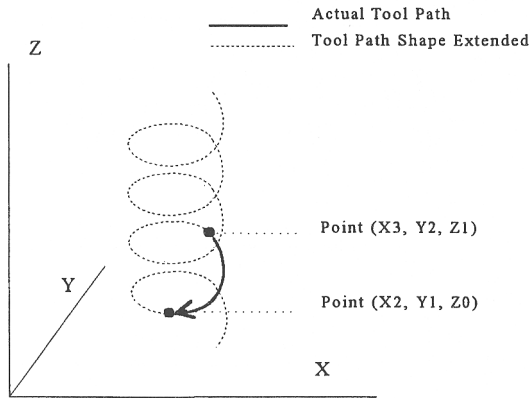
K = Z center - Z start (valid for G18 & G19)

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

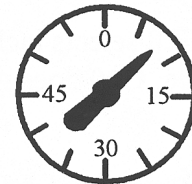


Helical motion

```
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:

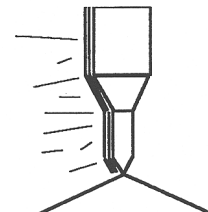
```
G0 X1 Y1 ; rapid to X1 Y1
G4 P2.51 ; pause for 2.51 seconds
X2 Y2
```

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.

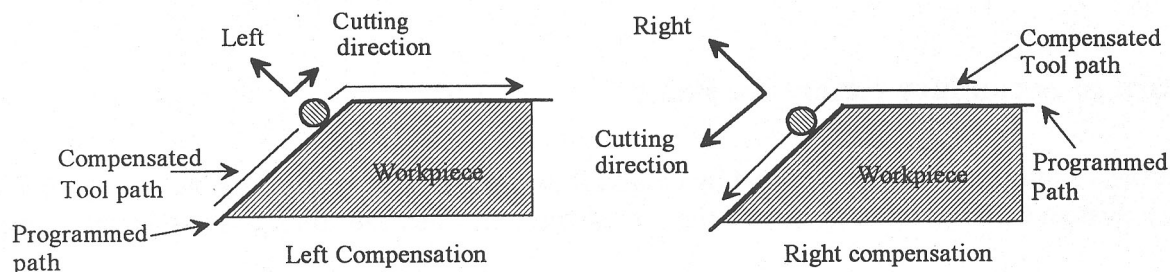
G41 offsets the cutter tool one half of the tool diameter selected with a D code, to the left of the workpiece, relative to the direction of travel.

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:

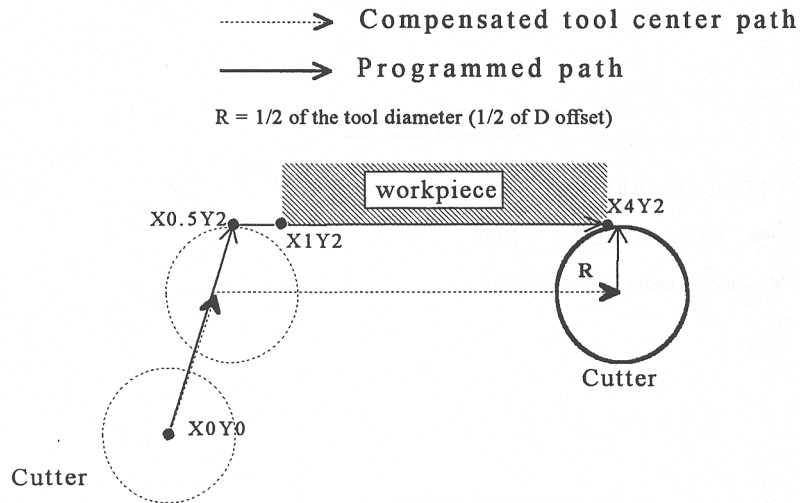
```
G41 D03      ; Tells the machine to compensate left half of the  
            ; Diameter of the amount that corresponds to D03 in the  
            ; Tool Library
```



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.

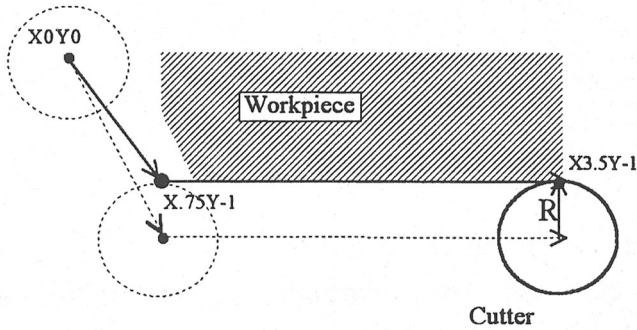
- NOTE: The diameter compensation statement G42 is placed before G0 X.5 Y2. As a result, the compensation is applied before the cutter reaches the starting cutting point X.5 Y2.
2. If the cutter is down, then the cutter compensation lead-in must always come from an appropriate direction. Otherwise, the 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
```

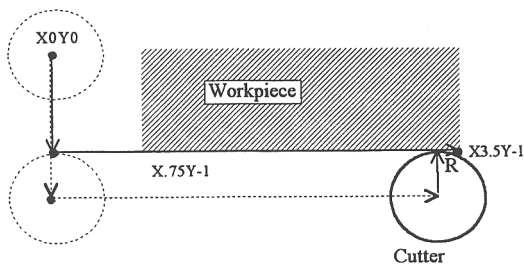
* NOTE: This problem could have been avoided by selecting a transitional point between X0 Y0 and X.75 Y-1. A transitional point such as X-1 Y-1 would properly modify the lead-in path, keeping the cutter from damaging the corner of the workpiece. Example 3 shows the correct way of performing this operation.

Example 3:

CORRECT WAY

-----> Compensated tool center path
———> Programmed path

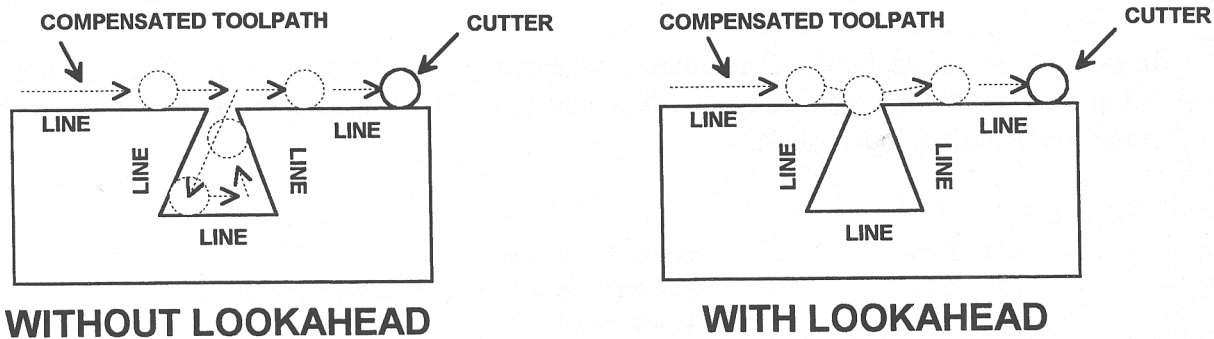
R = 1/2 of the tool diameter (1/2 of D offset)



```
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
```

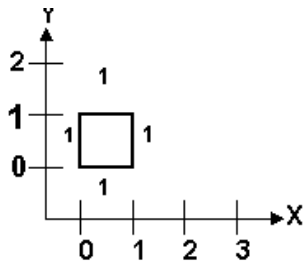
G50, G51 - Scaling / Mirroring (Optional)

G50 and G51 scales program G-codes relative to a scaling center point defined as position (X, Y, Z).

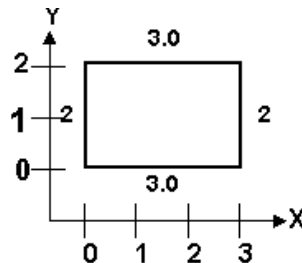
A G51 applies scaling/mirror to all positions, lines, and arcs following this G-code until a G50 is entered. Specify scaling factors with a value I, J, K. The X, Y, and Z parameters are the coordinates of the scaling center. If the scaling center is not specified, the default scaling center is the current 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 ;cancel scale
```



Original Box

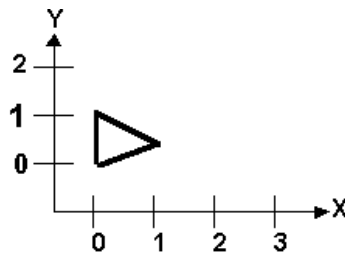


Scaled Box

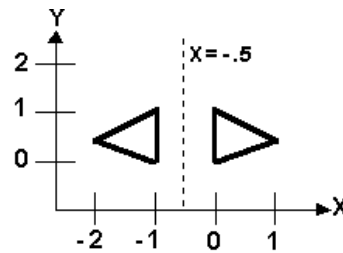
For this G51, the following program lines were scaled 3:1 in the X direction, 2:1 in the Y direction, and 1:1 in the Z direction. If no scale factor is specified, the default is 1:1 for all axes.

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
```

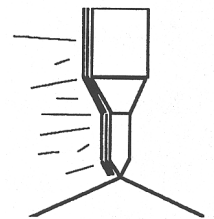
Using Extended Work Coordinate Systems: There are actually total of 18 workpiece origins. The extra workpiece origins are **not** accessible on the WCFG menu; they can only be set using Set Part 0/Position. In G-codes, the 12 additional workpiece origins may be selected with either “G54 P1” through “G54 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).

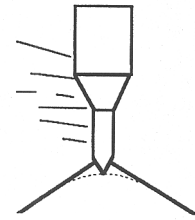
Example:

```
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 091xx to designate the beginning of the macro and M99 to end it. CNC7 will read the macro and generate a file 091xx.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:

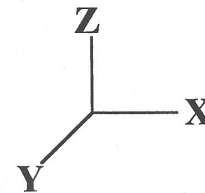
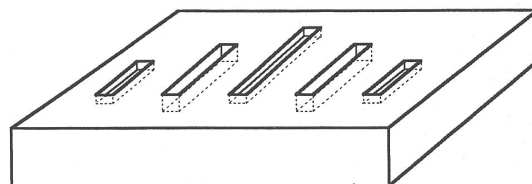
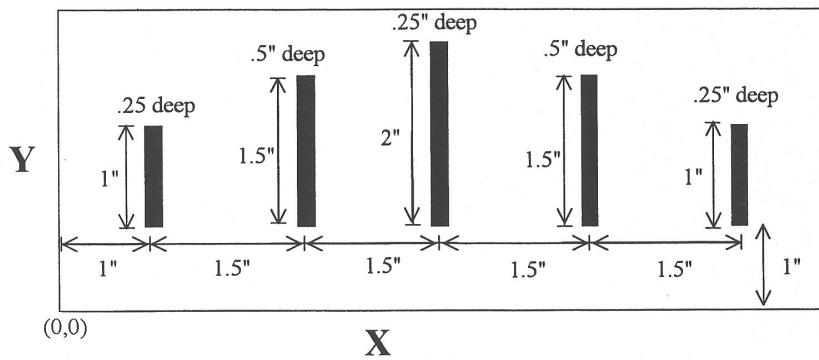
```
G1 X#4 Y#5 F#6  
G1 X#7 Y#8 F#9  
G1 X#10 Y#11 F#12
```

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

```

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)
G80	-----	-----	-----	Cancel canned 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, spotfacing
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

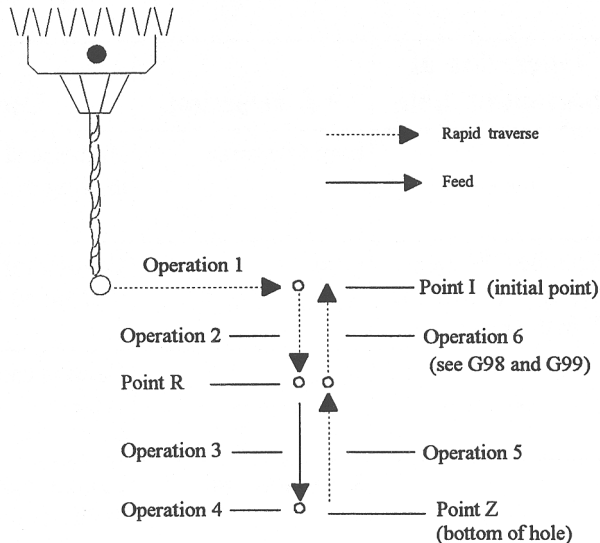
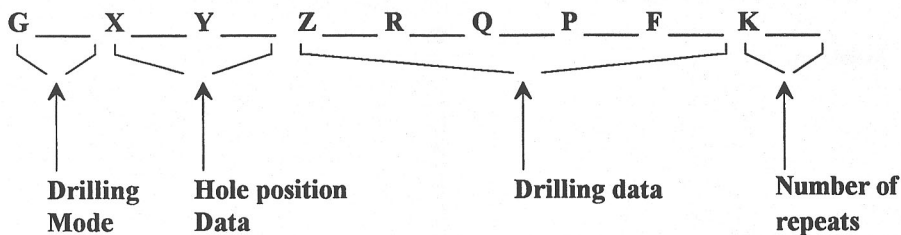


Figure 1. Drilling cycle operation

- 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.

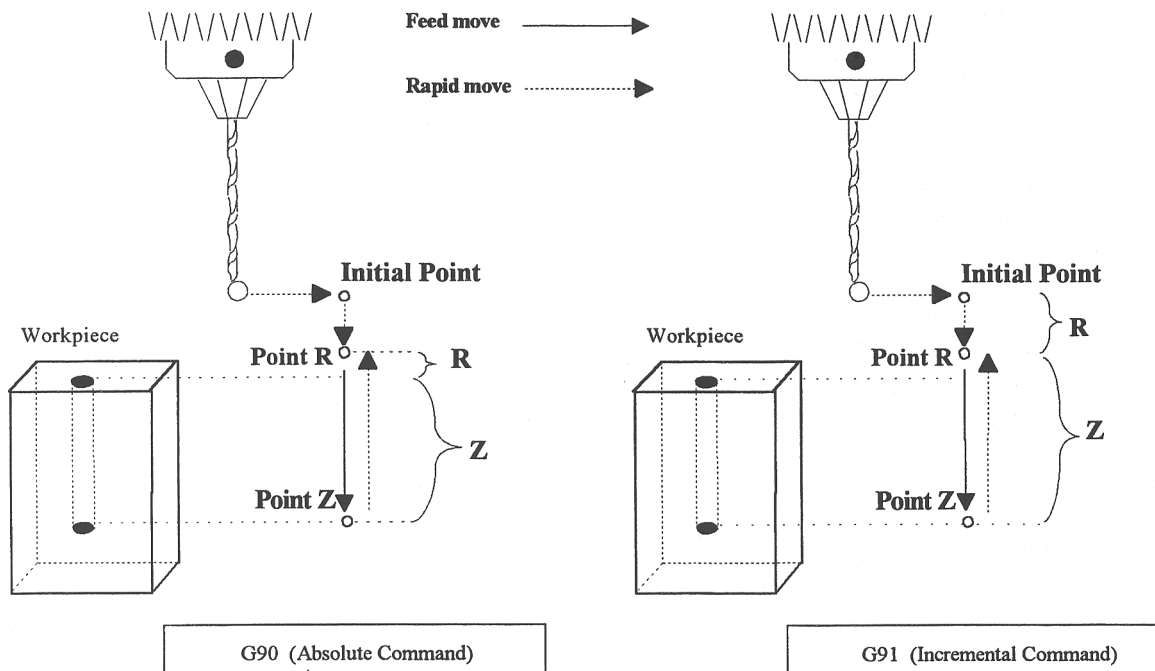


Figure 2: Canned Cycle Absolute and Incremental modes

* NOTE: In incremental mode the Z depth of the hole is measured from R, and R is measured from the initial tool position.

Example:

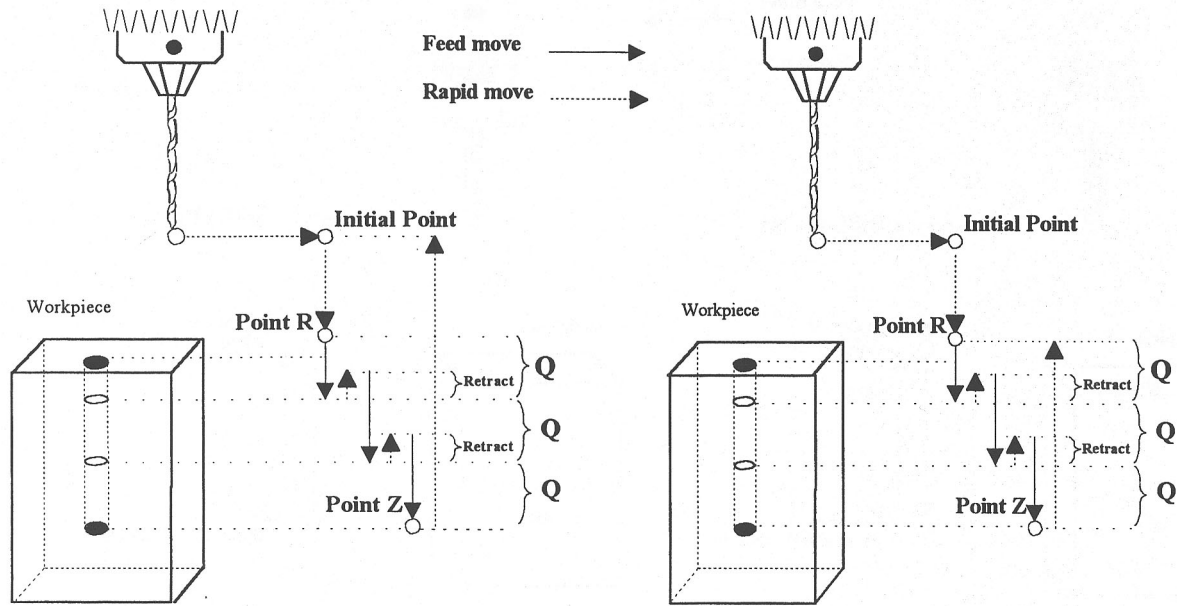
(Part surface height is Z = 0, initial tool position is X.50 Y1.0 Z.625. Drill 0.50 deep hole at X1.0 Y1.0; clearance height (R) is 0.10 above surface.)

```

; 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.

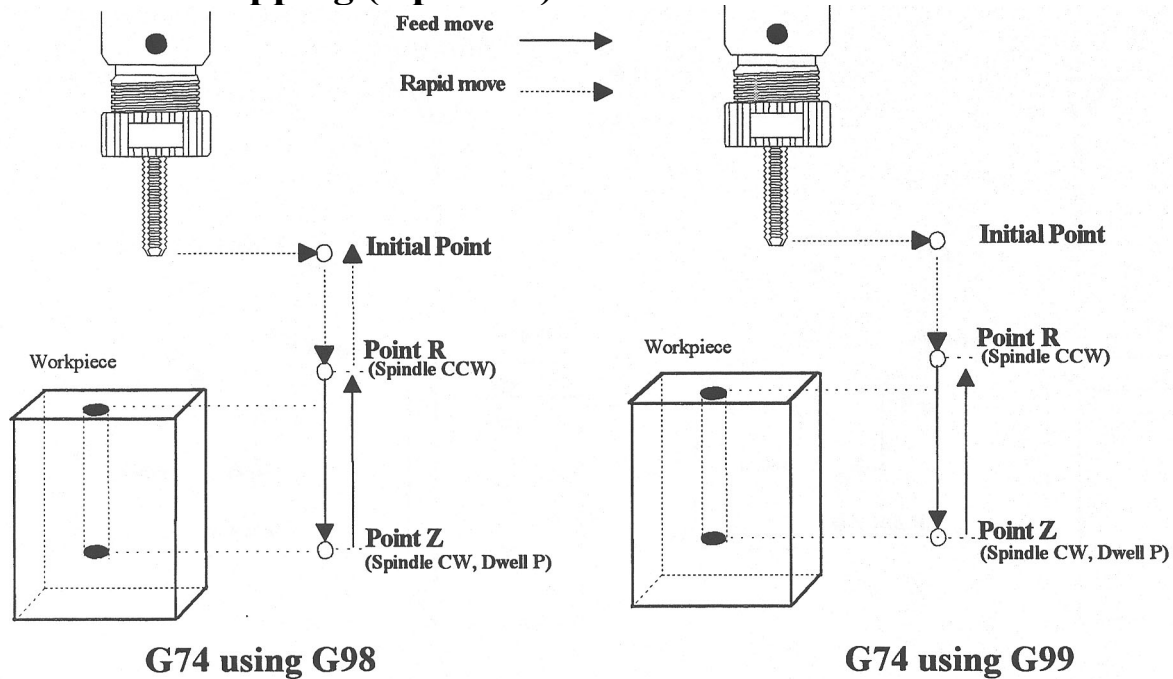
Example:

```

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.

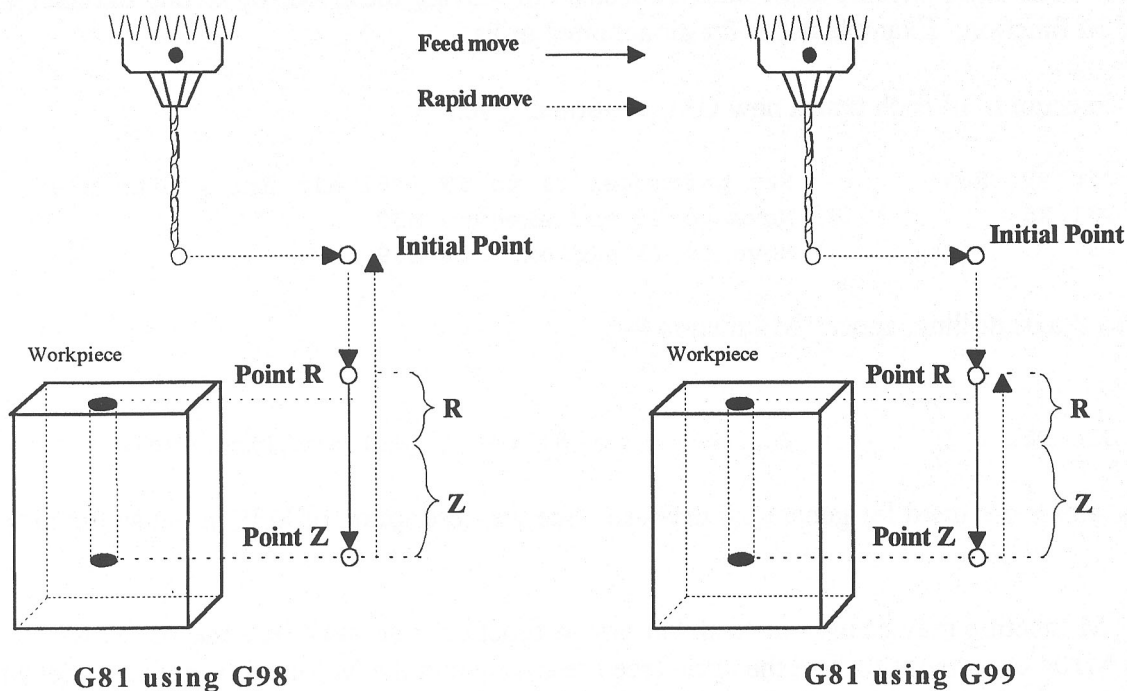
Note: If rigid tapping is enabled, a Q may be used to set the thread lead or pitch. However, because Q is not modal in the case of Rigid Tapping, you must specify Q on every line at which Rigid Tapping is to occur.

*** 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)
```

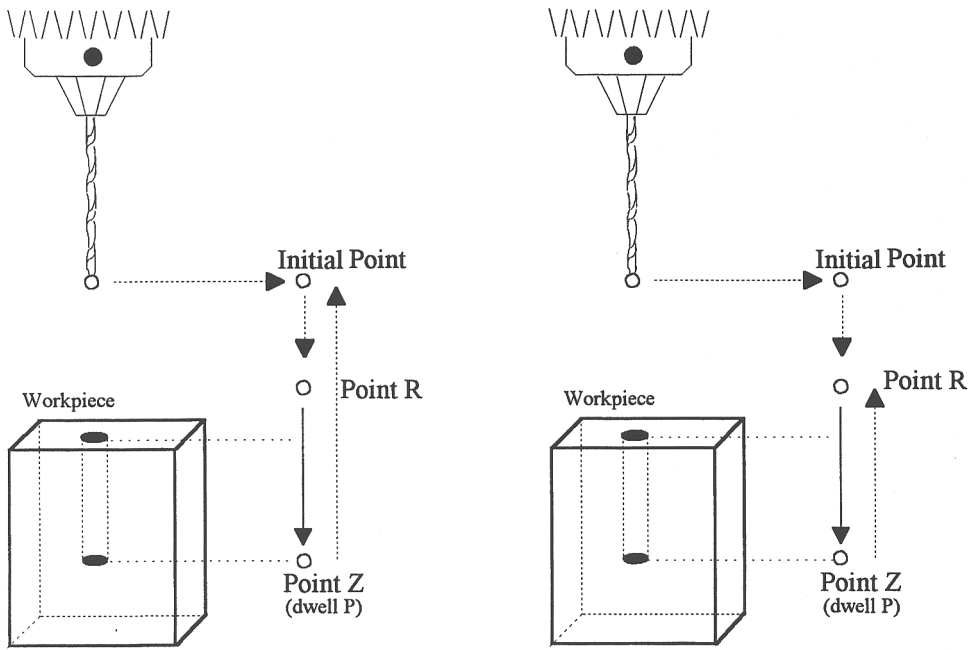
M function #39 is designed for general air drill use. See the description of M39 in the M functions section.

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 the next chapter.

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

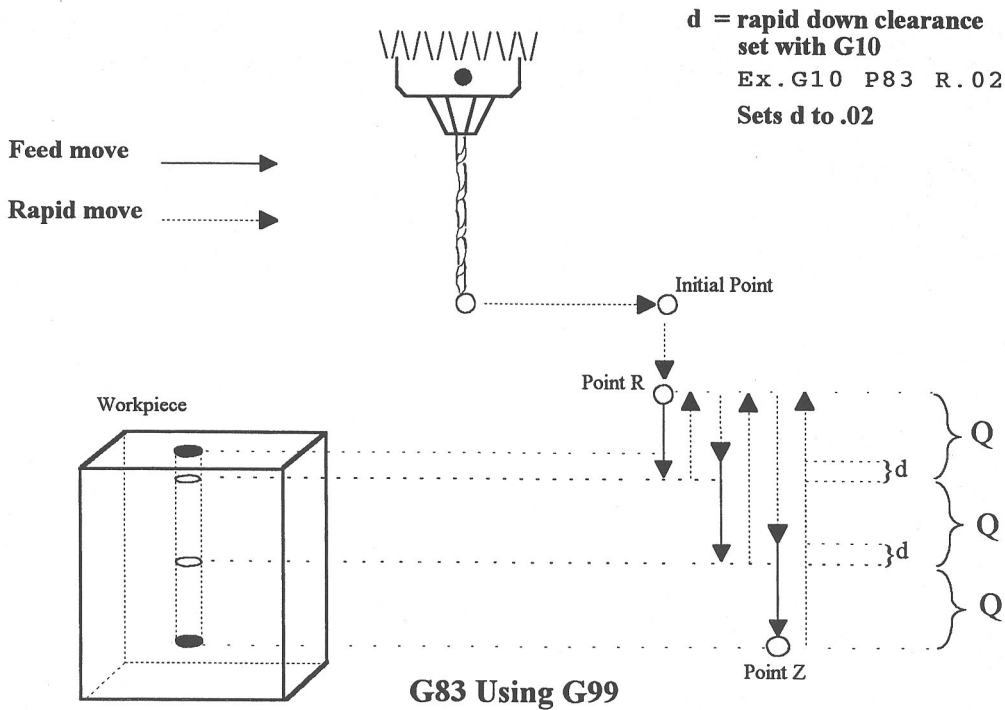
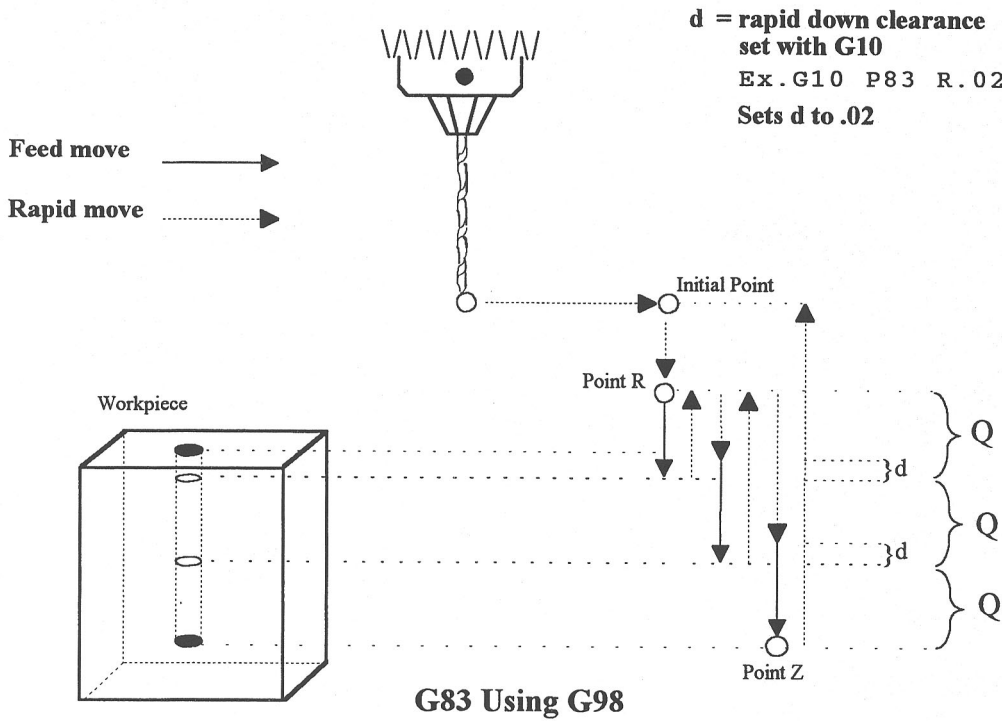
G82 Using G99

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).

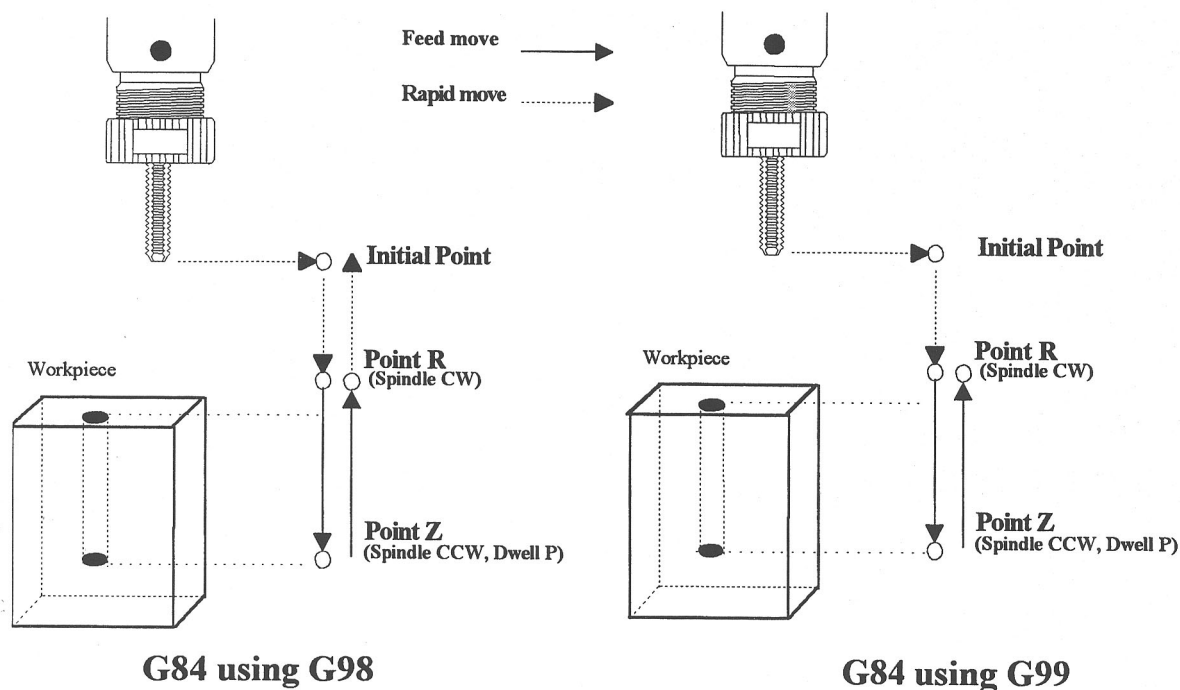
Example:

```
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
G80 ; cancel canned cycle
```

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.

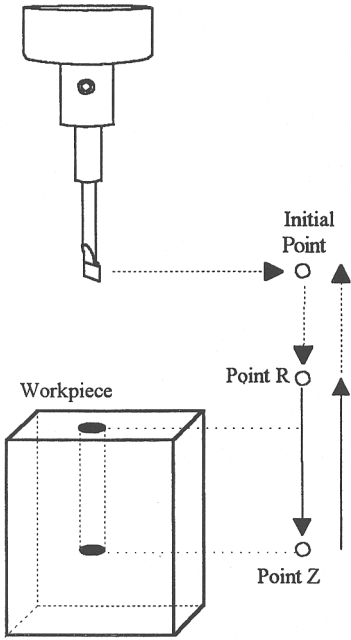
Note: If rigid tapping is enabled, a Q may be used to set the thread lead or pitch. However, because Q is not modal in the case of Rigid Tapping, you must specify Q on every line at which Rigid Tapping is to occur.

* WARNING: Do not press FEED HOLD or CYCLE CANCEL while the tap is in the hole.

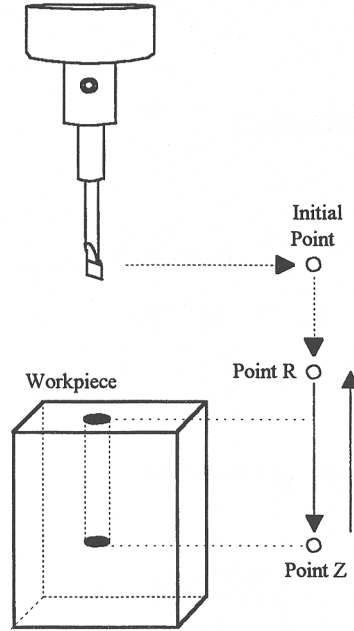
Example:

```
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.

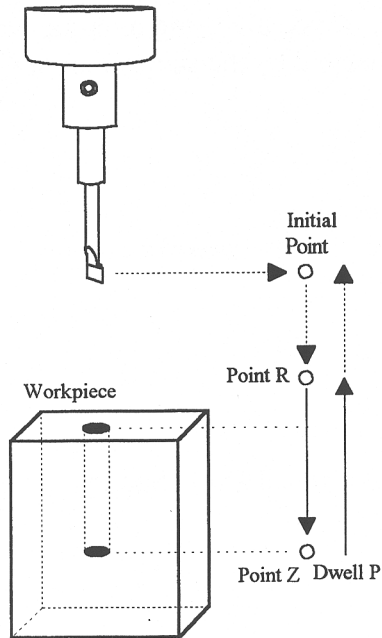
Example 1:

```
G85 X1 Y1 R.1 Z-.5      ; bore a 0.5" hole at X1 Y1  
G80                    ; cancel canned cycle
```

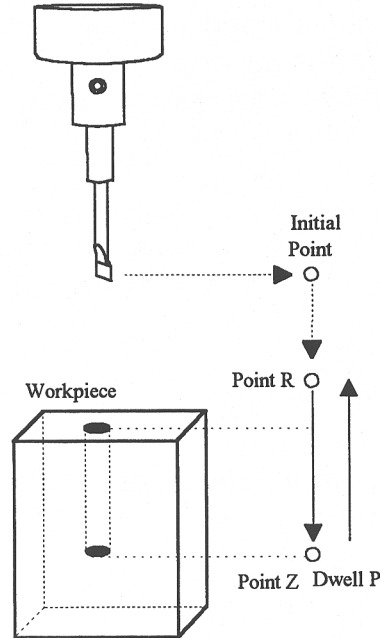
Example 2:

```
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

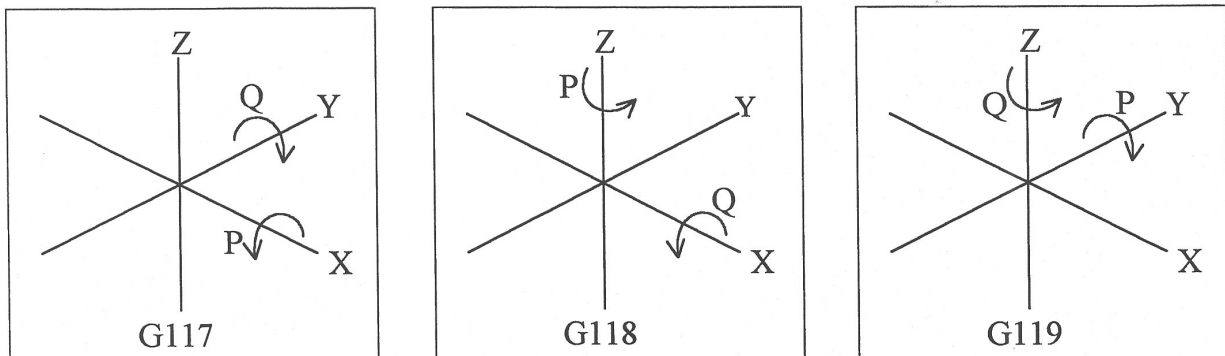
G98 sets the +Z return level to point I as pictured in Figure 1 in the Canned Cycle Section. (G98 is the default setting)

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 14

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
```

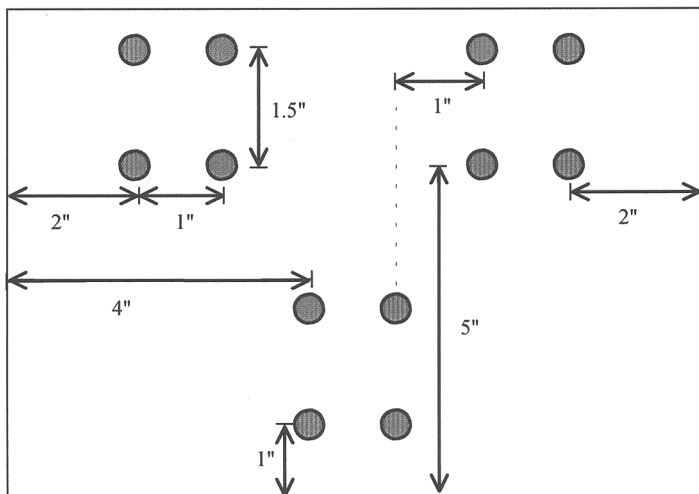
Where *xxxx* is the subprogram number (in file *Oxxxx.CNC*, 0000-9999 allowed, leading 0's required in filename), *rrrr* is the repeat value, and "program.cnc" is the name of the subprogram file.

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 00001.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

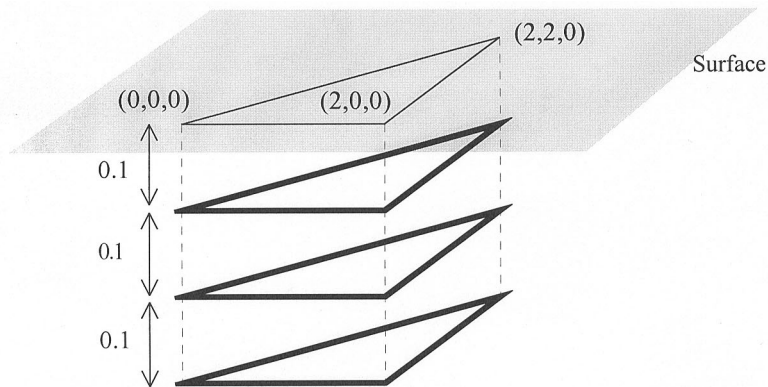
```

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 acknowledge 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
```


CHAPTER 15

Operator Panels

M-400 Operator Panel / M-39 Jog Pendant

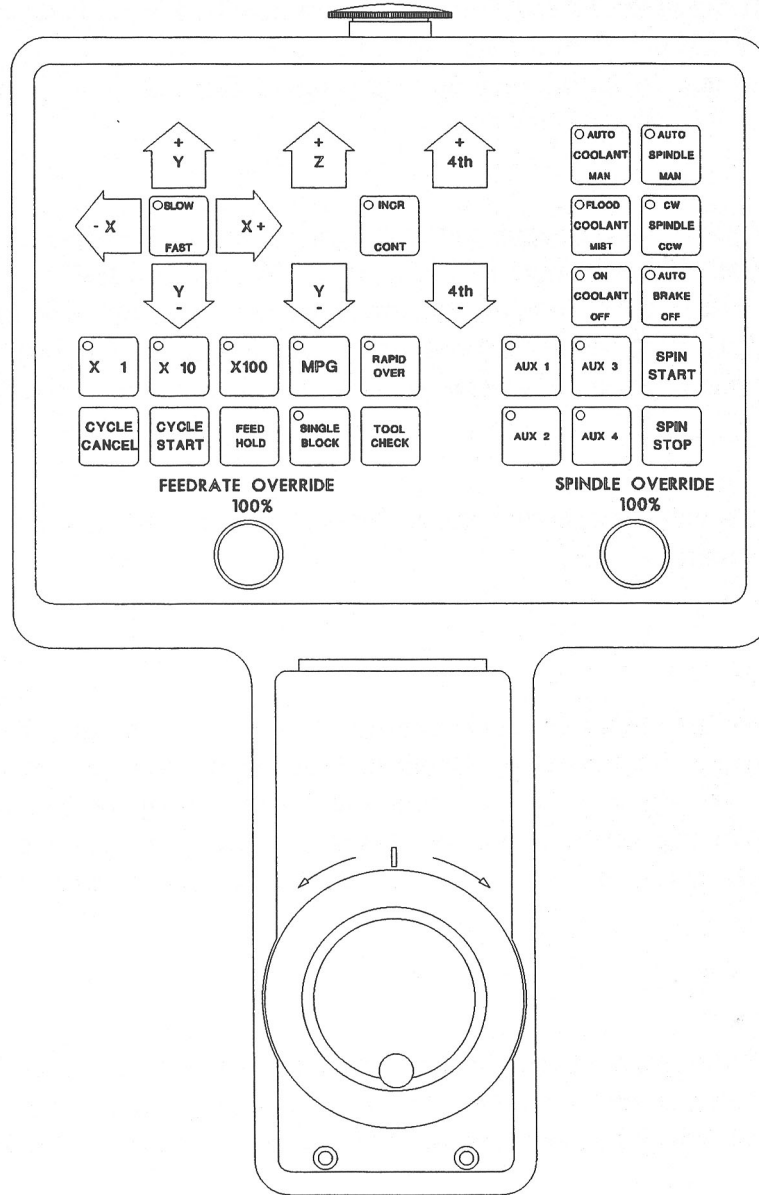


Fig 1 - M-39 Jog pendant Shown with Optional MPG Handwheel

The M-39/M-400 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 tell you the status of the machine functions.

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

SLOW/FAST selects between fast and slow jogging. 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 16 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

There are 2 styles of optional MPG handwheels available. Either style is housed in the pendant housing (M-39) or in a separate hand-held unit (M-400). If an M-39 is ordered with no handwheel, a blank cover plate is provided. These MPG handwheels can be replaced or installed in the field.

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.

Auxiliary Function Keys (AUX1 - AUX4)

The AUXILIARY FUNCTION KEYS have no set definition; the machine installer can program them. Your machine installer will provide you with documentation explaining the functions (if any) the auxiliary keys and/or LEDs perform.

COOLANT On/OFF

The COOLANT ON/OFF key is only operational when manual coolant mode has been selected. If the COOLANT ON/OFF LED is lit, coolant is ON. Pressing this button turns the coolant on and off in the manual mode.

Coolant Auto/Man

Press this key to select the coolant mode. The COOLANT AUTO/MAN LED will be lit if the auto coolant mode is selected. If the COOLANT AUTO/MAN LED is not lit, the M-Series Control is in manual coolant mode.

Coolant FLOOD/ Mist

The COOLANT FLOOD/MIST key is used to select between flood coolant and mist coolant. You can select between flood and mist coolant at any time, either in auto or manual coolant mode. The COOLANT FLOOD/MIST LED will be lit if flood coolant is selected. If the COOLANT FLOOD/MIST LED is not lit, then the mist coolant will be selected.

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 (CW/CCW)

The SPINDLE CLOCKWISE/COUNTERCLOCKWISE key determines the direction the spindle will turn if it is started manually. If the spindle is started automatically, the direction key is ignored and the spindle runs according to the program. When the LED is lit, the spindle will spin clockwise, and when the LED is not lit, the spindle will spin counterclockwise. The current direction of the spindle will be indicated by the LED when the spindle is running in automatic mode. Pressing the SPINDLE key toggles it from CW to CCW and vice versa. The default is CW.

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.**

M-15 Operator Panel

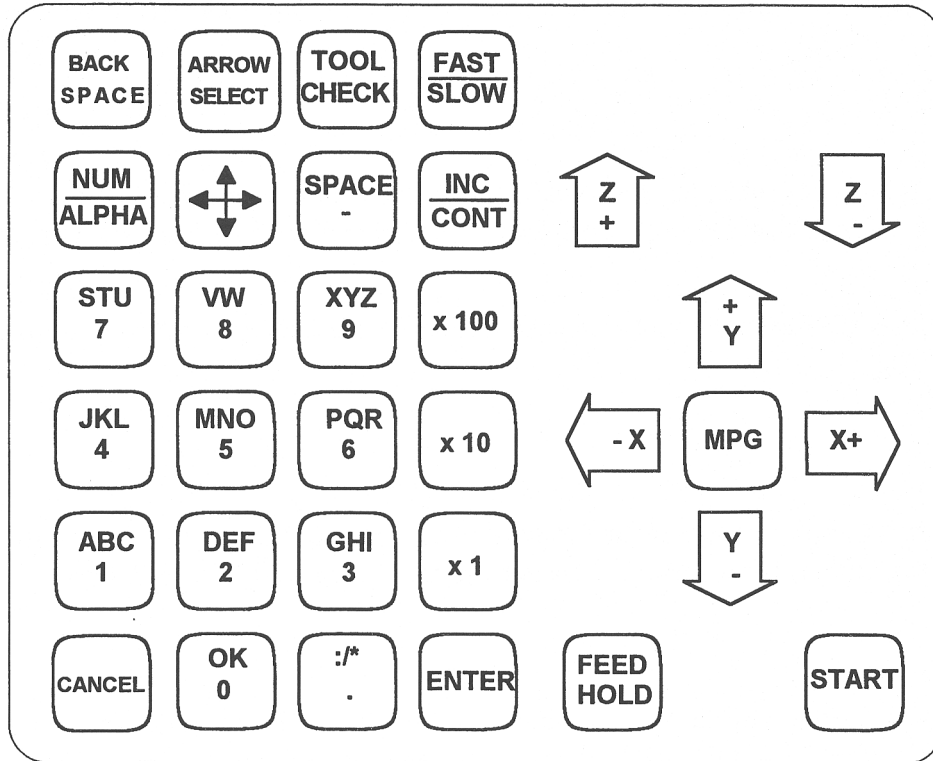


Fig 2 - M-15 Operator's Panel

The M-15 operator panel is a sealed membrane keyboard that enables you to control various machine operations and functions. The panel contains momentary membrane switches. The keys are grouped according to function (alpha-numeric, jogging controls, etc.).

Num/Alpha Key, Alphanumeric Keys

These keys work together to allow you to type numbers, letters, and selected punctuation marks. To type numbers, select numeric mode by pressing the NUM/ALPHA key until you see a '1' in the upper right corner of the screen. Then, type normally using the 10 digit keys, the '.' key, and the '-' key. To type letters, select alpha mode by pressing the NUM/ALPHA key until you see an 'A' in the upper right corner of the screen. Press the key with the letter you want once, twice, or three times (to select the first, second, or third letter displayed on that key). When the letter you want appears, press the OK key (0).

```
Job Name: FLANGE . CNC      ↓ 1
Feedrate: 100%
Fast Inc .0010"
Feed Hold: Off
```

```
Job Name: FLANGE . CNC      ↓ A
Feedrate: 100%
Fast Inc .0010"
Feed Hold: Off
```

Arrow Select Key, Cursor Arrow Key

These keys work together to allow you to move the cursor around the screen. The control has a single arrow key, which serves as the Up, Down, Left, and Right arrows. The ARROW SELECT key determines which of the four is active. The current arrow mode is indicated in the upper right corner of the screen, next to the NUM/ALPHA indicator.

```

Job Name: FLANGE . CNC      ↓ 1
Feedrate: 100%
Fast Inc .0010"
Feed Hold: Off

```

```

Job Name: FLANGE . CNC      ↓ A
Feedrate: 100%
Fast Inc .0010"
Feed Hold: Off

```

Axis Jog Buttons

X+ X- Y+ Y- Z+ Z-

The yellow X, Y, and Z keys are momentary switches for jogging each of the three axes of the machine. There are two buttons for each axis (+/-). Only one axis can be jogged at a time.

Jog Speed Selection

FAST/SLOW

FAST/SLOW selects between fast and slow jogging. When FAST jog is selected and a jog button is pressed, the axis will move at the fast jog rate. If slow jog is selected, the axis will move at the slow jog rate. See Chapter 16 for information on setting the fast and slow jog rates for each axis.

The current Jog Speed is displayed in the status window in the upper right corner of the screen.

```

Job Name: FLANGE . CNC      ↓ 1
Feedrate: 100%
Fast Continuous
Feed Hold: Off

```

```

Job Name: FLANGE . CNC      ↓ 1
Feedrate: 100%
Slow Continuous
Feed Hold: Off

```

Jog Mode Selection

INC/CONT

INC/CONT selects between continuous and incremental jogging. Pressing the key will toggle between those two modes. When the INC jog is selected, as indicated on the screen, 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. If CONT jog is selected and an axis jog button is pressed, the axis will move continuously until the button is released.

The current Jog Mode is displayed in the status window.

```

Job Name: FLANGE . CNC      ↓ 1
Feedrate: 100%
Fast Inc .0100"
Feed Hold: Off

```

```

Job Name: FLANGE . CNC      ↓ 1
Feedrate: 100%
Fast Continuous
Feed Hold: Off

```

*NOTE: The jog buttons will not operate if the control is PAUSED, the M-Series CNC software is **not** running, or a job (a CNC program) is running.

Jog Increment Keys

x1, x10, x100

To set the jog increment, select any one of these keys. The amount you select here is the distance the control will move an axis if you make an incremental jog. You may select only one jog increment at a time, and the current jog increment is indicated on the screen. 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.

If incremental jog mode or MPG mode is selected, the current Jog Increment is displayed in the status window.

Job Name: FLANGE . CNC	↓ 1
Feedrate: 100%	
Slow Inc .0100"	
Feed Hold: Off	

Job Name: FLANGE . CNC	↓ 1
Feedrate: 100%	
Slow Inc .0001"	
Feed Hold: Off	

MPG

This key was used in earlier controls to select between using the Jog Keys and using the MPG. It has no effect on the Control. It will allow either the Jog Keys or the MPG to be used without switching modes.

TOOL CHECK

Pressing TOOL CHECK while **no** program is running will move the Z-axis to its home position. However, pressing TOOL CHECK while a program is running will cause the M-15 Control to abort the current 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, the user will be able to change tool settings (height offsets, diameter offsets, etc.) and resume the job with the new tool settings.

START

When the START button is pressed, it will cause the M-15 Control to immediately begin processing the current program at the beginning and prompt you to press the START button again to begin execution of the current program. After an M0, M1, M2, or M6 is encountered in the program, the message

Press START to continue

will be displayed on the screen, and the M-15 will wait until the operator presses the START button before program execution will continue.

* NOTE: Pressing START will cause the M-15 Control to start moving the axes immediately without further warning. Be certain that you are really ready to start the program when you press this button. Pressing the FEED HOLD button or the CANCEL button will stop any movement if START is pressed accidentally.

CANCEL

Pressing CANCEL while a program is running will cause the M-15 Control to abort the current program. The control will stop moving immediately, clear all M-functions, and return to the main screen. It is recommended that FEED HOLD be pressed first before CANCEL. If you press CANCEL, program execution will stop; if you wish to restart the program you may rerun the entire program or use the search function. See search function operation in Chapter 5.

FEED HOLD

FEED HOLD decelerates motion of the current movement to a stop, pausing the job that is currently running. Pressing FEED HOLD again will continue the movement from the stopped location.

FEEDRATE OVERRIDE

This knob is the percentage of the programmed Feedrate that you want to use during feedrate cutting moves: lines, arcs, canned cycles, etc. This percentage can be from 2% to 200%.

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-axis will sometimes fall due to the lack of power.

CHAPTER 16

Configuration

(F3 from Setup)

WCS #1 (G54)	Current Position (inches)	Job Name: FLANGE . CNC
X	+ 0 . 0 0 0 0	Feedrate: 100%
Y	+ 0 . 0 0 0 0	Spindle: 0
Z	+ 5 . 0 0 0 0	Feed Hold: Off
Configuration		
Stopped		
Press CYCLE START to start job		
Control F1	Machine F2	Params F3
PID F4	WCS F5	Exit ESC

General

The first five options, <F1> through <F5>, 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 #2 (G55)	Current Position (inches)	Job Name: FLANGE . CNC
X	+ 0 . 0000	Feedrate: 100%
Y	+ 0 . 0000	Spindle: 300
Z	+ 0 . 0000	Feed Hold: Off
		Stopped Processing... Waiting for PLC operation Stopped
Control Configuration		
DRO display units:	Inches	(Inches / Millimeters)
Machine units:	Inches	(Inches / Millimeters)
Max spindle (high range):	4200.0	(1.0 to 500000.0 RPM)
Min spindle (high range):	0.0	(0.0 to 500000.0 RPM)
Machine home at pwrup:	Yes	(No / Yes)
PLC type:	Normal	(Absent / Normal / Lite / Dual)
Console type:	M-400 10.4"	
Jog panel required:	Yes	(No / Yes)
Screen blank delay:	20	(1 to 200 minutes)
Remote Drive & Directory:	E:\CNC7\NCFILES	
Press SPACE to change		
		<input type="button" value="Save F10"/> <input type="button" value="Cancel ESC"/>

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 12).

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 whether the machine will automatically home itself at powerup, or will have to be manually jogged to the home position. Set Machine Home at Powerup to Yes 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 No 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 an RS232 null modem cable or 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. You must run the INTERSVR program (supplied with IBM DOS) on the attached computer in order for this feature to work using the RS-232 connection.

Machine Configuration

Pressing <F2> from the configuration screen will display the machine configuration screen in the edit window (see figure below). The machine configuration screen provides you with a method of changing machine dependent data.

WCS #1 (G54)	Current Position (inches)	Job Name: FLANGE . CNC						
X	+ 3 . 5 0 0 0	Feedrate: 100%						
Y	+ 2 . 7 4 3 0	Spindle: 2992						
Z	- 0 . 2 5 5 0	Feed Hold: Off						
Machine Configuration								
Axis	Slow Jog (in/min)	Fast Jog (in/min)	Max Rate (in/min)	Deadstart (in/min)	Delta Umax (in/min)	Travel (inches)		
1	24.0000	200.0000	240.0000	6.0000	3.0000	-0.0000		
2	24.0000	28.3465	28.3465	6.0000	3.0000	-0.0000		
3	24.0000	28.3465	28.3465	6.0000	3.0000	-0.0000		
4	24.0000	28.3465	28.3465	6.0000	3.0000	-0.0000		
Axis	Label	Motor revs/in	Encoder counts/rev.	Lash Comp. (inches)	Limit - +	Home - +	Dir Rev	Screw Comp
1	X	5.0000	4000	0.0000	1 2	1 2	N	N
2	Y	127.0000	4000	0.0000	3 4	3 4	N	N
3	Z	127.0000	4000	0.0000	5 6	5 6	N	N
4	N	127.0000	4000	0.0000	0 0	0 0	N	N
Jog F1		Motor F2		Fnd Hom F3		Set Hom F4		Exit ESC

If you wish to change a field, press <F1> or <F2> to select the Jog or Motor fields, 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).

There are two sets of data in the Machine Configuration edit window. The upper set contains jog and feedrate information. The lower set contains information about the motors, ballscrews, and switches installed on your machine.

F1 - Jog Parameters (Values should be recorded on the Control Parameters page at beginning of manual.)

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 (Inches): The maximum distance the axis can travel 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.)

The <F2> key is used to set the motor parameters for each axis. 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.

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.

* 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.

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.

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).

0	0.0000	20	72.0000	40	0.0001	60	0.0000	80	0.0000
1	0.0000	21	0.0200	41	0.2500	61	0.5000	81	1.0000
2	0.0000	22	0.0200	42	51.0000	62	115.0000	82	0.0000
3	0.0000	23	0.0200	43	0.0100	63	1.5000	83	0.0500
4	1.0000	24	0.0200	44	0.0000	64	0.0000	84	3.0000
5	0.0000	25	0.6800	45	0.0000	65	1.0000	85	0.0000
6	1.0000	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	0.0000
8	2.0000	28	0.6800	48	0.0000	68	0.0000	88	0.0000
9	0.0000	29	150.0000	49	0.0000	69	0.0000	89	0.0000
10	0.0000	30	180.0000	50	1.0000	70	0.0000	90	0.0000
11	15.0000	31	0.0000	51	0.0000	71	0.0000	91	0.0000
12	12.0000	32	19200.0000	52	0.0010	72	0.0000	92	0.0000
13	0.0500	33	1.0000	53	0.0100	73	0.0500	93	0.0000
14	10.0000	34	8000.0000	54	0.0000	74	1.0000	94	1.0000
15	1.0000	35	2.0000	55	0.0000	75	0.0000	95	2.0000
16	10.0000	36	0.0000	56	8000.0000	76	0.0000	96	2.0000
17	0.0000	37	0.0000	57	8000.0000	77	0.0000	97	2.0000
18	2.0000	38	0.0000	58	8000.0000	78	0.0000	98	2.0000
19	0.0000	39	200.0000	59	8000.0000	79	0.0000	99	0.0000

Save F10	Cancel ESC
-------------	---------------

* NOTE: Many machine parameters can also be set with the G10 G-code.

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 Vector Drive 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	200
40	Basic Jog Increment	0.0001
41	User Jog Increment	0.25
42	Password for Configuration Menus	0
54	Centroid Advertisement Suppression	0
56-59	Hard Stop Limiters	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

70	Offset Library Inc/Decrement Amount	.001"/.02mm
71	Part Setup Detector Height	0
73	Peck Drill Retract Amount	0.05
74	Spindle CCW M-Function	4
81	Air Drill M-Function	-1
83	Deep Hole Clearance Amount	0.05
84	Spindle CW M-Function	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	10

Parameter 1 – Y jog key orientation

This parameter configures the direction of movement that the Y+ and Y- jog keys would cause. This should always be set to 0 except for very special applications.

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

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 or Interlink drives. If this parameter is a 1, the Load Job menu will load a job file from a drive higher than C: by making a copy of it and placing it on the control's hard drive (in the C:\CNC7\NCFILES directory). When the job file is later run, the control will use the copy that resides on the control's hard drive. However, if this parameter is 0, that job file will not be copied to the control's hard drive. A reference to the file will be created instead. When this job file is later run, the control will read the job file from whichever drive it resides.

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	Unused	----

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 14. It also affects whether the ATC key is present in the Tool Offset Setup.

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	Decelerate to stop on M105 and M106?	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 = 1, 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	4 TH	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 Vector Drive type spindle controller, you should not set this parameter to 0.

Value	Meaning
-1	12 bits to CPU7 , then to Koyo PLC Direct, then to Vector Drive
0	8 bits to CPU7 controlled spindle (Not for Vector Drive)
1	To COM1 serial port, then to Vector Drive
2	To COM2 serial port, then to Vector Drive

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 Baldor Vector Drive or SPIN232 board. This parameter has meaning only if the above 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).

Parameter 41 - User Jog Increment

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. It has no effect on the new consoles.

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 54 – Centroid Corporation Advertisement Suppression

If this parameter is set to a 1, then the Centroid advertisements that are normally displayed upon start-up, on the Configuration Menu, and on the Utility Menu are suppressed.

Parameters 56-59 - Hard Stop Limiters

If your machine is set up to home off of hard stops (instead of limit switches or home switches), these parameters select, for each axis, how much force the control will apply before it determines it has reached the stop. Values range from 0 to 32,000. The default value is 8000, which will be used if the parameter is left at 0. If this parameter is too low, the homing sequence will stop short of the hard stop, setting machine home in the wrong location. If this parameter is too high the machine may damage the stop.

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.

Parameter 70 - Offset Library Inc/Decrement Amount

Sets the increment and decrement amount used in the offset library.

Parameters 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.

Parameters 73, 74, 81, 83, and 84 - Canned Cycle Parameters

These parameters are associated with the canned drilling and tapping cycles. For a complete description of the use of these parameters, refer to the G-code in which they are used (e.g. G73 uses Parameter 73).

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

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 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).

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.

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: FLANGE . CNC						
X	+3.5000	Feedrate: 100%						
Y	+2.7430	Spindle: 2992						
Z	-0.2550	Feed Hold: Off						
Press CYCLE START to start job								
PID Configuration								
Axis	Kp	Ki	Kd	Limit	Kg	Kv1	Ka	Accel. (Max. Vel.)
X	1.000	0.00391	20.000	32000	0	0	0	0.780 240.0
Y	1.000	0.00391	20.000	32000	0	0	0	0.780 28.3
Z	1.000	0.00391	20.000	32000	0	0	0	0.780 28.3
N	1.000	0.00391	20.000	32000	0	0	0	0.780 28.3
Axis	Error	Sum	Delta	PID Out	Abs Pos	Line	PID Collection Program	
X	0	0	0	OFF	0	1		
Y	0	0	0	OFF	0	2		
Z	0	0	0	OFF	0	3		
N	0	0	0	OFF	0	4		
						5		
PID Collection Axis: X			Density: 1		File Extension: DAT			
PID F1	Prog. F2	Collect F3	Save F4	Tune F5	Drag F6	Exit ESC		

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 Vel: 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 information in the lower left hand side of the edit window provides information to qualified technicians about the selected axis.

F4 - Save Data

This option allows qualified technicians to save any data that was collected after pressing <F3>. It uses the File extension field located at the bottom of the screen. The collected data is saved as SUMMARY.extension.

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.

CHAPTER 17 - 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.

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.

105. Message: CNC7.PLC file read error... cannot continue (text mode).
Cause: Missing or error in CNC7.PLC.

- | | |
|--|--|
| <p>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.</p> <p>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.</p> <p>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.</p> | <p>Cause: ESC or CYCLE CANCEL pressed.
Effect: Job canceled.
Removed: Start of new job.</p> <p>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.</p> <p>309. Prompt: Waiting for CYCLE START button.
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.</p> |
|--|--|

Messages and Prompts in the Operator Status Window

Status messages

- | | |
|---|--|
| <p>301. Message: Stopped.
Cause: No operations in progress.
Effect: None.</p> <p>302. Message: Moving...
Cause: Motors are moving while a CNC program is running.
Effect: None.</p> <p>303. Message: Paused...
Cause: Motion is paused while a CNC program is running (FEED HOLD).
Effect: None.</p> <p>304. Message: MDI...
Cause: CPU7 running in MDI mode.
Effect: None.</p> <p>305. Message: Processing...
Cause: CPU7 running in a mode other than MDI.
Effect: None.</p> <p>306. Message: Job finished.
Cause: Normal end of CNC program.
Effect: None.</p> | <p>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.</p> <p>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.</p> <p>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.</p> |
|---|--|

307. Message: Operator abort: job canceled.

- | | |
|---|---|
| <p>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.</p> <p>314. Message: Input search data.
Cause: Run/search key pressed.
Effect: None.
Removed: After search data input.</p> <p>315. Message: Searching...
Cause: Run/search in progress.
Effect: None.</p> <p>316. Message: Search complete. Processing...
Cause: Run/search mode. Search successful. Preprocessing job.
Effect: None.</p> <p>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.</p> | <p>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.</p> <p>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.</p> <p>406. Message: Emergency Stop detected.
Cause: CPU7 stopped with no fault bits set.
Effect: 'Fault: Job canceled.' prompt.
Removed: when Emergency Stop released.</p> <p>407. Message: X+ limit (#1) tripped.
Cause: CPU7 stopped with limit switch status.
Effect: Job canceled.
Removed: Start of new job, when limit cleared.</p> <p>408. Message: Programmed action timer expired.
Cause: M103 time expired before M104 encountered.
Effect: Job canceled.
Removed: Start of new job.</p> |
|---|---|

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 (RTK2) has returned on line.
Effect: None.

404. Message: Spindle drive fault detected.

409.	<p>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.</p>	<p>motion is stopped, power to the motors is released (all servo drive commands cease) and the CNC program is aborted.</p> <p>The probable causes of this error are:</p> <ol style="list-style-type: none"> 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.
	<p>Effect: Job canceled</p>	<p>Effect: Job canceled</p>
	<p>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.</p>	<p>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.</p>
	<p>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.</p>	<p>Check the motor cabling paying particular attention to the ground connections. Replace the cable if it is damaged or repair the motor connections.</p> <p>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.</p>
	<p>Message: _ axis position error. Cause: A position error > .25 inches is detected on any axis. All axis</p>	<p>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.</p>
	<p>Message: _ axis position error. Cause: A position error > .25 inches is detected on any axis. All axis</p>	<p>Start of new job.</p>

411.	<p>Message: _ axis full power without motion.</p> <p>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.</p> <p> The probable causes of this error are:</p> <p> 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.</p> <p>Effect:</p> <p> 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.</p> <p> 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.</p> <p> 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.</p> <p> Use slow jog to move opposite the direction causing the error and clear all limit switches. Jog toward the direction causing the</p>	<p>error, if no motion occurs then a servo drive failure is indicated.</p> <p>Removed: Start of new job.</p>
412.	<p>Message: _ axis encoder connection is bad.</p> <p>Cause: Axis is enabled but a differential encoder signal is not detected. May indicate a loose or severed encoder cable or a bad encoder.</p> <p>Effect: All axis motion is stopped and the CNC program is aborted.</p> <p>Removed: Reconnect encoder or repair encoder and/or encoder cable.</p>	<p>Removed: Start of new job.</p>
413.	<p>Prompt: CPU Failure #01: power down.</p> <p>Cause: CPU7 has experienced a problem with the PC reset line.</p> <p>Effect: Power down, then power up the system. The error should disappear.</p> <p>Removed: Never - system must be powered down.</p>	<p>Removed: Start of new job.</p>
414.	<p>Prompt: CPU Failure #02: power down.</p> <p>Cause: CPU7 detected CPU failure.</p> <p>Effect: Power down, then power up the system. The error should disappear.</p> <p>Removed: Never - system must be powered down.</p>	<p>Removed: Start of new job.</p>
415.	<p>Prompt: CPU fault #XX detected.</p> <p>Cause: invalid stop reason from CPU7.</p> <p>Effect: Power down, then power up the system. The error should disappear.</p> <p>Removed: Never - system must be powered down.</p>	<p>Removed: Start of new job.</p>
416.	<p>Prompt: Motion fault #XX detected.</p> <p>Cause: Invalid motion status from CPU7.</p> <p>Effect: Power down, then power up the system. The error should disappear.</p> <p>Removed: never - system must be powered down.</p>	<p>Removed: Start of new job.</p>
417.	<p>Prompt: Abnormal end of job.</p> <p>Cause: Job ended without reason.</p> <p>Effect: Job canceled.</p> <p>Removed: Start of new job.</p>	<p>Removed: Start of new job.</p>

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.
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.
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).
Effect: The current job is canceled and power is released.
Removed: Start of new job (after motors have cooled below warning temperature).
422. Message: Jog Panel Offline.
Cause: Jog panel failure or loose cable.
Effect: All buttons on jog panel are inoperative.
Removed: By reconnecting jog panel cable and appearance of next message.
423. Message: Jog Panel Online.
Cause: Loose jog panel cable has been reconnected.
Effect: All buttons on jog panel are operative.
424. Message: Feedrate Override Offline.
Cause: Jog panel failure or loose cable.
- Effect: Feedrate knob and some jog panel keys are inoperative.
Removed: By appearance of next message.
425. Message: Feedrate Override Online.
Cause: Loose jog panel cable has been reconnected.
Effect: Feedrate Override knob is operative.
426. Message: Spindle Override Offline.
Cause: Jog panel failure or loose cable.
Effect: Spindle knob and some jog panel keys are inoperative.
Removed: By appearance of next message.
427. Message: Spindle Override Online.
Cause: Loose jog panel cable has been reconnected.
Effect: Spindle Override knob is operative.
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.
430. Message: CPU7 PIC Offline.
Cause: Power supply or hardware problem.
Effect: Power down, then power up the system. The error should disappear.
431. Message: CPU7 PIC Online.
Cause: CPU7 is back on line.
Effect: None.
432. Message: External PLC Offline.
Cause: Koyo PLC Direct failure or loose cable.
Effect: None.
Removed: When PLC failure removed or cable reconnected.

433. Message: External PLC Online.
Cause: PLC failure corrected.
Effect: None.
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.
Effect: Power to motors is released.
Removed: Start of new job.
435. Message: _ axis runaway: Check motor wiring.
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: Power to motors is released.
Removed: Start of new job (after wiring or servo drive failure has been removed).
436. Message: Servo drive shutdown.
Cause: This error message is produced by hardware detection of a physical error.

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
437. Message: Servo power removed.
Cause: Any of the axes were moving more than 120 in/min while power was supposed to be off.

The DSP originates this error message if it detects motion when it should be stopped. This message is displayed concurrently with message #435.

Effect: Power to motors is released.
Removed: Start of new job (after wiring or servo drive failure has been removed).

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 value on line NNNNN.
Cause: Value out of range (T, H, D).
Effect: Job canceled.
Removed: Start of new job.
505. Message: Invalid parameter on line NNNNN.
Cause: Invalid or missing number after letter.
Effect: Job canceled.
Removed: Start of new job.

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. | <p>Message: Error: No R point on line NNNNN.</p> <p>Cause: No R-value specified for canned cycle.</p> <p>Effect: Job canceled.</p> <p>Removed: Start of new job.</p> | 902. | <p>Message: No prior G28 or G30 on line NNNNN.</p> <p>Cause: G29 with no preceding G28 or G30.</p> <p>Effect: Job canceled.</p> <p>Removed: Start of new job.</p> |
| 802. | <p>Message: Error: Q = 0 on line NNNNN.</p> <p>Cause: Q value of 0 specified (Q used for G73 and G83 only).</p> <p>Effect: Job canceled.</p> <p>Removed: Start of new job.</p> | 903. | <p>Message: Warning: No coordinates for G92 on line NNNNN.</p> <p>Cause: G92 with no axis coordinates to set.</p> <p>Effect: Remainder of line processed; job continues.</p> <p>Removed: When next message appears.</p> |
| 803. | <p>Message: Error: No Z point on line NNNNN.</p> <p>Cause: No Z value specified for canned cycle.</p> <p>Effect: Job canceled.</p> <p>Removed: Start of new job.</p> | 904. | <p>Message: Invalid plane for arc on line NNNNN.</p> <p>Cause: I, J, or K specified with wrong plane (e.g. K with G17, or I with G19).</p> <p>Effect: Job canceled.</p> <p>Removed: Start of new job.</p> |
| 804. | <p>Message: Error: Ggg invalid on line NNNNN (gg = 76, 86, 87, 88).</p> <p>Cause: Unimplemented canned cycle requested.</p> <p>Effect: Job canceled.</p> <p>Removed: Start of new job.</p> | 905. | <p>Message: Warning: 0 radius arc on line NNNNN.</p> <p>Cause: Arc move was specified with zero radius.</p> <p>Effect: Move is done as a linear move; job continues.</p> <p>Removed: When next message appears.</p> |
| 805. | <p>Message: Error: No Q value on line NNNNN.</p> <p>Cause: Q value not specified for G73 or G83.</p> <p>Effect: Job canceled.</p> <p>Removed: Start of new job.</p> | 906. | <p>Message: Warning: unknown arc on line NNNNN.</p> <p>Cause: Position of arc move could not be determined from parameters (e.g. G91 G2 X0 Y0 R1).</p> <p>Effect: Move is done as a linear move; job continues.</p> <p>Removed: When next message appears.</p> |
| 806. | <p>Message: Error: No P value on line NNNNN.</p> <p>Cause: P value (dwell time) not specified for G82 or G89.</p> <p>Effect: Job canceled.</p> <p>Removed: Start of new job.</p> | 907. | <p>Message: _ axis travel exceeded on line NNNNN.</p> <p>Cause: Software travel limit would be exceeded by the requested move.</p> <p>Effect: Job canceled.</p> <p>Removed: Start of new job.</p> |

Miscellaneous errors

- | | |
|------|---|
| 901. | <p>Message: Ref. point invalid on line NNNNN.</p> <p>Cause: G30 with invalid P value (must be 1 or 2).</p> <p>Effect: Job canceled.</p> <p>Removed: Start of new job.</p> |
|------|---|

- | | |
|--|---|
| <p>908. Message: Option not available on line
NNNNN.
Cause: A code for an extra-cost option
was specified, but the option has
not been licensed.
Effect: Job canceled.
Removed: Start of new job.</p> <p>909. Message: Program too long: job canceled.
Cause: Attempt to run a job over 640K
in length, without the unlimited
program size option.
Effect: Job canceled.
Removed: Start of new job.</p> <p>910. Message: No subroutines in MDI.
Cause: Specified O9100 - O9999 in
MDI, which would begin an
embedded subprogram.
Effect: MDI canceled.
Removed: Start of new job.</p> <p>911. Message: Illegal recursion.
Cause: Attempt to execute a
subprogram or macro that calls
itself, either directly or
indirectly.
Effect: Job canceled.
Removed: Start of new job.</p> <p>912. Message: Too many subprogram calls.
Cause: Attempt to run a job with 20 or
more levels of subprogram
nesting.
Effect: Job canceled.
Removed: Start of new job.</p> <p>913. Message: Could not open file <i>filename.ext</i>.
Cause: Attempt to call a subprogram or
macro, but the subprogram file
does not exist.
Effect: Job canceled.
Removed: Start of new job.</p> | <p>1002. Message: Invalid scaling center on line
NNNNN.
Cause: Invalid parameter specified (X,
Y, Z).
Effect: Job canceled.
Removed: Start of new job.</p> <p>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.</p> <p>1004. Message: Turn scaling off before
rescaling.
Cause: Tried to rescale while scaling is
turned on.
Effect: Job canceled.
Removed: Start of new job.</p> <p>1005. Message: Cannot scale arcs with different
scale factors.
Cause: Scaling factors of the arc axes
are different.
Effect: Job canceled.
Removed: Start of new job.</p> |
|--|---|

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.

