CENTROID™

M-SERIES
Operator's Manual

CNC10 Version 2.32
Rev. 070111

U.S. Patent #6490500
© 2004-2006 Centroid Corp. Howard, PA   16841

CNC Control Information Sheet

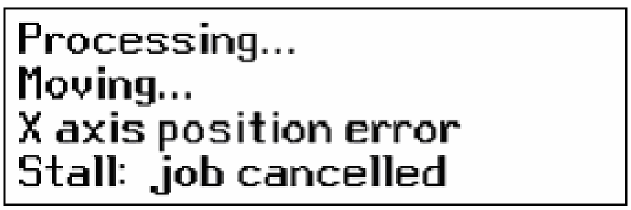
Fill out the following and fax back to Centroid Tech support **814-353-9265.** Date: _________
Company name: _________________________ Your name: ____________________________
Address: ______________________________ City: _________________ State:____ Zip: ________
Phone #: ________________ Fax #: __________________
Control Serial Number K_________ Software Version: _______ Approx. purchase date: __________
Dealer: ____________________________ Machine Brand: __________________________

Machine type (check below)

- ☐ Knee Mill
- ☐ Bed Mill
- ☐ Router
- ☐ Machining Center
- ☐ Lathe
- ☐ 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 CNC10 software (this is the screen that appears when your system is first turned on).
2. Press **F1 - Setup** to enter the Setup screen.
3. Press **F3 - Config** to enter the Configuration screen.
4. Type "137" in the window which asks for the password. Press **ENTER** to accept this.
5. Press **F2 – Mach** for Machine Configuration.
6. Press **F1 – Jog** for Jog Parameters or **F2 – Motor** for Motor Parameters

Jog Parameters

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

Motor Parameters

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

PID parameters: Press **ESC** at the machine configuration screen. Then press **F4 - PID**

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

To obtain the control configuration info, press **ESC** at the PID screen, and then press **F1 - Contrl**

DRO display units:_______
Machine units:__________
Max spindle (high range):__________
Min spindle (high range):__________
Machine home at powerup:________

PLC type:__________
Console type (Jog Panel Type):__________
Jog panel required:____________
Screen blank delay:___________
Remote Drive & Directory: ________________

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

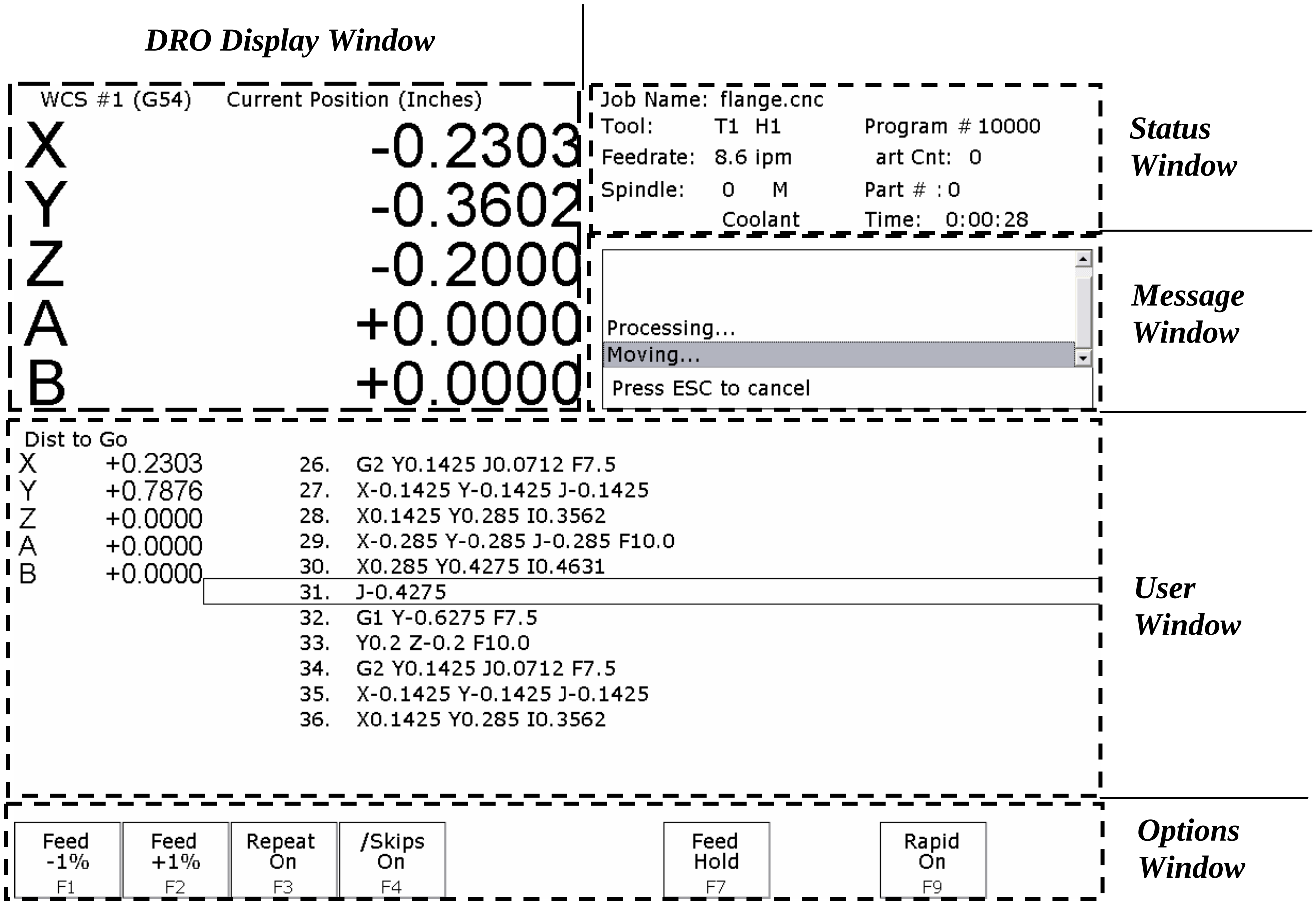
Param #	Value	Param #	Value	Param #	Value	Param #	Value	Param #	Value	Param #	Value
21		23		29		40		56		58	
22		24		30		41		57		59	

TABLE OF CONTENTS

Chapter 1
Introduction

Window Description

The CNC10 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 14). The bars under each axis are the load meters and represent the amount of power being supplied to the drive for that axis. The display of axis load meters is configured by machine parameter 143 – see Chapter 14 for specific information.

Distance to Go DRO

The distance to go DRO is located below the main DRO. This display shows the distance to go to complete the current movement. The display of distance to go is controlled by parameter 143. See Chapter 14 for details. See also “Hot Keys” in chapter 2.

Status window

The first line in the status window contains the name of the currently loaded job file. Below the job name are the Tool Number, Program Number, Feedrate Override, Spindle Speed, and Feed Hold indicators. The Feedrate Override indicator displays the current override percentage set on the Jog Panel. The Feedrate label will turn RED if the rapid override is turn off. If your machine is equipped with a variable frequency spindle drive (inverter), the Spindle indicator will display the current spindle speed. The Feed Hold indicator displays the current status (on/off) of FEED HOLD. See Chapter 2 for descriptions of the Feed Hold Button, Feedrate Override Knob, and Spindle controls. For a description of the Program Number see G65 in Chapter 12 or M98 in Chapter 13.

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

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

Message window

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

The message section is the top four text lines of the message window. This section will display warnings, errors, or status messages. The newest messages always appear on the lowest of the four lines. Old messages are shifted up until they disappear off the top of the message window. When old messages scroll out of view, a scroll bar will appear on the right side of the window. When the scroll bar is visible you may use the up and down arrow keys to view older messages. See Chapter 15 for a description of the CNC10 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 - CAM** 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.

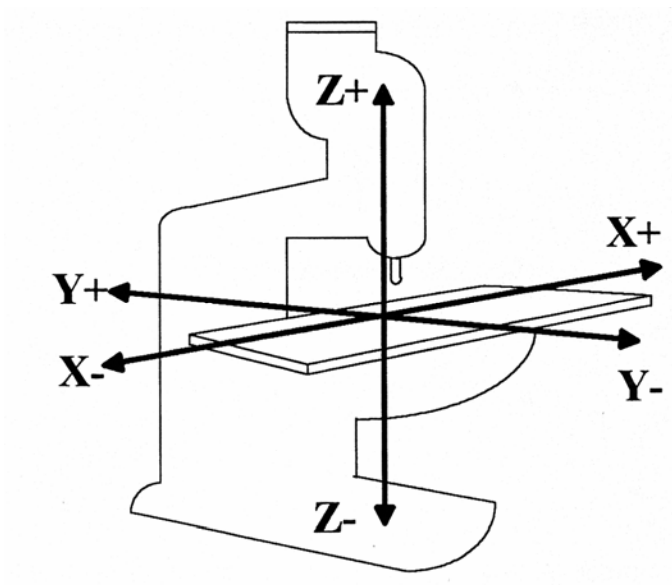
Conventions

*Bold capitalized characters represent keystrokes. For example, the A key is written as **A,** and the enter key is written as **ENTER**. The "Escape" key is written as **ESC**. Key combinations such as **ALT- D** mean that you should press and hold **ALT** then press **D**.

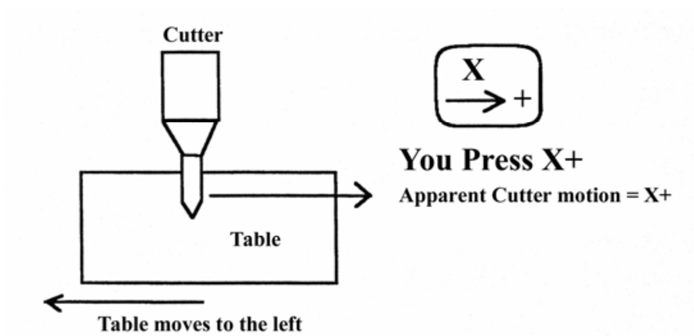
*All data entry screens in the M-Series Control use **F10** to save changes.

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

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



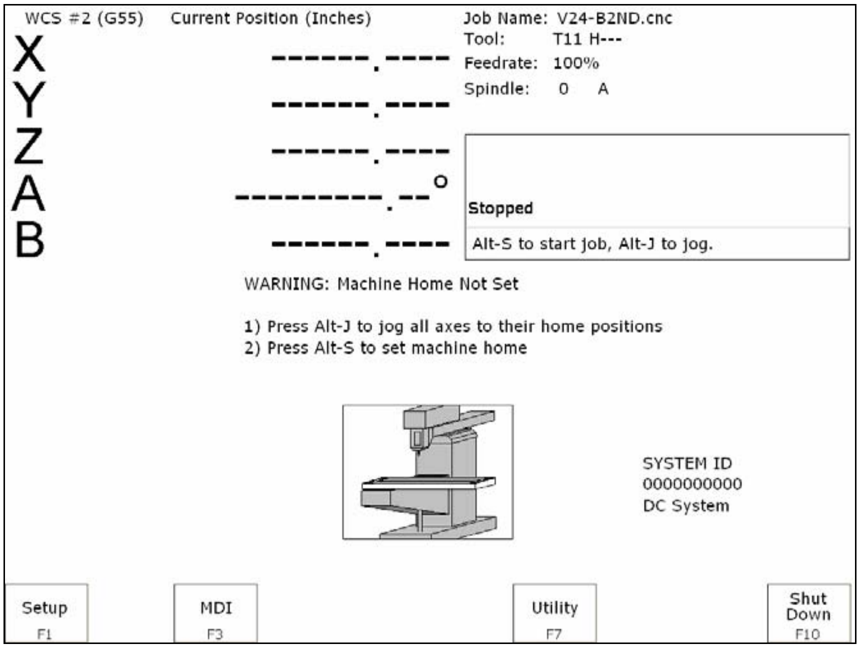
*The direction of motion is defined by the CUTTER motion, not the TABLE motion.



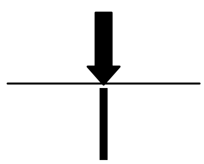
*CW stands for clockwise, and CCW stands for counterclockwise.

Machine Home

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



Before you can run any jobs, you must set the machine home position. If your machine has home/limit switches, reference marks or safe hard stops, the control can automatically home itself. If your machine has reference marks, jog the machine until the reference marks are lined up, (see below), before you press **CYCLE START** to begin the automatic homing sequence. The control will execute the G-codes in a file called cnc10m.hom in the c:\cnc10 directory. By default, this file contains commands to home Z in the plus direction, then X in the minus and Y in the plus direction.



Typical Reference Marks

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

WARNING: Machine Home Not Set

1) Jog all axes to their home positions
2) Press CYCLE START to set machine home

In this case you must move the machine to it's 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.

Mill M and G Codes

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

M00	Stop for operator	G00	Rapid Positioning
M01	Optional Stop for operator	G01	Linear Interpolation
M02	Restart Program	G02	Circular or Helical Interpolation CW
M03	Spindle on CW	G03	Circular or Helical Interpolation CCW
M04	Spindle on CCW	G04	Dwell
M05	Spindle off	G09	Exact Stop
M06	Start Tool Change	G10	Parameter Setting
M07	Mist Coolant on	G17	Circular Interpolation Plane Selection XY
M08	Flood Coolant on	G18	Circular Interpolation Plane Selection ZX
M09	Coolant off	G19	Circular Interpolation Plane Selection YZ
M10	Clamp on	G20	Select Inch Units
M11	Clamp off	G21	Select Metric Units
M14	Swing Arm Pot Up	G28	Return to Reference Point
M15	Unclamp tool, air on	G29	Return from Reference Point
M16	Unclamp tool, air off	G30	Return to Secondary Reference Point
M18	Home tool changer	G40	Cutter Compensation Cancel
M19	Orient spindle	G41	Cutter Compensation Left
M20	Pick up tool	G42	Cutter Compensation Right
M21	Move head up	G43	Tool Length Compensation (+)
M22	Move head to ATC level	G44	Tool Length Compensation (-)
M23	Rotate carousel	G49	Tool Length Compensation Cancel
M24	Start tool put back	G50	Scaling/Mirroring Off (Optional)
M25	Move to Z home	G51	Scaling/Mirroring On (Optional)
M26	Set axis home	G52	Offset Local Coordinate System Origin
M39	Air drill	G53	Rapid Position in Machine Coordinates
M50	Index tool plus	G54	Select Work Coordinate System #1
M51	Index tool minus	G55	Select Work Coordinate System #2
M60	Probing macro	G56	Select Work Coordinate System #3
M80	Carousel in	G57	Select Work Coordinate System #4
M81	Carousel out	G58	Select Work Coordinate System #5
M91	Move to minus home	G59	Select Work Coordinate System #6
M92	Move to plus home	G61	Exact Stop Mode
M93	Release motor power	G64	Cutting Mode
M94	Turn on input X	G65	Call Macro
M95	Turn off input X	G68	Rotate
M98	Call subprogram	G69	Cancel Rotate
M99	Return from subprogram	G73	High Speed Peck Drilling
M100	Wait for input to open	G74	Counter Tapping
M101	Wait for input to close	G80	Canned Cycle Cancel
M102	Restart program	G81	Drilling and Spot Drilling
M103	Programmed action timer	G82	Drill with Dwell
M104	Cancel programmed action timer	G83	Deep Hole Drilling
M105	Move minus to switch	G84	Tapping
M106	Move plus to switch	G85	Boring
M107	Output BCD tool number	G89	Boring with Dwell
M108	Enable override controls	G90	Absolute Positioning Mode
M109	Disable override controls	G91	Incremental positioning Mode
M115	Protected probing move	G92	Set Absolute position
M116	Protected probing move	G98	Initial Point Return
M120	Open data file (overwrite existing file)	G99	R Point Return
M121	Open data file (append to existing file)	G117	Rotation of Plane Selection XY
M122	Record position(s) and/or comment in data field	G118	Rotation of Plane Selection ZX
M123	Record value and/or comment in data field	G119	Rotation of Plane Selection YZ
M125	Protected probing move		
M126	Protected probing move		

How to unlock software features or unlock your Control

The following are necessary to unlock software features:

1. If at the demo mode expired screen, start at step7.
2. Go to the Main screen of the Control software.
   (Setup, Load, MDI … across the bottom of the screen)
3. Press **F1-Setup** to enter the Setup screen.
4. Press **F3-Config** to enter the Configuration screen.
5. Type “**137**” (default password) in then window which asks for the password. Press **Enter** to accept.
6. Press **F3-Params** to the Parameters screen.
7. Press **F1** to bring up the special parameter box and enter the parameter number.
8. Enter the parameter value that is beside the parameter number just entered and press **Enter**.
9. Repeat steps 7 and 8 for each new parameter value.
10. When finished entering all parameters press **F10-Save** to save the parameters entered.

Chapter 2
Operator Panels

M-Series Jog Panel

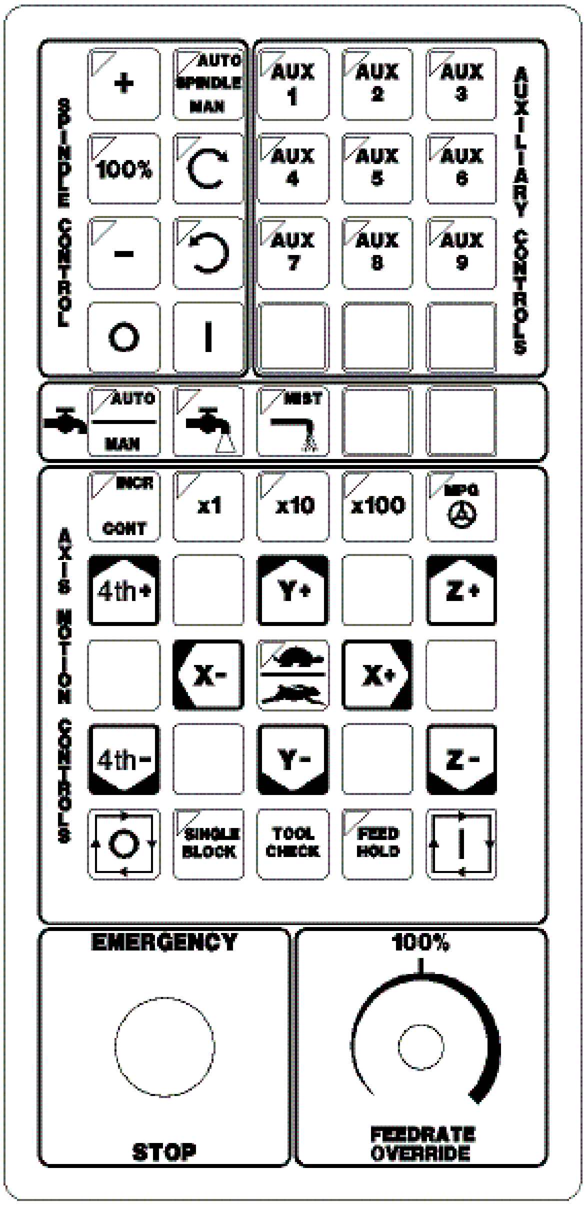


Fig 1 - M-Series Jog Panel

The M-Series operator panel is a sealed membrane keyboard that enables you to control various machine operations and functions. The panel contains momentary membrane switches, which are used in combination with LED indicators to indicate the status of the machine functions.

The M-Series operator panel is a sealed membrane keyboard that enables you to control various machine operations and functions. The panel contains momentary membrane switches. The M-Series jog panel can be customized as to the location of various keys. The jog panel displayed in the figure above is representative of a default configuration found on most M-series controls.

Axis Jog Buttons
X+ X- Y+ Y- Z+ Z- 4TH+ 4TH-

The yellow X, Y, Z, and 4TH keys are momentary switches for jogging each of the four axes of the machine. There are two buttons for each axis (+/-). Only one axis can be jogged at a time.

Slow/Fast

The **slow**/**fast** key is located in the center of the Axis Motion Controls section and is labeled with the turtle and rabbit icon shown to the right. The turtle represents slow jogging mode. When SLOW jog is selected (LED on) and a jog button is pressed, the axis moves at the slow jog rate. If FAST jog is selected, the axis will move at the fast jog rate. See Chapter 14 for information on setting the fast and slow jog rates for each axis.

Inc/Cont
INC/**CONT** selects between incremental and continuous jogging. Pressing the key will toggle between these two modes. The LED is lit when INC is selected. When set to INC jog and a jog button is pressed, the axis will move the current jog increment distance and stop. The jog button must be released and then pressed again before any further axis movement can occur. The LED is not lit when set to CONT. If CONT jog is selected and an axis jog button is pressed, the axis will move continuously until the button is released.

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

x1, x10, x100
Press any one of these keys to set the jog increment amount. The amount you select is the distance the control will move an axis if you make an incremental jog (x1=0.0001", x10=0.0010" and x100=0.0100"). You may select only one jog increment at a time, and the current jog increment is indicated by the key that has a lit LED. The jog increment you select is for all axes; you cannot set separate jog increments for each axis. The jog increment also selects the distance the control will move an axis for each click of the MPG handwheel.

MPG

The MPG is housed in a separate hand-held unit. Press the **MPG** key to set the control jog to respond to the MPG handwheel, if equipped. When selected, the LED will be on. Select the Jog Increment and desired axis and slowly turn the wheel. When the LED is not lit, the MPG is disabled and the jog panel is on.

* WARNING: Do not spin the handwheel too quickly. Damage to the machine or part may result.

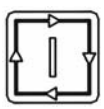
Tool Check

Press **TOOL CHECK** while no program is running to move the Z-axis to its home position/G28 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).

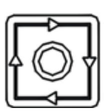
Cycle Start



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.

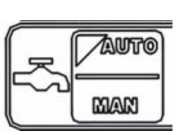
Cycle Cancel



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 3.

Coolant Control Keys

The coolant control keys are located in a single row between the Spindle Control section and Axis Motion Controls section of the jog panel.



Coolant **Auto**/**Manual** selection. This key will toggle between automatic and manual control of coolant. In automatic mode, M7 (Mist) and M8 (Flood) can be used in G-code programs to select the coolant type to be enabled. In manual mode, flood coolant and mist coolant are controlled by separate keys

When switching from automatic to manual mode, both flood and mist coolant are turned off automatically.

Coolant Flood

In manual coolant control mode, flood coolant can be toggled off and on by pressing this key. The LED will be on when flood control is selected in either automatic or manual mode.

Coolant Mist

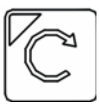
In manual coolant control mode, mist coolant can be toggled off and on by pressing this key. The LED will be on when mist control is selected in either automatic or manual mode.

Auxiliary Function Keys (AUX1 – AUX12)

The M-Series jog panel has nine auxiliary keys, some of which may be defined by customized systems.

Spindle Controls

Spindle (CW/CCW)




The **SPINDLE CLOCKWISE**/**COUNTERCLOCKWISE** keys determine the direction the spindle will turn if it is started manually. If the spindle is started automatically, the direction keys are ignored and the spindle runs according to the program. The default direction is CW.

Spindle Override Controls

Speed increase. Pressing this key will increase the spindle speed by 10% of the commanded speed in Auto spindle mode, limited by the maximum speed or 200% of commanded speed, whichever is less. For manual spindle mode, the spindle speed is increased by 5% of the maximum spindle speed (up to the maximum speed). The LED is on if the spindle speed is set above the 100% point.

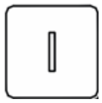
Pressing this key will set the spindle speed at the 100% point, which is defined as the commanded speed in Auto spindle mode, or ½ the maximum spindle speed in manual mode. The LED will be on when the spindle is at the 100% point.

Speed decrease. Pressing this key will decrease the spindle speed by 10% of the commanded speed in Auto spindle mode, limited to 10% of commanded speed. For manual spindle mode, the spindle speed is decreased by 5% of the maximum spindle speed down to 5% of maximum. The LED is on if the spindle speed is set below the 100% point.

Spindle (Auto/Man)

This key selects whether the spindle will operate under program control (automatic) or under operator control (manual). When the LED is lit, the spindle is under automatic control. If the LED is off, the spindle is under manual control. Pressing the **SPINDLE** (**AUTO**/**MAN**) key will toggle it from AUTO to MAN and back again. The default is AUTO mode.

Spin Start



Press the **SPIN START** key when manual spindle mode is selected to cause the spindle to start rotating. Press **SPIN START** when automatic mode is selected to restart the spindle if it has been paused with **SPIN STOP**.

Spin Stop

Press the **SPIN STOP** key when manual spindle mode is selected to stop the spindle. Press **SPIN STOP** when automatic mode is selected to pause spindle rotation and can be restarted with **SPIN START**.

*** WARNING: SPIN STOP should only be pressed during FEED HOLD or when a program is** NOT **running.**

Feedrate Override

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

Feed Hold

Feed Hold decelerates motion of the current movement to a stop, pausing the job that is currently running. Pressing **CYCLE START** will continue the movement from the stopped location.

Emergency Stop

EMERGENCY STOP releases the power to all the axes and cancels the current job immediately upon being pressed. **EMERGENCY STOP** also resets certain faults if the fault condition has been fixed or cleared.

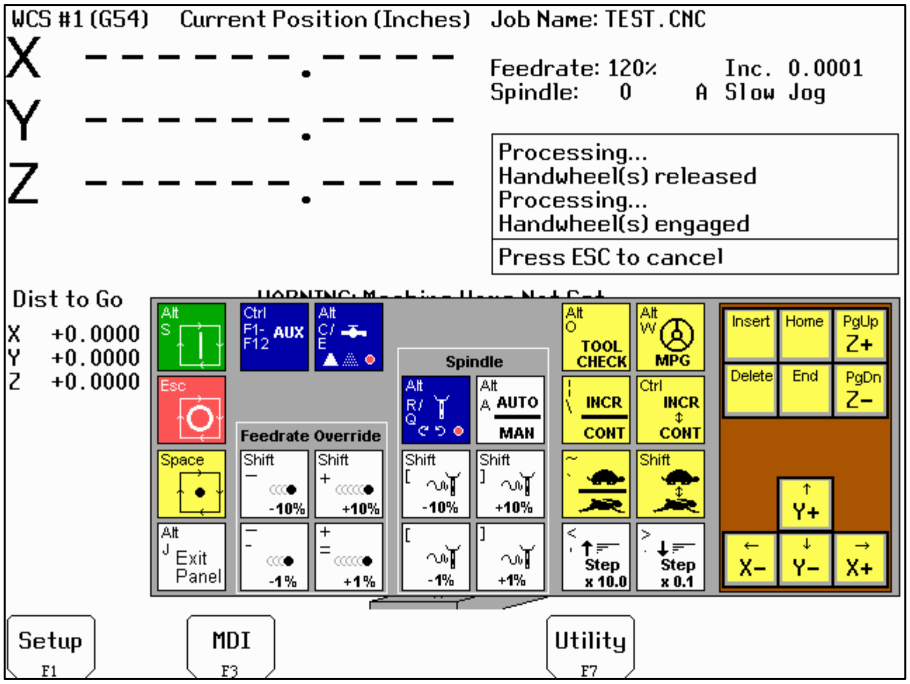
*** WARNING: On some machines, the Z or W axis will sometimes fall due to the lack of power.**

Notes about operator panels

The behavior of the control system in response to the functions listed above for the M-Series jog panel is dependant upon optional software options, the PLC program, machine parameters, and hardware wiring of the system. It is possible that the functioning explained in this chapter does not apply to a particular control system or that it may differ in some aspects.

Keyboard Jog Panel

The keyboard may be used as a jog panel. Press **Alt-J** to display and enable the keyboard jog panel. The jog panel appears as shown below:



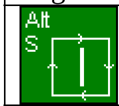
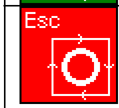
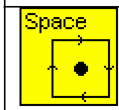
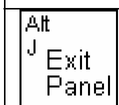
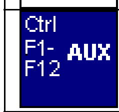
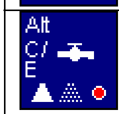
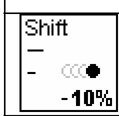
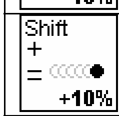
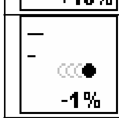
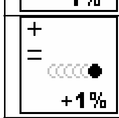
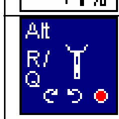
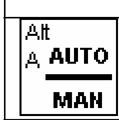
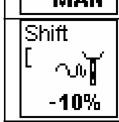
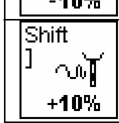
For full functionality of the keyboard jog panel, “Keyboard” must be selected as the console type in the Console Configuration menu.

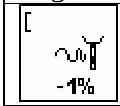
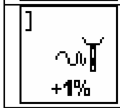
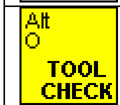
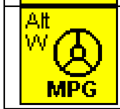
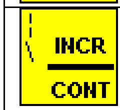
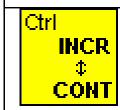
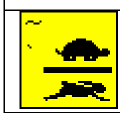
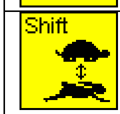
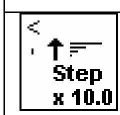
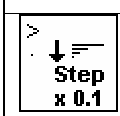
The jog panel shows the mapping of keys to jogging functions. Normally, the keyboard performs menu navigation and data entry functions. The keyboard can jog the axes only when the keyboard jog panel is displayed. Ctrl and Alt functions are available, for the most part, even when the jog panel is not shown.

The status window in the upper right corner of the screen displays the jogging mode (continuous/incremental), incremental step size, and jog speed (fast/slow). In continuous mode, the jog keys start movement when pressed and movement stops when you release the key. In incremental mode, the axis will move the indicated incremental step amount.

As shown in the picture above, the jog keys are located in the cursor key block to the right of the main keyboard and to the left of the numeric keypad. If a jog key controls an axis, it will be overlaid with the axis symbol (“X”, “Y”, etc.) The jog keys are the arrow keys, **Insert**, **Delete**, **Home**, **End**, **Page Up**, and **Page Down**.

The remaining keys are described below:

Legend	Key(s)	Function	Description	Availability (Notes)
	Alt S	Cycle Start	Same as Cycle Start.	Always, with few exceptions. (1)
	Esc	Cycle Cancel	Same as Cycle Cancel.	During a job run; otherwise, Esc is used to exit CNC10 menus.
	Space or Alt H	Feed Hold	Turns Feed Hold on and off	The space key may be used for editing and may not be available at all times. Alt-H is always available.
	Alt J	Start/Exit Panel	Invokes or exits the jog panel.	Always, with few exceptions. (1)
	Ctrl F1 - Ctrl F12	Aux 1 – Aux 12	Executes the corresponding Aux function and signals the PLC. A custom PLC program is required to act upon jog panel signals.	Always, with few exceptions. (1,3)
	Alt C and Alt Q	Flood Coolant and Mist Coolant	Alt C turns flood coolant on and off. Alt E turns mist coolant on and off. Both flood and mist may be on at the same time. Either key automatically selects manual coolant mode. If requested by CNC10, Alt C and Alt E will select "Auto Coolant Mode". Press either when prompted.	Always, with few exceptions. (1,3)
	Shift – or Shift _	Feed Rate Override –10%	Decreases the feed rate override by 10%.	Jog panel, job run, graphing, and some other times. (2,4)
	Shift = or Shift+	Feed Rate Override +10%	Increases the feed rate override by 10%.	Jog panel, job run, graphing, and some other times. (2,4)
	-	Feed Rate Override –1%	Decreases the feed rate override by 1%.	Jog panel, job run, graphing, and some other times. (2,4)
	=	Feed Rate Override +1%	Increases the feed rate override by 1%.	Jog panel, job run, graphing, and some other times. (2,4)
	Alt R and Alt Q	Spindle On/Off CW/CCW	Alt R turns the spindle on clockwise if the spindle is off; otherwise, it turns the spindle off. Alt Q is similar except counter-clockwise. Either will automatically select manual spindle operation.	Always, with few exceptions. (1,3)
	Alt A	Spindle Auto/ Manual	Toggles between automatic and manual spindle operation.	Always, with few exceptions. (1,3)
	Shift [or Shift {	Spindle Override –10%	Decreases the spindle override by 10%.	Only in jog panel, and during a job. (2,4)
	Shift] or Shift }	Spindle Override +10%	Increases the spindle override by 10%.	Only in jog panel, and during a job. (2, 4)

Legend	Key(s)	Function	Description	Availability (Notes)
	[	Spindle Override –1%	Decreases the spindle override by 1%.	Only in jog panel, and during a job. (2,4)
	]	Spindle Override +10%	Increases the spindle override by 1%.	Only in jog panel, and during a job. (2,4)
	Alt O	Tool Check	Performs a tool check.	Always, with few exceptions. (1)
	Alt W	MPG on/off	Turns MPG (handwheel) control on and off.	Available most times that jogging is available.
	| or \	Incremental/ Continuous Jog Selection	Selects incremental or continuous jog mode. Press again to select the opposite mode.	Only in jog panel.
	**Ctrl** (as modifier)	Incremental/ Continuous Jog	Fast and temporary incremental/continuous mode switch. Hold down simultaneously with a jog key. (This is like holding down the Shift key to type a capital letter instead of pressing Caps Lock.)	Only in jog panel.
	‘ or ~	Fast/ Slow Jog Selection	Selects fast or slow jog mode. Press again to select the opposite mode.	Only in jog panel.
	**Shift** (as modifier)	Fast/Slow Jog Selection	Fast and temporary fast/slow mode switch. Hold down simultaneously with a jog key. (This is like holding down the Shift key to type a capital letter instead of pressing Caps Lock.)	Only in jog panel.
	, or <	Increase Jog Step 10x	Changes incremental jog step from .0001 to .001 to .01, etc. (The “1” moves to the left in the status window.) This also selects handwheel speed.	Only in jog panel.
	. or >	Decrease Jog Step 10x	Changes incremental jog step from .1 to .01 to .001, etc. (The “1” moves to the right in the status window.) This also selects handwheel speed.	Only in jog panel.
	F1 – F10	F key pass-thru	Exits the jog panel and executes the corresponding F key.	Where F keys are visible.

Notes:
Hot key. In general, the key can be used at any time. Some CNC10 menus may prevent the use of certain keys.
The console type in the console configuration menu must be set to “Keyboard” to use this key.
The PLC program must be programmed to support this key. Keyboard only systems have this support built-in. Systems with other jog panels may not have this support.
Available if not in use by CNC10. For example, feed rate override can be adjusted from the main menu. If you are editing a value in a table or menu, you cannot adjust feed rate override.

MDI and the Keyboard Jog Panel

Many of the keys used by the keyboard jog panel are also possible commands to MDI. To use the keyboard jog panel functions in MDI, you must press **Alt J**. You may jog, use the handwheels, or any other jog panel function. Press **Alt J** or **Esc** to return to MDI.

Keyboard Shortcut Keys

A computer style keyboard is supplied with most systems. This keyboard can be used as a jog panel. The keyboard jog panel has many "hot keys". Hot keys are keys that can be used at almost any time, with few exceptions. (Some menus may prohibit their use.) CNC10 has many other hot keys in addition to the jog panel hot keys. The hot keys are listed below.

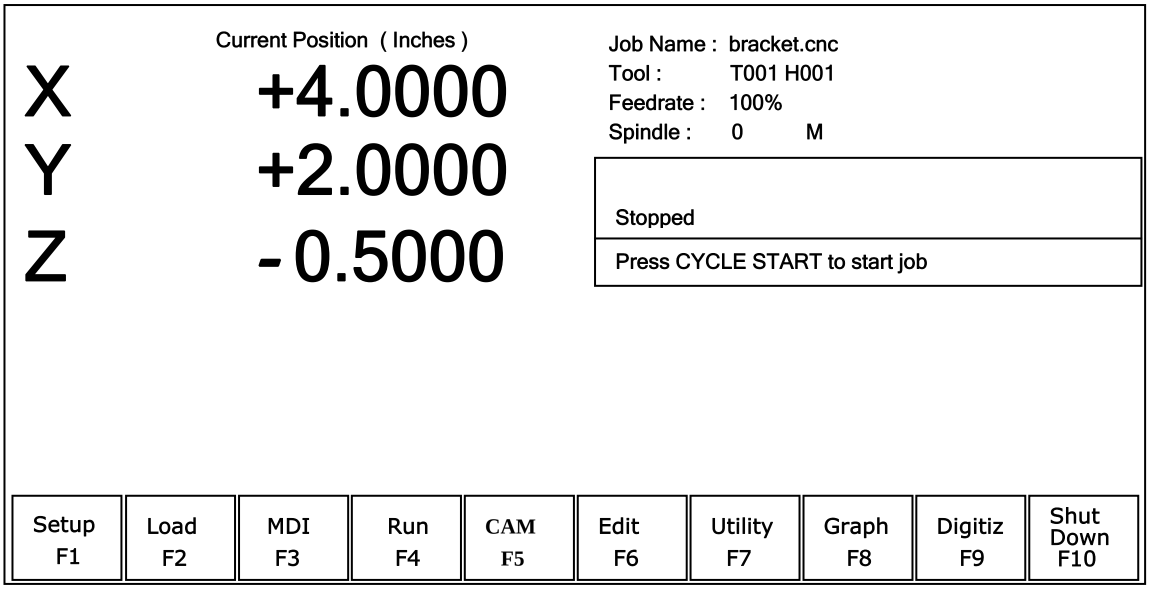
Hot Keys

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

Notes:

* This is a keyboard jog panel function.
** Not available during jobs, in jog panel or while handwheels are engaged.
*** plcstate.txt file is saved in the cnc10 directory. It will be attached to the report when created.

Chapter 3
CNC10 Main Screen



Menu Options:

F1 – Setup: Use the Setup menu to set part zeroes, set or change tool offsets and change the control configuration.
F2 – Load: Use this menu to load a job
F3 – MDI: The MDI menu allows you to a single line command such as: G1 X2 Y3 F20
F4 – Run: Use the Run menu to search and run a job from a specific line, resume a job after it has been canceled or to change the way a job runs.

F5 – CAM : Use the CAM menu to program parts.

F6 – Edit: Brings up a G-code (text) editor that allows you to edit the currently loaded job.

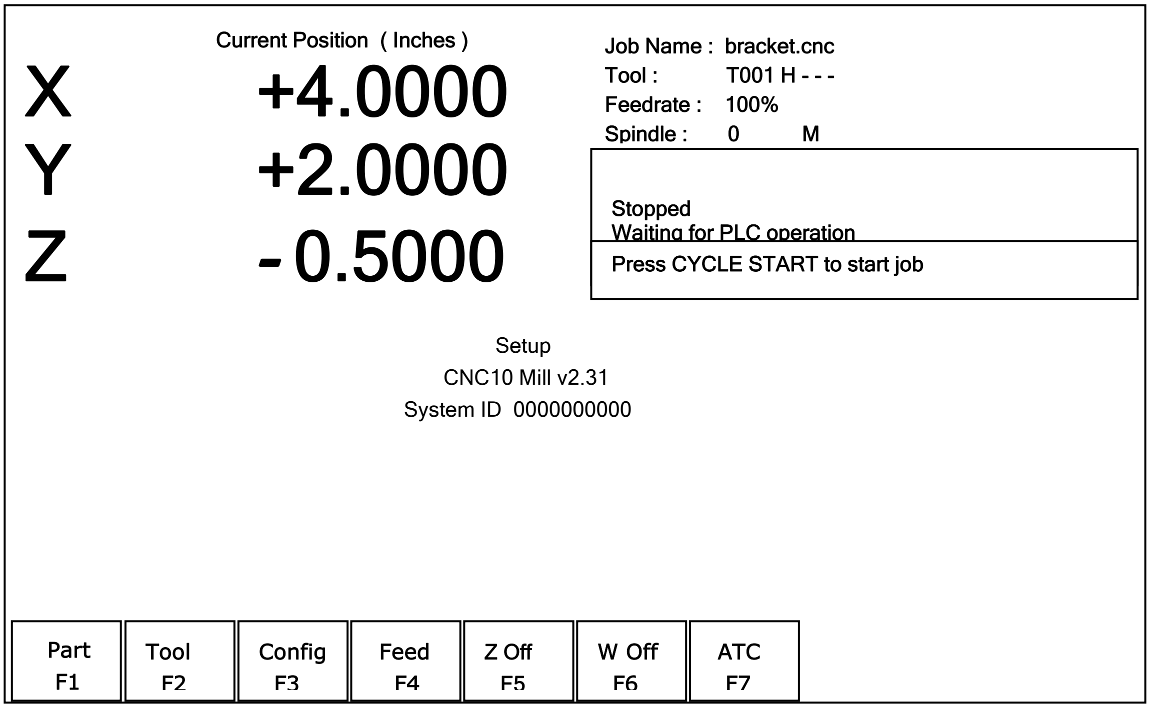
F7 – Utility: From the utility menu you can view available software options, backup part and configuration files, create new directories and import or export files to and from external locations.

F8 – Graph: Graphs the toolpath of the currently loaded part program.

F9 – Digitize: Displayed only if Digitizing option has been purchased. Used to Digitize (reverse engineer) parts.

F10 – Shut down: Power off control. Shutting down your machine without using this menu may damage your control.

F1- Setup Menu



F1 – Part This key displays the Part Setup menus that are explained in Chapter 4.

F2 – Tool This key displays the Tool Setup menus that are explained in Chapter 5.

F3 – Cfg This key displays the Configuration menu that is explained in Chapter 14.

F4 – Feed This key displays the Feed menu that is discussed in Chapter 6.

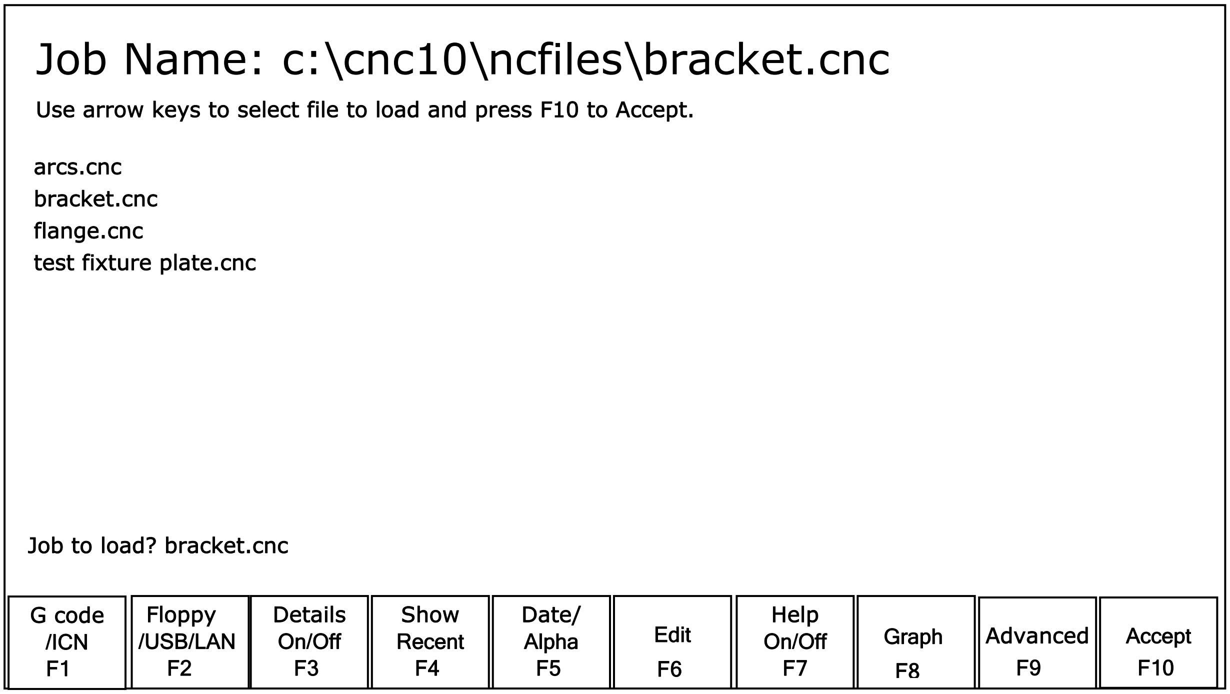
F5 – 3rd Axis Toggle This key will only be displayed if Machine parameter 130 is set. See Chapter 14 for configuration options.

F6 – 4th Axis Toggle This key will only be displayed if Machine parameter 131 is set. See Chapter 14 for configuration options.

F7 – ATC

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

F2- Load Job Menu



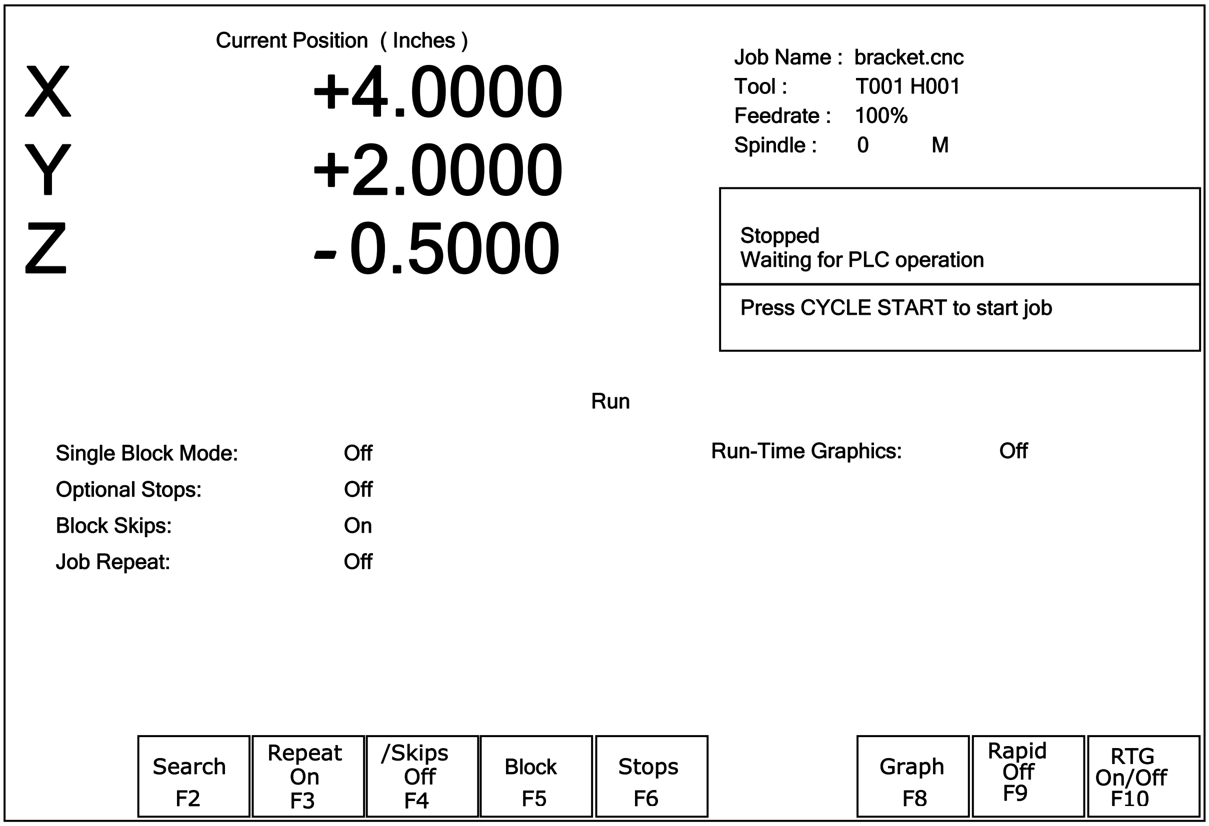
F1 – G code /ICN	Allows the user to change which types of files are displayed.
F2 – Floppy /USB/LAN	Select a different drive from which to load files.
F3 – Details	Displays file details including: Programmer, Description and Date Modified.
F4 – Show Recent	Displays a list of the 15 most recently loaded jobs.
F5 – Date/Alpha	Toggles the current view of files to be sorted alphabetically or by date modified.
F6 – Edit	Opens selected file in editor.
F7 - Help	Displays on screen help for the load screen.
F8 - Graph	Backplots (graphs) the selected file.
F9 - Advanced	Displays a unified file and device browser similar to Windows Explorer.
Page Up	Move the cursor backward one page.
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.

*Note: The path and/or file name may also be selected by typing the path or path and file name. A window will open automatically when you begin typing.

F3 –MDI - MDI mode allows you to directly enter M and G-codes one line at time. After entering the M and G-codes you wish to run, press cycle start to have the controller execute the command. When the command has finished executing the command, it will 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? M92 /Z ; Move the Z to the positive limit.
Block? M26 /Z ; Set the current Z position as Z home

F4 – Run Menu



Run job options:

F2 - Search Resume job by searching for line, tool or block number.
F3 - Repeat Toggles Job Repeat. Repeats the current program when job is finished.
F4 - Block Skips Turns on and off block skips in part programs.
F5 - Single Block Program runs in single block mode when turned on.
F6 - Stops Turns on and off optional stops (M01) in part programs.
F8 - Graph Graphs toolpath of currently loaded program
F9 - Rapid Turns on and off rapid override function
F10-RTG Turns on and off Run Time Graphics

For more information on these options, please see chapter 6.

F5 – CAM

Choose F5 CAM to program parts. The default part programming system is Intercon. Intercon is a conversational programming system that allows you to quickly and easily create a part program. Intercon features many easy to use canned cycles for most common machining operations such as: rectangular, circular and irregular pockets, pockets with islands, bolt hole circles, frames, thread milling operations and much more. When you are finished programming your part in Intercon, exit to return to the M-Series Control Main Screen and the posted Intercon program will be automatically loaded into CNC10 and ready to run. For more information on Intercon, see chapter 9.

F6 – Edit

Loads the current job into a text editor for editing. Some of the commands available in the editor are:

Alt-f = Opens the File Menu	**Ctrl-o = Open file**
Alt-e = Opens the Edit Menu	**Ctrl-n = New file**
Alt-s = Opens the Search Menu	**Ctrl-s = Save file**
Alt-p = Opens the Preferences Menu	**Ctrl-q = Quit**
Alt-c = Opens the Macro Menu	
Alt-w = Opens the Window Menu	

Shift-Ctrl-f = Find
Shift-Ctrl-g= Find next
Shift-Ctrl-r = Replace
Shift-Ctrl-l = Goto line number **Note: Alt key combos work only when Num Lock is OFF.**

When you exit the text editor, you will return to the CNC10 Main Screen. Attempting to edit files that contain non-printable characters may cause unexpected results. DO NOT edit the CNC10 files *cnc10m.cfg, cnc10m.prm, cnc10m.job, cnc10m.tl, cnc10m.ol,* and *cnc10m.wcs*. These files will be destroyed and all information lost if they are edited.

F7 – Utility From the utility menu you can view available software options, perform diagnostics, backup part and configuration files, create new directories and import or export files to and from external locations. **For further information please see chapter 6.**

F8 – Graph This option plots the tool path of the current program loaded. Canned drilling cycles are shown in gray. Rapid traverse movements are shown in red. Feedrate movements are shown in yellow.

F1 - 2D/3D - Press this key to view your part isometrically (3D). An axis pointer indicates the current direction of the view. To return back to the tri-planar view, press F1 again.

F2 – View/Rotate – Press this key to change the planar view of your part. The view is indicated by TOP, RIGHT, or FRONT shown at the top of the screen. In 3D Mode, use this key in 3D mode to rotate your part. using the keyboard arrow keys to rotate any in direction.

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

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

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

F6 - Pan - 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 - Zoom In- Press these keys to zoom into the part relative to the center of the screen.

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

F9 - Zoom All- Press this key to view the entire part fit inside the screen.

Use 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 - Digitize to bring up the Digitize screen. This screen allows you to set up and run touch probe digitizing. **See Chapter 8 for a detailed description of the digitizing operation.**

F10 - Shutdown

Press F10 – Shut Down to enter the Shutdown menu. This menu allows you to park the machine, poweroff the control, start a command window or exit CNC10.

Shutdown Menu

F1 Park

Press F10 - Park to park the machine at the end of the day for quicker machine homing at startup. Once F10 - Park is selected, The CYCLE START key must be press to start machine movement. The park feature homes each axis, at the maximum rate, to ¼ motor revolution from its home position.

F2 Poweroff

Press F2 – Power Off to properly shutdown the control. With most controls, this action turns off the control once the system has prepared itself to be shutdown. Just like a desktop computer, the control should be properly shutdown before turning off the power in order to reduce the risk of corrupting data on the hard drive.

NOTE: This option will only turn off the control. The machine itself will still need to be manually turned off.

F6 System Prompt

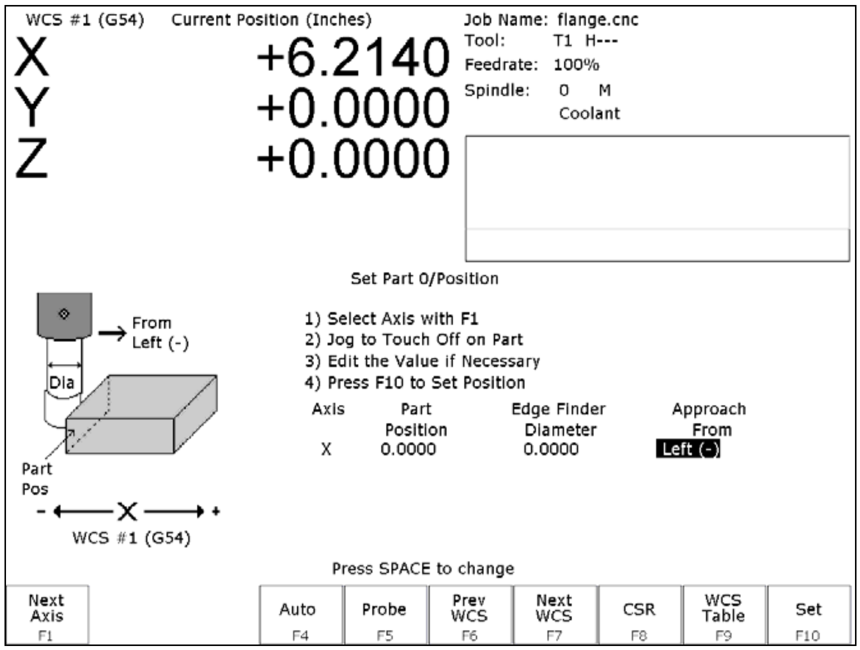
Press F6 – System Prompt to start a command window. From this window you can type CNC Linux commands at a prompt. Type the command exit to exit the command window.

F9 Exit CNC10

Press F9 – Exit CNC10 to exit CNC10 software. Exiting CNC10 starts the CNC10 start menu. From this menu, you can restart CNC10 by pressing F1 – CNC10.

Chapter 4
Part Setup
(F1 from Setup)

General



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

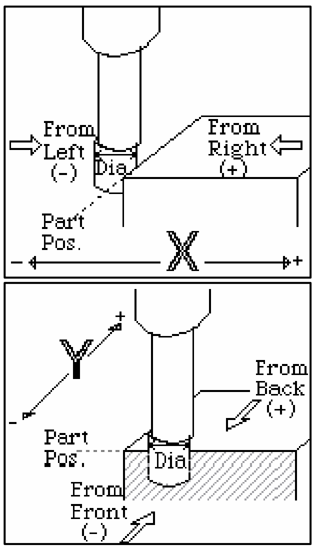
F1 – Next Axis: will toggle to the next axis. If changes were made to the current axis, but not yet accepted, they will be discarded.
F4 – Auto: Uses probe to automatically measure and set part position. Make sure your probe height and diameter offsets are set for the tool number you assigned to the probe and that parameter 12 is set to that tool number. See Chapter 8 for more details.
F5 – Probe: will open the probing operations menu. See Chapter 8 for details.
F6 – Prev WCS: will select the previous work coordinate. The position being set will only affect the currently selected work coordinate.
F7 – Next WCS: will select the next work coordinate. The position being set will only affect the currently selected work coordinate.
F8 - CSR: will open the CSR menu, which can be used to automatically detect coordinate system rotation. This function key appears only when the software option for Coordinate System Rotation is unlocked.
F9 – WCS Table: 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 – Set: 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.

Operation Description

Setting the part position 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,4th, and 5th Axes. 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 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 (-)

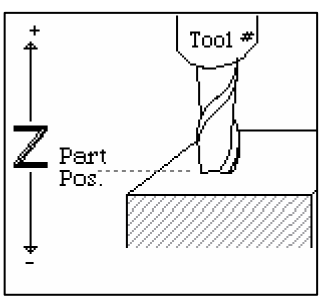
Part Position: enter the value of your part zero position or the offset.

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

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

*Note – Use the arrow keys to toggle between Part Position, Edge Finder Diameter, and Approach From options.

Setting up the Z AXIS



Set Part 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 value of your part zero position or the offset.

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.

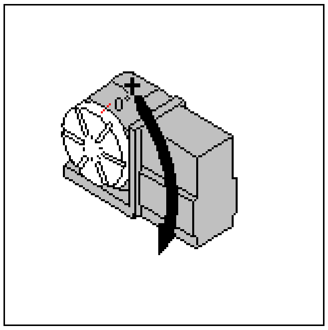
Example 1 (You are using the reference tool to find the Z-axis part zero):
Set Tool Number to 0: setting the Tool Number to zero tells the controller that you are using the reference tool.

Example 2 (You are using a tool other than the reference tool, and not a ball nose cutter):
Set Tool Number to a number tool that is assigned in the tool library (make sure its height offset is set).

Example 3 (You are using a ball nose cutter, other than the reference tool):
Set Part Position to the position of the surface plus the nose radius of the ball nose cutter, set Tool Number to the number this tool is assigned in the tool library.

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

Setting up the 4th or 5th AXIS***



Set Part 0/Position

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

Axis	Part Position	Standoff Distance	Approach from
B	0.0000	0.0000	+

Position: enter the value of your part zero position or the offset.

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.

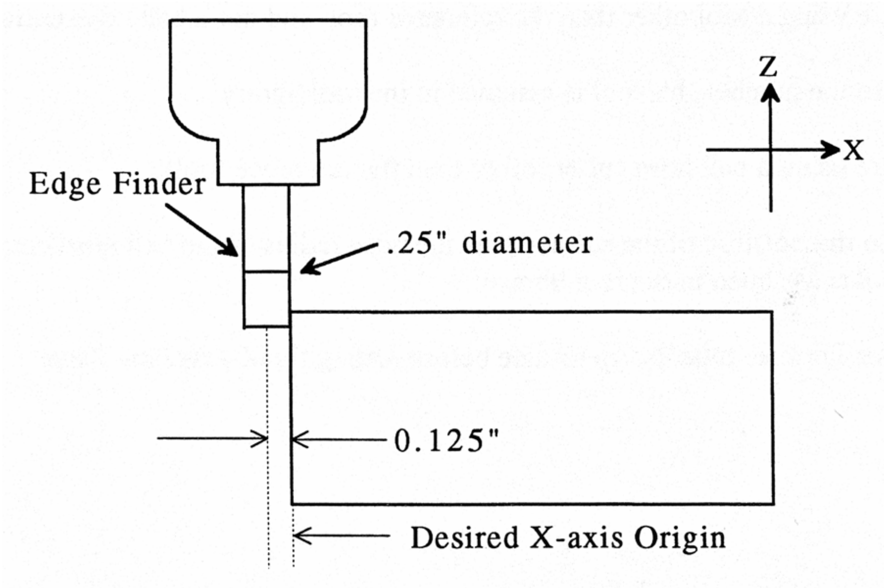
Using Multiple Work Coordinate Systems

If you will be using multiple work coordinates, you must set the part position separately for each work coordinate. Follow the instructions above to set the position for each axis in the first coordinate system. Then move to the next fixture and press **F6 – Prev WCS** to select the previous work coordinate or **F7 – Next WCS** to select the next work coordinate. 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 work coordinate, see the Work Coordinate System Configuration section later in this chapter.

WARNING This procedure does NOT apply to tilt table setup.

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 – Next Axis** 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 - Set** to accept the values

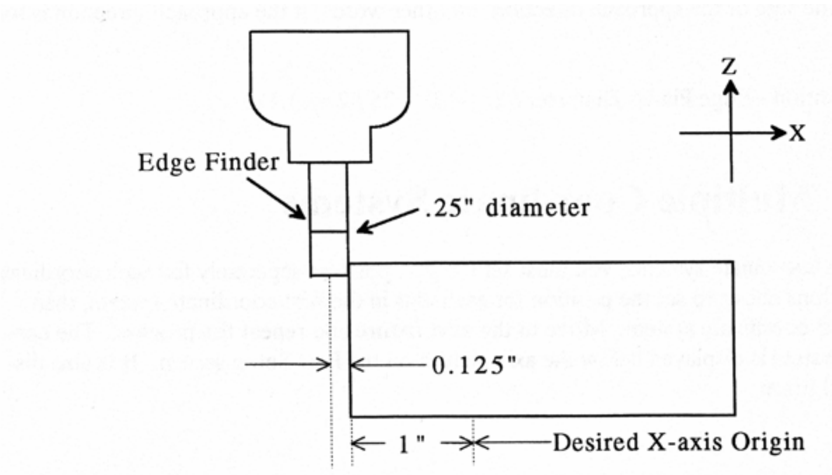
Axis	Part Position	Edge Finder Diameter	Approach From
X	0	0.25	Left (-)

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 - Set** 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 – Next Axis** until the axis field displays 'X'
3. Move the cursor to the Part Position field
4. Type -1 and press **ENTER**
5. Type .25 and press **ENTER**
6. Press **SPACE** until Left (-) is displayed
7. Press **F10 - Set** to accept the value

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

The Position value is relative to the current position of the Edge Finder. Part 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 Part 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 - Set** 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:

Position - Edge Finder Diameter / 2 = -1.0 - .25 / 2 = -1.125

Work Coordinate Systems (WCS) Configuration

Press **F9 – WCS Table** from the Part Setup screen to display the Work Coordinates System (WCS) menu. The Work Coordinate Systems screen provides access to reference return points, coordinate system origins, and work envelope. 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 - Reference Return Points

The **F1 - Return** key is used to set the reference return points for the machine.

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

F2 - Origin

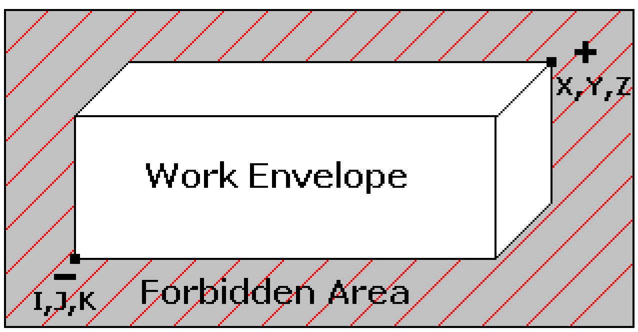
Use the **F2 - Origin** key to specify the locations (in machine coordinates) of the origins for all 18 work coordinate systems. Pressing **F1 – Next Table** will allow you to view the other WCS (6 per page). This option is a convenience and is not an absolute necessity for setting work coordinate system origins.

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

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

F3 – Work Envelope

Use the **F3 – Work Envel** key to specify the '+' and '-' work envelope locations (in machine coordinates) used in conjunction with the G22 G code. The X, Y, Z and I, J, K parameters specified in the G22 G code are stored here, so subsequent G22 codes do not need to specify the limits unless they change.

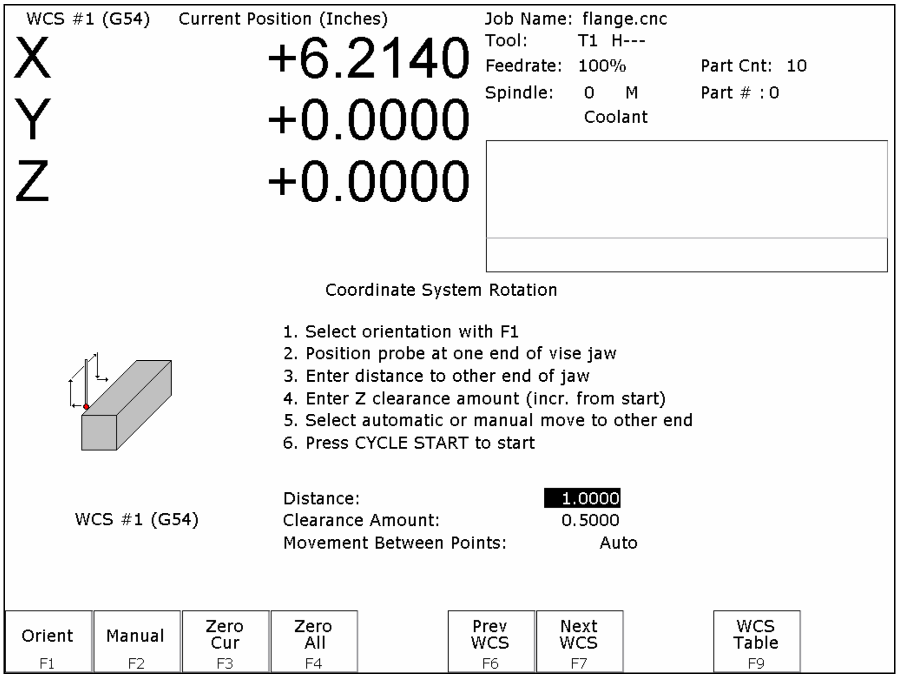


Note: The work envelope will only work in programmed moves. You will still be able to jog outside the work envelope.

Coordinate System Rotation (CSR)

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

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



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

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

F3 - Zero Cur is used to set the CSR angle for the current WCS to zero.

F4 - Zero All is used to set all CSR angles to zero.

F6 - Prev WCS and **F7 - Next WCS** are used to cycle through the available WCS systems.

F9 - WCS Table is a shortcut to the Work Coordinate System Configuration Screen described above.

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

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

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

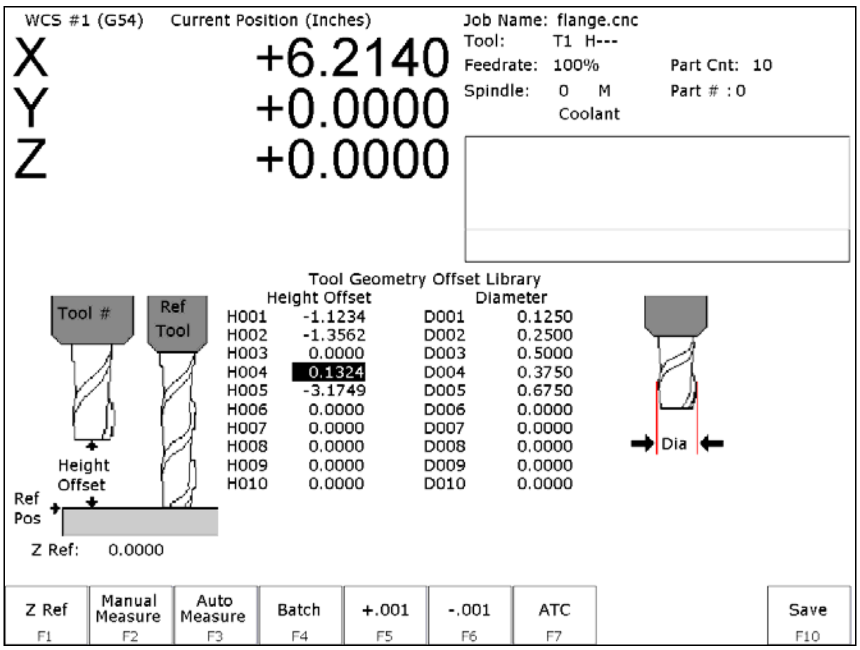
Movement Between Points can be toggled between Jog and Auto modes. In Auto mode, the clearing moves are made automatically as well as the move to the second point. In Jog mode, a prompt will be displayed in the center of the screen after the first point is probed.

Chapter 5
Tool Setup
(F2 from Setup)

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



Press **F1 – Z Ref** to set the Z reference height. Press **F2 - Manual** to manually measure tools. If you purchased the Automatic Tool Measurement (TT1) option, press **F3 - Auto** to automatically measure tool lengths. Press **F5 - +.001** or **F6 - -.001** to adjust the selected offset. If you have an automatic tool changer installed, press **F7 - ATC** to change tools. Press **F10 - Save** to save changes and exit, or **ESC** to exit without saving changes. If you have both purchased the Automatic Tool Measurement (TT1) option and also have an automatic tool changer installed, then you can press **F4 - Auto** 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 CNC10 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:

<u>Establishing the Z reference position</u>
Press **F1 – Z Ref** 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 - Save** to save this Z Position as the Reference Position.

<u>Measuring each tool height (Z position for tool minus Z position for Reference tool)</u>
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 –Manual Measure** to measure the height.
If the tool is a ball nose or bull nose cutter, press **F2 – Manual Measure** 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 - Save** to save the Offset Library and Exit.

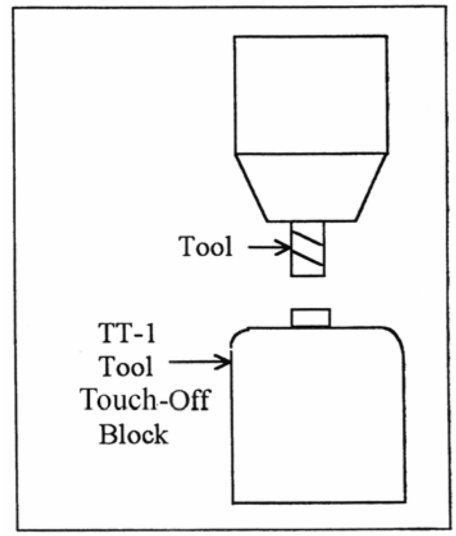
Examples (assuming Z Reference = -1.5):
If the tool position is -1.75, then the tool height = -0.25
If the tool position is -1.75 and nose radius is .25, then the tool height = -0.50
If the tool position is -2.25, then the tool height = -0.75
If the tool position is -2.75 and nose radius is .125, then the tool height = -1.375

<u>Diameter</u>
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, CNC10 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 - +.001** and **F6 - -.001**. Use the arrow keys to highlight the value to be adjusted. Press **F5 - +.001** to increase the offset value by 0.001" (or 0.02 mm in Metric mode). Press **F6 - -.001** to decrease the offset by the same amount. If the cut parts are undersized, use **F5 - +.001** to cut less material. If the cut parts are oversized, use **F6 - -.001** 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 9, 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- Z Ref**, 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. Parameter 3 bit 1 is used to set Z reference to Z home position. See the parameter section in Chapter 14 for more info.

Setting the Tool Height Offsets:

Pressing **F3 – Auto Measure** 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 – Auto Measure** 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 4). 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 – Auto Measure**.

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

Batch Tool Height Offset Measurement Process:

If you have both purchased the Automatic Tool Measurement (TT1) option and also have an automatic tool changer installed, then you can press **F4 - Batch** to measure multiple tools in one process. After pressing **F4 - Batch**, you will be prompted with the following dialogue box:

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

After entering a list of tool numbers, you can press **CYCLE START** to perform the batch tool measurement process. This process is similar to the single tool height offset measurement (accessed via **F3 – Auto Measure**) 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 turned down (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 – Z Ref**.
3. Press **F3** and then **CYCLE START**; the probe will find the Z Reference.

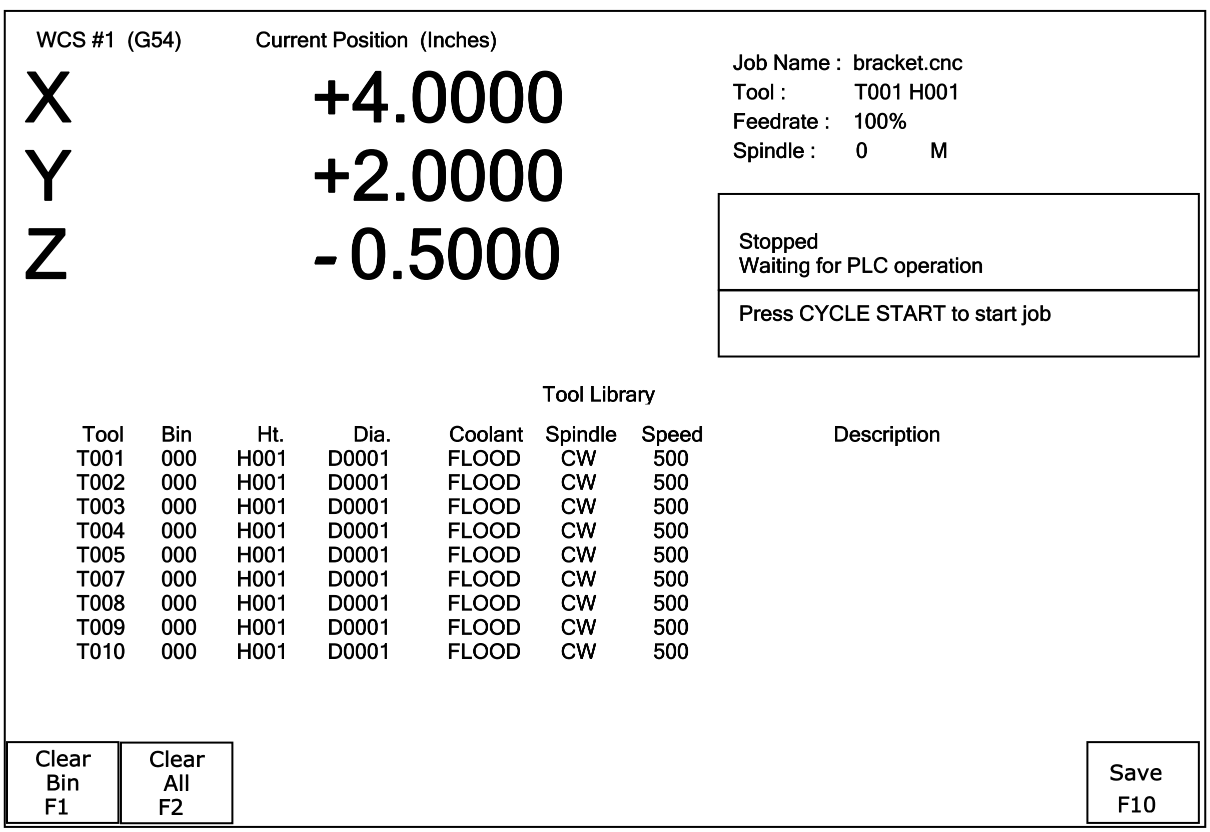
At this point, the Z Reference is now entered into the Offset Library and is the reference height for all other tools. Remove the probe and measure any other tool offsets manually as described earlier in this chapter.

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. Also be sure that parameter 44 is set correctly. This is the input number for the TT1. 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 – Z Ref**, then **F3 – Auto Measure** 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 – Auto Measure** 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 - Save** to save them. Otherwise, press **ESC** to cancel any changes.

Tool Library



The definitions in the Tool Library associate tool (T) numbers with height offset (H) and diameter (D) numbers, the default coolant type, spindle direction, and spindle speed for the tool, and a text description of the tool. This information is used by the Intercon-programming package (described in Chapter 10) to provide defaults whenever a tool change is selected. For enhanced ATC features, the (T) numbers are also associated with bin numbers. See Chapter 14 for more information about enhanced ATC features (parameter 160).

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

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

Bin

This field specifies which bin location, or ATC carousel position, that the tool is occupying. Valid values are –1 (shown as dashes "---") through the maximum number of tools specified by machine parameter 161. A value of 0 indicates that the tool is in the spindle. The **F1-F2** keys will work when the cursor is in the Bin column.

- **F1 - Clear Bin** – places dashes "---" into the bin field (same as entering –1).
- **F2 - Clear All** – places dashes into every bin field.

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

Height

This field specifies a default Height Offset (H) number to use with each tool. Possible values are 1 to 200. Intercon uses this information to provide a default H value at each tool change. CNC10 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 5).

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

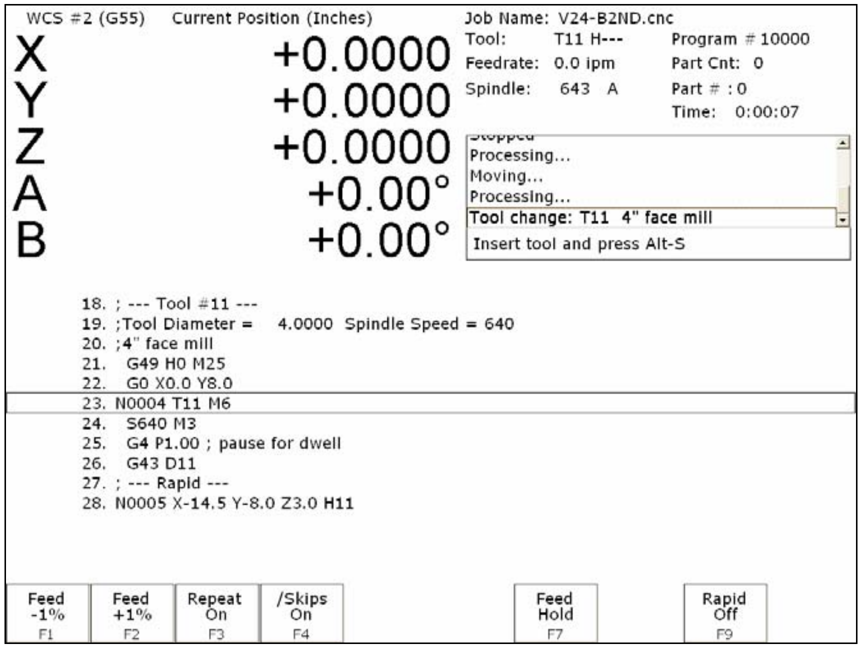
This field specifies a default spindle speed to use with each tool. Possible values are 0 to 500000. Intercon uses this information to automatically insert an S code after a tool change. To change the value, type a new number and press **ENTER**.

Description

This field contains a text description of the tool. The description will appear in a prompt message on the screen when CNC10 reaches a tool change (M6).

Chapter 6
Running a Job

To run the current job, press the **CYCLE START** button on the jog panel. See Chapter 2 for a description of the **CYCLE START** button. If your control is not equipped with a jog panel, press **ALT-S** on the keyboard. The following menu is available, while the job is running.



Job running menu

The following keys are available while the job is running with the G-Code displayed.

F1 – Feed (-1%)
Decrease feedrate override by 1%. This key only appears if jog panel is set to keyboard jogging.

F2 – Feed (+1%)
Increase feedrate override by 1%. This key only appears if jog panel is set to keyboard jogging.

F3 – Repeat On/Off
Toggle job repeat property.

F4 – Skips On/Off
Enable/Disable block skips.

F5 – Auto
Disable single block mode. This key only appears if Single Block Mode is turned on.

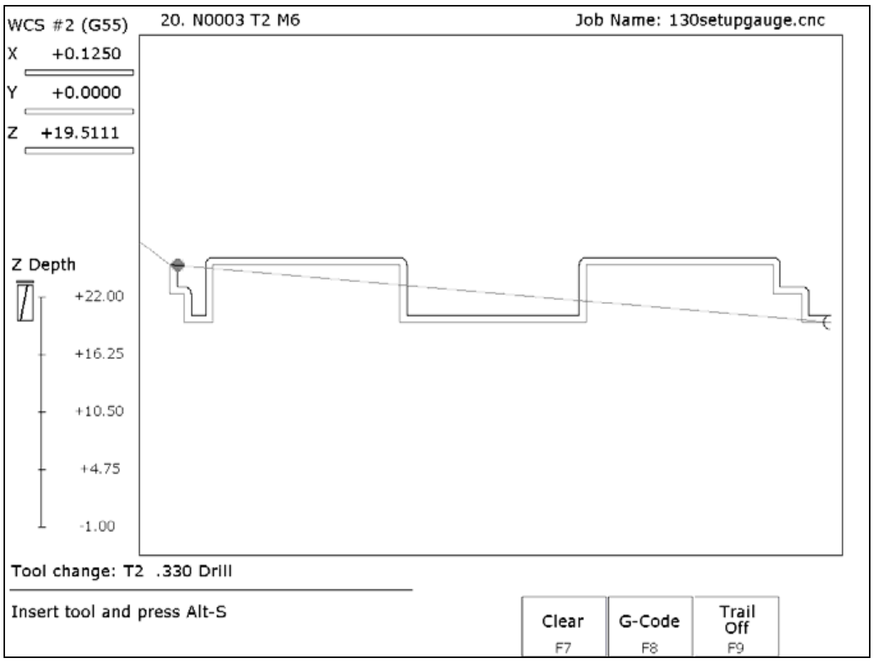
F6 – Stops off
Disable optional stops. This key only appears if Optional Stops is turned on.

F7 – Feed Hold
Turn feed hold on/off. This key only appears if jog panel is set to keyboard jogging.

F8 – Graph
Return to run-time graphics screen. This key only appears if the run-time graphics option is turned on.

F9 – Rapid On/Off
Turn rapid override on/off. The Rapid on/off key controls rapid override. If rapid override is on, the FEEDRATE OVERRIDE knob applies to rapid (GO) moves and to jogging. If the rapid override is off, the FEEDRATE OVERRIDE knob will have no effect on rapid moves and jogging.

F10 - Edit
Start the G-code editor. Press **ALT Tab** to switch between the editor and CNC10 as the job is running.



The following keys are available while the job is running in Run Time Graphics.

F7 – Clear
Clears the trail up to the tools current position in the program.

F8 – G-Code
Return to theG-Code screen.

F9 – Trail On/Off
Return to run-time graphics screen. This key only appears if the run-time graphics option is turned on.

Canceling a Job in Progress

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

CYCLE CANCEL

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

TOOL CHECK

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

EMERGENCY STOP (E-Stop)

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

Resuming a Canceled Job

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

CYCLE START

Pressing the **CYCLE START** button will restart the job at the BEGINNING of the part program.

Resume Job – F1 from the Run menu

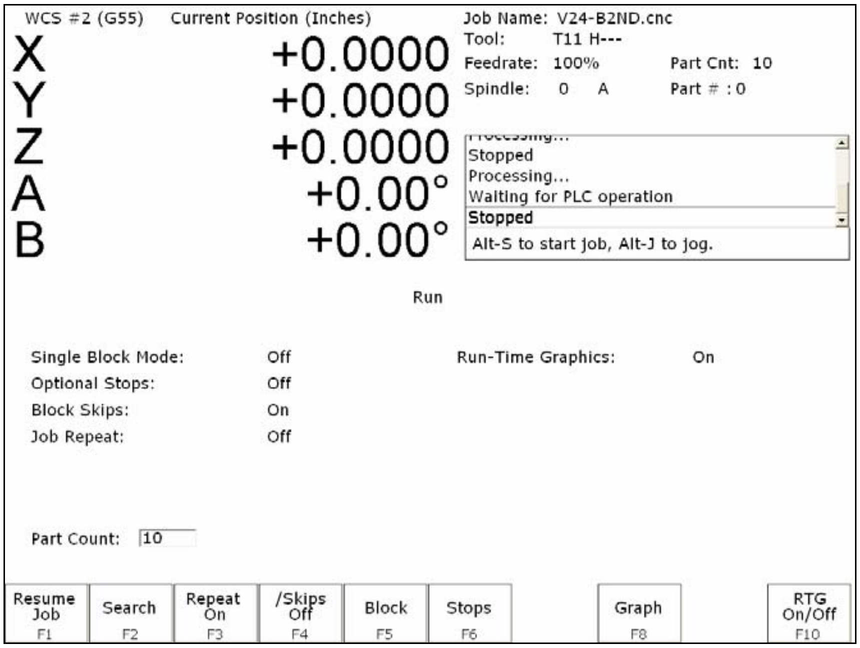
Restart the canceled job at or near the point of interruption. See the next section in this chapter entitled "**Run menu**" for more information.

Search – F2 from the Run menu

Restart at a specified point in the part program. See the next section in this chapter entitled "**Run menu**" for more information.

Run menu

Press **F4 - Run** from the main screen to access the Run menu. From this menu, the operator can restart a canceled job or change the way the job will run.



F1 - Resume Job

Access the resume job screen by pressing **F4 - Run** on the main screen to go to the run screen, and then pressing **F1 – Resume Job** in the run screen to go to the resume job screen. If the job was canceled by pressing **TOOL CHECK**, the control will go to the resume job screen automatically. From this screen, the user can modify tool offsets and the tool library, turn block mode on and off, turn optional stops on or off, graph the partially completed job, or start the partially completed job.

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

Loading a new job.

Running a job to completion.

Parse errors in the job.

Editing or reposting the job file.

Loss of power while a job is running.

F2 - Search

Invoking this option will bring you to the “Search and Run” menu. This menu will allow you to specify the <u>program line,</u> <u>block number,</u> or <u>tool number</u> 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 – Tool Change** “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.

NOTE: You cannot search into a subroutine.

F3 – Repeat On/Off

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

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

F4 - /Skips On/Off

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

F5 - Block Mode

Turns single block mode on and off. This is similar to pressing **AUTO**/**BLOCK**. If single block mode is on, CNC10 will stop after each block in your part program and wait for you to press **CYCLE START**. The current state is indicated in the user window above.

F6 - Optional Stops

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

F8 - Graph

Graphs the part. For more information, see the ***"F8 - Graph"*** section in chapter 3. If this feature is invoked from the Run and Search screen or the Resume Job screen, then the graphics will show exactly where the searched line or block begins. Dotted lines indicate the portion of the part that is skipped. Solid lines indicate the portion of the part that will be machined.

F9 – Rapid On/Off

This function key toggles Rapid Override. The On or Off label indicates the state to which the Rapid Override feature will toggle to when pressed. It does not indicate the current state. It has the same effect as the Rapid Over key discussed in Chapter 2.

F10 – RTG On/Off

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

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.

Power Feed

Press **F4 - Feed** from the Setup menu to access the Power Feed screen. This 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 - ABS** to move an axis to an absolute position, at a specified feedrate.

F2 - Incremental Power Feed

Press **F2 - INC** to move an axis an incremental distance, at a specified feedrate.

F3 - Free XY

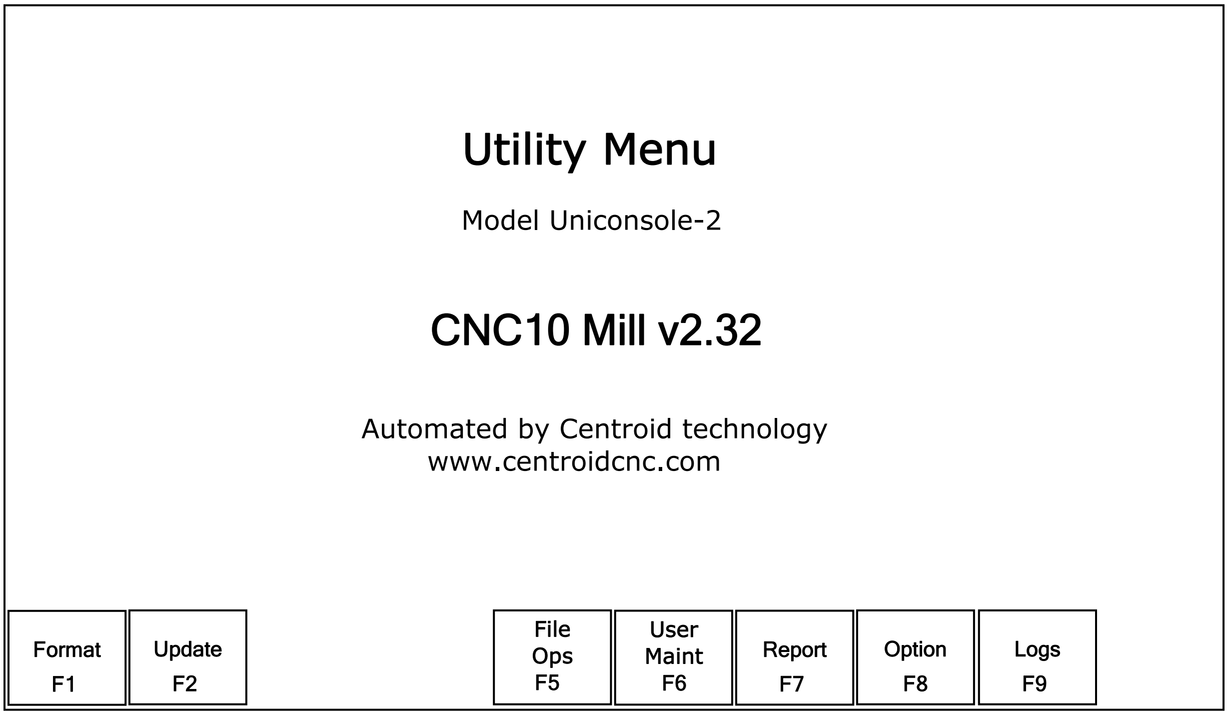
Press **F3 - Free** to release power to the X and Y motors, allowing you to use your machine manually for these two axes.

F4 - Power XY

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

Chapter 7
The Utility Menu

To get to the Utility Menu, press **F7 - Utility** at the CNC10 Main Screen. The model will vary depending on your M-Series Control model.



F1 - Format

Press **F1 - Format** to format a high-density floppy disk.

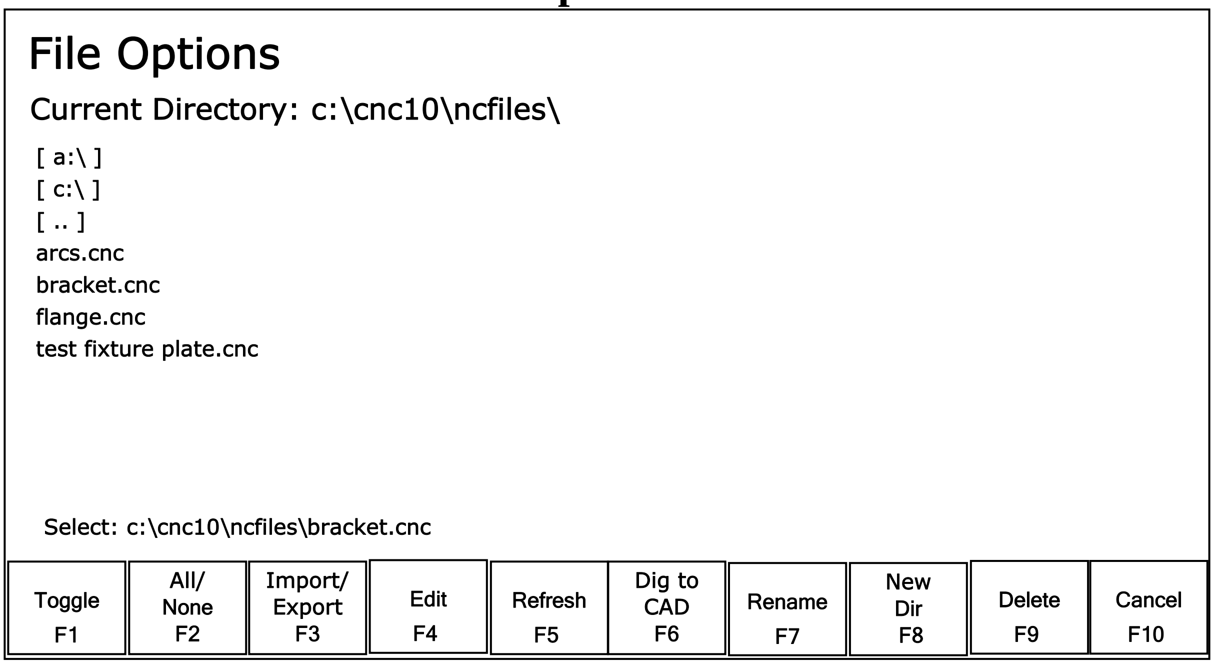
F2 - Update

To update your control software from a floppy disk or USB Storage device, put the update disk in the floppy disk drive (or attach the USB storage device) and press **F2 - Update**. Choose the floppy drive or USB storage device as location of the update. See “Using the Location Chooser”, below. The new software will then be automatically loaded onto the hard drive. Once the new software is loaded, you may be required to power down the controller before using the new software. Failure to do this may cause unpredictable errors.

F5 – File Ops

Use this menu to perform file and directory operations such as: Importing and Exporting files to and from the control, rename or delete files, create or delete directories or convert digitized data to CAD data.

File Ops Menu



F1 – Toggle	Press once to select or press again to unselect a single file.
F2 – All / /None	Press once to select all or press again to unselect all files.
F3 – Import/ Export	Import or Export selected files.
F4 – Edit	Opens selected file in editor.
F5 – Refresh	Refresh file list. Use after inserting a new USB device or floppy disk
F6 – Dig to CAD	Translates digitized files to CAD data.
F7 - Rename	Rename selected file or directory.
F8 - New Dir	Create a new directory in the current folder. Displays a unified file and device browser similar to Windows Explorer.
F9 - Delete	Deletes selected file or directory.
Page Up	Move the cursor backward one page.
Page Down	Move the cursor forward one page.
End	Select the last file in the list.
Home	Select the first fil in the list.
Arrow Keys	Move the cursor in the selected direction.

F6 – User Maint

Use this menu to perform user maintenance such as checking an axis for excessive drag or setting backlash

F1 – Drag

The Drag Factor utility is used to determine if an axis has an excessive amount of drag. To run a drag test, use the F1 key to select the axis which you wish to test, position the axis at or near the home position and press cycle start. The axis will move back to the home switch then traverse the entire range of travel for the axis moving to the opposite limit and returning to home while moving the slow jog rate. If excessive friction (drag) is encountered and error message will be displayed. When the test completes, use F8 Graph to display the results. The red horizontal lines indicate the bounds acceptable limits for the machine as it is currently configured.

F2 - Lash

Backlash Compensation – In order to insure an accurate measurement, always set the backlash compensation in the control to zero before attempting to measure the physical lash in an axis.

F7 - Report

Generates a backup of system configuration files called *report.zip* and copies it to the specified location. Your dealer may then use the disk for servicing and troubleshooting purposes. To restore the configuration files from the report disk, press **F2 - Update** from the Utility menu.

F8 - Options

Shows the software options that you have purchased or added to your control. This page will also display the PLC programs, PIC type, and System ID # at the bottom of the screen.

F9 - Logs

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

F1 - Errors

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

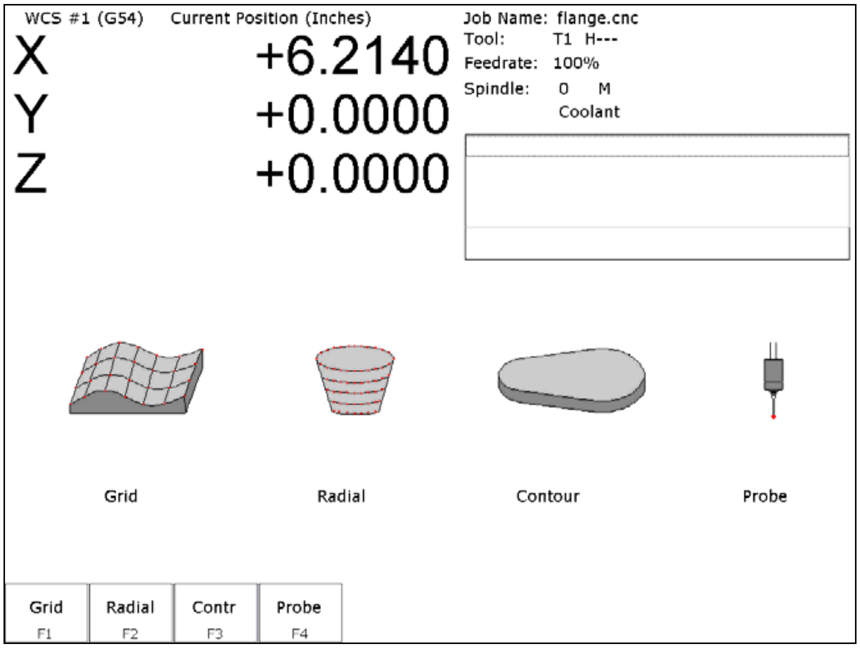
F2- Stats

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

F3 – Export

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

Chapter 8
Digitize
(F9 from Main Menu)



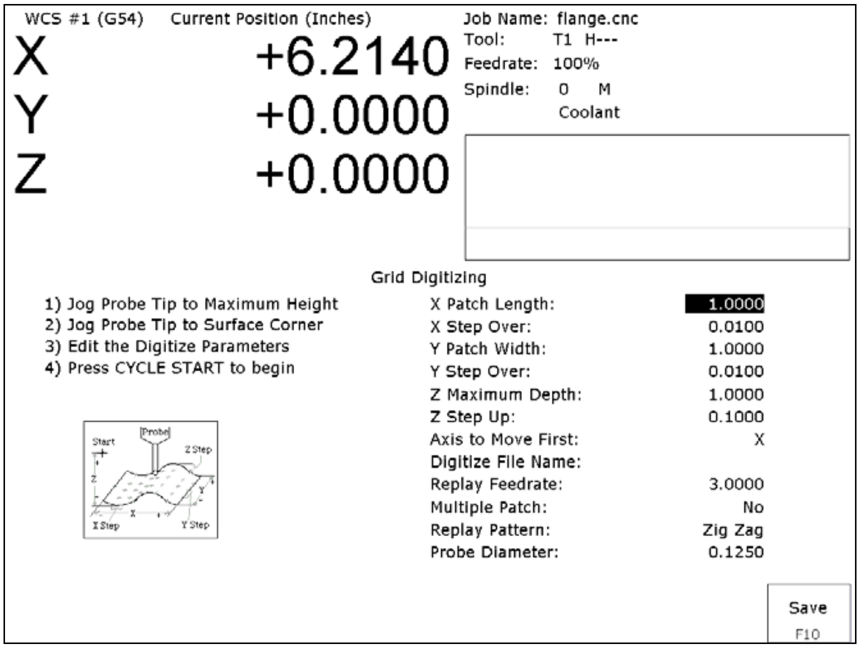
The Digitize feature of CNC10 is used to digitize a rectangular surface area (grid), an inside circular bore area (radial), or a contour. The digitizing process creates a file with M & G codes that represent the digitized surface. If the digitizing probe tip is chosen to match the milling cutter size, the digitized file can be loaded and run to produce an exact copy of the digitized part.

To digitize rectangular surface areas, press **F1 - Grid** (see grid digitize section). To digitize the inside of a bore, press **F2 - Radial** (see radial digitize section). To digitize the contour of a part, press **F3 - Contr** (see contour digitize section). Press **F4 - Probe** to select from the Probing Cycles (See Chapter 9 of this manual).

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

Brushless motor note: If you experience excessive vibration on a brushless drive system, use Parameter 10 to select smooth deceleration in digitizing probing moves. See Chapter 14 for more information.

Grid Digitize
(F1 from Digitize Screen)



<u>Grid Digitize Run Setup</u>
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.

<u>Grid Digitize Parameters</u>
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.

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.

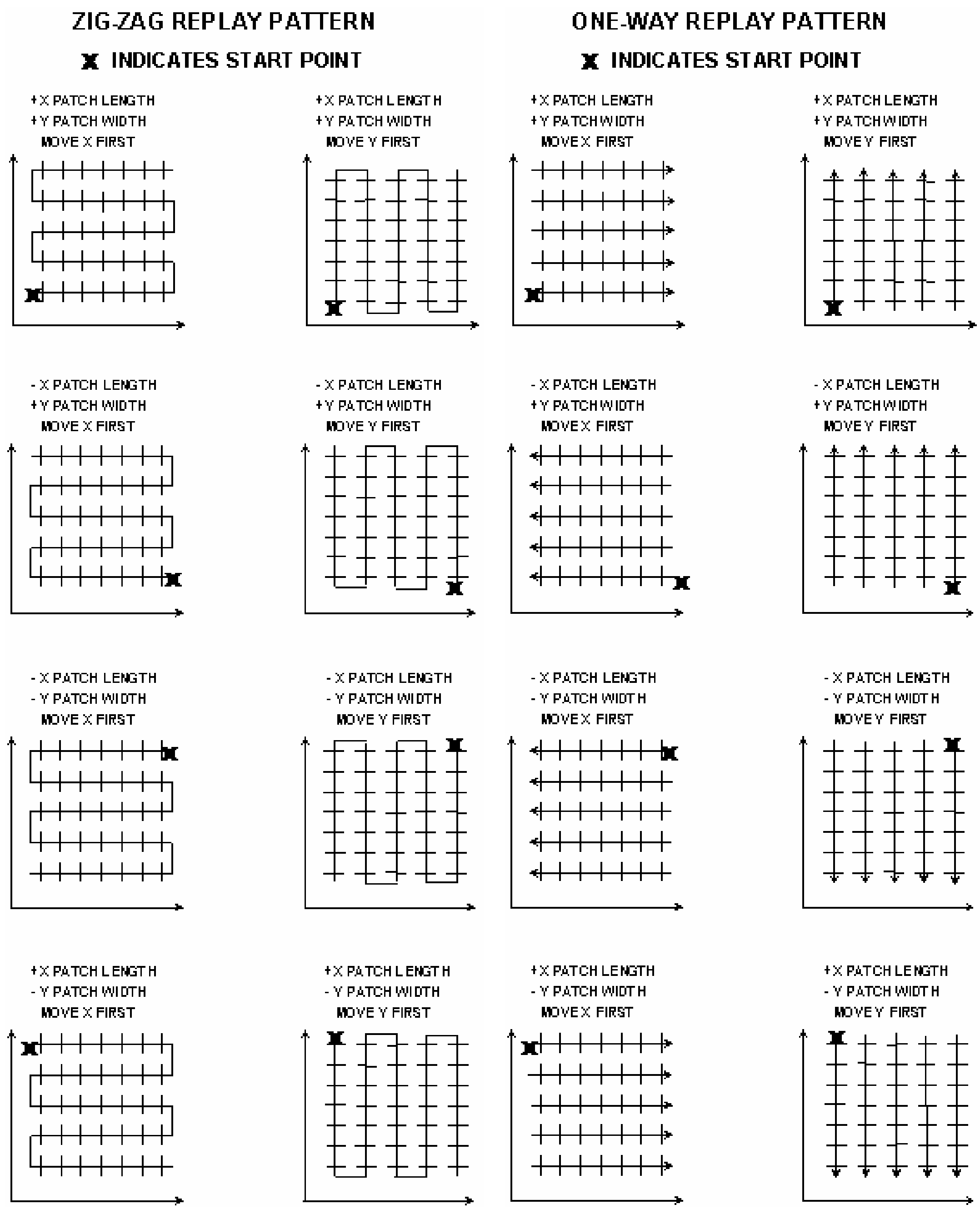
Multiple Patch: Indicates whether or not this digitizing is a continuation of an earlier digitizing. Choose NO if the current digitizing is the first or only digitize run for the part to be digitized. Choose YES if the current digitizing is not the first digitize run for the part. If Yes is selected,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

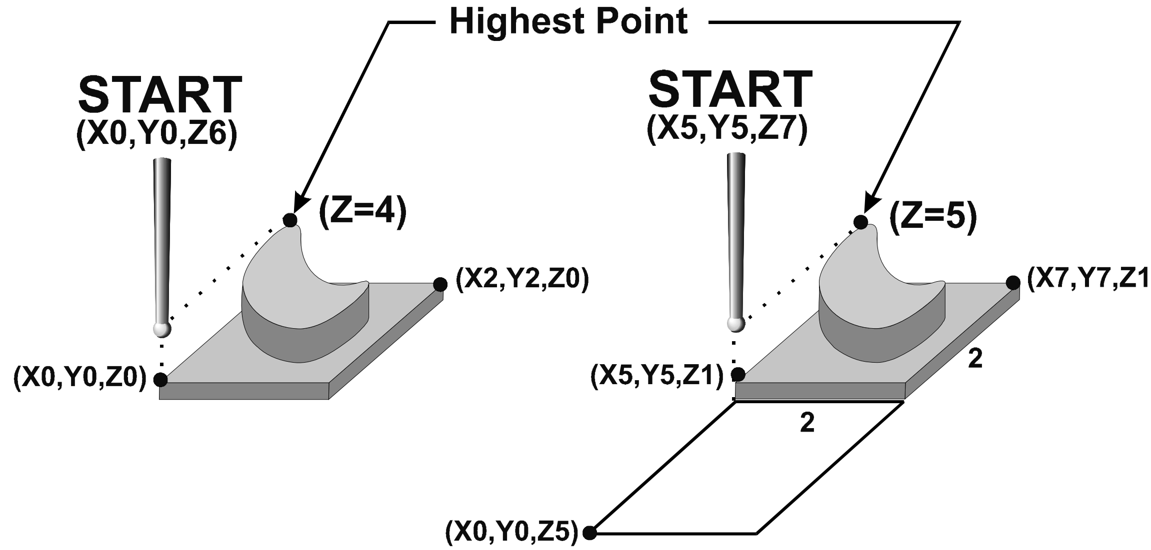
Probe Diameter: Indicates the probe tip diameter.

Grid Digitize Notes

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

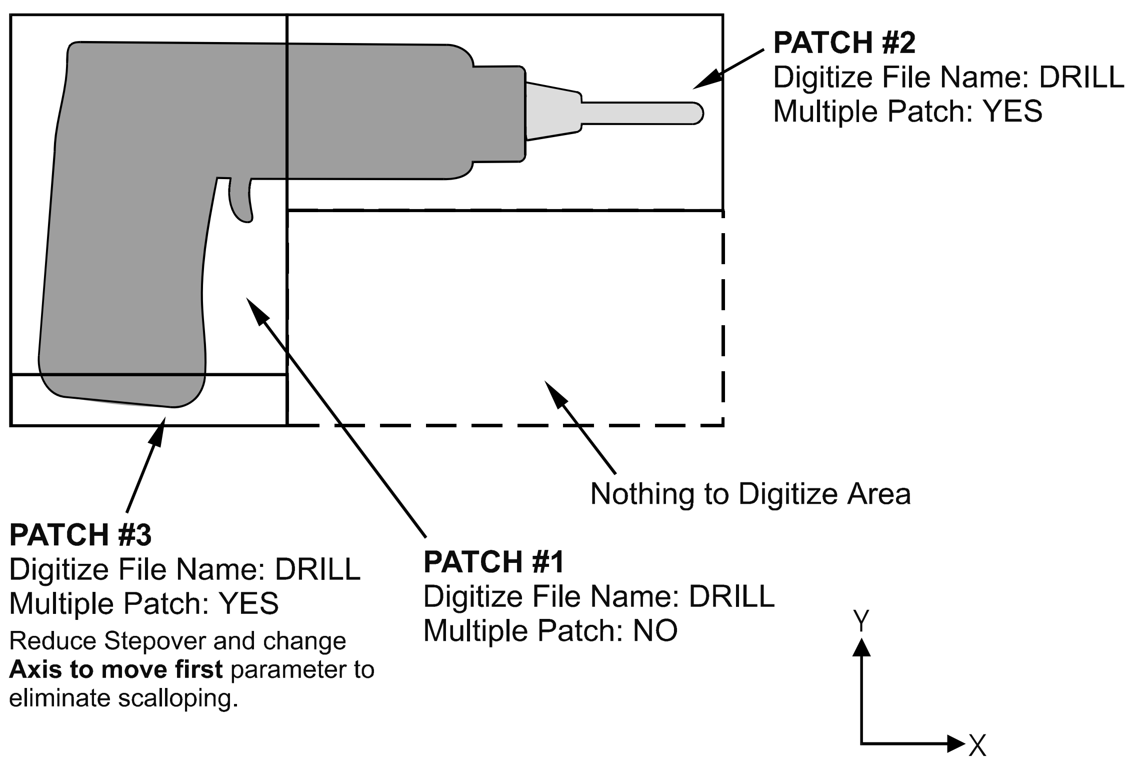


2. A digitizing patch can be located anywhere in the coordinate system. The digitizing starting point is referenced from the part zero. For example, setting up digitizing, as shown in the figure on the right below, will record the first point at (X5, Y5, Z1) and the last point at (X7, Y7, Z1). If the 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 4.



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

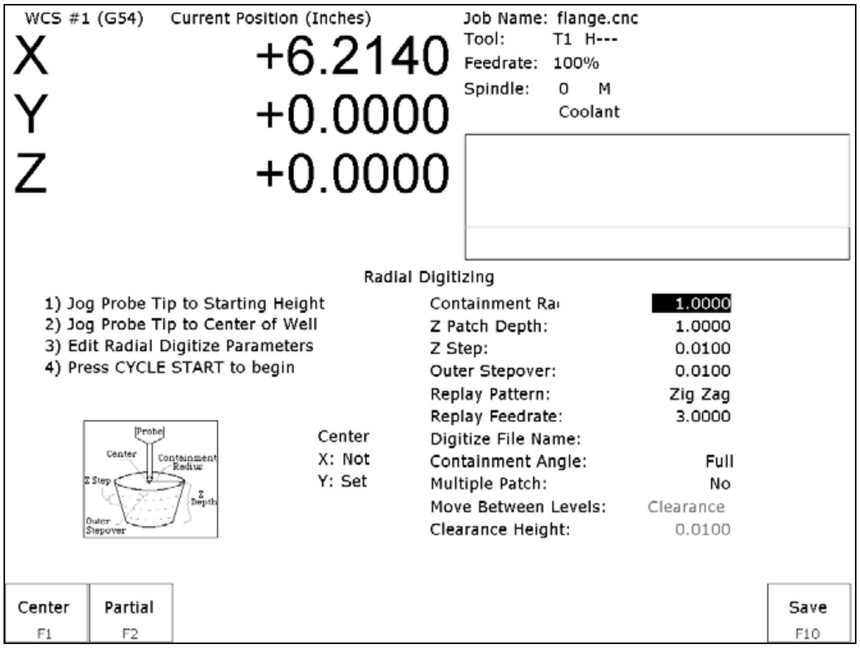
4. Multiple patches are useful in the following situations: completing a canceled digitize run, digitizing parts with large areas that contain nothing to be digitized (shown below), and patching vertical walls to eliminate scallops caused by the cutting tool.



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

Digitizing the entire part and then adding multiple small patches along the walls can avoid vertical wall scalloping. If a vertical wall extending along the X axis needs to be cleared of scallops, just add a small patch running the length of the wall. Set the “Axis to Move First” parameter to Y. This will clear the scallops.

Radial Digitize
(F2 from CNC10 Digitize Screen)



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 bore to be digitized. Then press **F1 - Center** 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 - Partial** 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 later in this section.

Radial Digitize Parameters

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

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

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

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

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

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

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

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

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

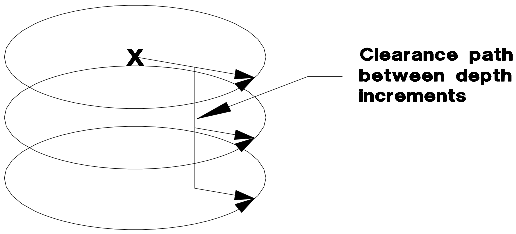
Move Between Levels: This field is enabled only if Partial and CCW or CW option is selected. It indicates the move between Z levels on replay of a partial sector radial digitize file. This move may now be done in three different ways: Clearance, which goes to the clearance height as in previous versions,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

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

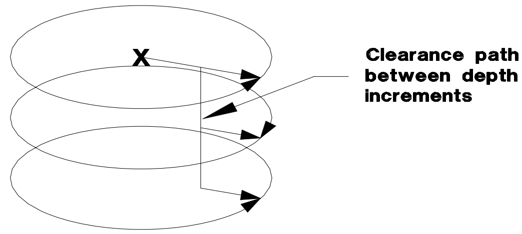
Radial Digitize Notes

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

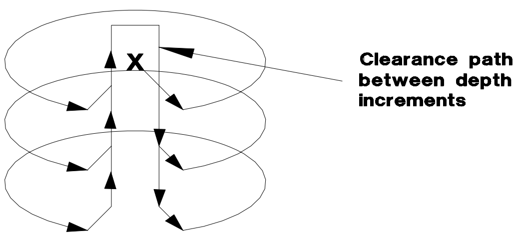
FULL CONTAINMENT ANGLE
ONE WAY REPLAY PATTERN



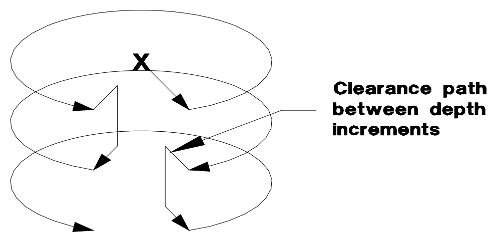
FULL CONTAINMENT ANGLE
ZIG ZAG REPLAY PATTERN



PARTIAL CONTAINMENT ANGLE
ONE WAY CLEARANCE REPLAY PATTERN

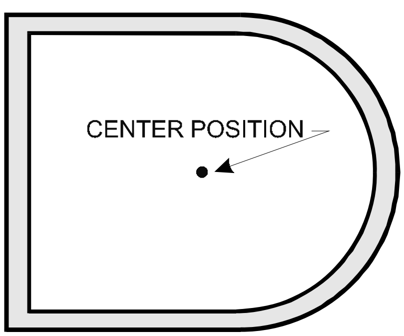


PARTIAL CONTAINMENT ANGLE
ZIG ZAG DIRECT REPLAY PATTERN

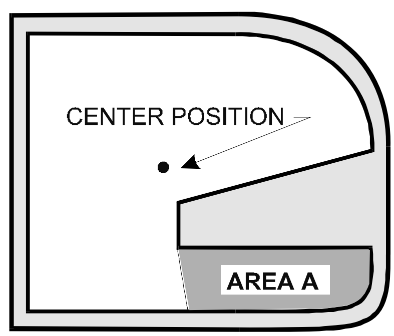


2. When radial digitizing, make sure the probe can fully retract to the center position without obstructions. Observe the two parts below. The cross section on the left has no obstructions that could keep the probe from full retraction to the center position. The cross section on the right does not allow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

IDEAL PART

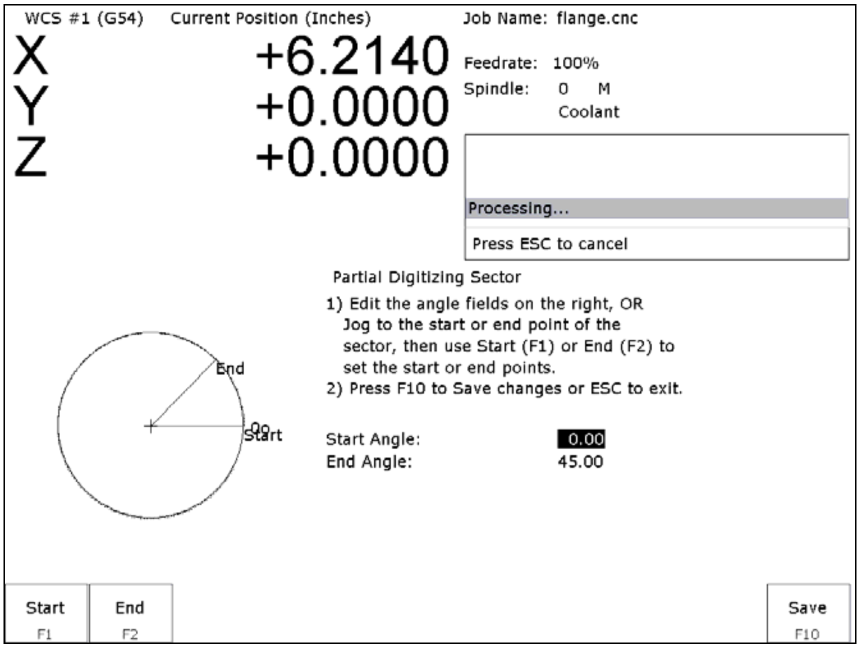


PROBLEM PART



Partial Digitizing Sector Setup

If you set the Radial Digitize Containment Angle to "Partial" then you must set up the Digitizing Setup by pressing **F2 - Partial** from CNC10 Radial Digitize Screen.



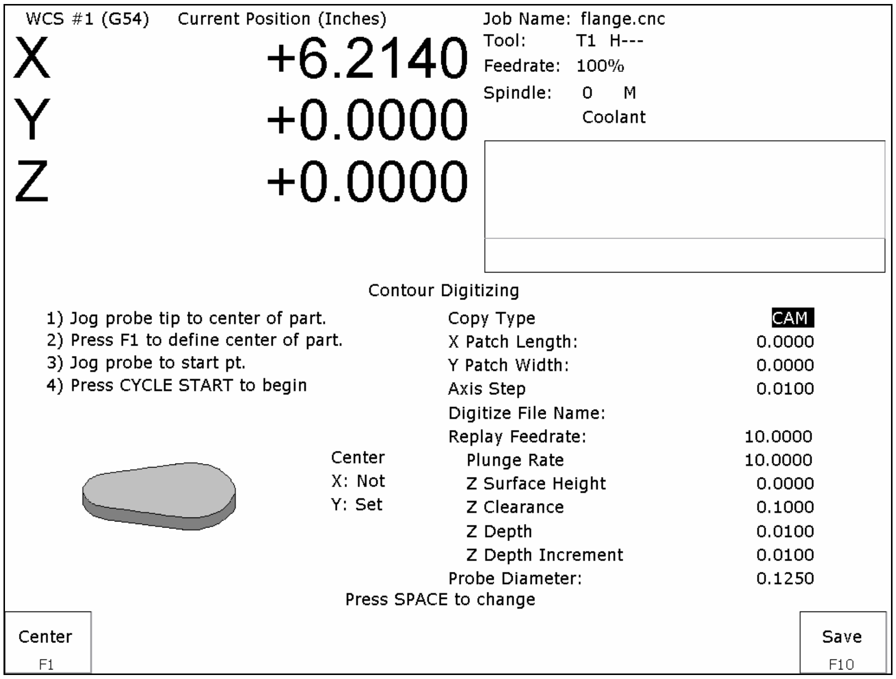
The partial sector can be setup one of two methods:

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

The second method involves jogging the probe tip and touching off the digitize surface. By moving the probe tip to positions on the digitizing surface, one can set the angles. To set the start angle, jog the probe tip to the position on the digitizing surface where the digitizing is to begin and press **F1 - Start** 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 - End** instead to set the end angle.

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

Contour Digitize
(F3 from CNC10 Digitize Screen)



<u>Contour Digitize Run Setup</u>

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

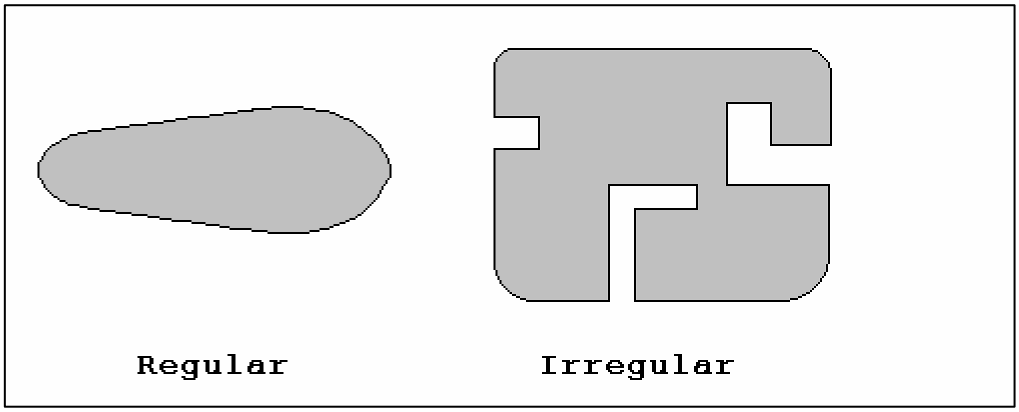
Select CAM for a true CAM shape contour, or Wall for irregular shapes for wall following. Enter the rest of the parameters for the part and digitizing job as shown below.

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

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

Contour Digitize Parameters

Copy Type: Toggle between CAM or Wall. Use CAM for regular shapes (no extreme direction changes) and use Wall following for contours with irregular shapes. See example below.



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

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

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

Digitize File Name: The base name of the file in which the digitize data is stored. The file has an extension of .cam for CNC replay and is stored in the c:\cnc10\ncfiles directory.

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

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

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

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

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

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

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

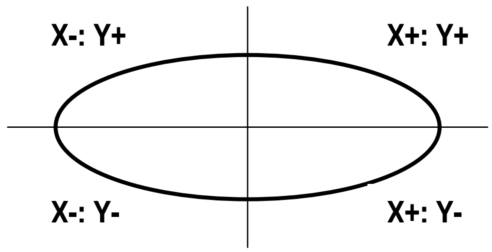
Contour Digitize Notes

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

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

Table 1 – Probe direction by starting quadrant

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



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

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

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

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

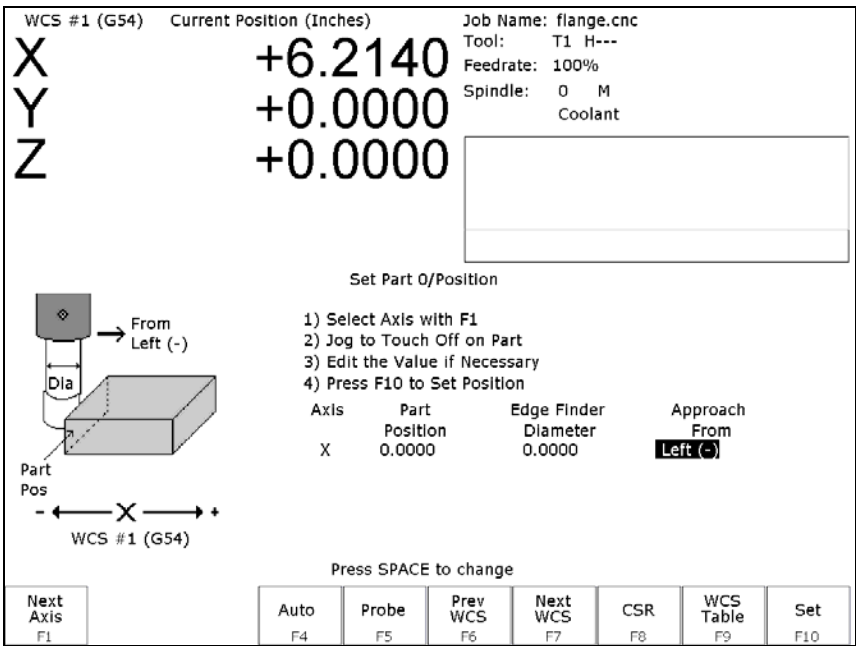
Chapter 9
Probing

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

Part Setup with Probing

Single axis, single surface probing is available on the Set Part 0/Position screen using the **F4 - Auto** 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 **F5 - Probe** key or from **F4 - Probe** in the digitize screen. These allow you to locate the center points and corners of differently shaped parts.

Brushless motor note: If you experience excessive vibration on a brushless drive system, use Parameter 10 to select smooth deceleration in probing moves. See Chapter 14 for more information.



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. Make sure your probe height and diameter offsets are set for the tool number you assigned to the probe and that parameter 12 is set to that tool number. The Edge Finder Diameter will be set automatically.

To set part 0 using the probe:

1. Select the current work coordinate system by pressing **F6 – Prev WCS** or **F7 – Next WCS**. Then select the axis you want to probe by pressing **F1 – Next Axis**.

2. Manually jog the probe about 1/2 inch away from the surface you wish to define. Make sure the approach direction to the part is set properly. Probe the selected axis by pressing **F4 - Auto**. When the surface is found, the control will assume this point to be the new axis 0.

3. If you want this probed surface to be something different than 0, enter the value by the using arrow keys to highlight Part Position. Type in the value and press **F10 - Set**.

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

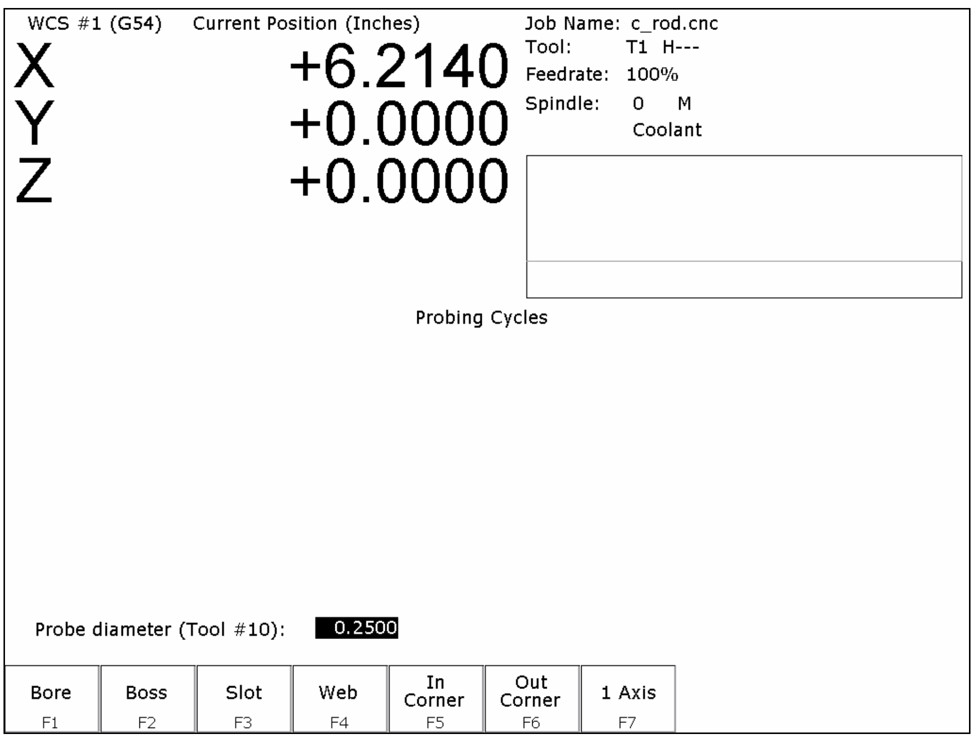
To enter the Probing Cycles screen, press **F5 - Probe** from the Set Part 0/Position screen. You can locate the center of a bore, boss, slot, or web. You can also find an inside corner, outside corner, or a single axis. The corner points don't even have to be right angles! The Edge Finder Diameter doesn't need to be entered since these cycles place the probe directly over the center or corner of the part.

Calibrating the Probe Tip Diameter

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

Probing Cycles

You can enter the Probing Cycles screen from either the Set Part 0/Position screen (**F5 - Probe**) or the Digitize menu (**F4 - Probe**). The Probing Cycles screen is shown below:

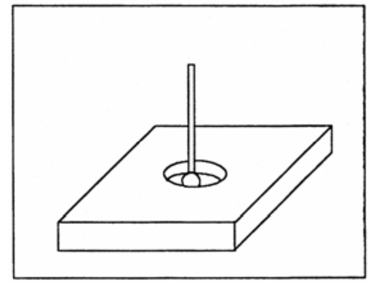


The probing cycles will report the location and dimensions, as applicable, of the probed feature in a floating dialog box. The dimensions are adjusted to compensate for the probe tip diameter, entered in the Offset Library (see Parameter 12). For your convenience, you can edit the probe diameter on this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

F1 – Bore

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

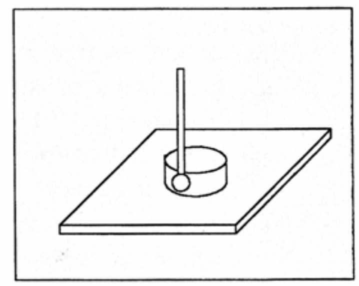
1 - Make sure the probe is clear of any obstacles.
2 - Manually jog the probe inside the hole. The probe tip should be just below the top edge of the surface.
3 - Press **CYCLE START** to start the probing.
At the end of probing, the probe will be positioned at the center of the bore, and the X and Y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

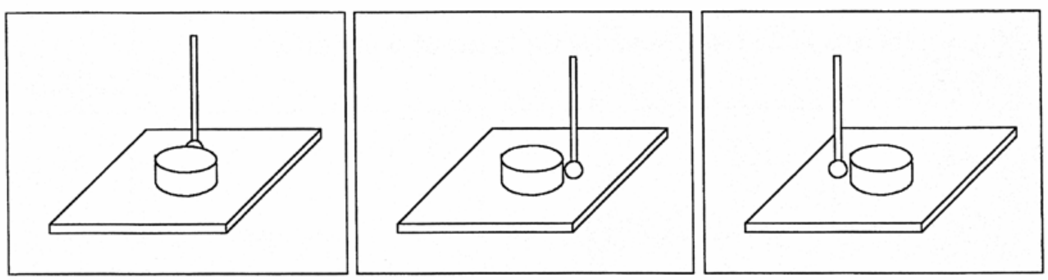


F2 - Boss

Press **F2 - Boss** 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 - Orient** 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. Enter approximate distance (in the Z direction) the probe must move to lift up over the Boss.
5 - Press **CYCLE START** to start the probing cycle.



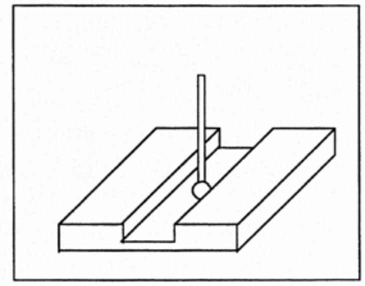


If the Z clearance you entered is too small, the probe will stop and show an error message. Correct the problem by repeating the previous steps. If the approximate diameter you entered is too small, the probe will bounce by 10 percent of its diameter across the top surface until it either finds the correct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

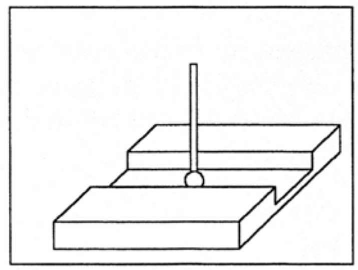
Once the probing cycle is complete, the probe will be positioned at the center of the boss at the Z clearance level entered. Press **ESC** to return to the Set Part 0/ Position screen or digitize screen.

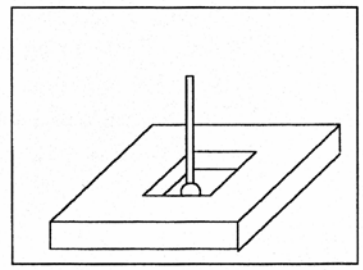
F3 - Slot

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



1 - Press **F1 - Orient** 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 **CYCLE START** to begin probing.



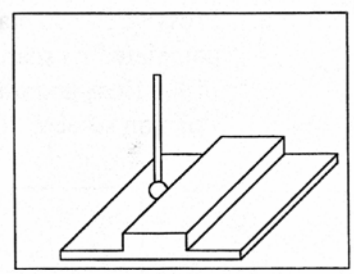


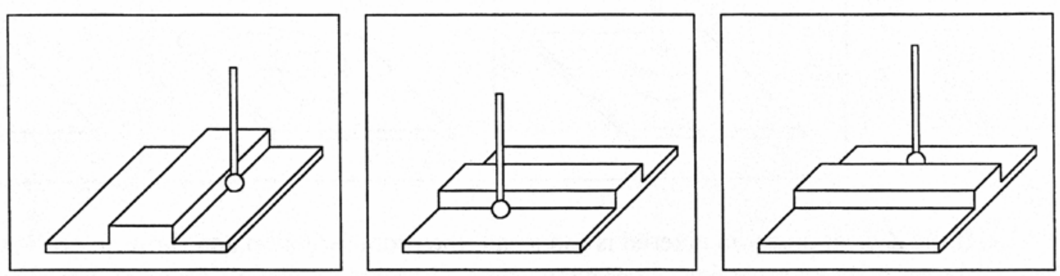
Once the cycle is finished, the probe will be located at the center of the slot. Press **ESC** to return to the Set Part 0/ Position Screen or digitize screen.

F4 - Web

Press **F4 - Web** 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 - Orient** 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 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 **CYCLE START** to start the probing cycle. Once the probe has completed its search, it will automatically position to the centerline of the web.



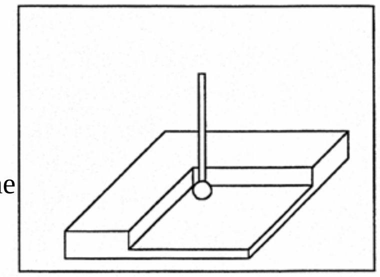


If the Z clearance you entered is too small, the probe will stop and show an error message. Correct the problem by repeating the previous steps. If the approximate width you entered is too small, the probe will bounce by 10 percent of its width across the top surface until it either finds the correct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

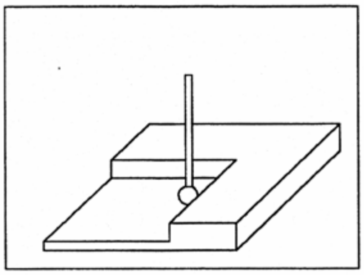
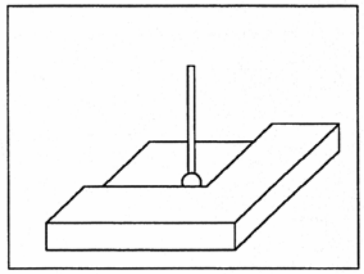
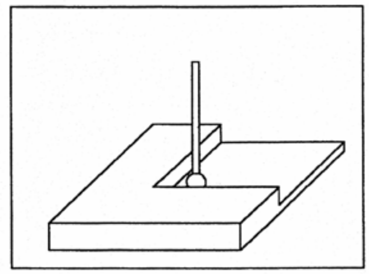
Once the probing cycle is complete, the probe will be positioned at the center of the web, at the Z clearance level entered. Press **ESC** to return to the Set Part 0/ Position screen or digitize screen.

F5 - In Corner (Inside Corner)

Press **F5 - In Corner** to enter the Inside Corner screen. One of the pictures will appear with instructions. This cycle is similar to that of a slot cycle, the main difference is that you need to enter a clearance amount.



1 - Press **F1 Orient** and the screen will cycle through one of the probe orientations shown here.
2 - Enter the clearance Amount. This is an approximate distance (in the Z direction) the probe must move to clear the corner
3 - 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).
4 - Press **CYCLE START** to start the probing cycle.

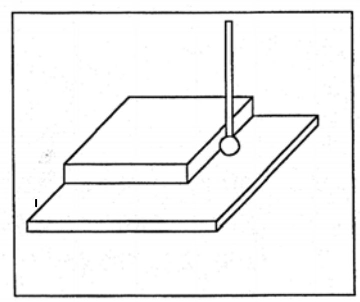
At the end of the probing cycle, the probe will be positioned above the corner at the Z clearance level entered. Press **ESC** to return to the Set Part 0/ Position screen or digitize screen.

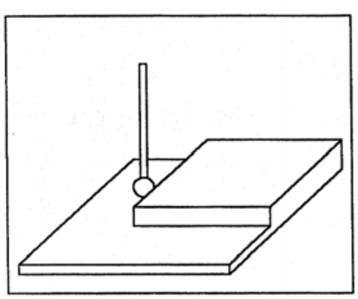
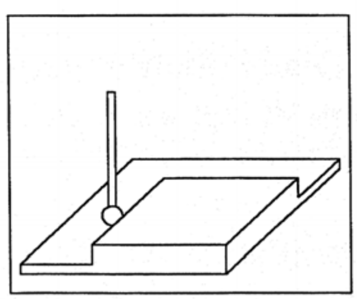
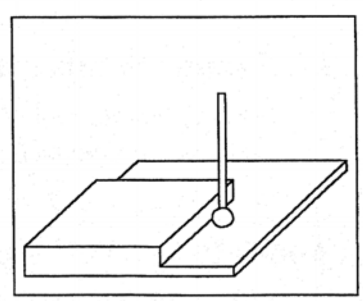
F6 - Out Corner (Outside Corner)

Press **F6 – Out Corner** 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 - Orient** to select the orientation of the probe with respect to the Corner. You will see one of the pictures shown.
2 - Press **F2 - Side** 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 **CYCLE START** to start the probing cycle.



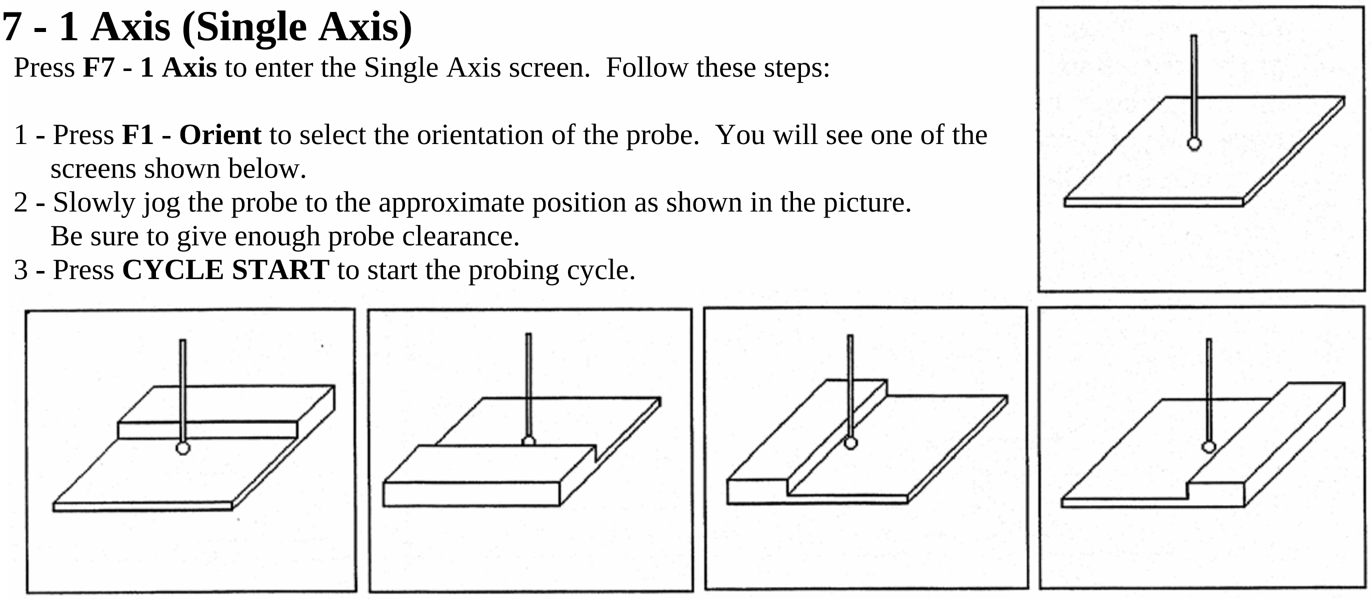




Once the probe has completed its search, it will be positioned above the corner at the Z clearance you specified. Press **ESC** to return to the Set Part 0/ Position screen or digitize screen.

F7 - 1 Axis (Single Axis)

Press **F7 - 1 Axis** to enter the Single Axis screen. Follow these steps:

1 - Press **F1 - Orient** 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. Be sure to give enough probe clearance.
3 - Press **CYCLE START** to start the probing cycle.



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

Probe Parameters

Various probing parameters can be set on the Machine Parameters screen (see Chapter 14). Make sure you enter these parameters before you begin using the probe. If these parameters are not entered properly, damage to the probe may result.

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

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

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

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

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

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

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

Chapter 10

Intercon Software

Introduction

Centroid's Intercon Conversational Software allows you to quickly create a 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.

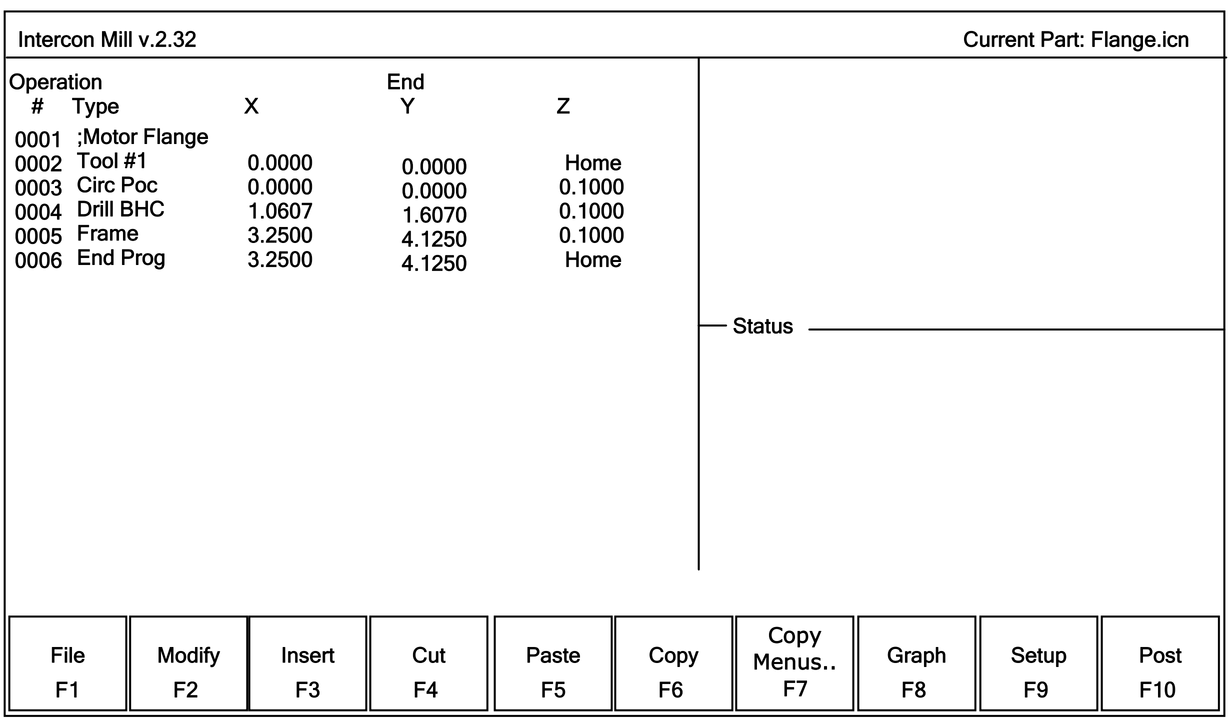
Intercon generates a G-code program from the geometric information you have entered. This is an advantage in several ways:

The G-code program generated by Intercon can be edited using the built-in Centroid G-code editor (**F6 - Edit** from the main screen). Intercon programs can be interrupted and restarted even in the middle of a canned cycle.

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

Intercon Main Screen

When you access Intercon through the **F5 - CAM** option in the CNC10 Main screen, the part program will be displayed if the current job loaded in CNC10 had an associated Intercon program. If the job file in CNC10 did not have an associated Intercon program, the **F1 - File** menu will be displayed to load or create a file.



When the part program is displayed, different operations can be navigated and highlighted for additional actions by using the arrow keys, and the **HOME**, **END**, **PAGE UP** and **PAGE DOWN** keys.

F1 – File Choosing **F1 - File** will display the screen below. Intercon stores part programs with an extension of .icn. For example, if you choose to name your new part program flange, Intercon will save the program as flange.icn. ICN files are only readable by Intercon.

Intercon Mill v2.32		Intercon File Menu	Current Part: Flange.icn
Directory: c:\intercon			
File	Programmer	Description	Date Modified
[a:\]		Drive	
[c:\]		Drive	
[d:\]		Drive	
[j:\]	Sandisk	Drive	
[..]		Parent directory	
flange	John Q. Public	Demo Flange Part	06-Oct-2006
helix	John Q. Public	Demo Helix Part	06-Oct-2006
irreg	John Q. Public	Demo Pocket Cleanout	06-Oct-2006
lineclr	--	--	19-Nov-2006
thread	John Q. Public	Demo Thread	06-Oct-2006
__oscp__	One Shot Circ Pocket	* DO NOT MODIFY *	06-Oct-2006
__osda__	One Shot Drill Array	* DO NOT MODIFY *	06-Oct-2006
__osdb__	One Shot Drill BHC	* DO NOT MODIFY *	06-Oct-2006
__osdr__	One Shot Drill Cycle	* DO NOT MODIFY *	06-Oct-2006
__osfa__	One Shot Face	* DO NOT MODIFY *	06-Oct-2006
__osfr__	One Shot Frame	* DO NOT MODIFY *	06-Oct-2006
__osrp__	One Shot Rect Pocket	* DO NOT MODIFY *	06-Oct-2006

New F1	Load F2	Save F3	Save As F4	Delete F5	Details On/Off F9

File Menu

F1 – New To program a new part, choose **F1 – New.** A prompt will be displayed where the name of the new program can be typed, followed by the **F10 - Accept** or **ENTER** key to accept the new name can be entered. You can enter who the programmer is, a program description, the units of measure and the date.

F2 – Load When you press **F2 - Load**, the screen below is displayed. (Shown with details "ON")

Load file from CNC hard drive c:\intercon

Use arrow keys to select file to load and press F10 to Accept.

File	Programmer	Description	Date Modified
[..]		Parent directory	06-Oct-2006
flange	John Q. Public	Demo Flange Part	06-Oct-2006
helix	John Q. Public	Demo Helix Part	06-Oct-2006
irreg	John Q. Public	Demo Pocket Cleanout	19-Nov-2006
thread	One Shot Circ Pocket	Demo Thread	06-Oct-2006
__oscp__	One Shot Drill Array	* DO NOT MODIFY *	06-Oct-2006
__osda__	One Shot Drill BHC	* DO NOT MODIFY *	06-Oct-2006
__osdb__	One Shot Drill Cycle	* DO NOT MODIFY *	06-Oct-2006

Job to load? bracket.cnc

G code /ICN F1	Floppy /USB/LAN F2	Details On/Off F3	Show Recent F4	Date/ Alpha F5	Edit F6	Help On/Off F7	Graph F8	Advanced F9	Accept F10

Load Menu

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

It is also possible to start typing the name of the program to be loaded. When typing has started, the characters appear in the "File to load:" prompt above the function keys. Different drives and directories can be accessed by typing in the path at the "File to load:" prompt, or by pressing **F10** or **ENTER** on a bracketed directory name. When loading a new file, a prompt will be displayed asking whether to save the existing file if there was one.

Additional viewing and loading options are available through the F-Key menus which are detailed below:

F1 – G code/ICN Allows user to toggle the view between the Intercon files present in either c:\intercon or c:\cnc10\ncfiles.

F2 – Floppy USB/LAN Provides options for loading Intercon files from USB devices, floppy and LAN drives.

F3 – Details On/Off The **F3-Details On/Off** option changes the format of the display such that each file or directory is on a separate line and there are columns displayed for Programmer, Description, and Date Modified, i.e., the information that is contained in the program header operation.

F4 – Show recent Use the **F4–Show Recent** option to show the 15 most recently loaded Intercon and g-code files. It is important to remember that even though g-code files are displayed on this screen, ONLY Intercon files should be loaded from this screen. WARNING!!! Attempting to load a g-code file from the "Show Recent" screen will cause an error which will discard the current Intercon program. All unsaved changes will be lost. If you should accidently load a g-code file, press escape to return to the main Intercon menu.

F5 – Date/Alpha Use **F5-Date/Alpha** to view files either alphabetically or by date modified. By default, programs are listed in ascending alphabetical order.

F6 – Edit Opens the selected file in Intercon for editing.

F7 – Help On/Off Displays on screen help for the load menus.

F8 – Graph Graphs the selected file.

F9 – Advanced Displays file menu in a comprehensive "all in one" format similar to Windows Explorer

File Menu (continued from pg 10-2)

F3 - Save Press **F3 - Save** to save the current part program. The current program will be saved under the specified name.

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

F5 – Delete Press **F5 - Delete** to delete a file. After **F5 - Delete** is pressed, the screen will appear as in the **F2 - Load** option where you can be navigate the files. A yes/no prompt will appear after accepting a file for deletion for final confirmation.

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

Intercon Main Screen (continued from pg 10-1)

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

F5 – Help Displays a picture to help define operations. Can be turned on and off by pressing the **F5 – Help** key.

F6 - Math Help Provides a math assistance to solve trigonometric problems common in part drawings. For more information see the Math Help section later in this chapter.

F8 – Graph Choosing **F8 - Graph** will graph the current program. The graph is the same as what would be produced if the current program were translated into G-codes and graphed from CNC10 software. See Chapter 3 for more information about the Graph menu.

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

F3 – Insert Choosing **F3 - Insert** will insert a new operation before the operation that is currently highlighted, unless the highlighted operation is the first operation in which case the inserted operation will be inserted as the second operation. For more information on the operations see Insert Operations later in this chapter.

F4 – Cut Choosing **F4 - Cut** will cut (remove) the highlighted operation from the program. The operation that is cut is placed onto the clipboard stack.

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

F6 – Copy Choosing **F6 - Copy** will copy the highlighted operation into the clipboard stack and advance the cursor to the next operation.

F7 – Copy Menus… Choosing **F7 - Copy Menus…** will display these options:

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

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

F4 – Cut Choosing **F4 - Cut** will cut (remove) the highlighted operation from the program. The operation that is cut is placed onto the clipboard stack.

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

F6 – Copy Choosing **F6 - Copy** will copy the highlighted operation into the clipboard stack and advance the cursor to the next operation.

F9 - Clear Clipbrd Removes all operations in the clipboard stack.

F8 – Graph Choosing **F8 - Graph** will graph the current program. The graph is the same as what would be produced if the current program were translated into G-codes and graphed from CNC10 software. See Chapter 3 for more information about the Graph menu.

F9 – Setup Choosing **F9 - Setup** will display the Setup menu where certain options can be set. The Setup menu appears as below.

Intercon Mill v2.32 Current Part: Flange.icn

Intercon Setup

Comment Generation	:	Disabled
Clearance Amount	:	0.1000
Spindle/Coolant Delay	:	3.00
Corner Feedrate Overide	:	50%
Modal Linear	:	No
Modal Arc	:	No
Modal Drill/Bore/Tap	:	No
Rotary 4th Axis	:	No
DRO Units	:	Inches
Machine Units	:	Inches
Help Icons always on	:	No

Toggle F1

Accept F10

Use the up and down arrow keys to move. Clearance Amount, Spindle/Coolant Delay, and Corner Feedrate Override require a value to be typed in. The other fields have fixed values that may be toggled by using the **F1 - Toggle** key.

<u>Comment Generation</u>: 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.

<u>Clearance Amount</u>: 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.

<u>Spindle/Coolant Delay</u>: 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.

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

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

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

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

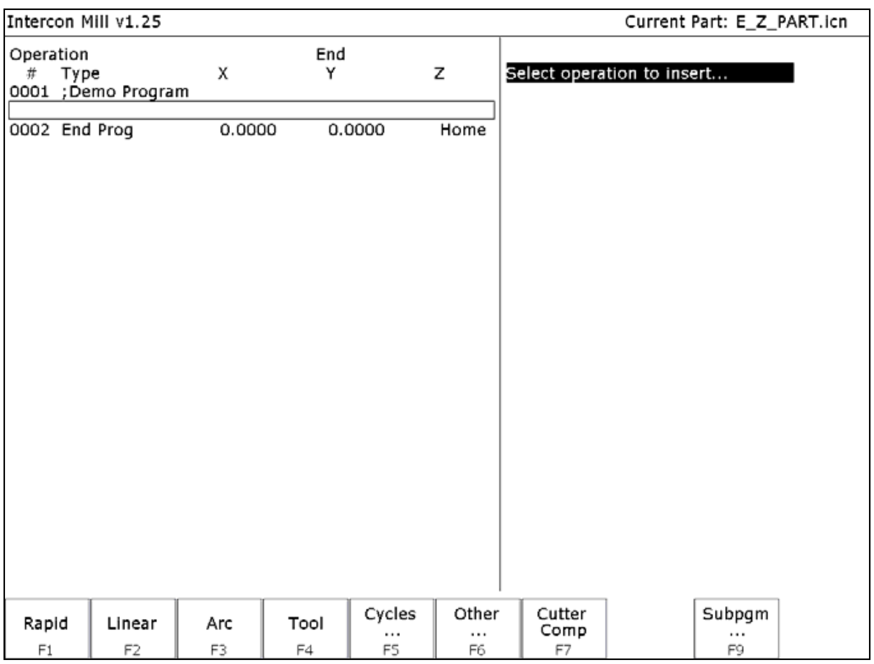
Machine Units: Specifies the Units used for machining. It affects the corresponding field in the Control Configuration of CNC10. The posted G-code will contain a G20 for Inches mode and a G21 for Metric mode.

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

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

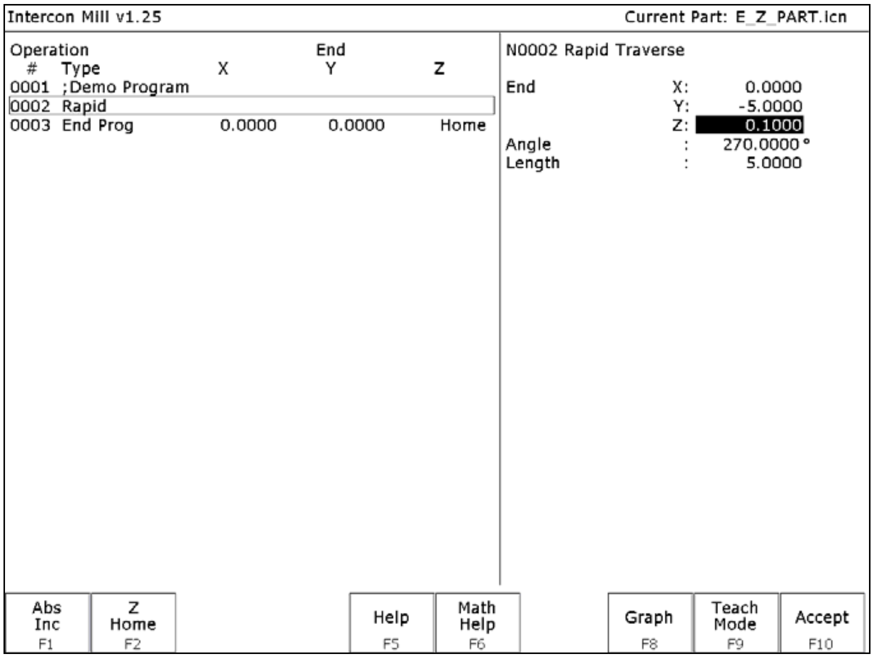
Insert Operation

When you press **F3 - Insert**, or when you choose New Part from the Main Screen, you will see the Insert screen:



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

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



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 – Abs/Inc** 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.

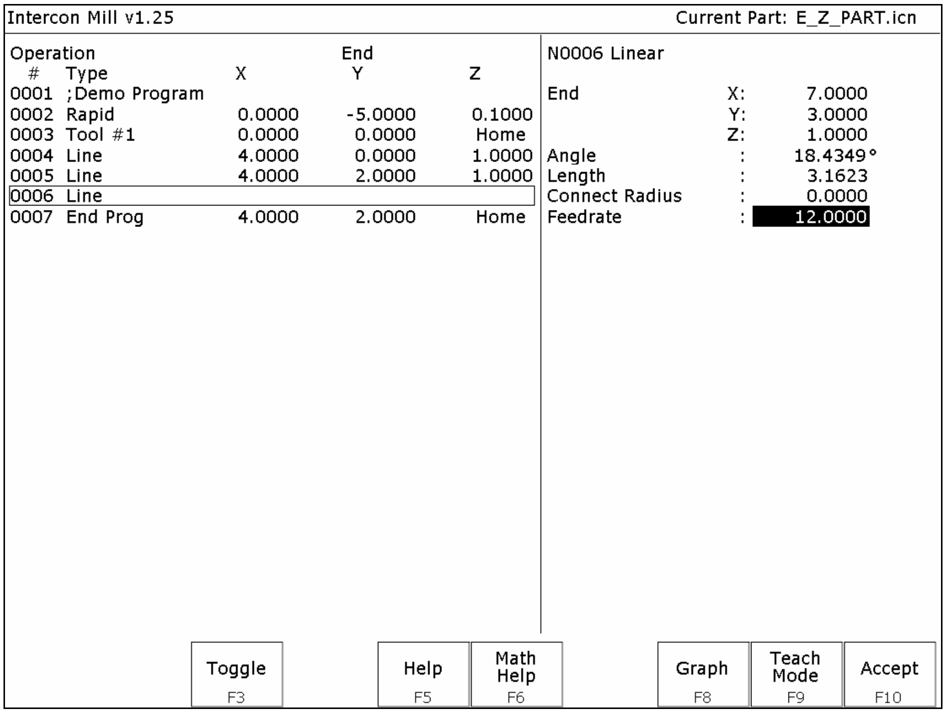
The **F2 - Z Home** key may be used on the Z destination field to tie the ending Z coordinate to the Z home position. This means that no matter what your Z home value is at the time that you run your program, the final Z position will be the Z home position.

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

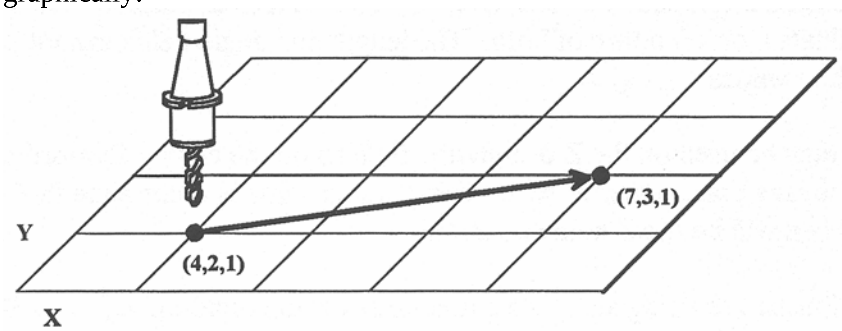
NOTE: When making rapid moves, if a Z destination higher than the current cutter position is specified, the cutter will first be raised to the destination Z position, and then move linearly in X and Y to arrive at the destination. If a Z destination lower than the current cutter position is specified,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

F2 - Linear Mill If you press **F2 - Linear** from the Insert Operation screen, a screen similar to the following appears:



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



<u>End:</u> 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.

<u>Angle:</u> 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.

<u>Length:</u> 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.

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

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

$\boxed{M}$ $\boxed{F}$

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

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.

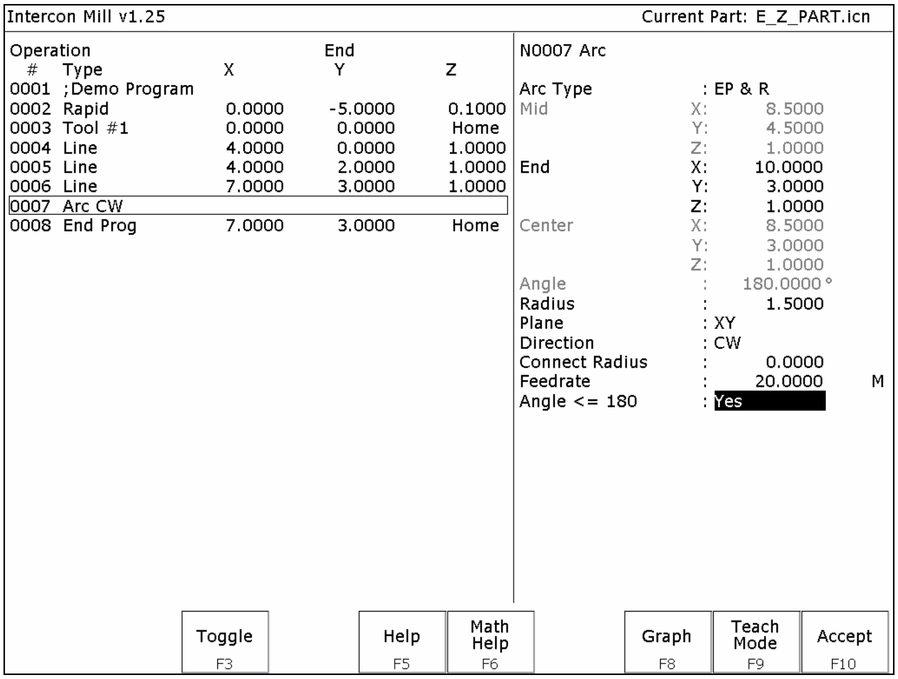
Seconds: The number of seconds you want to move the rotary axis. Values for this field are between 0 and 59.

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.

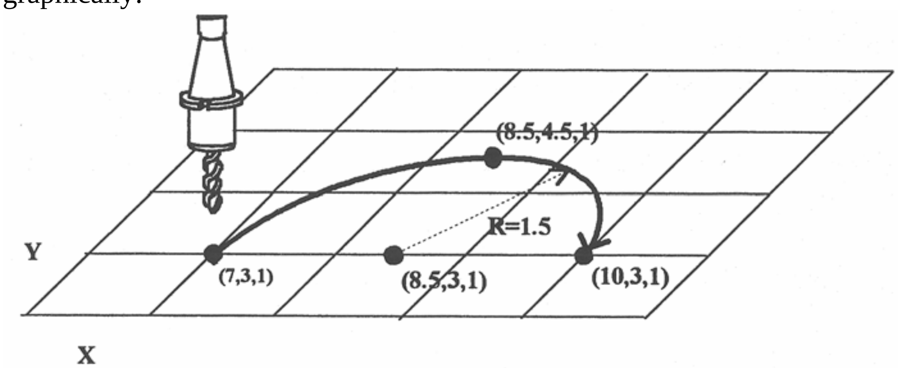
Rotary movement defaults to zero degrees, incremental. To enter an absolute (rather than incremental) rotary position, you must press **F1 - Abs**/**Inc** to toggle to absolute.

F3 - Arc Mill

If you press **F3 - Arc** for Arc Mill from the Insert Operation screen, you will see the following screen:



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 (3-Point). The Three Point arc is designed to be used in conjunction with Teach Mode. When specifying a particular kind of arc, you will not be able to modify certain fields. For example, if you are specifying an endpoint and a radius, you will not be able to modify the mid point, center point and angle fields. This is because Intercon calculates the correct values for these fields.

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

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

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

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

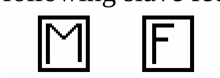
Radius: Distance from the center of the arc to its edge. This value must be greater than 0. You will only be able to edit this value if you are specifying an end point and radius (EP&R) arc.

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

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

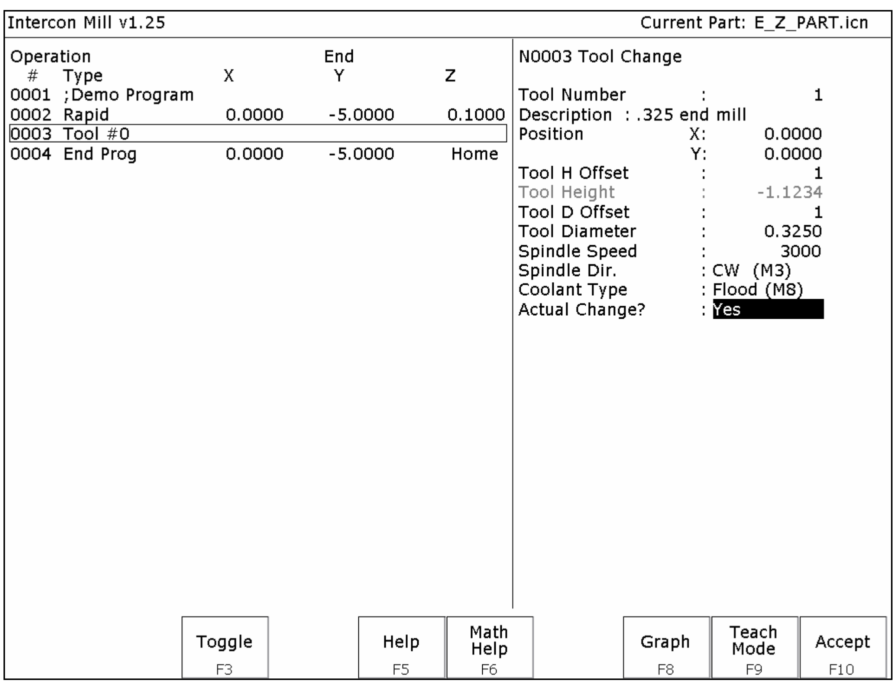
Connect Radius: This field works like the Linear Mill connect radius. It allows for the blending of an arc into the next line or arc operation.

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



Angle <= 180º: 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 - Tool** the following screen appears:



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 CNC10 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 - Toggle** or **SPACE** to toggle between CW, CCW, and Off.

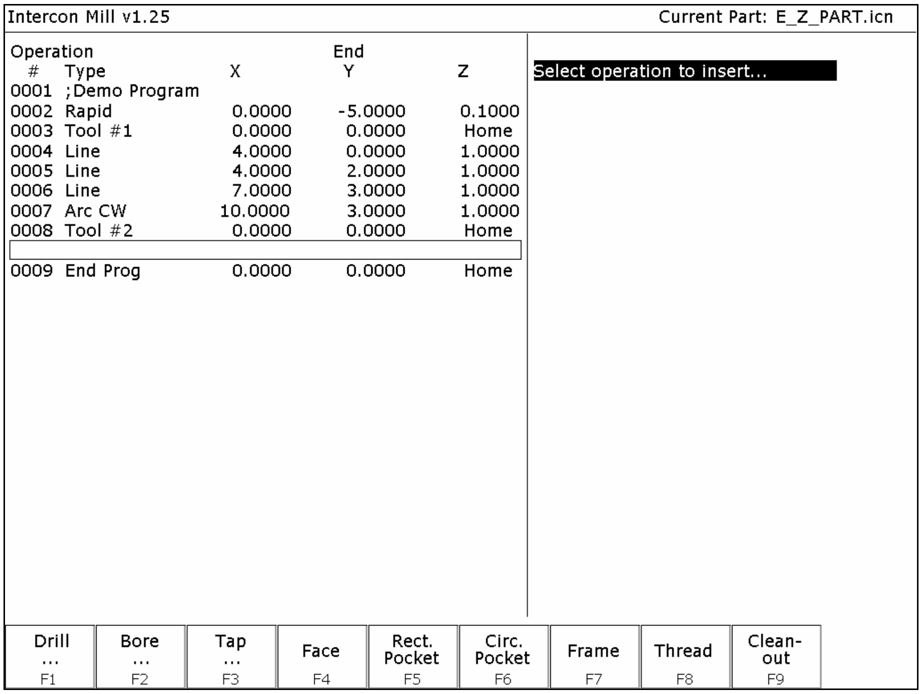
Coolant Type: Type of coolant to activate after the tool change. If this is set to Flood or Mist, the selected coolant system will be started automatically after the tool change. Press **F3 - Toggle** or **SPACE** to toggle between Flood, Mist, and Off.

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 - Accept**, **Page Up**, or **Page Down** when you are finished to accept these values and make changes to the tool library. If you have changed any field other than the Tool Number of the Actual Tool Change field or position, you will make changes to the CNC10 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.

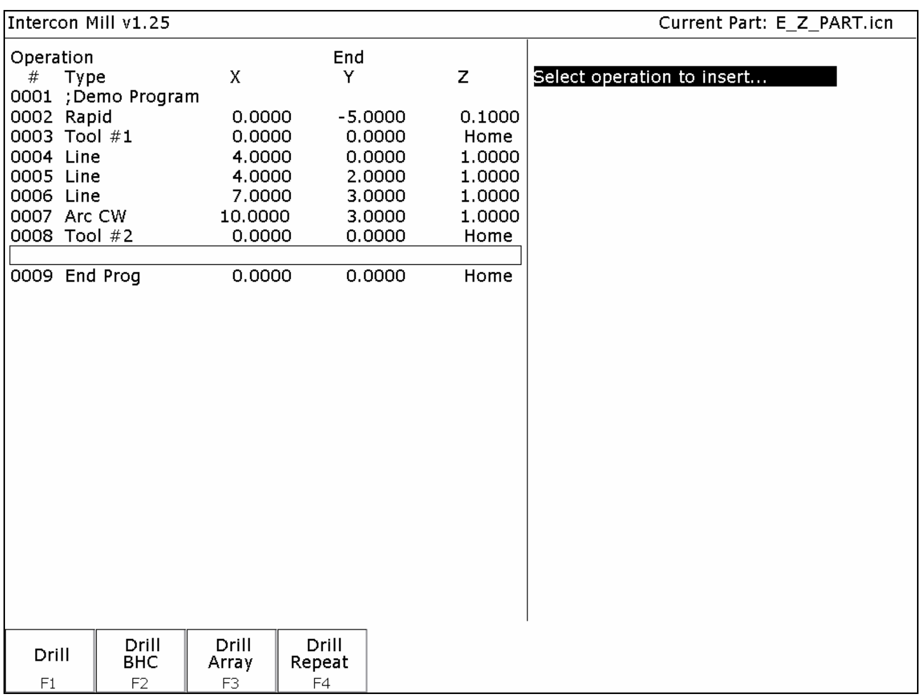
F5 - Canned Cycles

When you choose the Canned Cycle operation by pressing **F5 - Cycles**, the following screen appears:

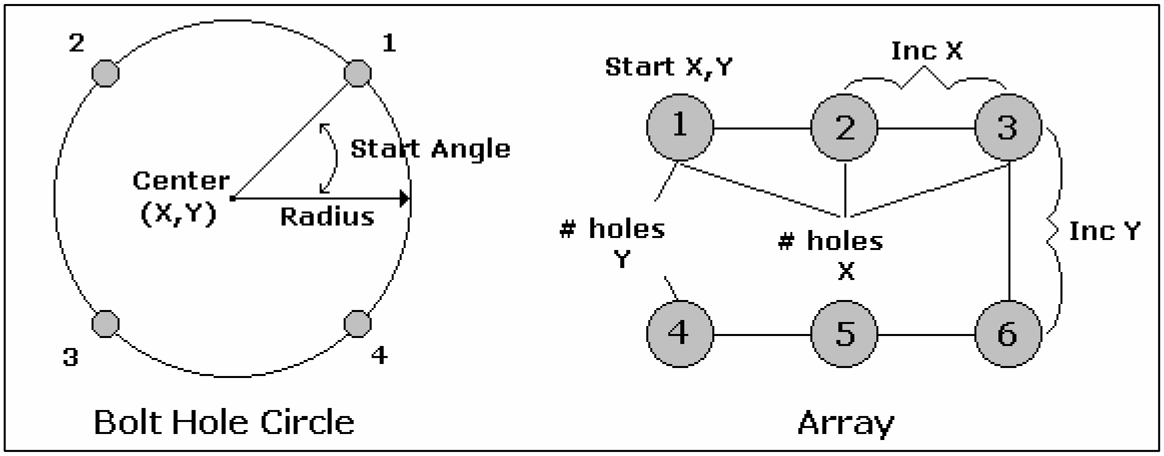


Canned Cycle Introduction #1: Using Pattern and Repeat (Drilling, boring, tapping)

Selecting **F1 - Drill** will give you four choices; **F1 – Drill, F2 – Drill BHC, F3 – Drill Array, or F4 – Drill Repeat**. **F2 - Bore** and **F3 - Tap** will have the same menu selections as drill except they will display Bore or Tap cycles

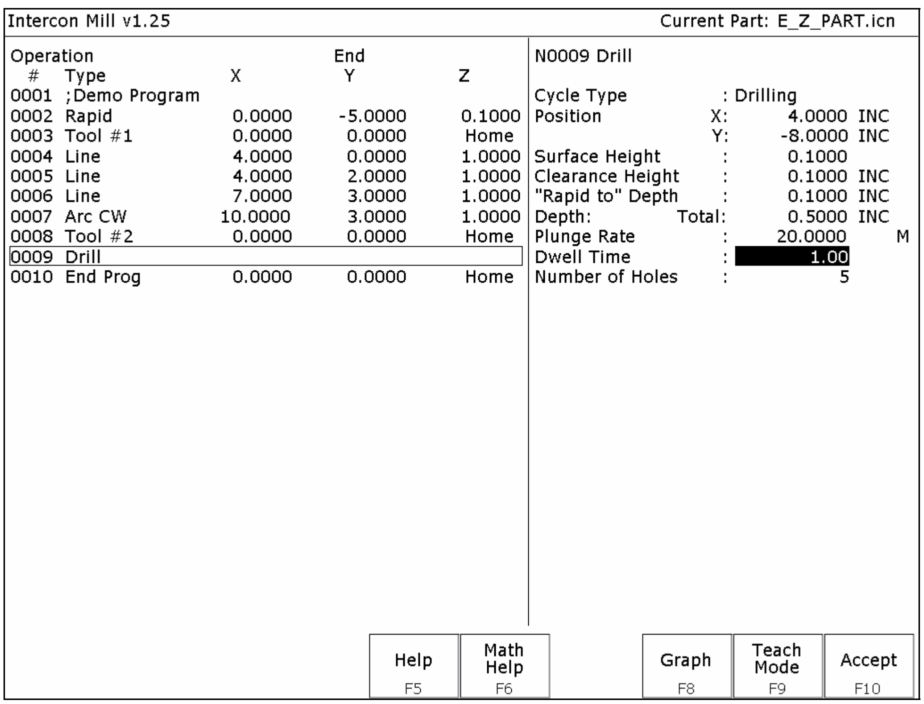


All canned cycle operations using the Drill BHC (Bolt Hole Circle) or Drill Array are identical to their equivalent using the **F1 - Drill** single hole selection. The use of the Drill BHC or Drill Array, however, offers the option to drill more than one hole in a pattern dictated by the new fields in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

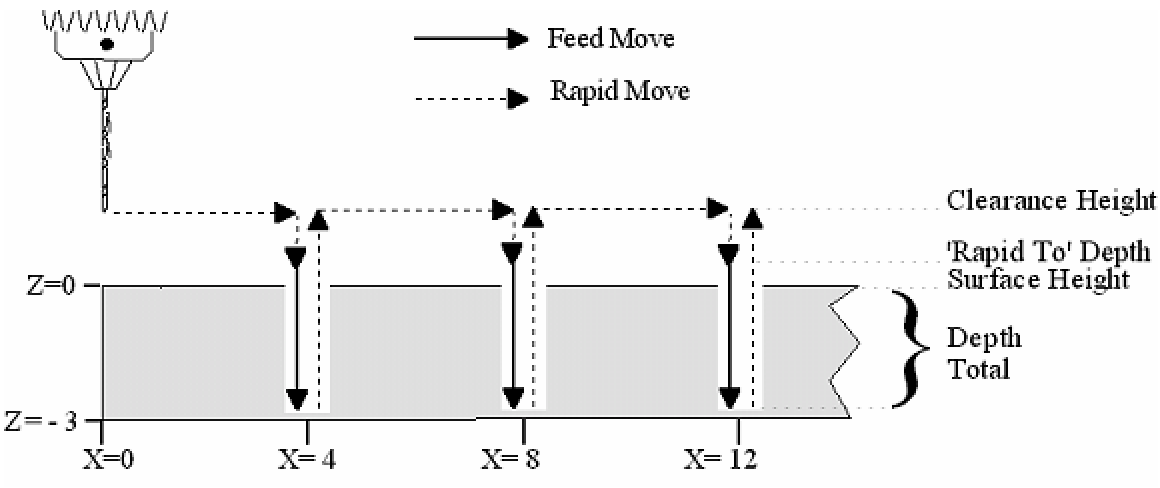


Canned Cycle Introduction #2: Linear Repetition Of Operations (Drilling, Boring, Tapping)

If you want to perform one operation several times in a linear pattern, simply define Position X, Y or both as incremental values. To do this, use the **F1 – Abs/Inc** Key. 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:

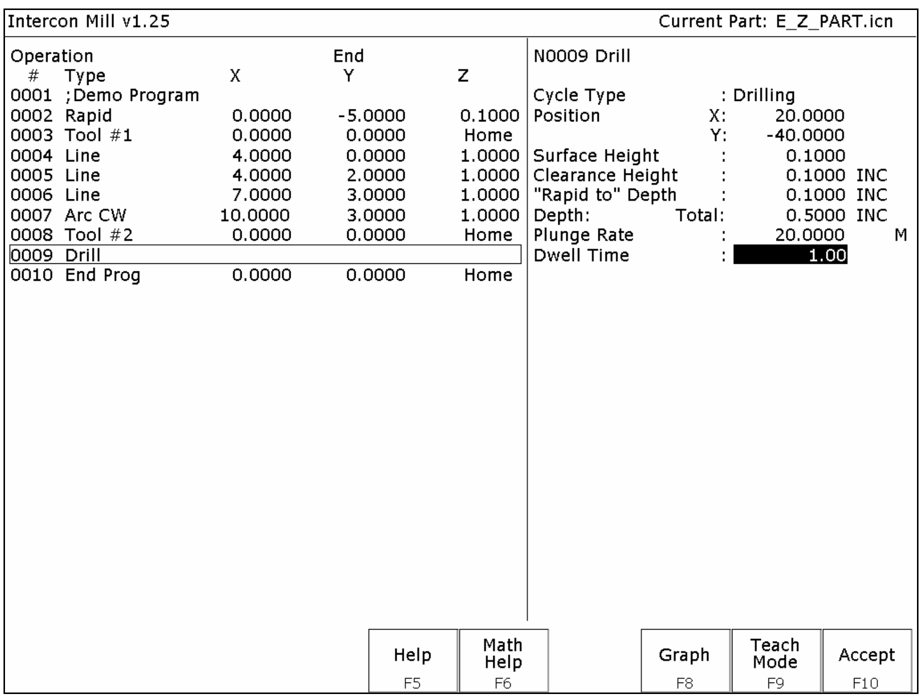


The numbers in the fields on the screen correspond to the following example, shown here graphically:

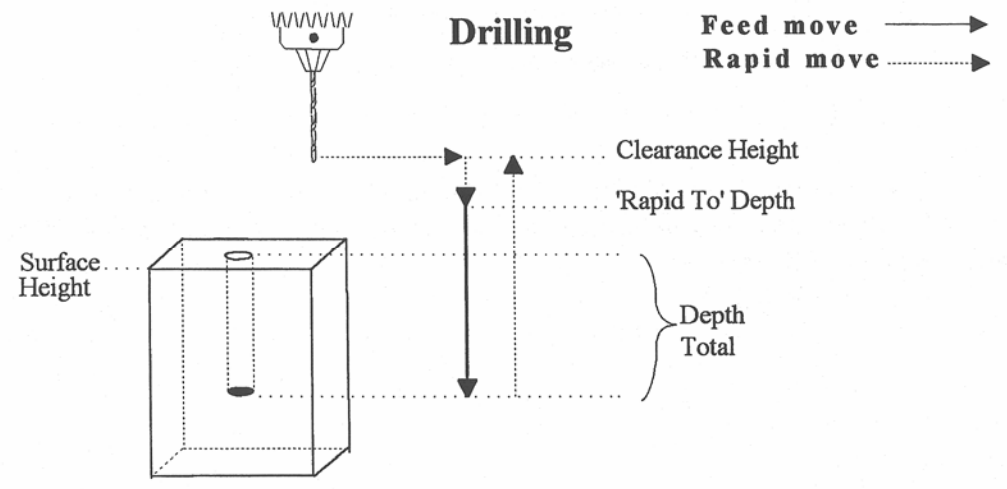


Drilling (F1 in the Canned Cycle Menu: option #1)

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 - Toggle** or **SPACE** toggles between all three. In this section we will examine the first option: Drilling.



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 - Toggle** 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 to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

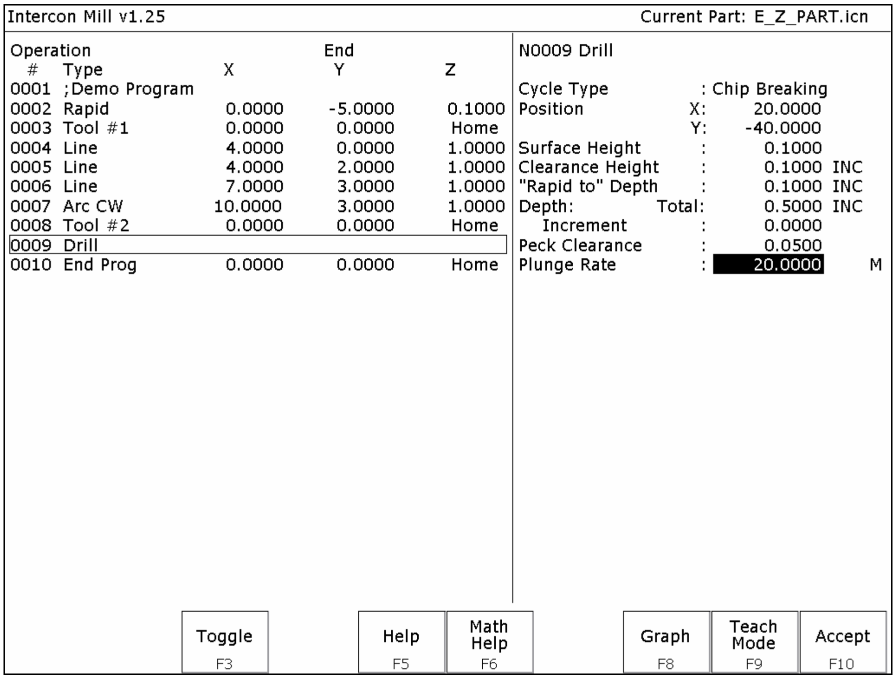
Depth: Total: Depth of hole (incremental) as measured from Surface Height.

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

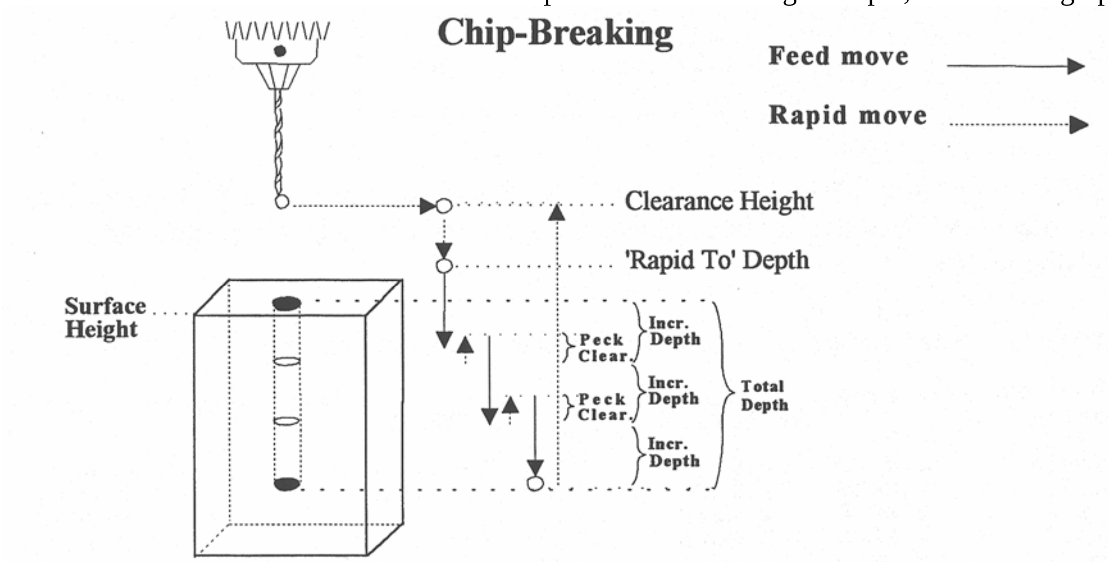
Dwell Time: Delay at bottom of hole before starting ascent.

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

Chip Breaking (F1 in the Canned Cycle Menu: option #2)
If you press **F1 - 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 - Toggle** or **SPACE** toggles between all three. In this section we will examine the second option: Chip Breaking.



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 - Toggle** 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 to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

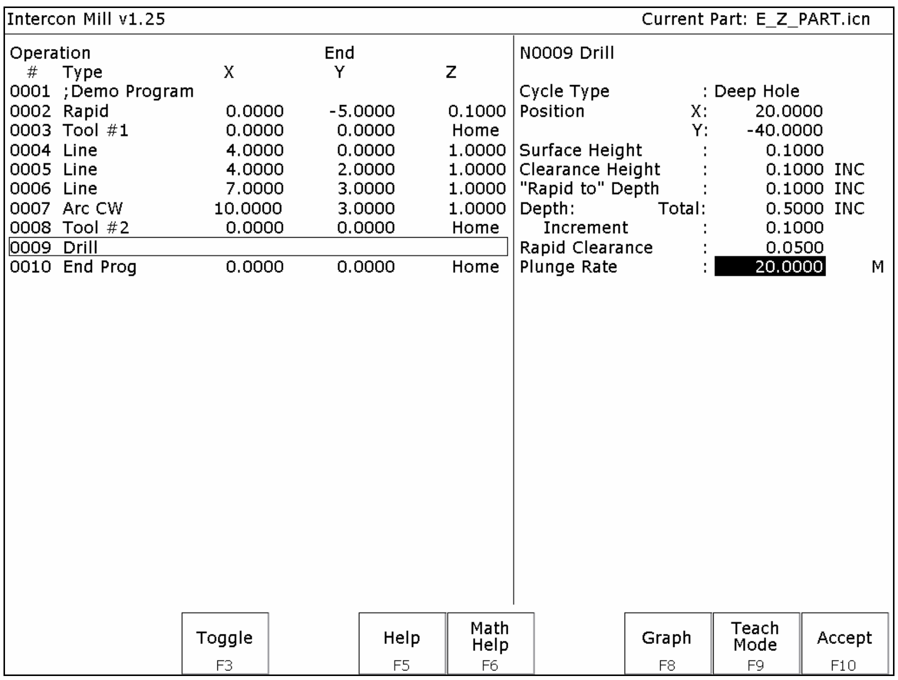
Depth: Total: Depth of hole (incremental) as measured from Surface Height.

Depth: Increment: Depth of each individual peck.

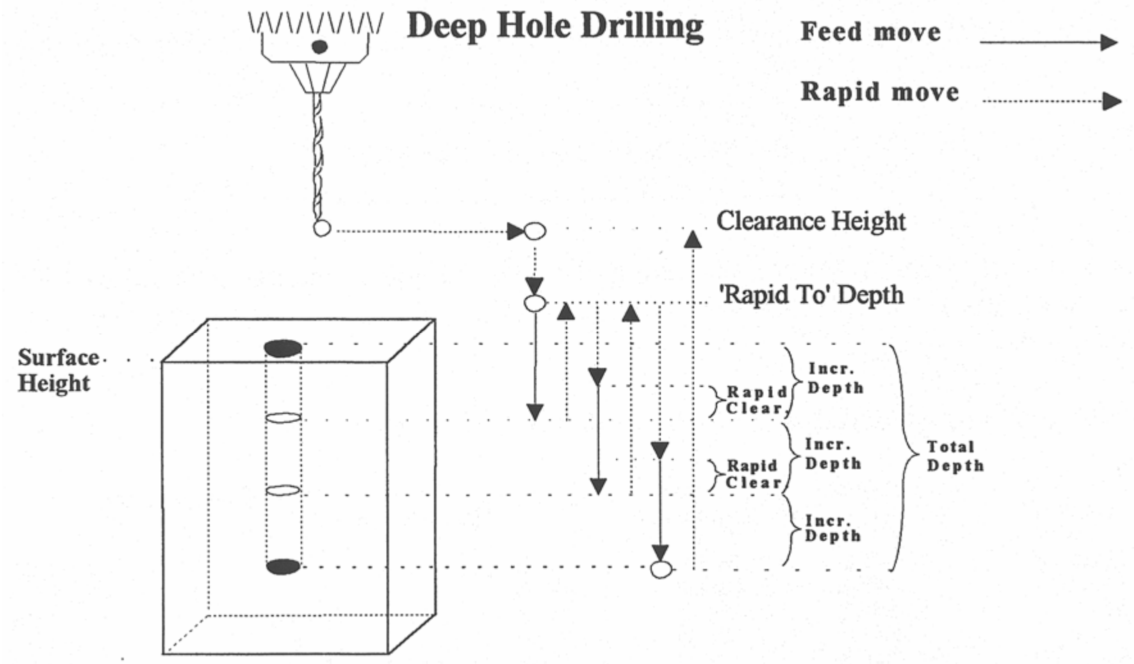
Peck Clearance: Distance the tool retracts before drilling the next peck.

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

Deep Hole Drilling (F1 in the Canned Cycle Menu: option #3)
If you press **F1 - 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 - Toggle** or **SPACE** toggles between all three. In this section we will examine the third option: Deep Hole drilling.



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 to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

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

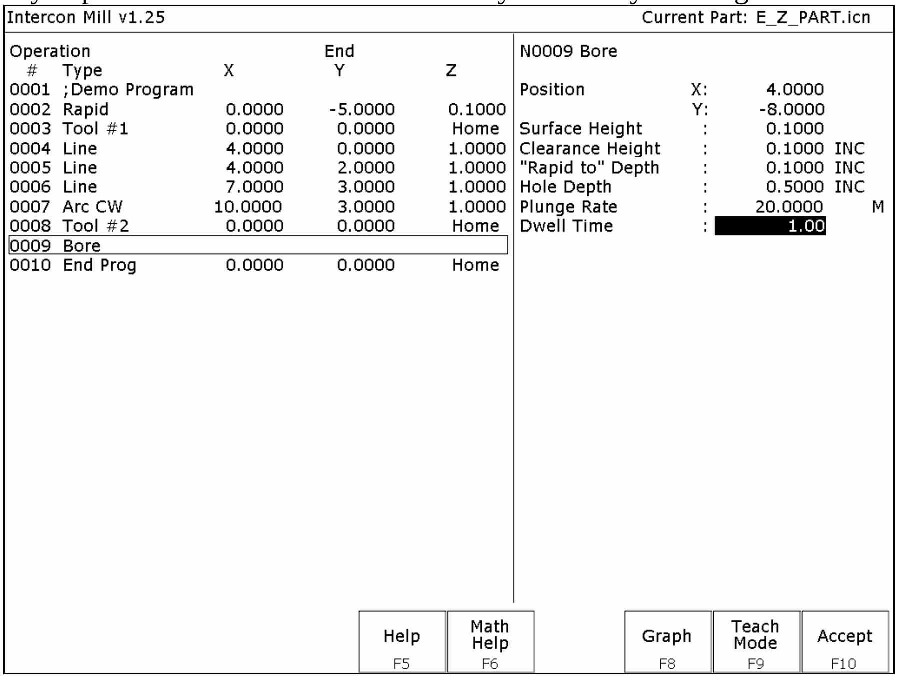
Depth: Increment: Depth of each individual step of the drilling.

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

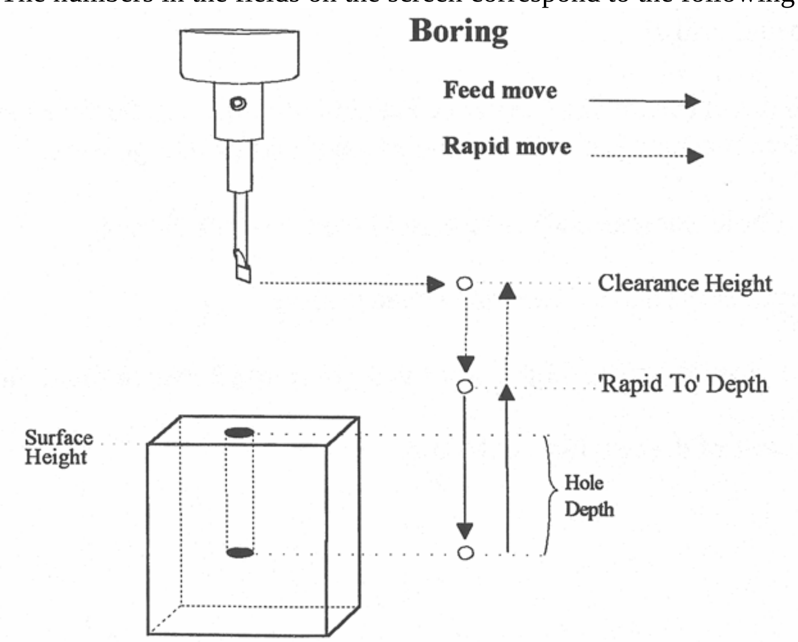
Plunge Rate: Z-axis speed of descent during drilling.
The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol. The slave plunge rate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

Boring (F2 in the Canned Cycle Menu)

If you press **F2 - Bore** from the Canned Cycle Menu you will gain access to the boring operation:



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.

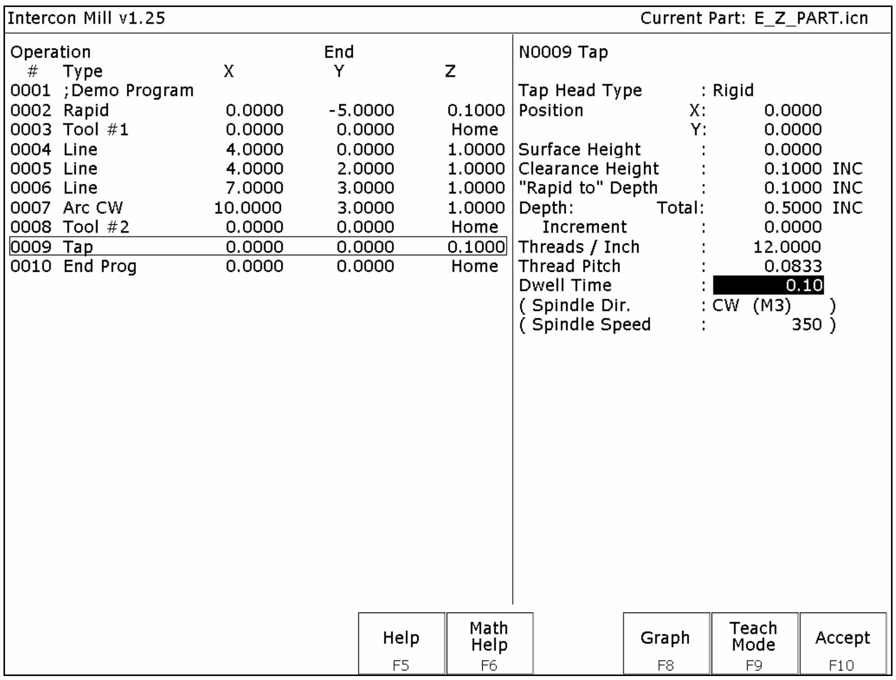
'Rapid To' Depth: The depth to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

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

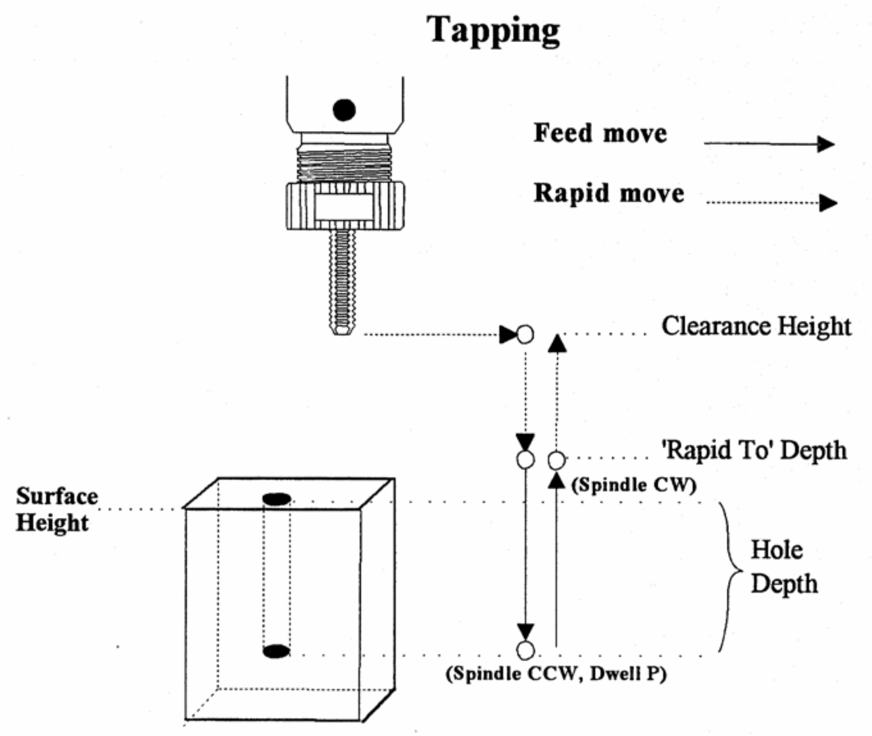
Plunge Rate: Z-axis speed of descent during drilling.
The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the “M” symbol or if it is fixed it will have the “F” symbol . The slave plunge rate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

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

Tapping (F2 in the Canned Cycle Menu)
If you press **F2 - Tap** from the Canned Cycle Menu you will gain access to the tapping operations:



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



Where:

<u>Tap Head Type:</u> 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.

<u>Spindle Direction:</u> 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.

<u>Position:</u> 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)

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

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

<u>'Rapid To' Depth:</u> The depth to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

<u>Depth Total:</u> Depth of hole (incremental) as measured from Surface Height.

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

<u>Threads / Unit:</u> 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).

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

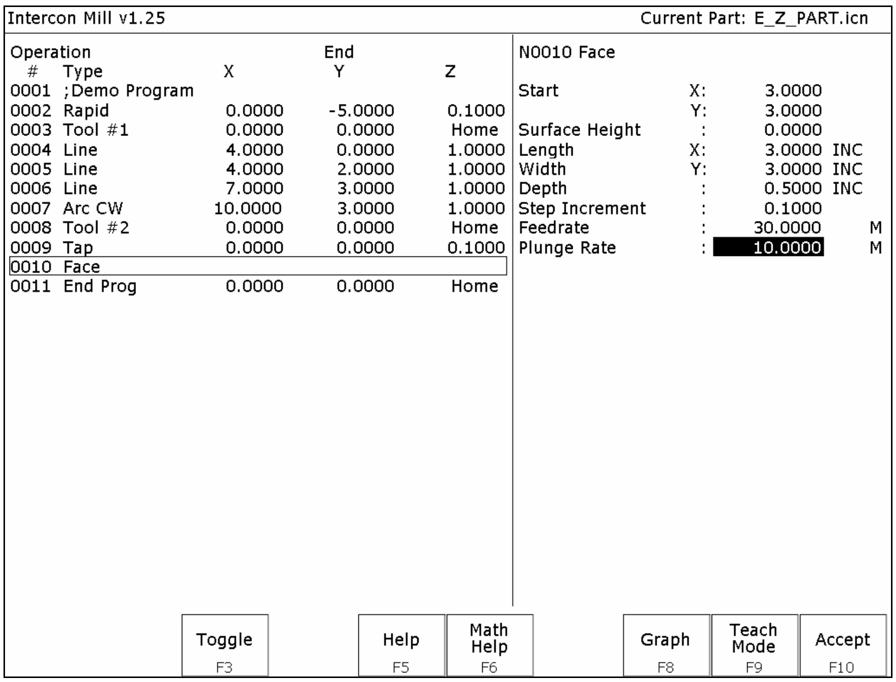
* **WARNING:** The spindle speed must be set before performing this operation.

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.

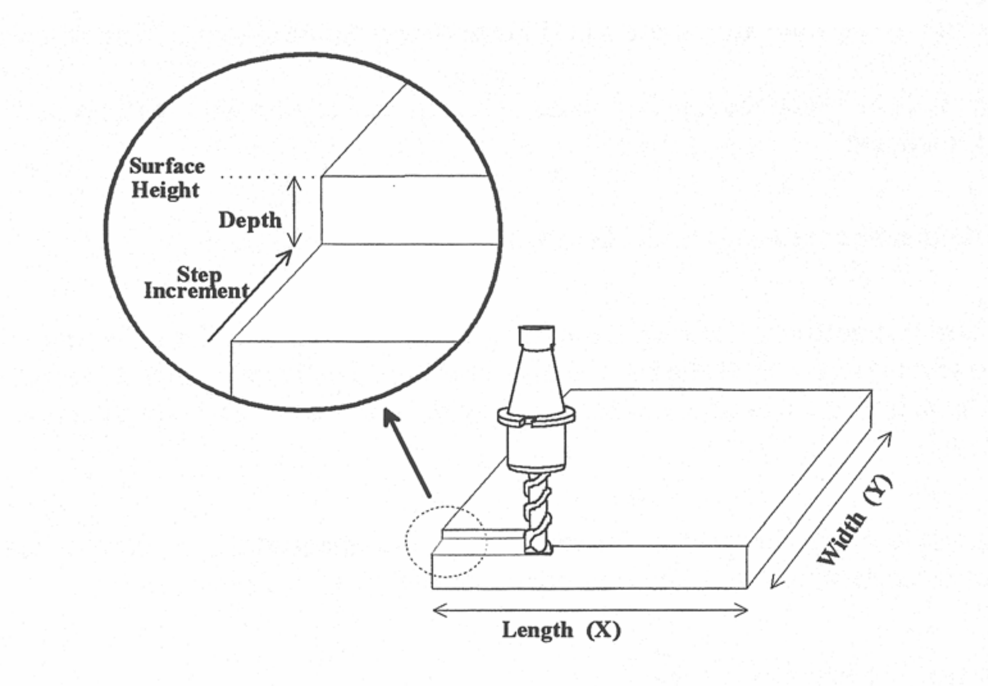
* **NOTE:** When using low gear for tapping, the spindle may turn opposite the direction specified. The operator is responsible for setting the correct spindle speed and direction.

Facing (F4 in the Canned Cycle Menu)

If you press **F4 - Face** at the Canned Cycle Selection Menu, the following screen is displayed:



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



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

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

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

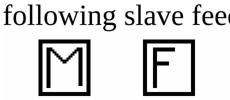
Width: Y-axis dimension of the area to be faced. If a negative value is entered for the width, the facing will occur in the negative Y-axis direction from the Y-axis start position; otherwise, facing will occur in the positive Y-axis direction from the Y-axis start position.

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

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

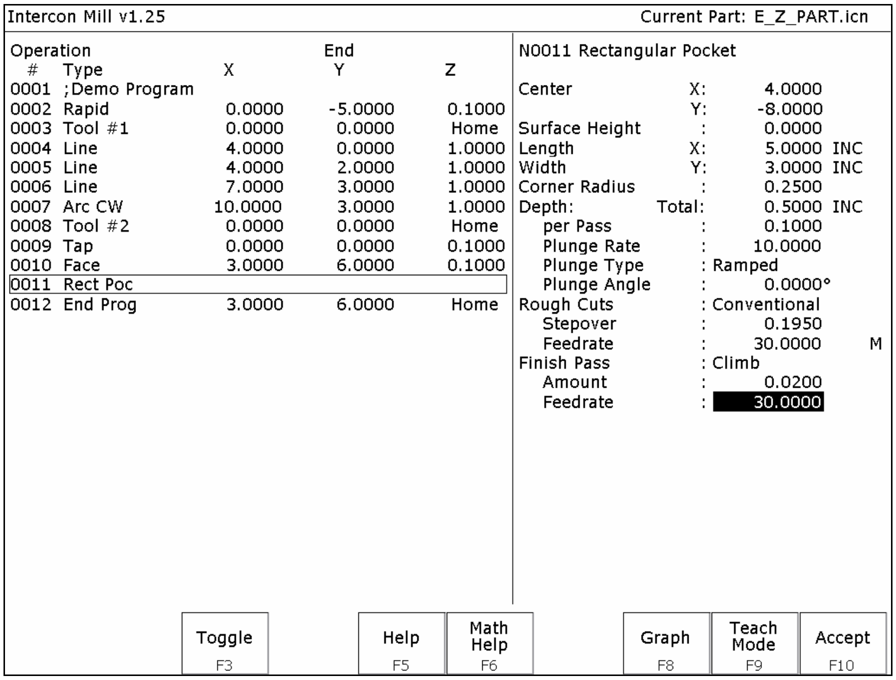
Plunge Rate: Z-axis speed of descent during facing.
The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the “M” symbol or if it is fixed it will have the “F” symbol. The slave plunge rate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

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

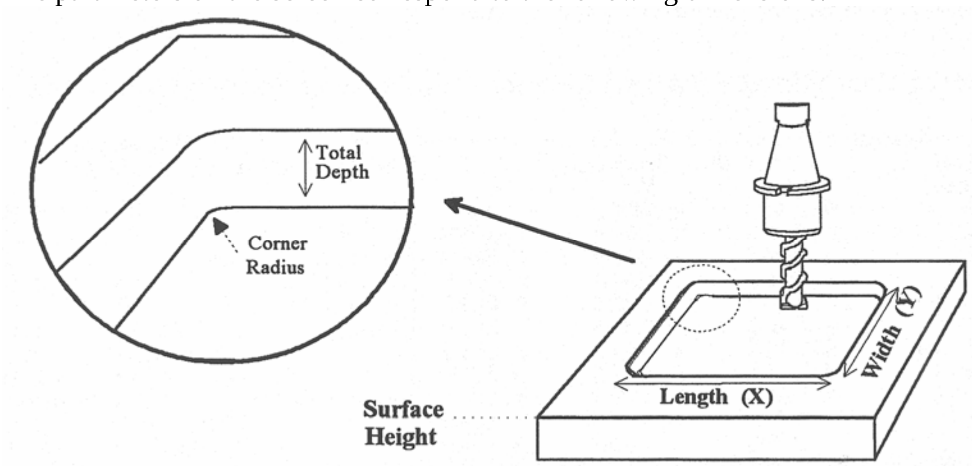


Rectangular Pocket (F5 in the Canned Cycle Menu)

Pressing **F5 - Rect. Pocket** from the Canned Cycle Selection Menu displays the following screen:



The parameters on the screen correspond to the following dimensions:



Where:

<u>Center:</u> X and Y coordinates of the center of the Rectangular Pocket.

<u>Surface Height:</u> Z-axis position from which each incremental depth is measured.

<u>Length:</u> X-axis dimension of the rectangular pocket.

<u>Width:</u> Y-axis dimension of the rectangular pocket.

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

<u>Depth: Total:</u> Total depth of the rectangular pocket.

<u>Depth: Per Pass:</u> Depth of each individual pass.

<u>Depth: Plunge Rate:</u> 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: This field means nothing if the Plunge Type is Straight.

Rough Cuts: Selects type of rough cut: conventional or climb. Use **F3 – Toggle** or **SPACE** to toggle between them.

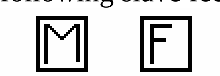
Rough Cuts: Stepover: Amount of material removed by cutter during each pass around the pocket.

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

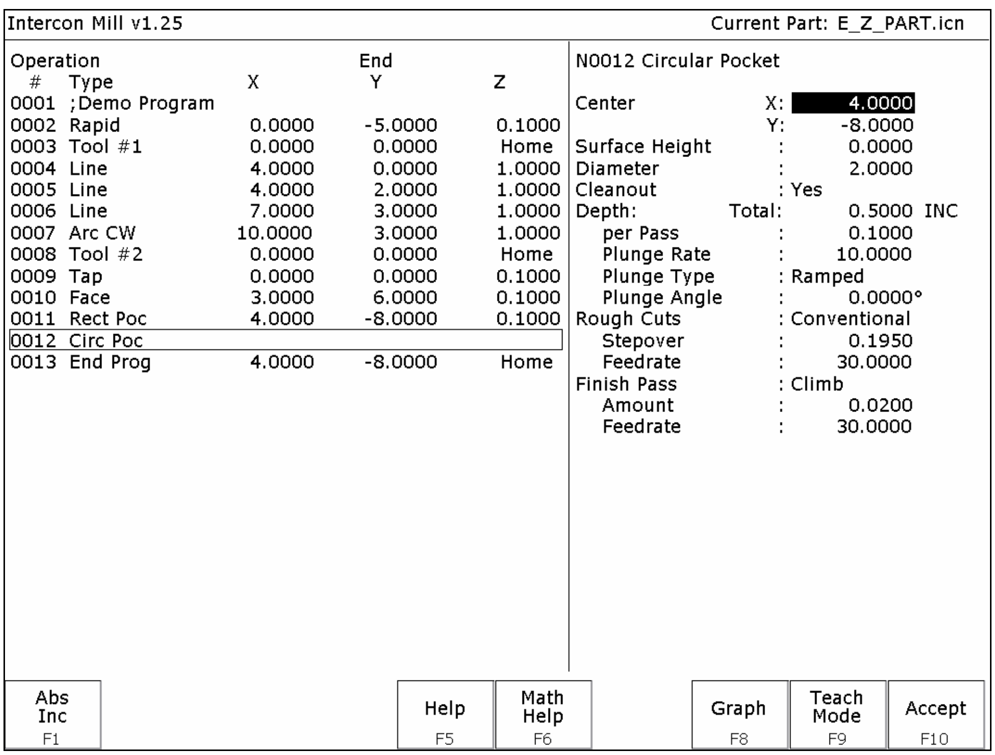
Finish Pass: Selects type of finish pass: climb, conventional or none at all. Use **F3 - Toggle** or **SPACE** to toggle between them.

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

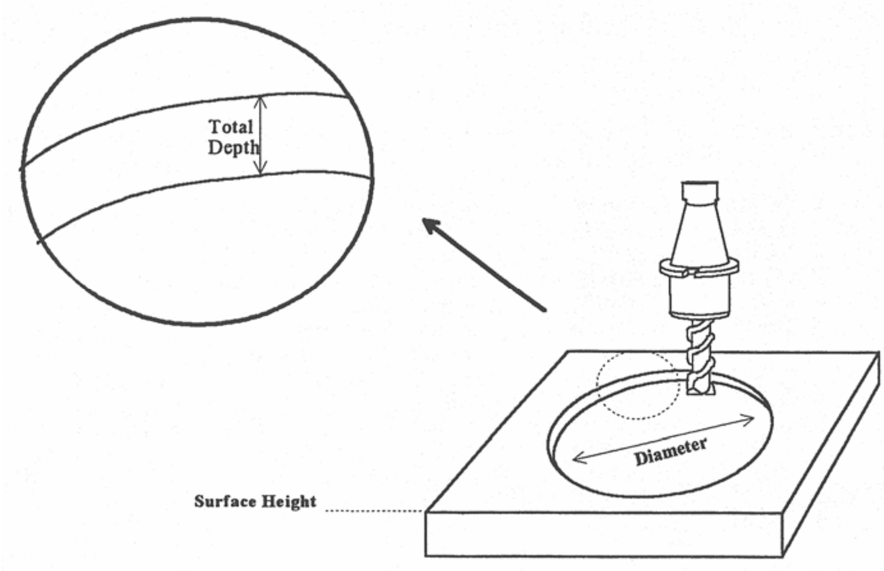
Finish Pass: Feedrate: Speed at which cutter performs finish pass.
The feedrate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the feedrate field. If the feedrate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol shown below. The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



Circular Pocket (F6 in the Canned Cycle Menu)
When you press **F6 - Circ. Pocket** from the Canned Cycle Selection Menu, this screen is displayed:



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, then all the material in the pocket will be removed. If cleanout is No, then all the material will not be removed. The cutter starts in the center of the pocket 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: This field means nothing if the Plunge Type is Straight.

Rough Cuts: Selects type of rough cut: conventional or climb. Use **F3 - Toggle** or **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.

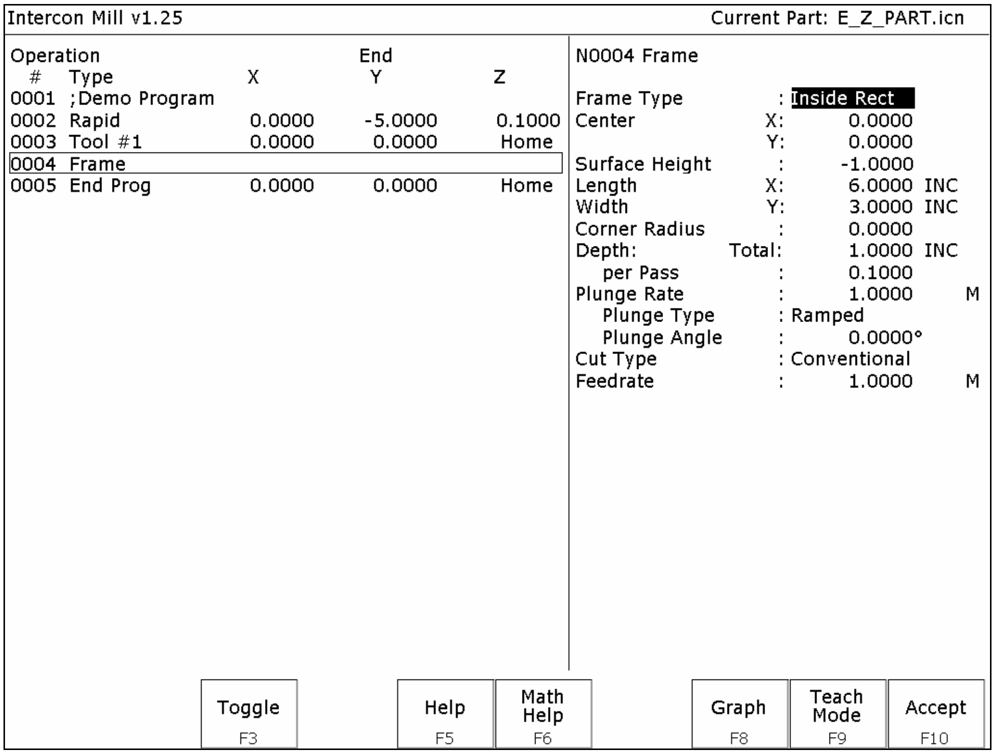
Finish Pass: Selects type of finish pass: climb, conventional or none at all. Use **F3 - Toggle** or **SPACE** to toggle.

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

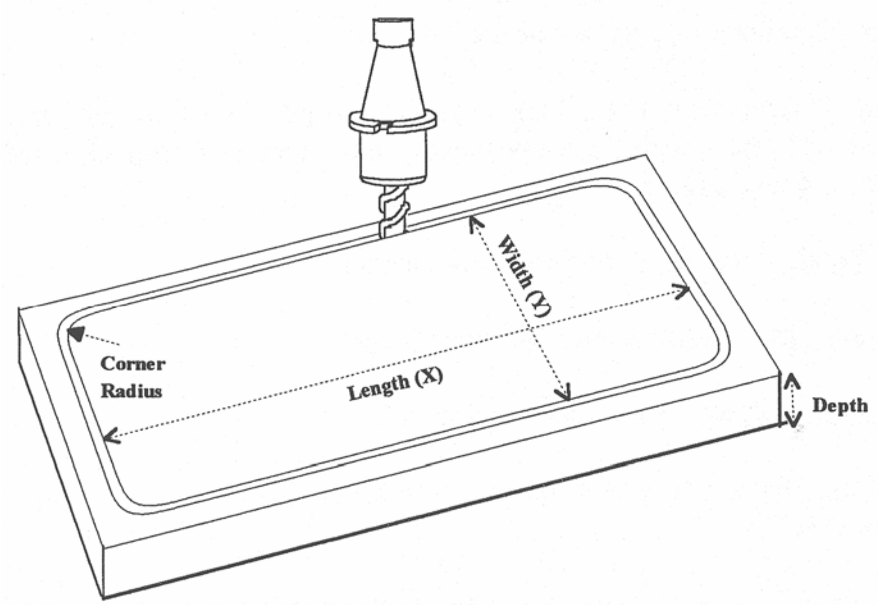
Finish Pass: Feedrate: Speed at which cutter performs finish pass.

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

When you press **F7 - Frame** from the Canned Cycle Selection Menu, the following screen is displayed:



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



Where:

Frame Type: Selects Inside Rectangle, Outside Rectangle, Inside Circle, and Outside Circle. Press **F3 - Toggle** or **SPACE** to toggle between them.

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

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

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

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

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

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

Depth: Total: Total depth of the frame mill.

Depth: Per Pass: Depth of each individual pass.

Plunge Rate: Z-axis speed of descent.

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

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

Entrance Type: Selects type of entrance: Arc On or Arc Off, use **F3 - Toggle** or **SPACE** to toggle between them. (Circular frame only.)

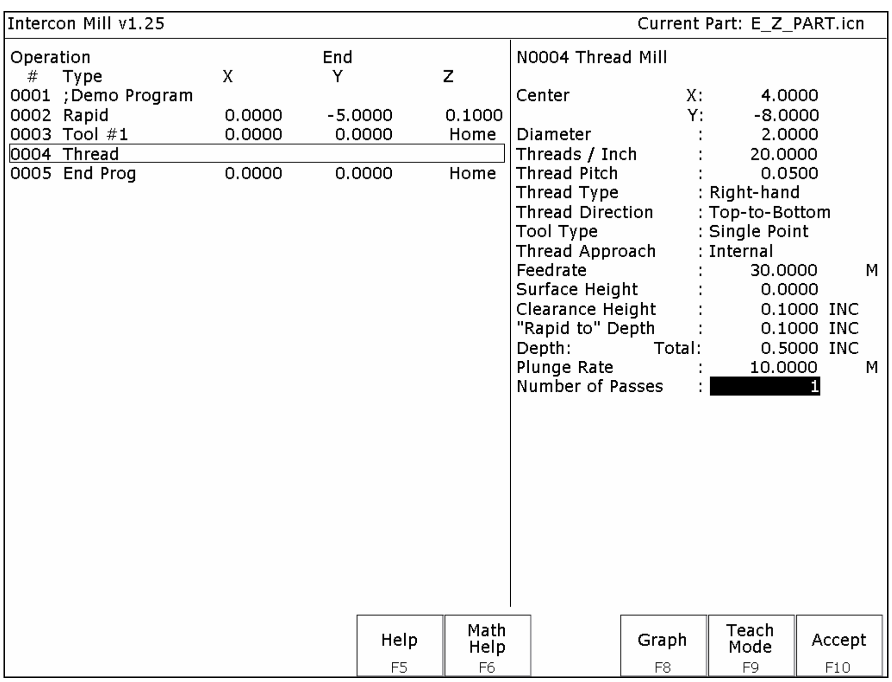
Cut type: Selects type of cut: conventional or climb, use **F3 - Toggle** or **SPACE** to toggle between them.

Feedrate: Speed at which the cutter performs frame mill.

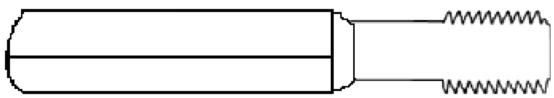
NOTE: To make a circular frame mill of radius R, specify R as the Corner Radius and set the Length and Width parameters equal to 2 x R.

Thread Milling (F8 in the Canned Cycle Menu)

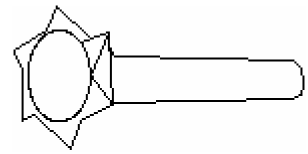
When you press **F8 - Thread** from the canned cycle menu, the following screen is displayed:



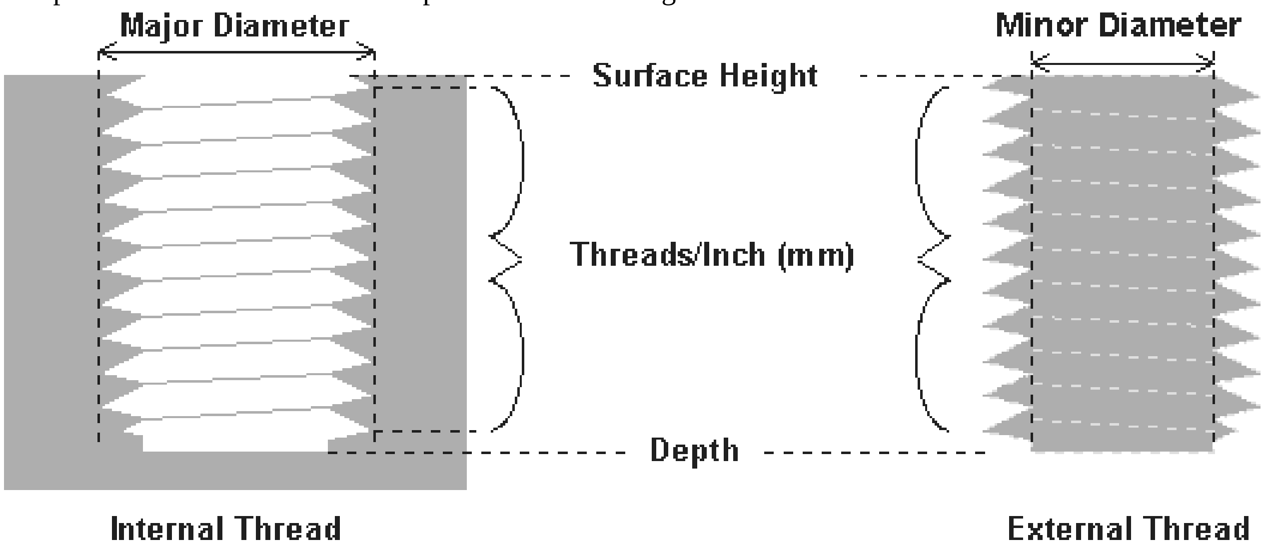
Multiple Thread Mill



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 internal thread milling and minor diameter for external 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.

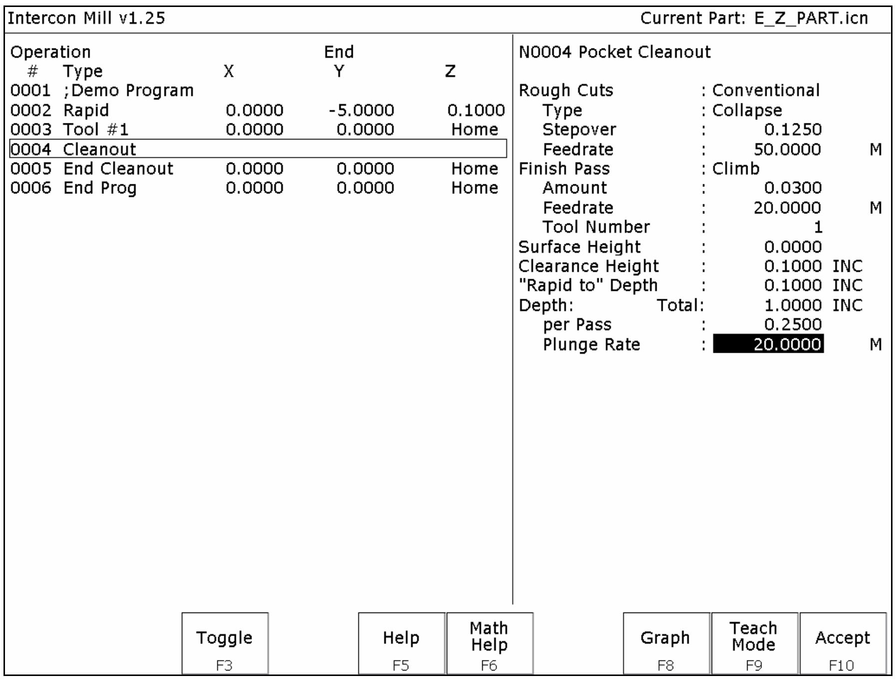
'Rapid to' Depth: The depth to which the cutter rapid moves before beginning to thread mill at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

Depth: The total depth of the thread.

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

Cleanout (F9 in the Canned Cycle Menu)

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



Where:

Rough Cuts: Selects type of rough cut. Use **F3-Toggle** or **SPACE** to toggle between Conventional and Climb.

Type: Selects type of cleanout to use. The choices are Collapse, which cleans out the pocket from the outside in, or Expand, which cleans out the pocket from the inside out. You can switch between the two by using the **F3-Toggle** or **SPACE** keys

Stepover: The distance between each step in the pocket cleanout. This value cannot be greater than 50% of the tool diameter.

Feedrate: Speed at which cutter performs rough cuts.

Finish Pass: Selects type of finish pass. Use **F3-Toggle** or **SPACE** to toggle between Conventional, Climb, or None.

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

Feedrate: Speed at which cutter performs finish pass.

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

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

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

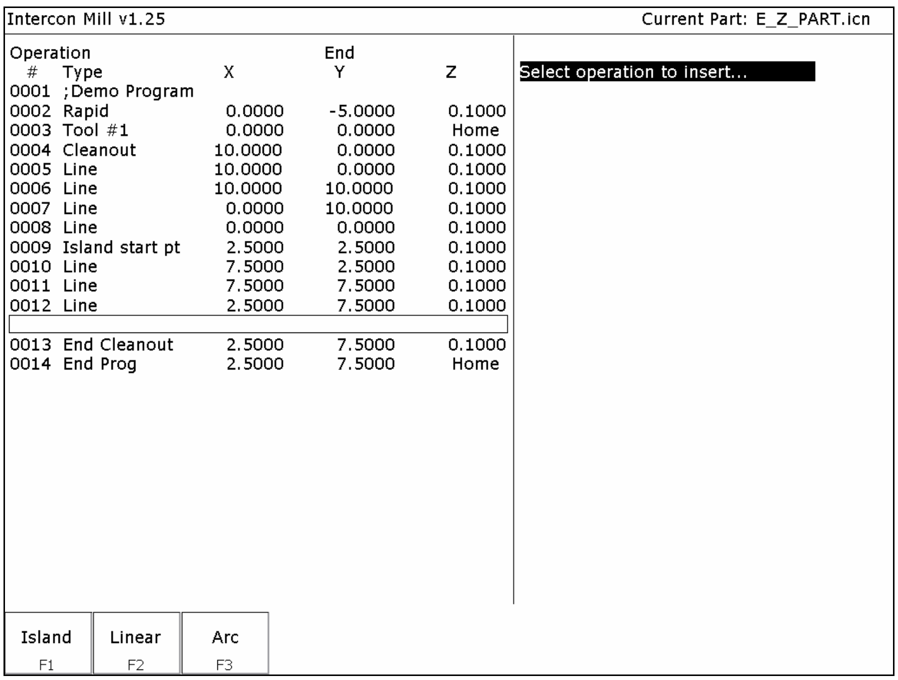
'Rapid To' Depth: The depth to which rapid positioning moves will be made to when moving the Z axis downward.

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

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

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

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



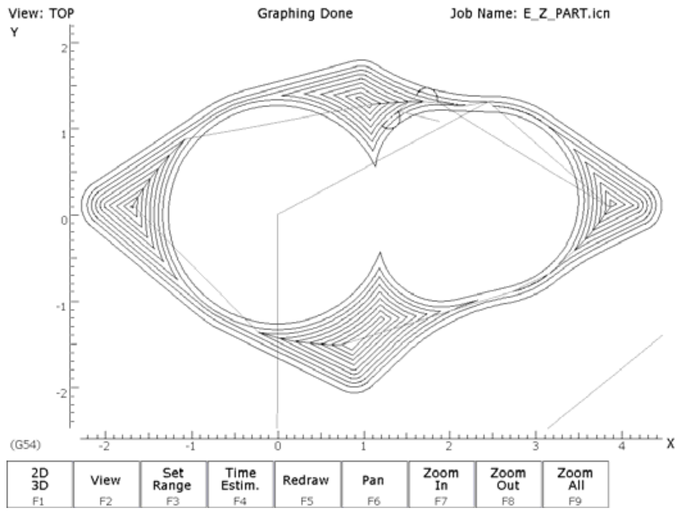
Key points about the Cleanout cycle:

- When creating a pocket the first move in your cleanout cycle **must** be a linear move.
- If the profile contour does not end at the start point, a linear segment will automatically be inserted to close the pocket.
- The last line of the contour will not include a connecting radius to the starting point.

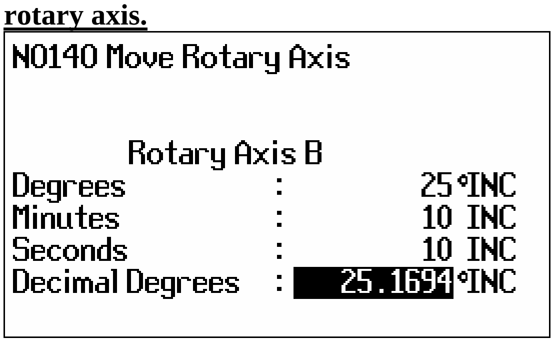
Once you have defined the specifics of your cleanout cycle, the shape of your pocket will be defined through a series of lines and arcs by choosing the **F2-Linear** and **F3-Arc** options in the cleanout cycle.

F1-Island (Island Avoidance)
Once you have defined a pocket in the cleanout cycle. There may be areas or islands that you don't want cleaned out. To create an island press **F1-Island**, enter the starting point of the island and then use **F2-Linear** and **F3-Arc** to create the island.

See an example of a completed cleanout backplot below.



F6 - Other
Choosing **F6 - Other** will display the following operations that may be inserted.
<u>F1 - Comment</u> - Enter a comment, up to 35 characters long, which will be displayed in the generated CNC program.
<u>F2 - Spindle</u> - Change the actual state of the spindle. Press **F3 - Toggle** or **SPACE** to toggle between CW, CCW, and OFF.
<u>F3 - Coolant</u> - Change the actual state of the coolant. Press **F3 - Toggle** or **SPACE** to toggle between FLOOD, MIST, and OFF.
<u>F4 - Clamp</u> - Turn the Clamp ON and OFF. Press **F3 - Toggle** or **SPACE** to change the clamp state.
<u>F5 - Z Home</u> - Send the Z-axis to its home position.
<u>F6 - M & G Code</u> - 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 turning the spindle off during a cut.
<u>F7 - Rotary - Insert a rapid rotary move. This operation requires a rotary fourth axis. 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.</u>



<u>F8 - Import DXF</u> - Insert Intercon operations created from objects in DXF files. For more information, refer to the “Importing DXF files” section later in this chapter.

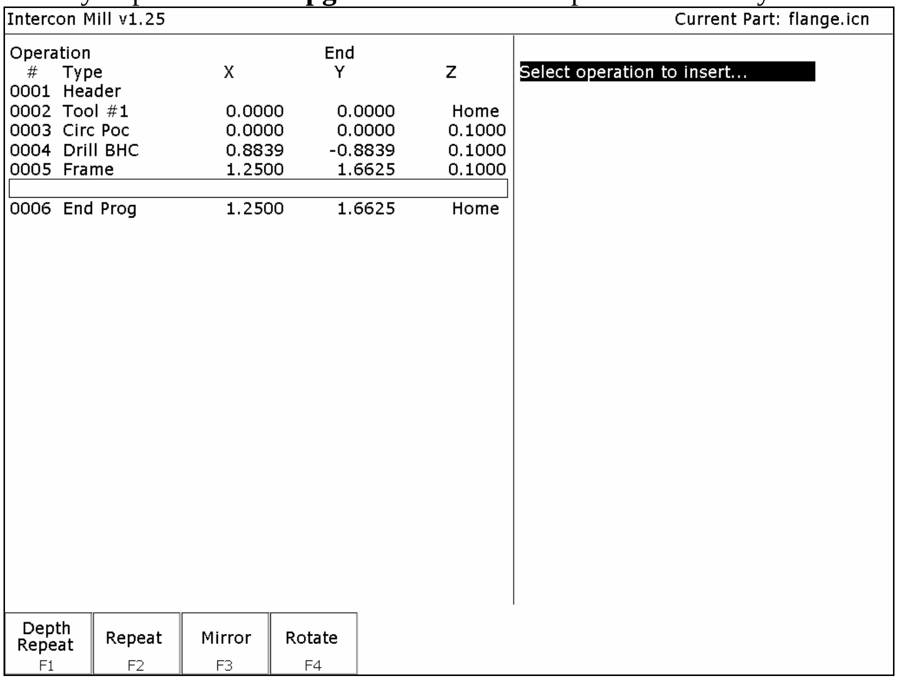
F7 - Cutter Compensation
Pressing **F7 - Cutter Comp** from the Insert Operation screen, will insert a cutter compensation command. Press **F3 - Toggle** or **SPACE** to select cutter compensation Left, Right, or Off. Cutter compensation may be used with Linear Mill, Frame Mill, and Rapid Traverse operations. For details on using cutter compensation, see Chapter 12 page 12-7 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.

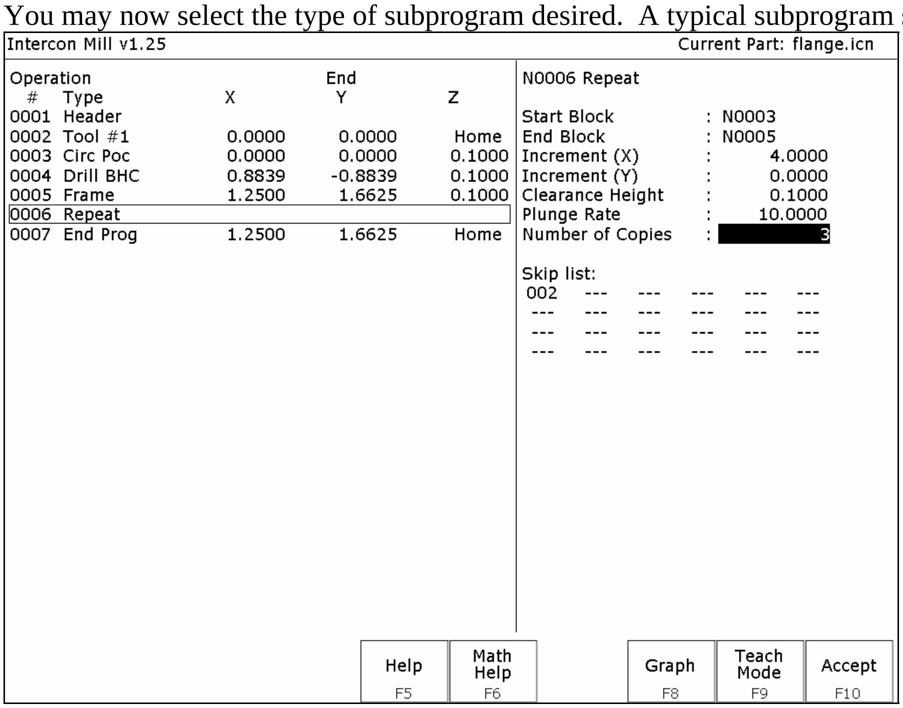
F9 - Subprograms

Intercon subprograms allow you to make additional copies of a programmed contour. The copies may be repeated in the x/y axes, depth repeat, rotated, or even 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 **F3 - Insert** 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 occur first.

When you press **F9 - Subpgm** from the Insert Operation screen you will see the Insert Subprogram screen:



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



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.

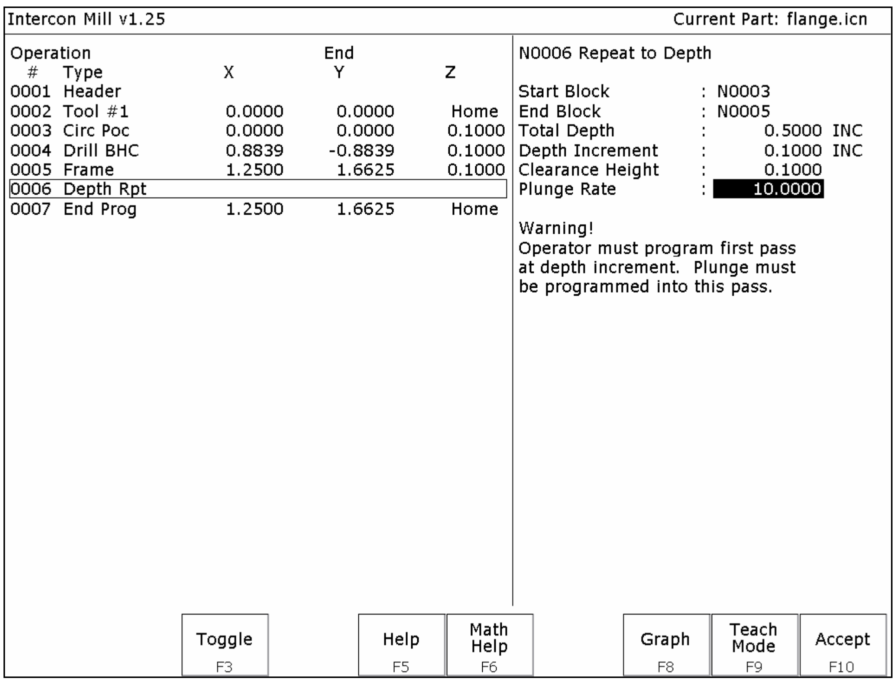
End Block: Selects the last operation in the block of operations to repeat. Again, this operation must lie prior to place in your program where you are trying to repeat operations, but not precede the start block.

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 - Z 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)



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.

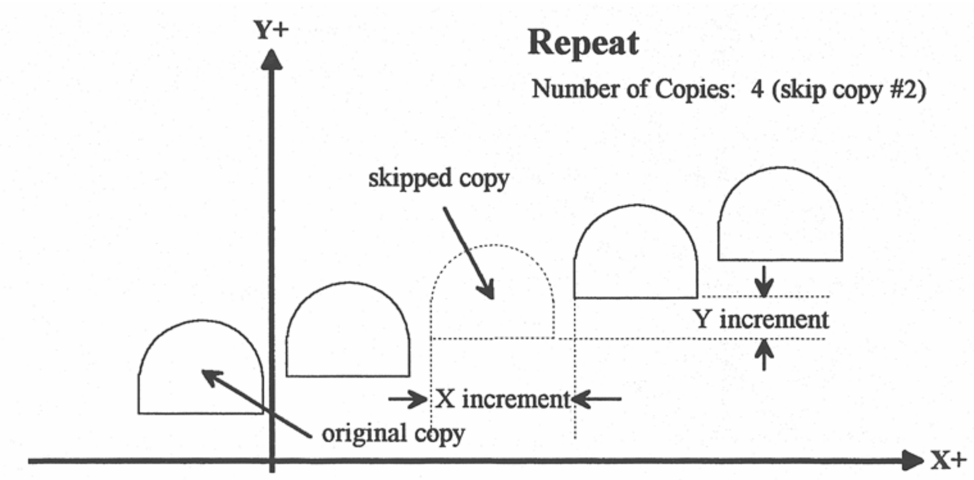
Depth Increment: Specifies the distance to drop each time the contour is repeated. This is a positive value that may not exceed the total depth of the operation.

When you have finished entering the required parameters, press **F10 - Accept** to accept them. An operation labeled "Depth 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 "Depth Rpt" operation, then press **ENTER**.

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

Linear Repeat (F2 in the Insert Subprogram Menu)

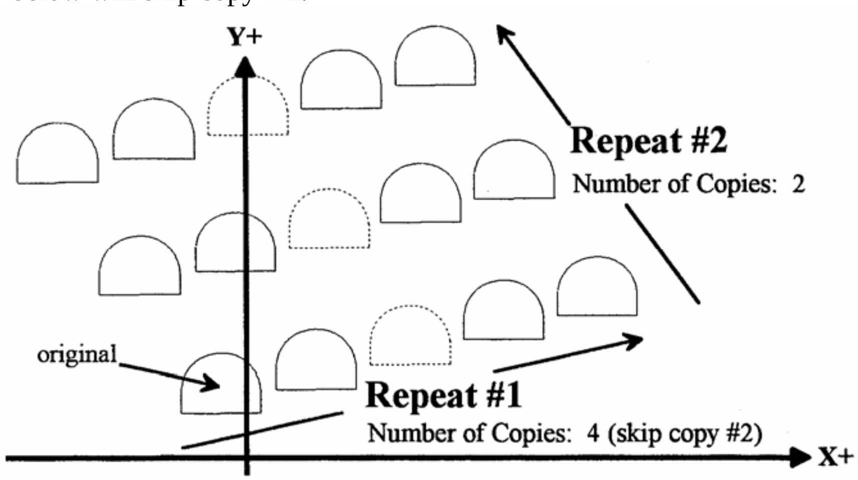
The Linear Repeat feature is useful for repeating a part contour multiple times along a straight line. The contour formed by these operations may either be closed or open.



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

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

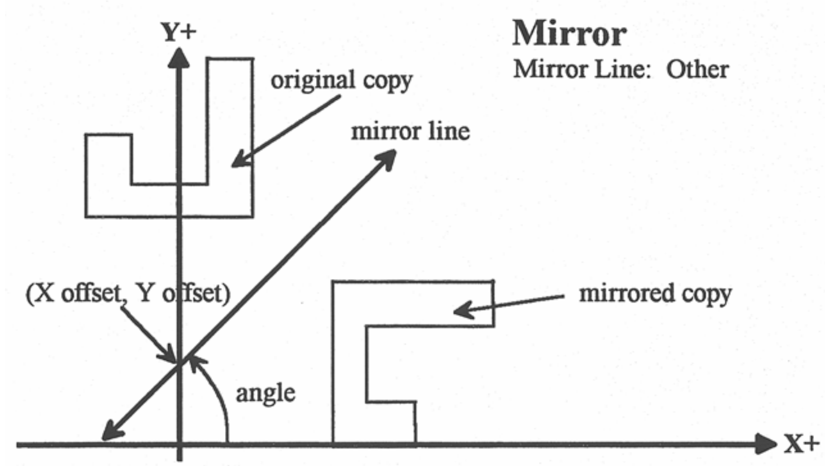
Skip List: List of copies that are skipped. Enter the number or numbers of the copies that you wish to skip. In the example below will skip copy # 2.



* 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).

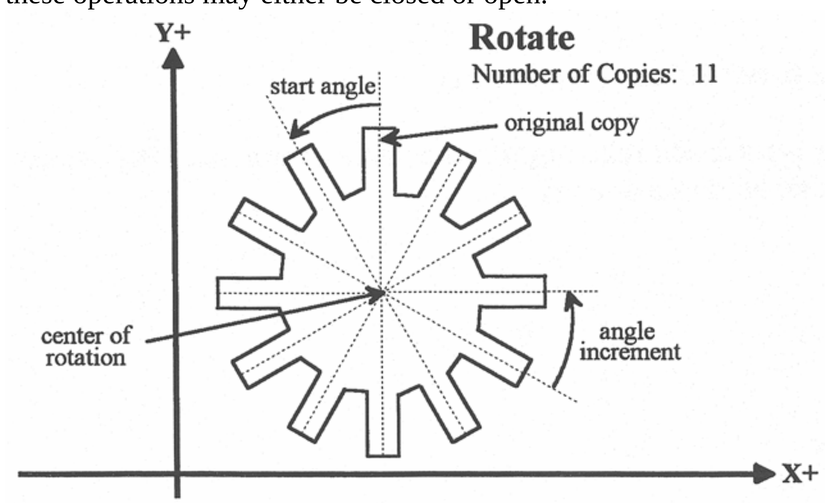
X Offset: Specifies the X coordinate on the Mirror Line. This field will not be visible for a horizontal mirror line.

Y Offset: Specifies the Y coordinate on the Mirror Line. This field will not be visible for a vertical mirror line.

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. Must have a value larger than 1 in the number of copies.

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.

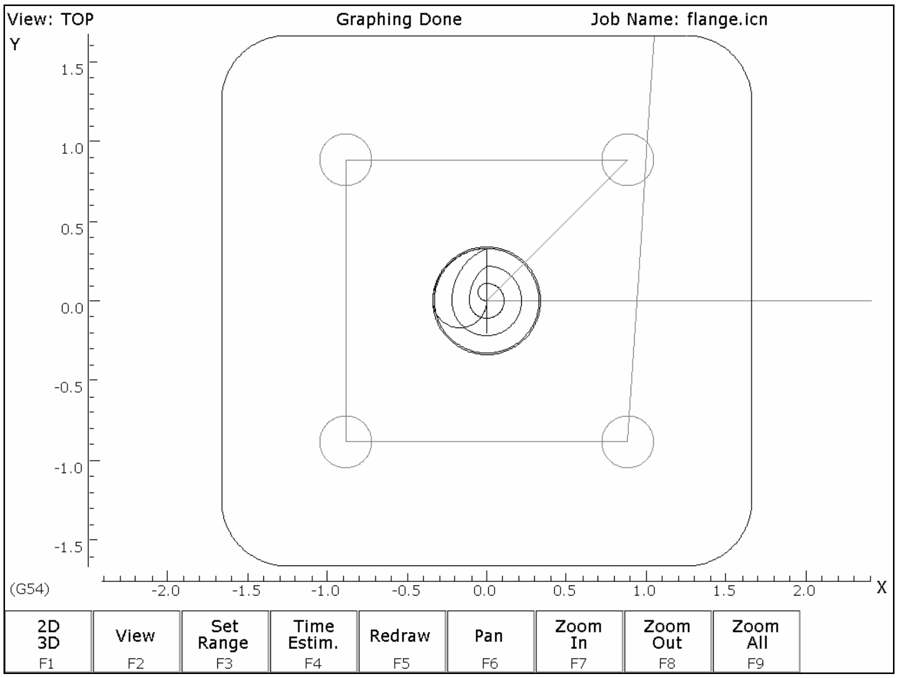
Skip List: List of copies that are skipped. Enter the number or numbers of the copies that you wish to skip.

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

Graphics

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

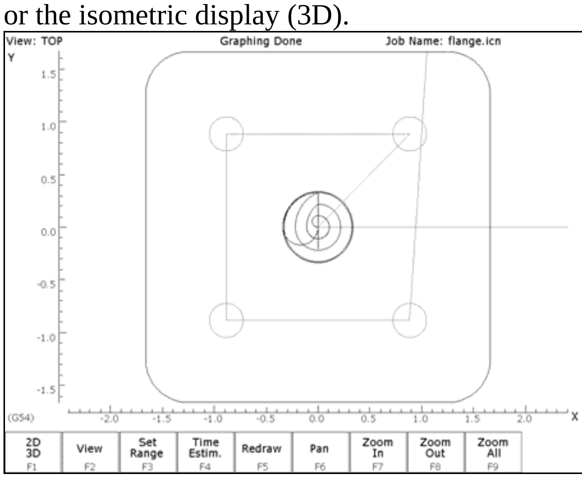
The format of the display will be similar to the following:



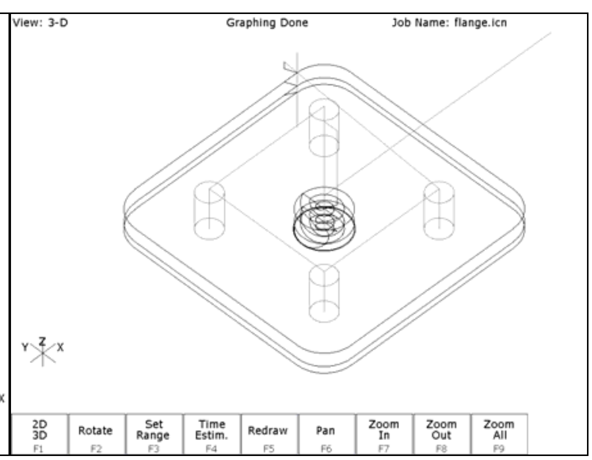
The display will consist of arcs and/or lines that make up the tool path followed. Rapid (G0) moves will appear in Red, linear (G1) and arc (G2, G3) moves will be Yellow, and compensated pass will appear in grey. Canned cycle operations (except the facing cycle) will also display a gray.

F1 – 2D/3D

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



Three-plane (2D)



Isometric (3D)

F2 – View/Rotate
In three-plane (2D) view, **F2 - View** switches the point of view to a different plane. In isometric, (3D) view, **F2 - Rotate** enables the arrow keys to rotate the figure. The arrow keys actually rotate a larger version of the YZX axes figure that shows the orientation in which the part will be redrawn. Press **F2 - Rotate** to redraw without leaving rotation mode. If you press **Enter** or **F5 - Redrew** after rotating the axes, the display of the axes will disappear. To rotate to a different angle you will have to press **F2 - Rotate** again. Press **Esc** to cancel rotation.

F3 – Set Range
Press **F3 – Set Range** to specify the range of operations to draw. You will be prompted for a start block and an end block.

F4 – Time Estimate
Press **F4 - Time Estim** to hide or display the time estimate in the upper left-hand portion of the screen.

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

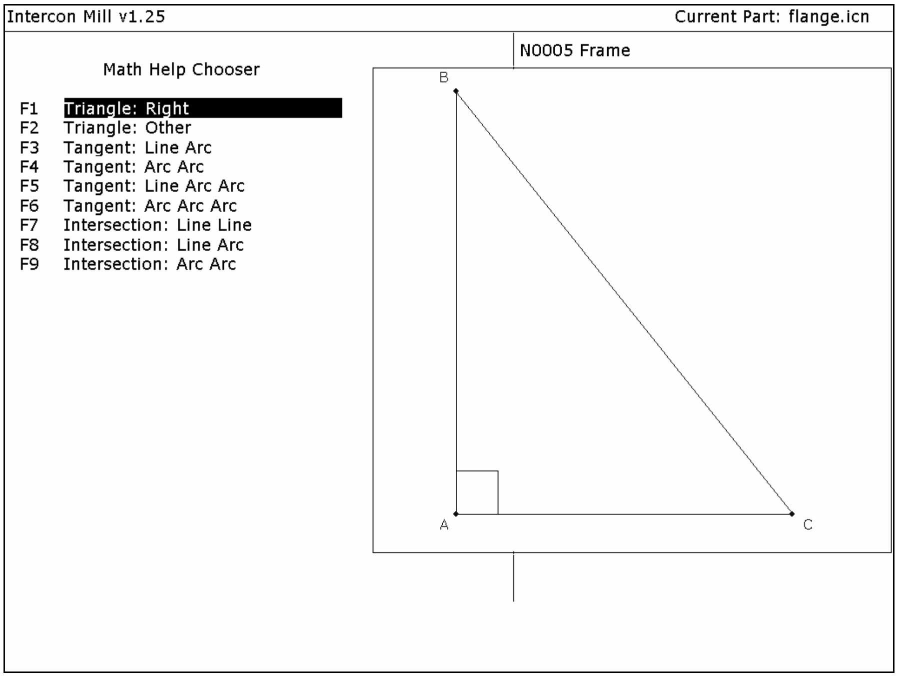
F6 - Pan
When using the pan feature, the project can be centered to the crosshairs in the display windows of the three-plane display, or rotated around the center of the isometric display screen. To enter pan mode, simply press the **F6 - Pan** key or press one of the arrow keys. A set of crosshairs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

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

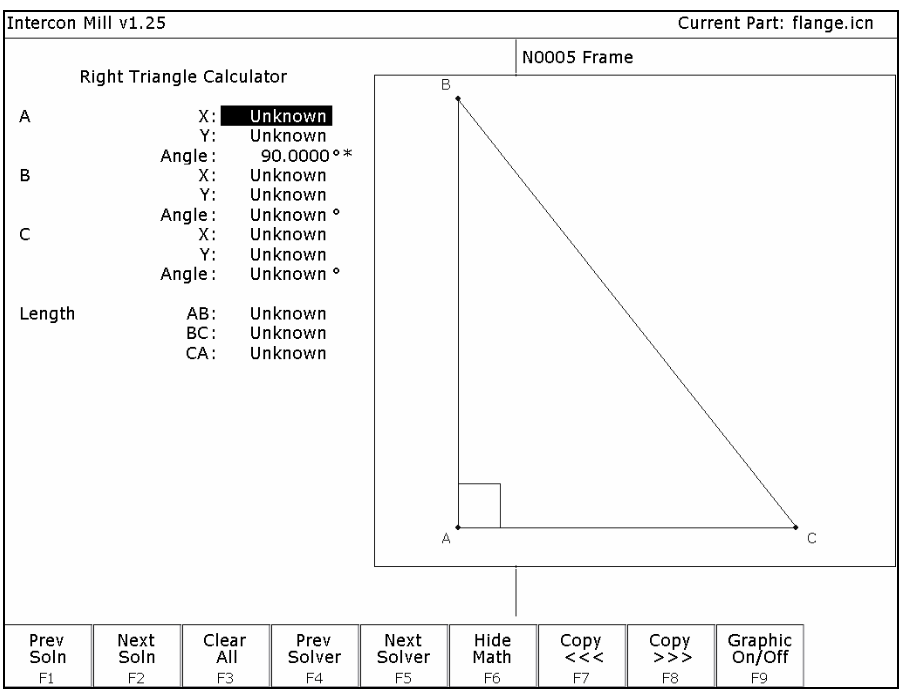
Number keys and Space bar – Feed Rate Override & Hold
If no jog panel is attached (or "Keyboard" has been selected as the jog panel type) the number keys 1 – 9 and 0 choose feed rate overrides 10% - 90% and 100%, respectively. 1 is 10%, 9 is 90%, and 0 is 100 %. If there is a jog panel attached you can use the feedrate knob to adjust the speed as well. The space bar toggles feed hold on and off.

Math Help

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



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



F1 – Prev Soln (Previous Solution)

F2 – Next Soln (Next Solution)

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

F3 – Clear All

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

F4 – Prev Solver

F5 – Next Solver

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

F6 – Hide Math

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

F7 – Copy <<<

F8 – Copy >>>

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

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

F9 – Graphic On/Off

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

↑ ↓ ← → (Arrow Keys) – Select Fields

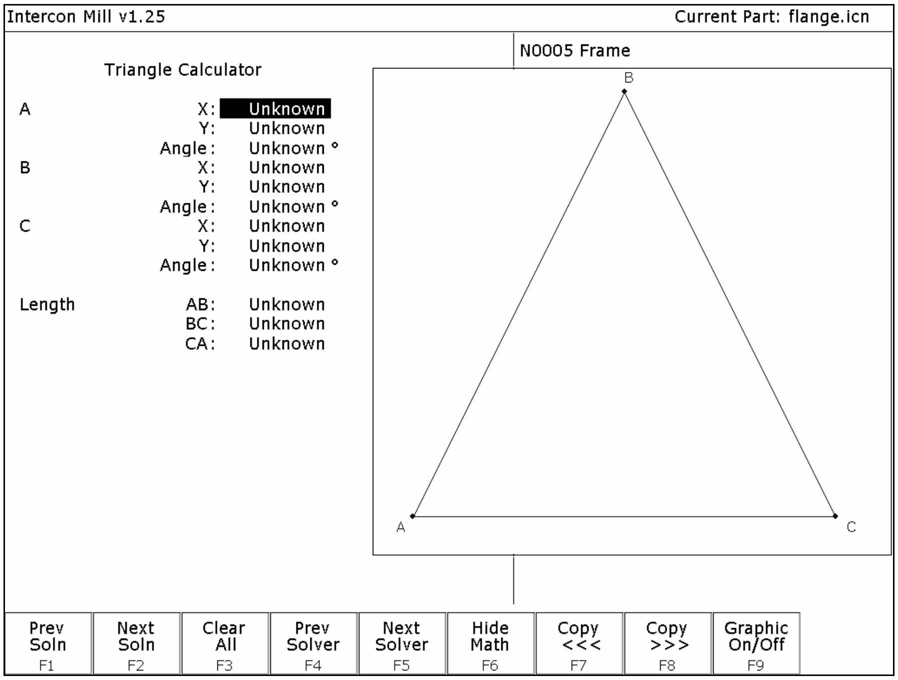
The **LEFT** and **RIGHT** arrow keys are used to navigate between the math menu and the edit menu. The **UP** and **DOWN** arrow keys are used to navigate within a menu. To choose fields for the "Copy" option, above, use the **UP** and **DOWN** arrow keys to highlight the desired field in the menu and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

Other features common to all Math Help operations

In some math help operations, there will be an asterisk '*' character that appears immediately to the right of a field. This character marks the field as a "given" field, which means that the value of this field will be held constant in the process of solving the math equations.

F1 –Triangle:Right

F2 –Triangle:Other

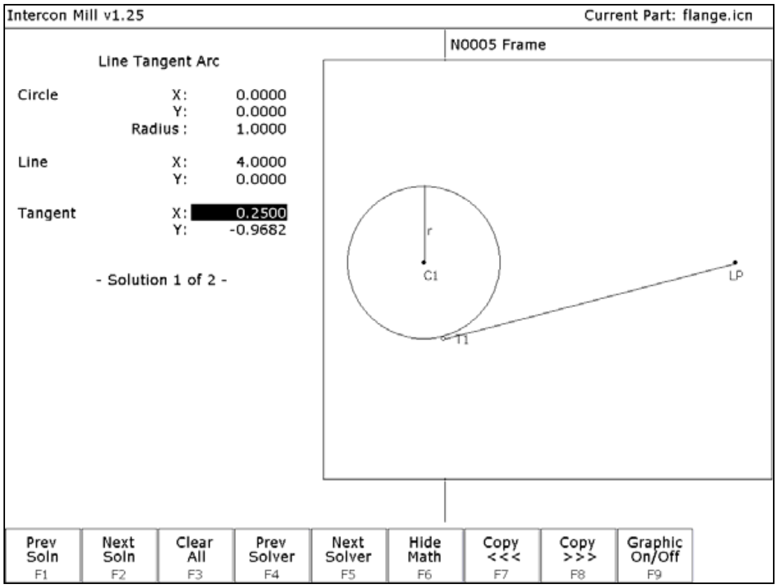


The screen will show UNKNOWN if the value of each parameter is not known. Math Help waits for known values to be entered, where:

Point a, b, or c is the coordinate value for each corner of the triangle.
Angle A, B, or C is the angle at each point of the triangle.
Length of values are the distances between the points indicated.

Continue adding all the known parameters. Select parameters using the arrow soft keys. When Math Help solves the remaining unknown values, the screen will display them.

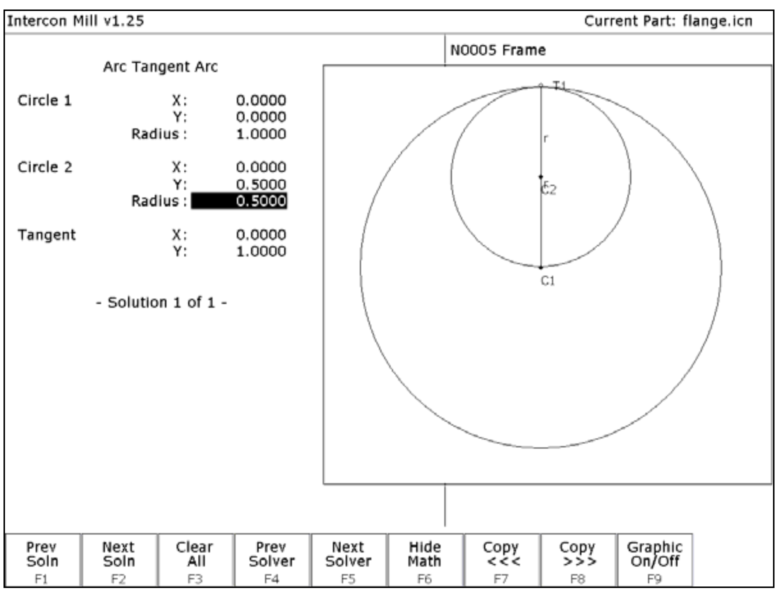
F3 – Tangent: Line Arc



Given the center (C1) the radius of an arc, and 1 point (LP) on a line, find the lines tangent to the arc (defined by the tangent point (T1)).

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

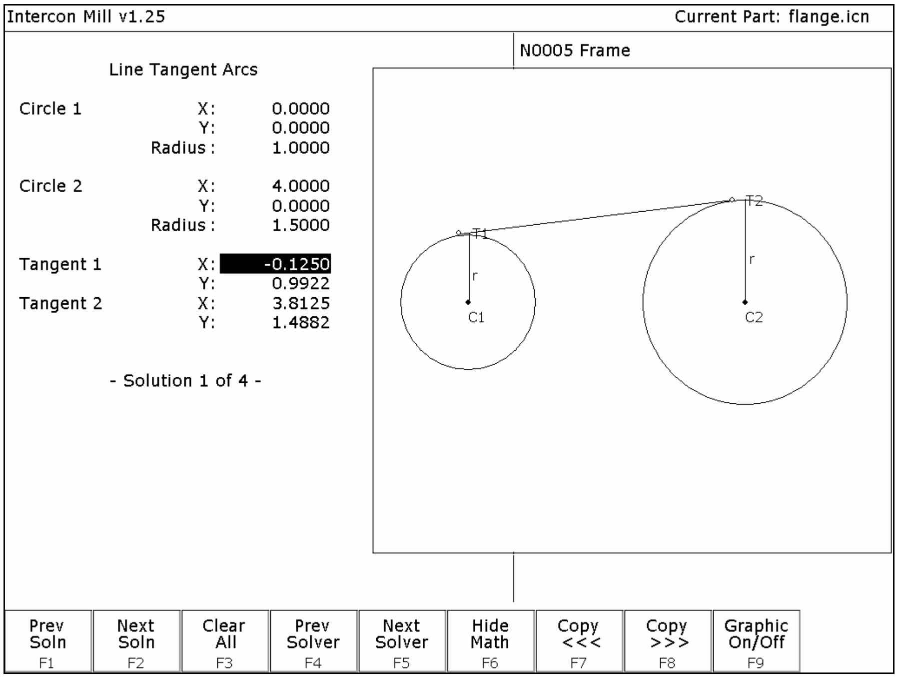
F4 – Tangent: Arc Arc



Given the center points (C1 and C2) and radii (R1 and R2) of two arcs, find the point (T) at which they are tangent.

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

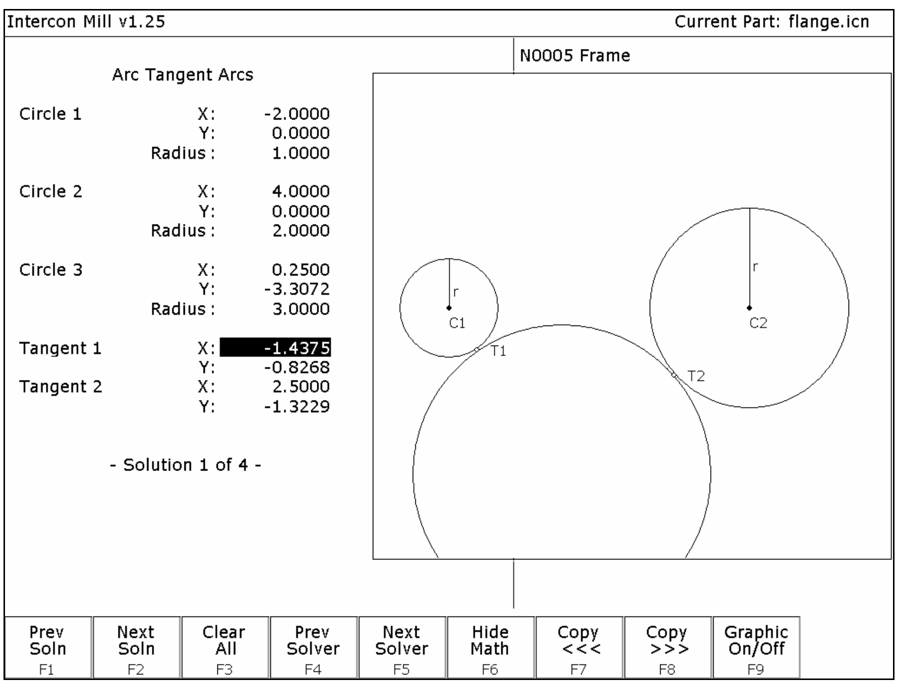
F5 – Tangent: Line Arc Arc



Given the center points (C1 and C2) and radii (R1 and R2) of two arcs, find the lines (defined by T1 – T2) tangent to both arcs.

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

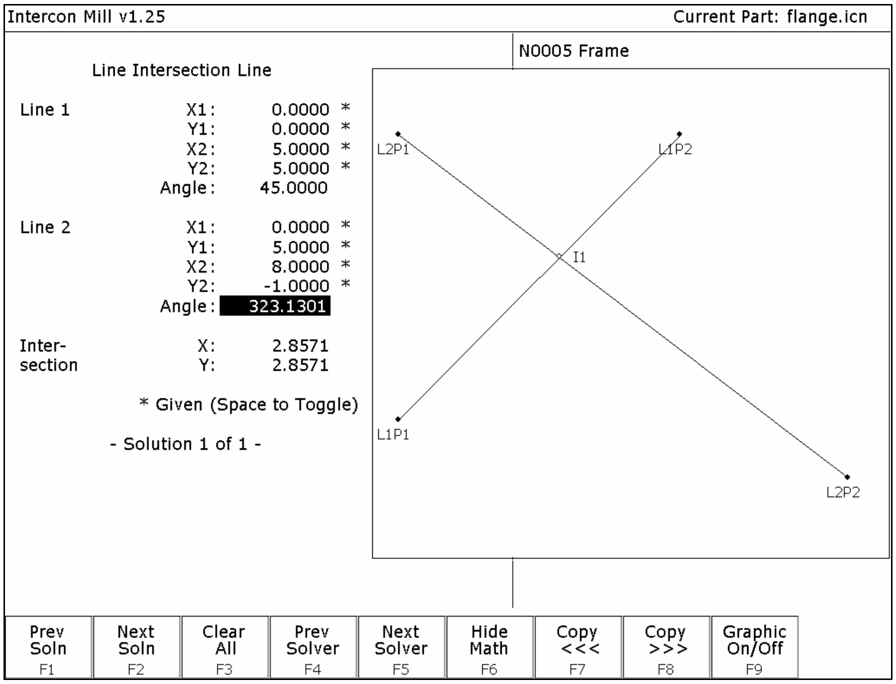
F6 – Tangent: Arc Arc Arc



Given the center points (C1 and C2) 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).

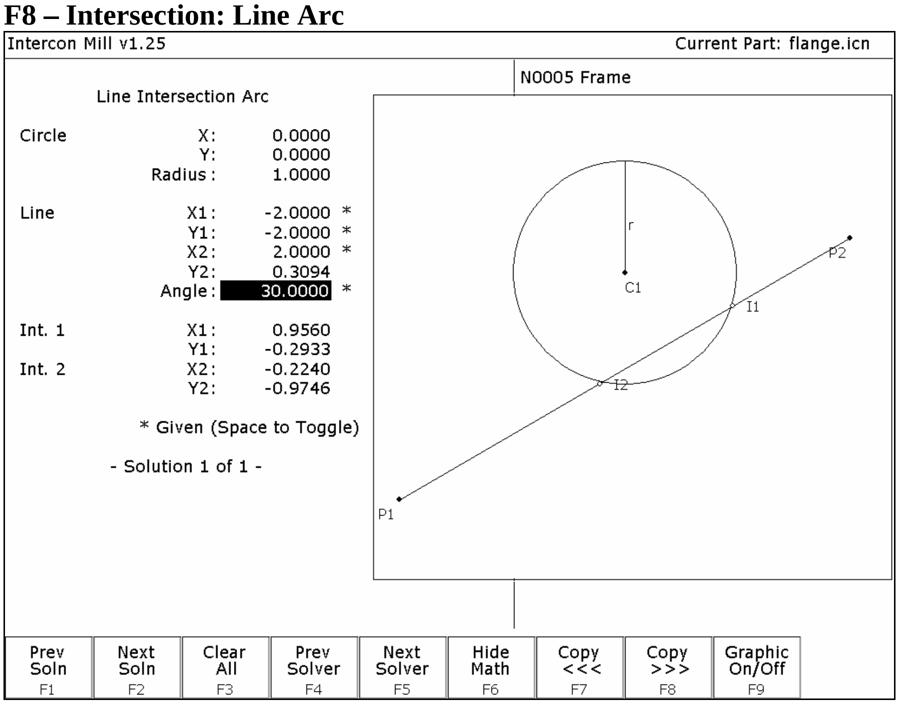
You must enter the radius of the tangent arc, the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

F7 – Intersection: Line Line



You must enter the X and Y coordinates for 1 point on each line, and also one of the following:
* the X and Y coordinates for a second point
* the X coordinate for a second point and the angle from horizontal
* the Y coordinate for a second point and the angle from horizontal
* the angle from horizontal only

F8 – Intersection: Line Arc

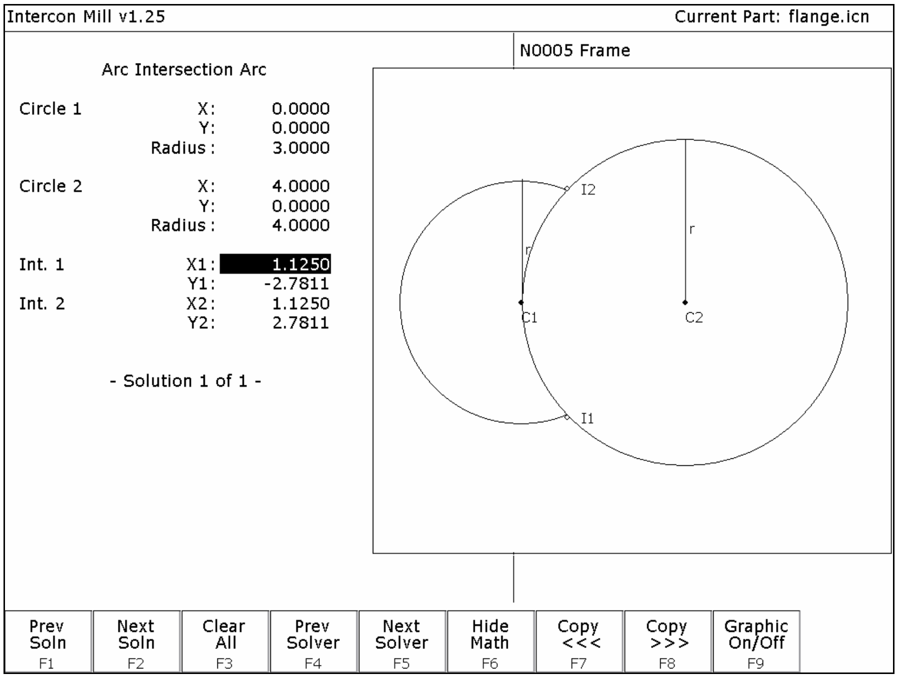


Given the center (C1) and radius (R) of an arc, 1 point (P1) and either a second point (P2) or one coordinate (P2 X or Y) and the angle from horizontal, find the intersection point(s) (I1 and I2).

You must enter the X and Y coordinates for the circle's center point, the circle's radius, the X and Y coordinates for one point on the line, and one of the following:

* the X and Y coordinates of a second point on the line
* the X coordinate of a second point and the angle from horizontal
* the Y coordinate of a second point and the angle from horizontal

F9 – Intersection: Arc Arc



Given the center points (C1 and C2) and the radii (R1 and R2) of two arcs, find the intersection point(s) (I1 and I2) of the arcs.

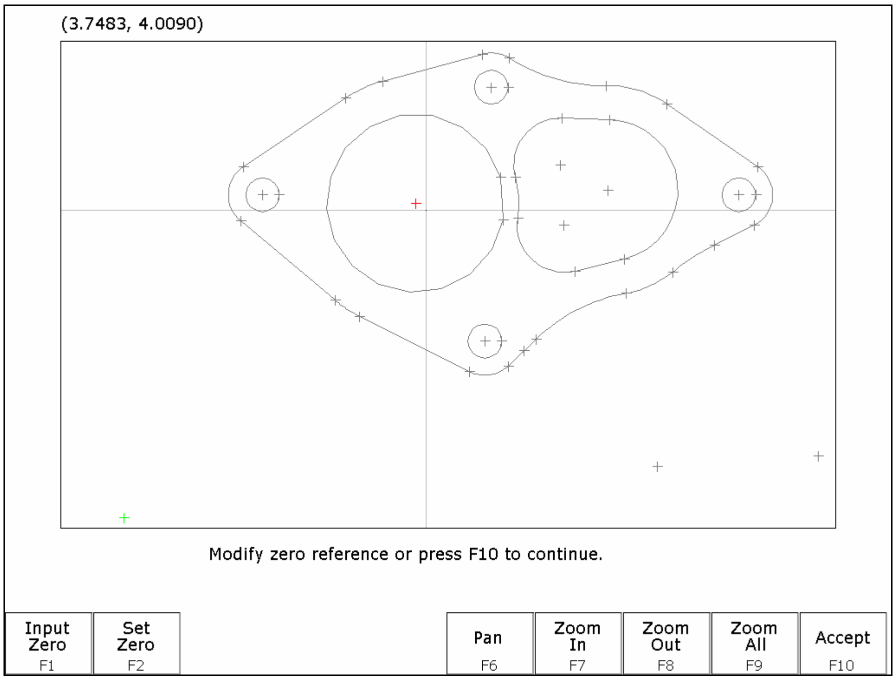
You must enter the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

Importing DXF files

Intercon allows you to convert geometry in DXF files to Intercon operations. To insert operations from a DXF file, press **F6 – Other** then **F8 - Import DXF**. If no DXF files have been loaded yet, the Intercon load file menu will appear. From here you may select the DXF file you wish to load. By default, Intercon expects DXF files to reside in the **c:\cnc10\ncfiles** directory. To change the default directory used by the Intercon load menu when loading DXF files, see the "User Specified Paths" section of Chapter 14.

Intercon reads DXF files up to and including version R14. At this time, only *point*, *line*, *arc*, *circle*, *polyline and lwpolyline* entities can be used to create Intercon operations. All other entities such as *text* must be converted to lines and arcs for them to appear in the Intercon Import DXF menu.

After a DXF file has been loaded, the Set zero reference menu appears.

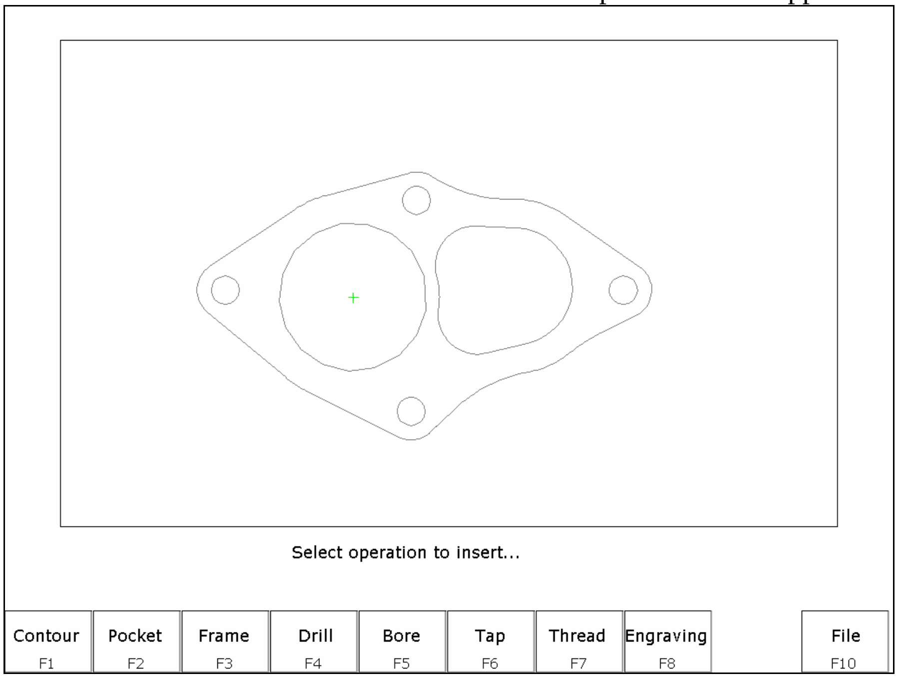


This menu allows you to change the absolute zero reference of the DXF file. The current zero reference appears as a green cross. All other points appear as gray crosses. A new zero reference may be defined by a combination the following methods:

- Press **F1 - Input Zero** to enter the coordinates of the new zero reference.
- Move the crosshairs with the arrow keys to highlight a point (represented by a gray cross) and when the crosshairs are close to a gray cross it will change red. Press **F2 - Set Zero** to set the zero reference to the position of that point.

When satisfied with the current location of the zero reference, press **F10 - Accept**. The zero reference may be changed multiple times before pressing **F10 - Accept**.

After the zero reference is set the Select Intercon operation menu appears.



This menu allows you to select the type of Intercon operation you wish to create using geometry from the DXF file.

F1 – Contour
Convert one or more connected lines and/or arcs to linear and arc operations.

F2 – Pocket
Convert a chain of lines and/or arcs to one of the Intercon pocket operations. The type of pocket depends on the geometry of the selected chain. A chain of arcs will be converted to a circular pocket if all arcs have the same center point and radius. A chain of four lines forming a rectangle will be converted to a rectangular pocket. All other chains will be converted into a cleanout operation.

F3 – Frame
Create a frame operation that surrounds a chain of lines and/or arcs. The height, width and center of the chain are used to define the frame.

F4 – Drill
Convert one or more points to a drilling operation.

F5 – Bore
Convert one or more points to a boring operation.

F6 – Tap
Convert one or more points to a tapping operation.

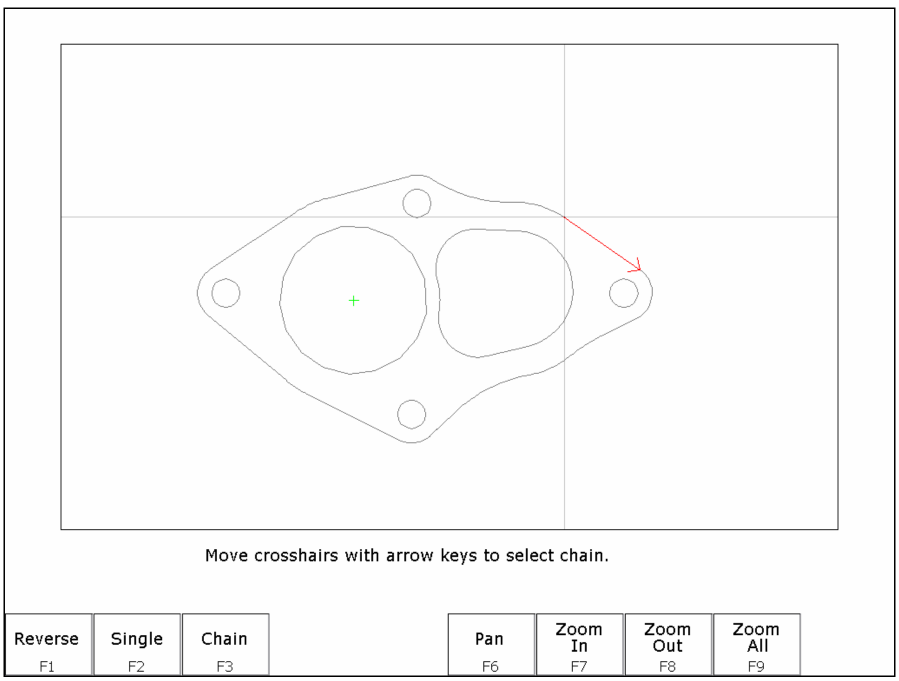
F7 – Thread
Convert one or more points to a threading operation.

F8 – Engrave (Optional)
Convert the entire file to engraving operations. Choosing this option displays the engraving menu. Set the options as desired for the engraving. Surface Height is the height of the surface to be engraved. Clearance Height is the height that the engraving tool will move up to clear the surface. Depth is the depth of the engraving. Set the Plunge Rate and Feedrate appropriately for the tool and material.

F10 – File
Pressing **F10 - File** displays the following options:
F1 - Load - Load a new DXF file.
F2 - Zero - Change the current zero reference.
F3 - Gap - Modify the current gap tolerance. Two lines or arcs are connected if the distance between their end points is less than the gap tolerance.

Selecting DXF geometry
After selecting an operation to from the Select Intercon operation menu, one of two menus appears. Contour, pocket and frame operations display the Select Chain menu and Drill, Bore, Tap and Thread operations display the Select Point menu. These menus allow you to select the geometry you wish to use to create the specified Intercon operation(s).

Select Chain menu
This menu allows you to select a chain of one or more lines and/or arcs. To select a chain, highlight the first line or arc with the crosshairs. Press either **F2 - Single** to accept only that object or press **F3 - Chain** to accept that object and create a chain of lines and arcs connected to that entity. When satisfied with the selected chain, press **F10 - Done** to edit the values for the chosen Intercon operation.



↑ ↓ ← → (Arrow Keys) – Move crosshairs
Use the arrow keys to move the crosshairs.

F1 – Reverse
When a line or arc is highlighted, an arrow appears indicating the direction of the object. Contour operations use this direction when cutting lines and arcs. To reverse the direction of the highlighted line or arc, press **F1 - Reverse**.

F2 – Single
Select the currently highlighted line or arc.

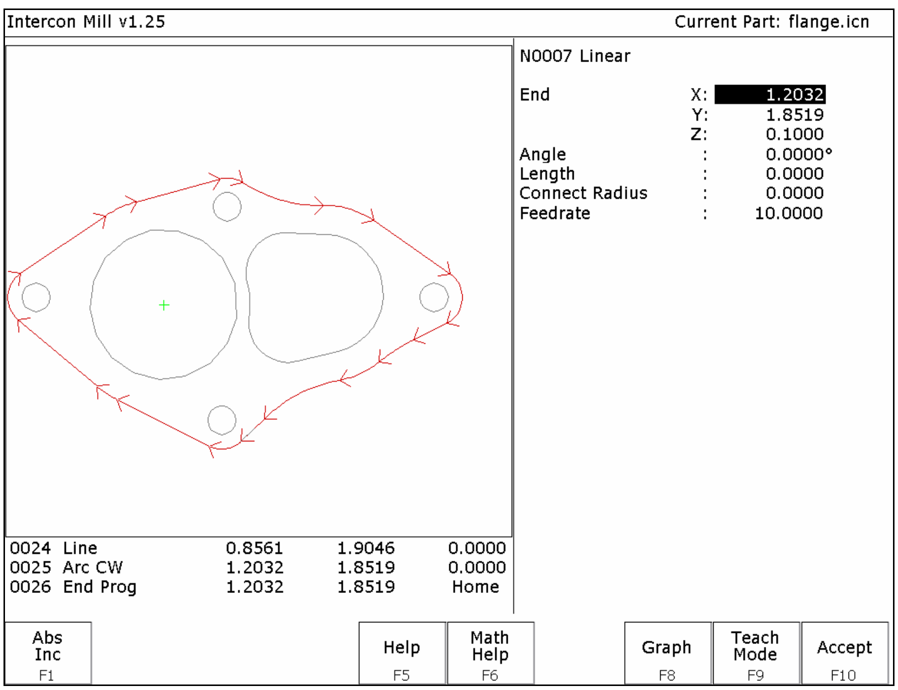
F3 – Chain
Select the currently highlighted line or arc and create a chain of connected lines or arcs in the direction of this selected object. Chaining will stop if there are no more connected, unselected lines or arcs, or the chain connects to the first object in the chain, or a branch point occurs. A branch point occurs when the last object in the chain is connected to more than one unselected lines or arcs. When this situation occurs, highlight the desired unselected line or arc that connects to the last object in the chain and press **F3 - Chain** to continue chaining.

F5 – Undo
Unselect the last line or arc in the chain.

F6 – Pan
Set the plot center to the center of the crosshairs.

F7, F8 & F9 - Zoom In, Zoom Out & Zoom All
F7 - Zoom In and **F8 - Zoom Out**, set the center of the plot to the center of the crosshairs and Zoom In and Zoom Out respectively. **F9 - Zoom All** redraws the part with its original scale.

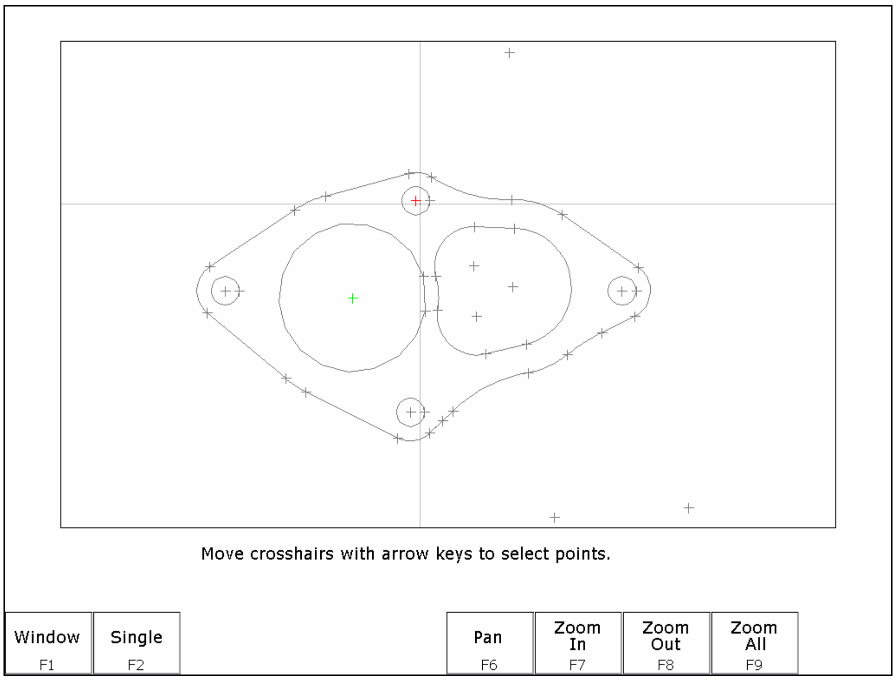
F10 – Done
Accepts the selected chain and proceeds to the Intercon operation edit menu. The operation edit menu allows you to enter values for fields such as feedrate and plunge rate. These values are copied to the rest of the lines and arcs in the chain where applicable.



The keys **F1 - Reverse**, **F2 - Single** and **F3 - Chain** only appear when a line or arc is highlighted. The keys **F5 - Undo** and **F10 - Done** only appear when one or more objects has been selected with **F2 - Single** or **F3 - Chain**.

Select Point menu

This menu allows you to select one or more points to be converted into Intercon Drilling/Threading operations. In this menu, all selectable points are displayed as gray crosses. Selectable points include point entities, line/arc endpoints and arc/circle center points. To select a point, position the crosshairs over the desired point until the cross turns red and press **F2 - Single** to accept the point. More points can be selected by highlighting them with the crosshairs and pressing **F2 - Single**. **F1 - Window** can be used to select all points within a specified window. When satisfied with the selected point(s), press **F10 - Done** to edit the Intercon operation parameters.



↑ ↓ ← → (Arrow Keys) – Move crosshairs
Press the arrow keys to move the crosshairs.

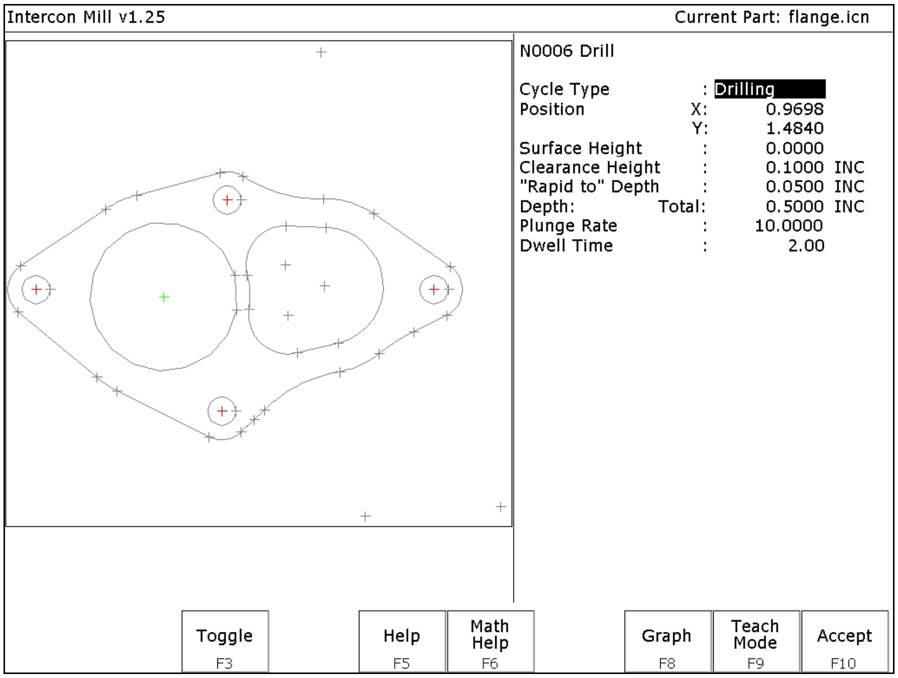
F1 – Window
This key allows you to select all points within a specified box. Press **F1 - Window** once to set the first corner of the box. Move the crosshairs to the desired location for the opposite corner of the box and press **F1 - Window**. All points within the box are selected.

F2 – Single
Accept the currently highlighted point.

F5 – Undo
Unselect the last selected point.

F6, F7, F8 & F9 – Pan, Zoom In, Zoom Out & Zoom All
These keys perform the same operations they do in the Select Chain menu.

F10 – Done
Accepts the selected points and proceeds to the Intercon operation edit menu. The operation edit menu allows you to enter values for fields such as plunge rate and depth. These values are copied to the rest of the selected operations where applicable.



The key **F2 - Single** only appears when a point is highlighted. The keys **F5 - Undo** and **F10 - Done** only appear when one or more points have been selected with **F1 - Window** or **F2 - Single**.

Using a mouse
In addition to the arrow keys, a mouse may be used to position the crosshairs in the DXF selection menus. Simply move the mouse pointer to the desired crosshair location and click the left mouse button. This action will move the crosshairs to the location of the mouse click and highlight the closest object.

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.

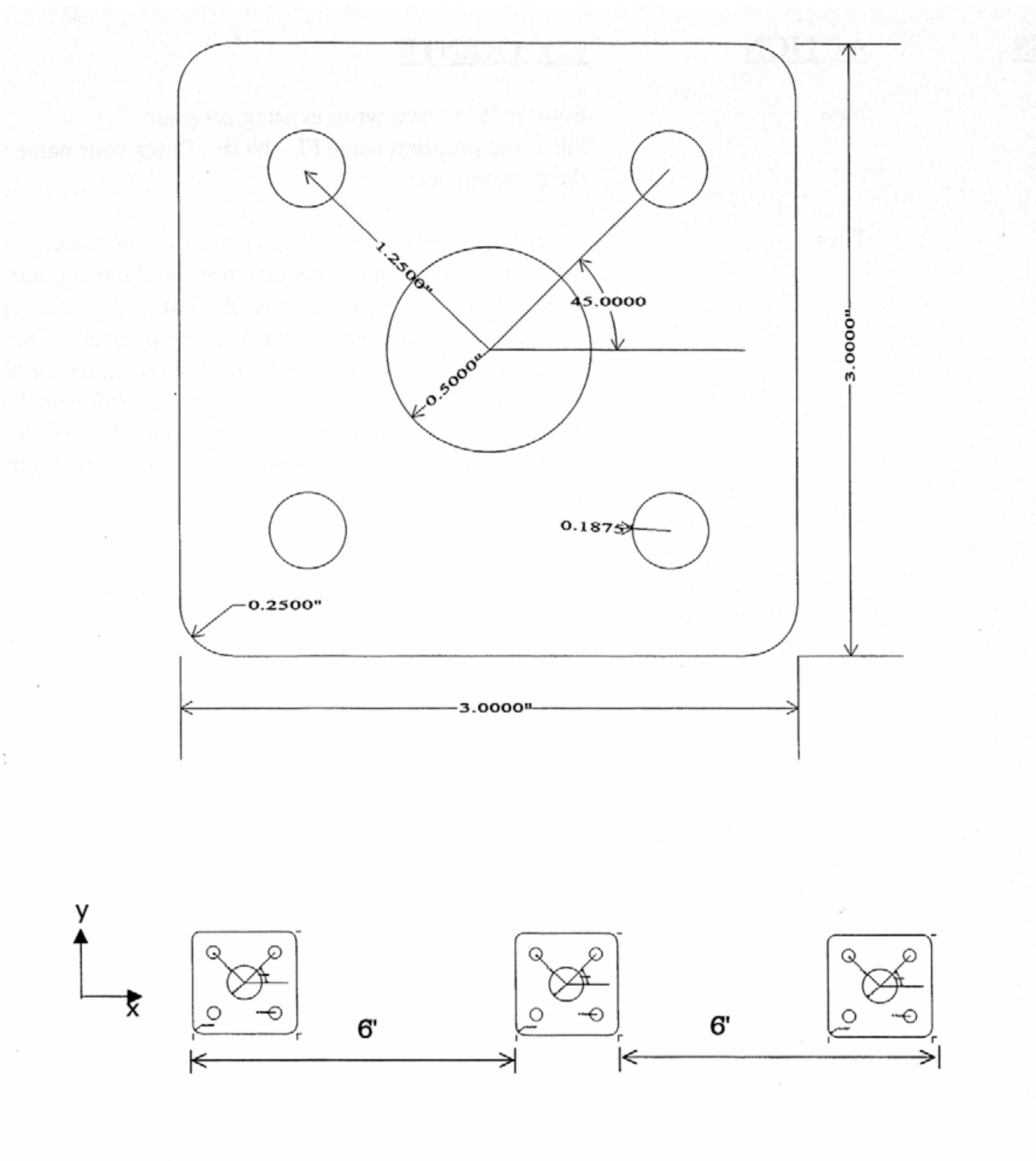


FIG. 1 Blueprint of flange part and the 3 fixtures.

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 **CNC10 Main** screen, press **F5 - CAM**). Beginning from the Intercon File Menu (press **F1 - File** if the file menu is not shown) the following series of keystrokes will describe the step-by-step process of designing the part shown in Figure 1.

PRESS	COMMENTS
F1 - New	Fill in the program name *flange*. Enter your name in as the programmer. Enter the description as "Intercon Tutorial #1".
F4 - Tool	Describe the tool below. The position values specify where to do the tool change. This position should be a point outside of the workpiece so that the last tool can be removed from the chuck and the new tool can be inserted. The Yes in the 'Actual Tool Change' field turns off the spindle and coolant upon reaching this spot. Use a 0.3750-inch diameter cutter. The length and diameter are updated based on the offsets. (The longest tool should have a 0.0000 length).

N0020 Tool change

Tool Number		:	1
Description		:	3/8" end mill
Position:	X	:	-2.0000
	Y	:	-2.0000
Tool H Offset		:	1
Tool Height		:	0.0000
Tool D Offset		:	1
Tool Diameter		:	0.3750
Spindle Speed		:	1000
Spindle Direction		:	CW (M3)
Coolant Type		:	Flood (M8)
Actual Tool Change		:	Yes

F10 - Accept	Keep selected values.
F5 - Cycles	Access the list of available Canned Cycles.
F6 - Circ. Pocket	Start with the 1.0000-inch diameter circular pocket. Enter the following values:

N0030 Circular pocket

Center:	X	:	0.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Diameter		:	1.0000	
Cleanout		:	Yes	
Depth:	Total	:	0.5000	INC
Per Pass		:	0.2500	
Plunge Rate		:	2.0000	[M]
Plunge Type		:	Ramped	
Plunge Angle		:	0.00°	
Rough Cuts		:	Conventional	
Stepover		:	0.2250	
Feedrate		:	20.0000	[M]
Finish Pass		:	Climb	
Amount		:	0.0020	
Feedrate		:	10.0000	[M]

F10 - Accept Keep selected values.

F5 - Cycles Access the list of available Canned Cycles.

F1 - Drill Select drilling cycles

F2 - Drill BHC Select the bolt hole circle type of drilling cycles

N0040 Drill bolt holes

Cycle Type		:	Drilling	
Center:	X	:	0.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Clearance Height		:	0.2500	INC
'Rapid To' Depth		:	0.1000	INC
Depth:	Total	:	0.5000	INC
Plunge Rate		:	2.0000	
Dwell Time		:	0.0000	
Number of holes		:	4	
Radius		:	1.2500	
Start angle		:	45.00°	

F8 - Graph Display a preview of the part up to this point. This preview can be used to detect problems that may occur if the part was cut now.

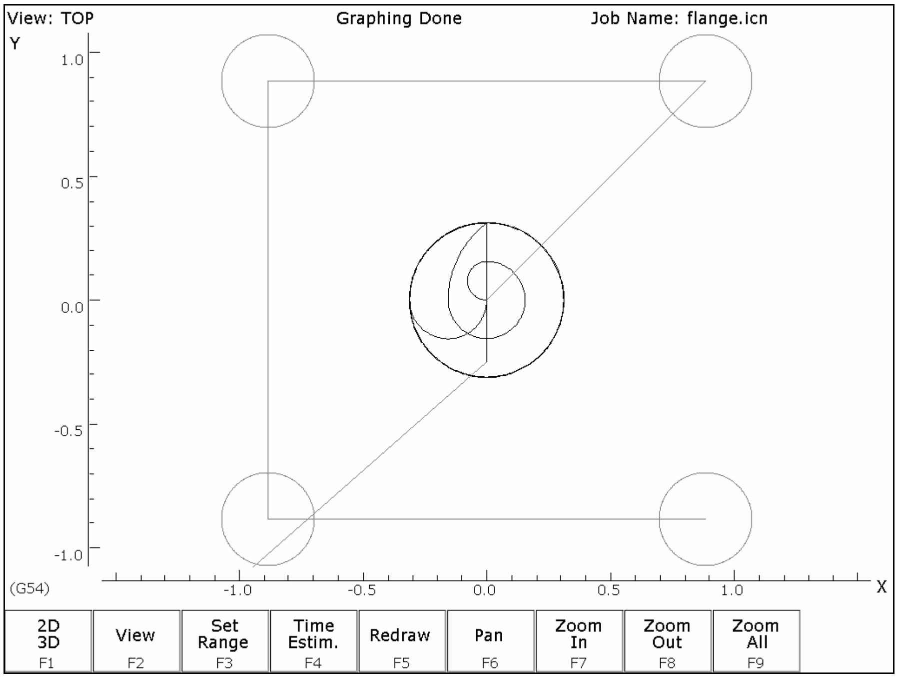


FIG. 2 - Graphics screen showing bolt holes and circular pocket

ESC/CANCEL	Return to the editing screen.
F10 - Accept	Keep selected values.
F5 -Cycles	Access the list of available Canned Cycles.
F7 - Frame	Now add an outside frame to cut the flange out of the material. The flange is 3.0000 inches long by 3.0000 inches wide, and has rounded corners with 0.2500-inch radii.

N0050 Frame mill

Frame		:	Outside Rect	
Center:	X	:	0.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Length	X	:	3.0000	INC
Width	Y	:	3.0000	INC
Corner Radius		:	0.2500	
Depth:	Total	:	0.5000	INC
per Pass		:	0.2500	
Plunge Rate		:	2.0000	
Plunge Type		:	Ramped	
Plunge Angle		:	0.00°	
Cut Type		:	Conventional	
Feedrate		:	10.0000	

F8 - Graph	Display a preview of the part up to this point. This preview can be used to detect problems that may occur if the part was cut now.

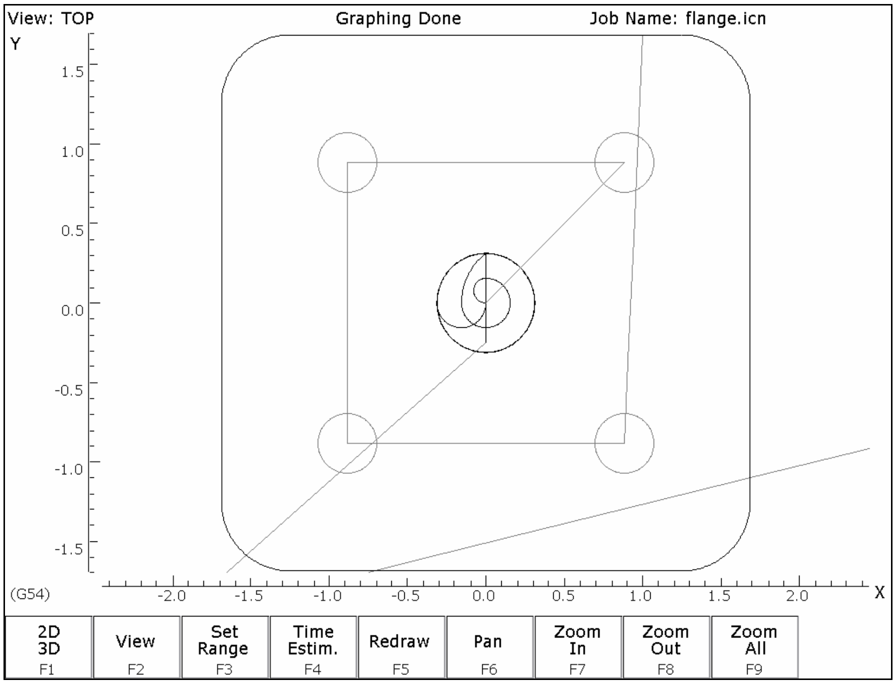


FIG. 3 - Graphics screen showing part with bolt holes and outer frame

ESC/CANCEL	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 **#0003** and ends with the linear mill in operation **#0005**. Press **F2 - Z Home** to enter “Home” for “Clearance Height.

N0060 Repeat

Start Block		:	N0003
End Block		:	N0005
Increment	X	:	6.0000
	Y	:	0.0000
Clearance Height		:	Home
Plunge Rate		:	2.0000
Number of copies		:	2

F8 - Graph	Display a preview of the parts. This preview can be used to detect problems that may occur if the part was cut now.
ESC/CANCEL	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 — Creation of the part is complete. Intercon programs automatically turn the spindle and coolant off at the end.

F1 - File — Press **F3 - Save** to save the part under its current name. Press **F4 - Save As** to save it under a new name.

F10 - Post — The CNC file needed to run this part on your mill will be generated at this time. The Intercon program displays the operation number of the part it is processing as it works through each operation in memory:

Generating CNC Program
Block 0050

As it processes each operation, it checks for values that, if used, will cause incorrect code to be produced. If such a value is found, a message will appear on the screen alerting you of the problem. For example, a problem with a Frame Mill may produce this message:

Message
Corner radius too small for Cutter...hit a key.

Changes to the part would then be required to allow proper code generation to proceed. If no problems are encountered during code generation, the following message appears:

Message
CNC code generation successful

Program Finished! — You are now finished designing your part. In order to run your part, you now need to return to the CNC10 software.

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	COMMENTS
JOG KEYS	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 (CNC10) 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	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	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 - Offset Lib.	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 and H001. D001 should be 0.375, H1 should be 0.0000 (the two inch tool). If either 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 in Chapter 5.
F10 - Save	Keep the updated tool offset library values.
F2 - Tool Lib.	Now you need to make sure that each tool uses the correct diameter and height offset values. Inspect the values for T001. T1 should use H001 and D001. 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	Leave Tool Setup. Return to the CNC10 Setup Screen.
ESC/CANCEL	Leave CNC10 Setup. Return to the CNC10 Main Screen.
CYCLE START	The **CYCLE START** button is located on your jog panel. This key will cause the mill to begin cutting your part

Tutorial Complete!

Intercon Tutorial #2

This demonstration will show you how to create a tool path for a part from a blueprint using the Math Help function of Intercon. The tool path to be created is for the part shown in Figure 1 below.

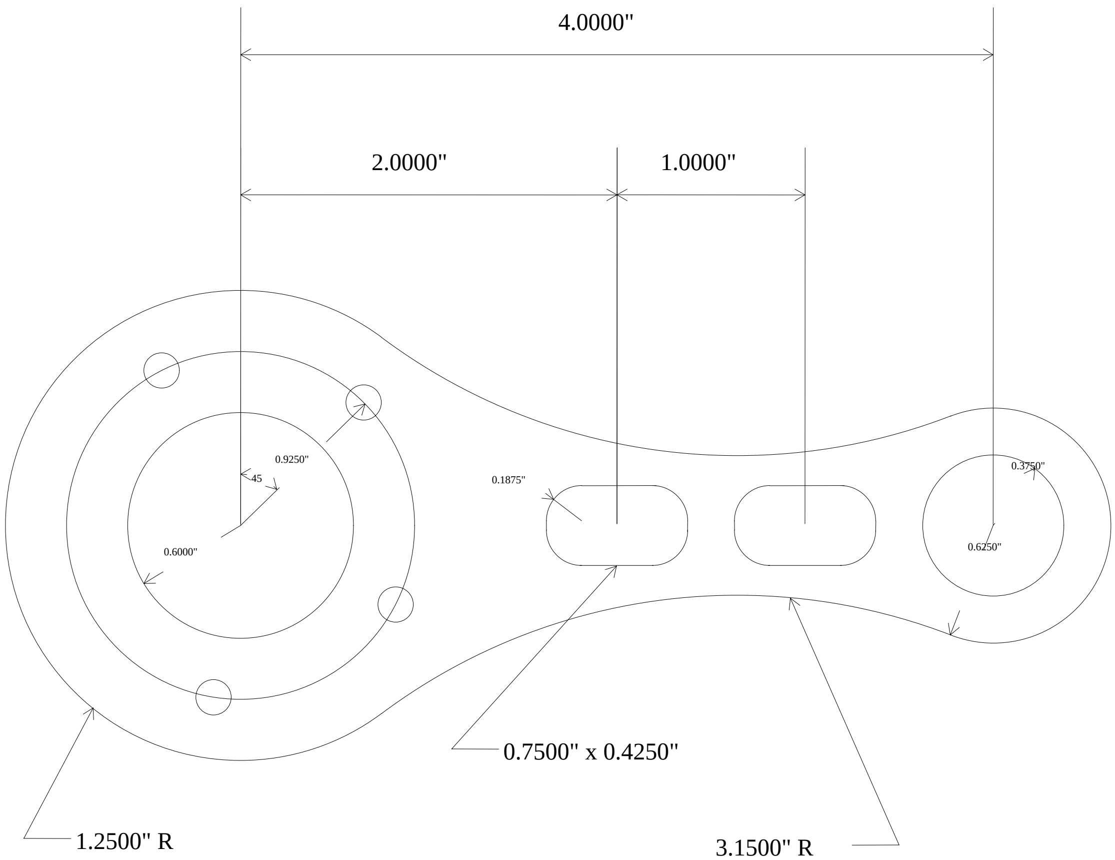


FIG. 1 - Part to be machined**.**

Part Creation

The process of creating a part is called part programming. Each feature of the part will become an operation in your program. Before beginning, decide where you want the **X0** and **Y0** reference. For this particular demo, the center of the Bolt Hole pattern was selected for convenience). Beginning from the Intercon File Menu (press **F1 - File** if the file menu is not shown) the following series of keystrokes will describe the step-by-step process of designing the part shown in Figure 1.

PRESS	COMMENTS
F1 - New	Create a new program by filling in the appropriate program name (we recommend *c_rod*) and your name. Press **Enter** or **F10 - Accept** to accept the new name. Enter “Intercon Tutorial #2” for the description. Press **F10 - Accept** to accept.
F4 - Tool	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 0.1875 inch drill. 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.

N0002 Tool change

Tool Number		:	1
Description		:	0.187” Drill
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 in Chapter 5 for more details.

F10 - Accept	Keep selected values.
F5 - Cycles	Access the list of available Canned Cycles.
F1 - Drill	Select drilling cycles
F2 - Drill BHC	Select a bolt hole circle operation.

The clearance height is the Z height from which the downward rapid traverse begins before each hole. It is also the Z height to which the tool returns upon completion of drilling the hole.

The 'Rapid To' depth is the Z height to which the tool rapid traverses before drilling a hole.

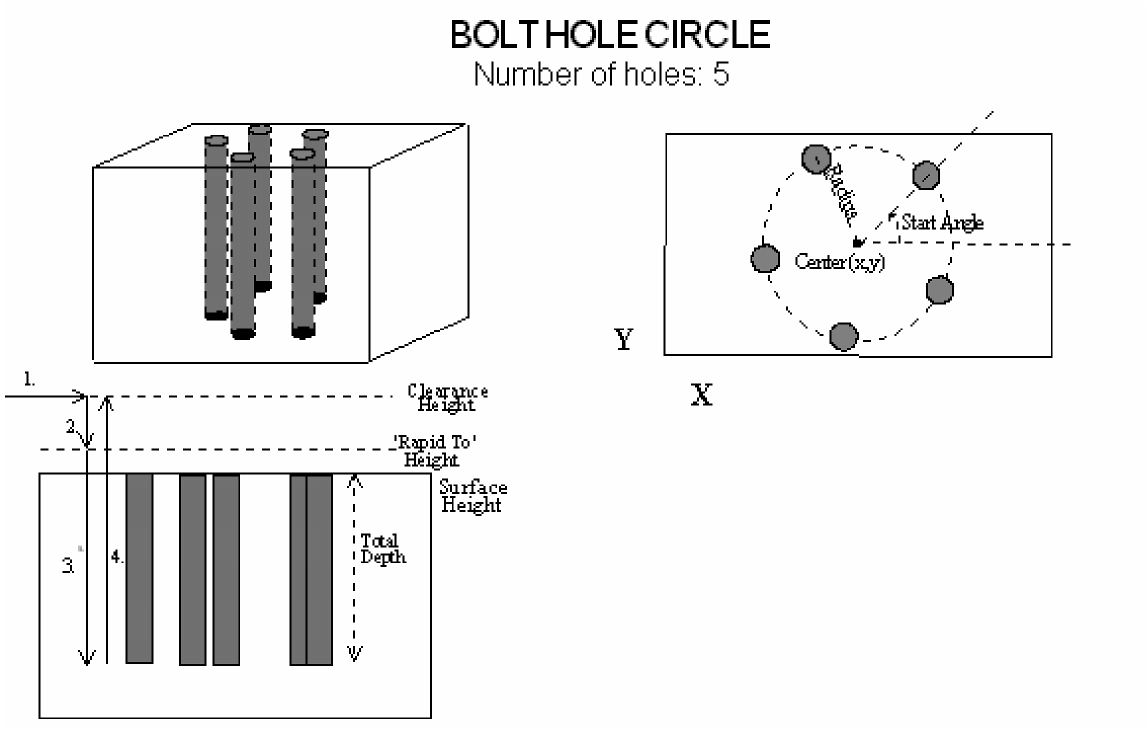


FIG. 2 - Bolt Hole Circle

N0003 Drill bolt holes

Cycle Type		:	Drilling	
Center:	X	:	0.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Clearance Height		:	0.5000	INC
'Rapid To' Depth		:	0.1000	INC
Depth:	Total	:	0.5100	INC
Plunge Rate		:	2.0000[M]	
Dwell Time		:	0.0000	
Number of holes		:	5	
Radius		:	0.9250	
Start angle		:	45.00°	

F10 - Accept Keep selected values.

F4 - Tool Use a 0.2500 diameter end mill now. Notice that the tool height shown below is a negative value. This value represents the difference in height between this tool and the longest tool being used. The longest tool used (in this case, operation N0020 above) has a height of 0.0000. Again, do not be alarmed if the Tool Height is not -1 for operation N0040. If this tool does not have the desired spindle (CW) and coolant (Flood) settings, you should also enter values specific to your machine setup.

N0004 Tool change

Tool Number		:	2
Description		:	0.250 Dia End Mill
Position:	X	:	0.0000
	Y	:	0.0000
Tool H Offset		:	2
(Tool Height)		:	(Your tool)
Tool D Offset		:	2
Tool Diameter		:	0.2500
Spindle Speed		:	1000
Spindle Direction		:	CW (M3)
Coolant Type		:	Flood (M8)
Actual Tool Change		:	Yes

F10 - Accept Keep selected values.

F5 - Cycles Access the list of available canned cycles.

F6 - Circ. Pocket Start with 1.2000 inch diameter Pocket.

N0005 Circular pocket

Center:	X	:	0.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Diameter		:	1.2000	
Cleanout		:	Yes	
Depth:	Total	:	0.5100	INC
Per Pass		:	0.2500	
Plunge Rate		:	2.0000	[M]
Plunge Type		:	Ramped	
Plunge Angle		:	0.00	°
Rough Cuts		:	Conventional	
Stepover		:	0.2000	
Feedrate		:	2.0000	[M]
Finish Pass		:	Climb	
Amount		:	0.1000	
Feedrate		:	2.0000	[M]

F10 - Accept Keep selected values.

F5 - Cycles Access the list of available canned cycles.

F6 - Circ. Pocket Repeat above pocket cycle. The center X value 4.0000 and the diameter is 0.7500 inches.

N0060 Circular pocket

Center:	X	:	4.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Diameter		:	0.7500	
Cleanout		:	Yes	
Depth:	Total	:	0.5100	INC
Per Pass		:	0.2500	
Plunge Rate		:	2.0000	
Plunge Type		:	Ramped	
Plunge Angle		:	0.00	°
Rough Cuts		:	Conventional	
Stepover		:	0.2000	
Feedrate		:	2.0000	
Finish Pass		:	Climb	
Amount		:	0.1000	
Feedrate		:	2.0000	

F10 - Accept Keep selected values.

F5 - Cycles Access the list of available Canned Cycles.

F5 - Rect. Pocket Cut the first rectangular pocket.

N0070 Rectangular pocket

Center:	X	:	2.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Length	(X)	:	0.7500	INC
Width	(Y)	:	0.4250	INC
Corner Radius		:	0.1875	
Depth:	Total	:	0.2500	INC
Per Pass		:	0.2500	
Plunge Rate		:	2.0000	
Plunge Type		:	Ramped	
Plunge Angle		:	0.00	°
Rough Cuts		:	Conventional	
Stepover		:	0.1000	
Feedrate		:	2.0000	
Finish Pass		:	None	
Amount		:	0.0000	
	Feedrate	:	2.0000	

F10 - Accept Keep selected values.

F5 - Cycles Access the list of available Canned Cycles.

F5 - Rect. Pocket Repeat above Pocket cycle. The center X value lies at 3.0000.

N0080 Rectangular pocket

Center:	X	:	3.0000	
	Y	:	0.0000	
Surface Height		:	0.0000	
Length	(X)	:	0.7500	INC
Width	(Y)	:	0.4250	INC
Corner Radius		:	0.1875	
Depth:	Total	:	0.2500	INC
Per Pass		:	0.2500	
Plunge Rate		:	2.0000	
Plunge Type		:	Ramped	
Plunge Angle		:	0.00	°
Rough Cuts		:	Conventional	
Stepover		:	0.1000	
Feedrate		:	2.0000	
Finish Pass		:	None	
Amount		:	0.0000	
Feedrate		:	2.0000	

F10 - Accept Keep selected values.

F7 - Cutter Comp Hit **Space** until Left cutter compensation is selected. The tool must move outside of the part outline at a distance at least equal to its radius so the part outline is the correct size. Cutter compensation should be turned on before a rapid to maintain proper line and arc travel.

N0009 Comp left

F10 - Accept Keep selected values.

F1 - Rapid Move to a location outside the part. The purpose of this move is to prepare to use cutter compensation on the tool.

N0010 Rapid traverse

End:	X	:	5.0000
	Y	:	0.5000
	Z	:	0.1000
Angle		:	14.0365°
Length		:	2.0615

F10 - Accept Keep selected values.

F3 - Arc 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.

N0011 Arc

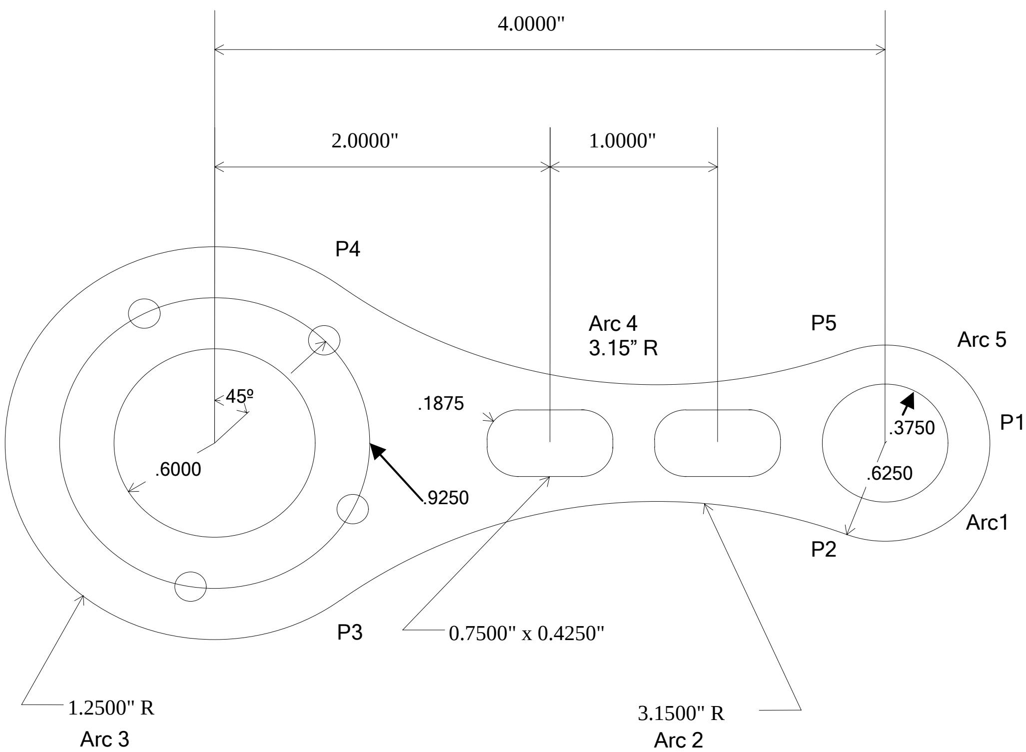
Arc Type:		:	EP&R
Mid	X	:	
	Y	:	
	Z	:	
End	X	:	4.625
	Y	:	0.0000
	Z	:	-0.0500
Center:	X	:	
	Y	:	
	Z	:	
Angle		:	
Radius		:	.5
Plane		:	XY
Direction		:	CCW
Connect Radius		:	0.0000
Feedrate		:	10.0000
Angle <= 180		:	Yes

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 The first arc to be cut is labeled as ARC 1 in Figure 3 below. The start point, labeled P1, is the end point of the previous move. The end point of the arc will be generated with Math Help. We will be using end point and radius (EP&R) arcs.

FIG 3. - Tangent point and arc reference.



N0012 Arc

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

F6 - Math Help

We are trying to find end points for the arcs that make up the outside edge of the part. Note: the main Math Help menu will listing all available Math Help solvers.

F6 - Tangent Arc Arc Arc

This scenario will generate tangent points P2 - P5 of Figure 3. Enter the values as shown below:

Arc Tangent Arcs:

Circle 1:			
	X	:	4.0000
	Y	:	0.0000
	Radius	:	0.6250
Circle 2:			
	X	:	0.0000
	Y	:	0.0000
	Radius	:	1.2500
	Radius	:	3.1500

Intercon will calculate the missing values for this scenario.

F2 - Next Soln

Find scenario that corresponds to the actual arcs being milled. Observe Figure 4. Point T1 is the one needed.

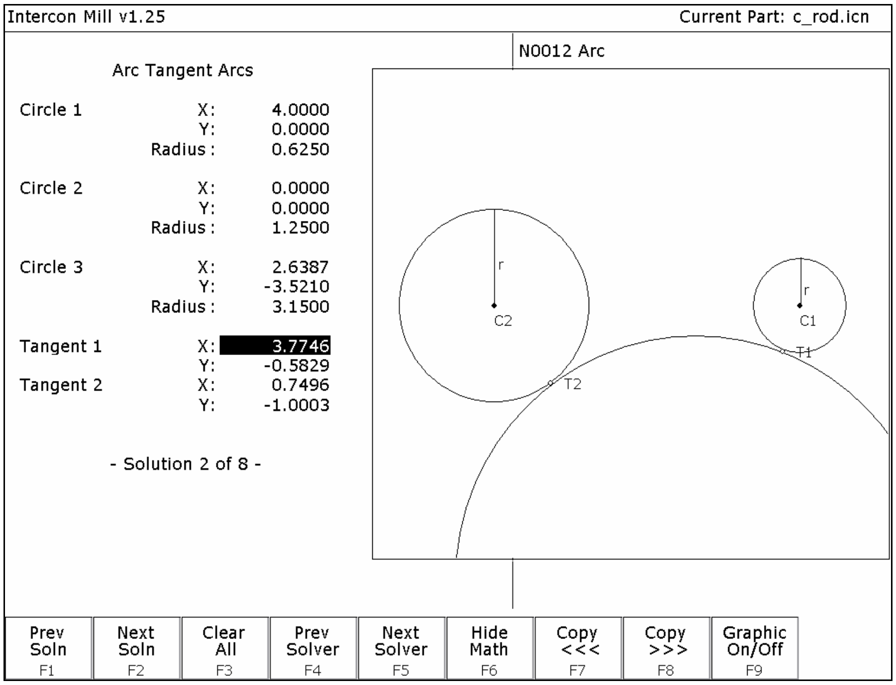


FIG. 4 - Screen showing Math Help Arc Tangent Arc solutions

ARROWS	If necessary, move the block cursor to the Tangent 1 X field as shown above. Note: Use only <↑> and <↓>. If you press the right arrow, press the left arrow to get back to the Math Help fields.
F9 - Graphic on/off	Press to hide the graphical display and reveal the arc operation behind it.
→ (ARROW)	Move the cursor to the arc operation. The solid block cursor on the left side of the screen will be replaced by an outlined rectangle and the solid block will appear in the arc operation on the right.
ARROWS	Move the block cursor to the End X field of the arc operation. As before, use only <↑> and <↓>.
F8 - Copy >>>	Transfer the tangent point T1 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically.
F8 - Copy >>>	Transfer the tangent point T1 value for Y into the end point Y coordinate.
ARROWS	Move down to the radius field and enter the radius of the arc labeled as ARC 1 in Figure 3. (This radius is 0.6250 in.).

Arc type		:	EP&R
Mid	X	:	
	Y	:	
	Z	:	
End	X	:	3.7746
	Y	:	-0.5829
	Z	:	-0.0500
Center:	X	:	
	Y	:	
	Z	:	
Angle		:	
Radius		:	0.6250
Plane		:	XY
Direction		:	CW
Feedrate		:	10.0000
Angle <= 180°		:	Yes

F6 - Hide Math — Hide Math Help temporarily. (We will return later to pick up the other tangent points.)

F8 - Graph — Observe Figure 5. The graphics show a preview of the part up to this point. This preview can be used to detect problems that may occur if the part was cut now.

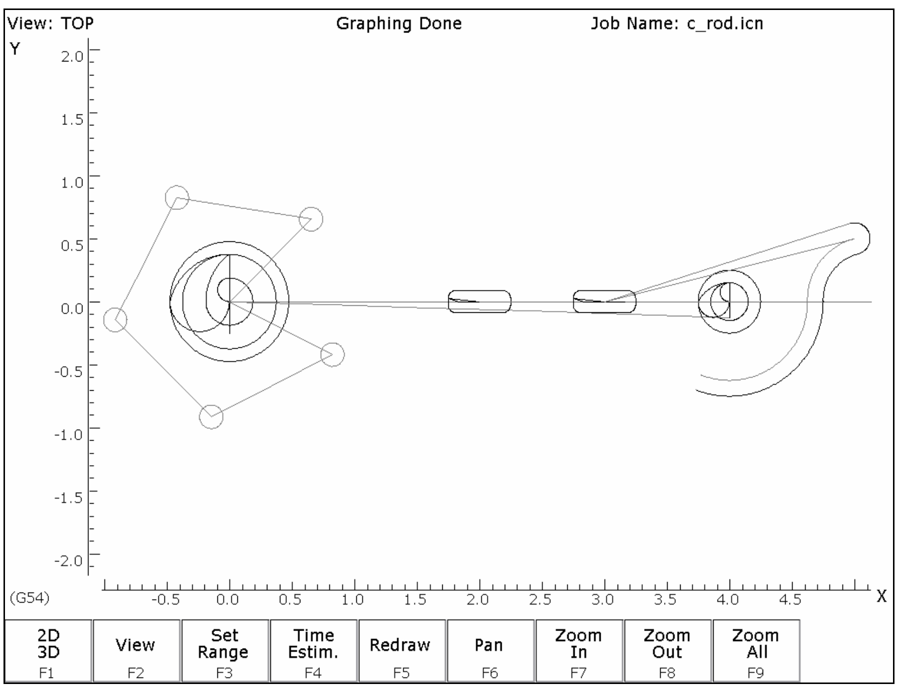


__FIG. 5 - Draw screen showing Bolt Holes, Pockets and first arc of part__

ESC/CANCEL — Return to the editing screen.

F10 - Accept — Keep selected values. The other arc values were calculated for you.

F3 - Arc

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.

N0013 Arc

↑↓(UP/DOWN)

Move down to the End X field. This selects End X as the destination of the Math Help solution.

F6 - Math Help

Redisplay the Math Help values calculated for the last arc. The screen will look like figure 6, below.

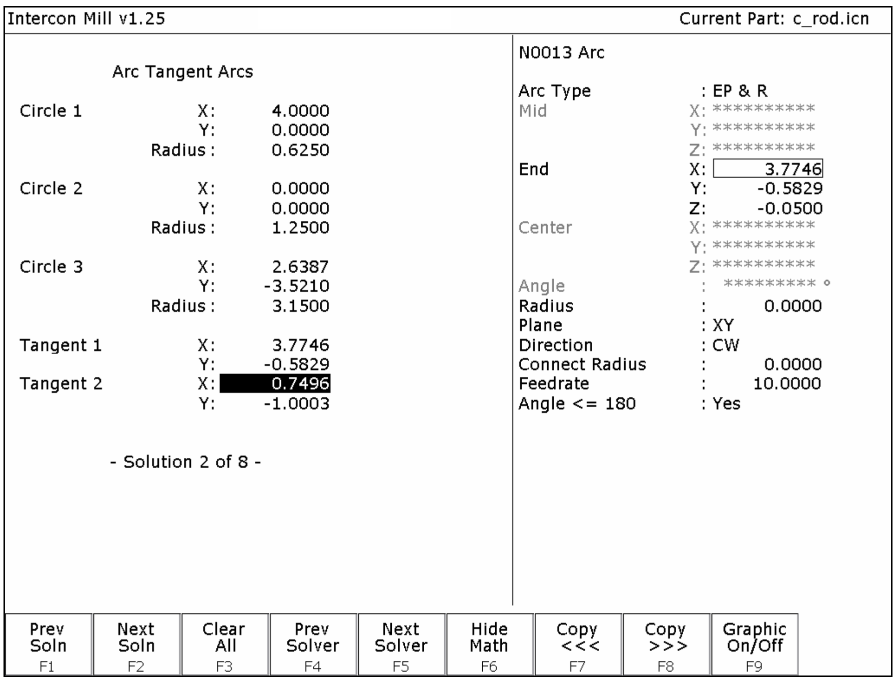


FIG. 6 – New arc 2 entry screen shown with solution for arcs 1 and 2 of Figure 3.

↑↓(UP/DOWN)

If necessary, move the block cursor to the Tangent 2 X field as shown above. The rectangle at End X shows that it will be the destination of the copy.

F8 - Copy >>>

Transfer the tangent point T2 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically.

F8 - Copy >>>

Transfer the tangent point T2 value for Y into the end point Y coordinate.

F6 - Hide Math

Hide Math Help temporarily.

↑↓(UP/DOWN)

Move down to the radius field and enter the radius of the arc labeled ARC 2 in Figure 3 (this radius is 3.1500 inches). Set the direction to "CCW".

```
Arc type       :  EP&R
Mid        X   :
           Y   :
```

	Z	:	
End:	X	:	0.7496
	Y	:	-1.0003
	Z	:	-0.0500
Center:	X	:	
	Y	:	
	Z	:	
Angle		:	
Radius		:	3.1500
Plane		:	XY
Direction		:	CCW
Feedrate		:	10.0000
Angle <= 180°		:	Yes

F10 - Accept	Keep selected values.
F3 - Arc	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. N0014 Arc
↑↓(UP/DOWN)	Move down to the End X field. This selects End X as the destination of the Math Help solution.
F6 - Math Help	Redisplay the Math Help values calculated for the last arcs.
F9 - Graphic on/off	Redisplays the diagram of the scenario selected to calculate arcs 1 and 2 on Figure 3.
F1 - Prev Soln	Continue pressing F1 until you arrive at the scenario showing arcs 3 and 4 in Figure 3 (in this case, solution #1 is the appropriate one)

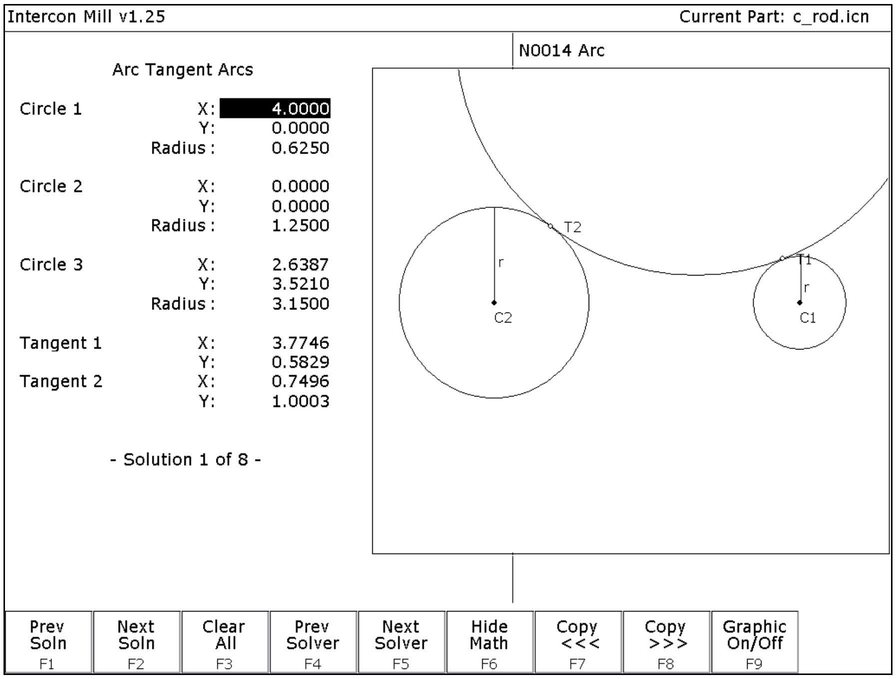


FIG. 7 - Math Help solution for arcs 3 and 4.

↑↓(ARROWS)	Press to highlight the needed tangent point X coordinate in Math Help. Tangent point T2 is the one you want this time.
→ (ARROW)	Press to remove the graphic display and move the cursor to the arc operation. (This shortcut saves you from pressing **F9 –- Graphic on/off** to hide the graphics each time.) The solid block cursor on the left side of the screen will be replaced by an outlined rectangle and the solid block will appear in the arc operation on the right.
↑↓(ARROWS)	Move the block cursor to the End X field of the arc operation.
F8 - Copy >>>	Transfer the tangent point T2 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically.
F8 - Copy >>>	Transfer the tangent point T2 value for Y into the end point Y coordinate.
↑↓(ARROWS)	Move 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 — 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.

N0015 Arc mill

↑↓(ARROWS) — Move down to the End X field. This selects End X as the destination of the Math Help solution.

F6 - Math Help — Re-display the Math Help values calculated for the last arc.

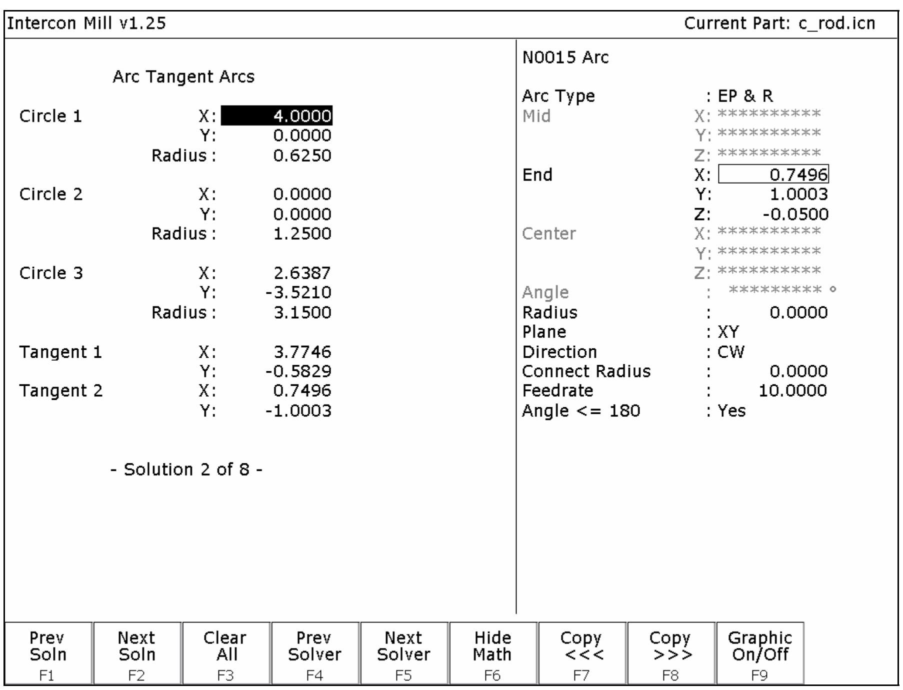


FIG. 8 - New arc 4 entry screen shown with solution for arcs 3 and 4 of Figure 3.

↑↓(UP/DOWN)	Highlight the needed tangent point X. Tangent point T1 is the one you want this time.
ARROWS	If necessary, move the cursor to the arc operation and select the End X field.
F8 - Copy >>>	Transfer the tangent point T1 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically.
F8 - Copy >>>	Transfer the tangent point T1 value for Y into the end point Y coordinate.
F6 - Hide Math	Hide Math Help.
ARROWS	Move down to the radius field and enter the radius of the arc labeled ARC 4 in Figure 3. (This radius is 3.1500 inches). Be sure to set the direction to CCW.

Arc type		:	EP&R
Mid:	X	:	
	Y	:	
	Z	:	
End:	X	:	3.7746
	Y	:	0.5829
	Z	:	-0.0500
Center:	X	:	
	Y	:	
	Z	:	
Angle		:	
Radius		:	3.1500
Plane		:	XY
Direction		:	CCW
Feedrate		:	10.0000
Angle <= 180°		:	Yes

F10 - Accept	Keep selected values.
F3 - Arc	Mill the arc labeled as ARC 5 in Figure 3 back to point P1.

N0016 Arc mill

Operation type		:	EP&R
Mid:	X	:	
	Y	:	
	Z	:	
End:	X	:	4.6250
	Y	:	0.0000
	Z	:	-0.5000
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 Move tool away from the edge of the part after the last arc.

N0017 Arc mill

Arc type		:	EP&R
Mid:	X	:	
	Y	:	
	Z	:	
End:	X	:	5.0000
	Y	:	-0.5000
	Z	:	0.1000
Center:	X	:	
	Y	:	
	Z	:	
Angle		:	
Radius		:	0.5000
Plane		:	XY
Direction		:	CCW
Feedrate		:	10.0000
Angle <=180°		:	Yes

F10 - Accept Keep selected values.

F9 - Subpgm Access the **Subprogram** screen.

F1 - Depth Repeat

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 N0011 and ends with the Linear Mill in operation N0017.

N0018 Repeat to Depth

Start Block	:	0011	
End Block	:	0017	
Total Depth	:	0.5100	INC
Depth Increment	:	0.0500	INC
Clearance Height	:	0.2500	
Plunge Rate	:	5.0000	

F10 - Accept

Keep selected values.

F7 - Cutter Comp

Hit the space bar until cutter compensation is turned '**Off**'. It is no longer needed.

N0019 Comp off

F10 - Accept

Keep selected values.

F1 - Rapid

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 CNC10 software to correct its position.

N0020 Rapid traverse

End:	X	:	0.0000
	Y	:	0.0000
	Z	:	3.0000
Angle		:	174.2894°
Length		:	5.0249

F10 - Accept

Keep selected values.

ESC/CANCEL

Creation of the part is complete. Intercon programs automatically turn the spindle and coolant off at the end.

F8 - Graph

Display a preview of the finished part. Just make sure that the finished part is going to look the way you want it to. The display shown in Figure 7 has rulers placed around the various view windows that are scaled to the same size as the part displayed to allow visual inspection of the part. Remember, this preview shows where the center of the current tool will move (cutter compensation is not represented except in pocket and frame displays).

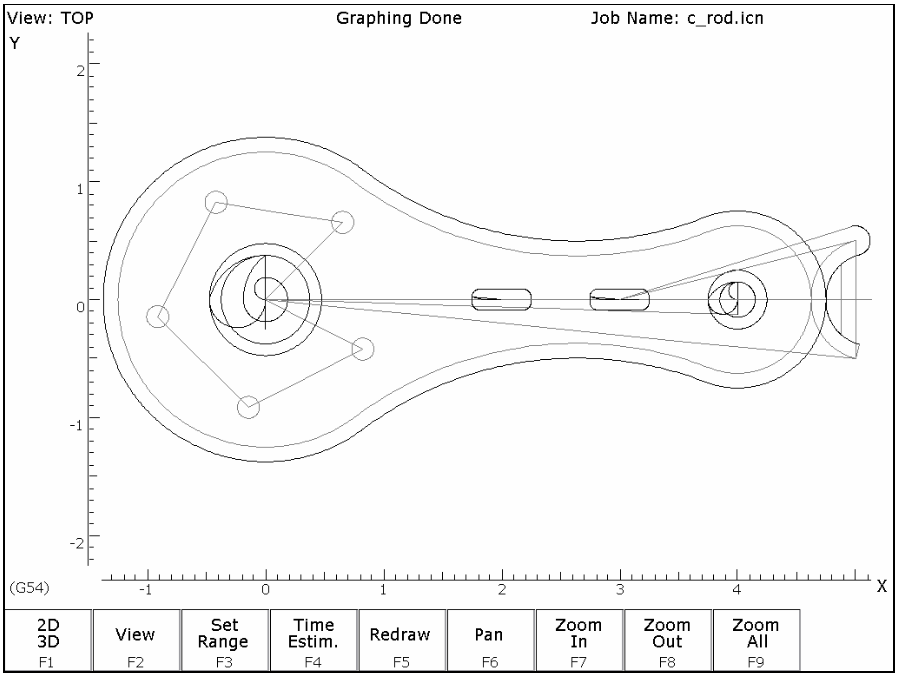


FIG. 9 - Draw screen showing complete part

ESC/CANCEL

Return to Main screen.

F1 - File

Go to the File Menu. Press **F3 - Save** to save under the current file name or press **F4 - Save As** to save the program under a different name.

F10 - Post

The CNC file needed to run this part on your mill will be generated at this time. The Intercon program displays the operation number of the part it is processing as it works through each operation in memory:

Generating CNC Program

Block 0050

As it processes each operation, it checks for values that, if used, will cause incorrect code to be produced. If such a value is found, a message will appear on the screen alerting you of the problem. For example, a problem with a rectangular pocket may produce this message:

Message

Corner radius too small for Cutter...hit a key.

Changes to the part would then be required to allow proper code generation to proceed. If no problems are encountered during code generation, the following message appears:

You are now at the main menu again.

You are now finished designing your part. In order to run your part, you now need to return to the CNC10 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	COMMENTS
JOG KEYS	Jog the table so that your tool rests on the stock at the location that will represent X0 and Y0.
F1 - Setup	Enter the CNC10 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	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	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 - Offset Lib.	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 in Chapter 5.**

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	Leave **Tool Setup**. Return to the **CNC10 Setup** Screen.
ESC/CANCEL	Leave **CNC10 Setup**. Return to the **CNC10 Main** Screen.
CYCLE START	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. For more information also see chapter 5.

PRESS	ACTION	COMMENTS
F1 - Setup		From the main screen enter the Setup.
F2 - Tool		Enter tool screen.
F1 - Offset Lib.		Enter the tool offsets screen.
JOG ARROWS		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 the quill up to the Z home position. Insert the first tool to measure.
ARROWS		Select height offset which holds the height of the first tool.
JOG ARROWS		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 - Manual) for each additional tool to measure.
F10 - Save		Store modifications to offset library of your tools.

Chapter 11
CNC Program Codes

General

The next three chapters contain a description of the CNC program codes and parameters supported by the M-Series Control. The M-Series Control has some G codes and parameters that are modal, and some that are "one shots." The G codes and parameters that are modal will stay in effect until a new G code or parameter is issued. One shots are effective for the current line only. For example, a movement command of G1, which is modal, will remain in effect until a different movement command is issued, such as G0, G2, G3, etc.

Miscellaneous CNC Program Symbols

D - Tool Diameter Offset Number

D is used to select the Tool Diameter Offset from the offset library. The D code values are stored in the Offset Library. Tool Diameter Offsets can be specified anytime before Cutter Comp is turned on (G41 or G42). Once specified, the offset amount is stored and will only be changed when another D code is entered therefore, D is modal. 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 E18 select among the 18 work coordinate systems. For more information on work coordinate systems see chapter 4.

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% for DC systems and 2 – 100% for AC systems). 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 therefore, H is modal. 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:

```
H1          ; Selects offset corresponding to H1.
G43 Z3      ; Moves to Z3 using H1 offset.
G1X0Y1      ;
H3          ; Selects offset corresponding to H3.
X1Y1.25     ;
G0H5        ; Selects offset corresponding to H5.
```

* NOTE: For editing instruction for the offset library see chapter 5. For information on length compensation functions see G43 and G44 in Chapter 12.

N - Block Number

Block numbers are used to identify CNC program lines. Block numbers are optional, but can be used as the destinations of GOTO statements (see Advanced Macros in this chapter) and targets of the Search Function (See Main Screen Search option in Chapter 3). 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       ; sets tool diameter #5 to 0.5” in the offset library
G81 X0 Y0 Z-.5 R.1 F15 ; drill to Z-.5 with return height of .1
```

S - Spindle Speed Setting

Specifying a spindle speed causes the automatic spindle speed setting to be immediately updated. Setting the spindle speed does not cause the spindle to start. The maximum spindle speed is used to compute the output value to the spindle speed control circuit.

Example:
```
S1400 M3    ; Starts the spindle CW at 1400 RPM
```

* NOTE: The Spindle Speed is used in conjunction with the maximum spindle speed to determine the actual spindle speed output to the PLC. Also, this only works when a VFD (Variable Frequency Drive) spindle drive is connected.

T - Select Tool

Prompts the operator to insert the proper tool or change tools, when M6 is encountered.

Example:
```
T1 M6  ; Prompt operator to load tool number 1
T2     ; no action
G0X0Y0 ; move to X0 Y0
M6     ; prompt operator to load tool number 2
```

: - Visible Comment Identifier

The colon (:) is used to indicate the start of a comment line within a CNC program. The colon must be the first character on the line.

Example:
```
: select absolute positioning
G90
: XY plane
G17
```
:Visible comments will be displayed on screen with the G-codes.

; - Internal Comment Identifier

The semicolon (;) is used to indicate the start of an internal comment within a CNC program line. All characters after the semicolon are ignored when the program is run. Internal comments are used to document NC programs or temporarily omit the remainder of a line.

Example:

```
G90    ; select absolute positioning
G17    ; XY plane
G1 X1 Y1 F10
G0 ; X0 Y0  ; G0  selected with no movement
```

[ ] – Numerical Expression

The left bracket '[' and right bracket ']' are used to delimit a numerical expression. Numerical expressions can contain floating-point numbers or user and system variables in combination with mathematical operators and functions. The left parenthesis '(' or bracket '[' and right parenthesis ')' or bracket ']' can be used between the first left bracket and last right bracket to force operator precedence or associatively. A bracketed numerical expression can be used anywhere a number would be used. Comparison operators ('eq', 'ne', etc.) have built in rounding specified by parameter 144. Without this rounding, 'eq' would usually return "false" when comparing two numbers calculated in different ways. Comparison operators and logical operators ('!', '&&', '||') return 1.0 for "true" and 0.0 for "false".

The mathematical operators and functions are:

+	Addition (or unary positive)	eq or ==	Equals
-	Subtraction (or unary negative)	ne or !=	Not equals
*	Multiplication	ge or >=	Greater than or equals
/	Division	gt or >	Greater than
^	Exponentiation	le or <=	Less than or equals
Mod or %	Modulo (remainder of a division)	lt or <	Less than
abs	Absolute value	not or !	Logical not
sin	Sine (degrees)	&&	Logical and
cos	Cosine (degrees)	||	Logical or
tan	Tangent (degrees)	and	Bit-wise and
sqrt	Square root	xor	Bit-wise exclusive or
#	Variable access	or	Bit-wise or
		~	Bit-wise complement

Examples:

```
G91 X[13/64] Z[1+3/8]  ; move the X axis 13/64 (0.2031) units
                       ; and the Z axis 1 3/8 (1.375) units incrementally
X[SQRT[ABS[SIN[#101]-COS[#102]]]] ; Move X as a function of #101 and #102
```

#, = - User or System Variable Assignment

The '#' character is used to reference a macro or a user or system variable. For variables that can be written, the '=' is used to assign to them.

Index	Description	Returns	R/W
1-3	Macro arguments A-C	The floating point value if defined by a G65 call, 0.0 otherwise.	R/W
4-6	Macro arguments I-K (1st set)		R/W
7-9	Macro arguments D-F or 2nd set of I-K		R/W
10	3rd I (G is invalid)		R/W
11	Macro argument H or 3rd J	These can be used as private,	R/W

12	3rd K (L is invalid)	local variables in any program or subprogram. (See examples.)	R/W
13	Macro argument M or 4th I		R/W
14	4th J (N is invalid)		R/W
15	4th K (O is invalid)		R/W
16	5th I (P is invalid)		R/W
17-18	Macro argument Q-R or 5th J-K		R/W
19-21	Macro arguments R-T or 6th set of I-K		R/W
22-24	Macro arguments U-W or 7th set of I-K		R/W
25-27	Macro arguments X-Z or 8th set of I-K		R/W
28-30	9th set of I-K		R/W
31-33	10th set of I-K		R/W
100 - 149	User variables	Floating-point value. Initialized to 0.0 at start of job processing	R/W
150 – 159	Nonvolatile user variables	Floating-point value saved in CNC10.JOB file.	R/W
2400, 2401-2418	Active WCS, WCS #1-18 CSR angles		R/W
2500, 2501-2518	Active WCS, WCS #1-18 Axis 1 values (X)	Floating point value	R/W
2600, 2601-2618	Active WCS, WCS #1-18 Axis 2 values (Y)		R/W
2700, 2701-2718	Active WCS, WCS #1-18 Axis 3 values (Z)		R/W
2800, 2801-2818	Active WCS, WCS #1-18 Axis 4 values (W)		R/W
3901	Parts Cut (Part #)		R/W
3902	Parts Required (Part Cnt)		R/W
4001	Move mode	0.0 (rapid) or 1.0 (feed)	R
4002	Constant surface speed mode (lathe only)	96.0 (on) 97.0 (off)	R
4003	Positioning mode	90.0 (abs) or 91.0 (inc)	R
4005	Feedrate mode (lathe only)	98.0 (units per min) or 99.0 (units per rev)	R
4006	Units of measure	20.0 (imp) or 21.0 (metric)	R
4014	WCS	54.0-71.0 (WCS#1-18)	R
4109	Feedrate (F)	Floating point value	R
4119	Spindle Speed (S)		R
4120	Tool Number (T)		R
4121	Mill: Current height offset number (H) Lathe: Current offset (“oo” in “Tttoo”)		R
4122	Current diameter offset number (D, mill only)		R
4201	Job processing state	0 = normal, 1 = graph	R
4202	Search mode (0 = search mode off)	0 = search mode off	R
5021-5024	Machine Position (X=5021, Y=5022, etc.)	Floating point value	R
5041-5044	Current Position (X=5041, Y=5042, etc.)		R
6001-6080	PLC Inputs 1 - 80	Least significant bit is lowest numbered PLC bit. 0 = closed, 1 = open	R
6900-6909	PLC Inputs, eight at a time.		R
7001-6080	PLC Outputs 1 - 80		R
7900-7909	PLC Outputs, eight at a time.		R
8001-6080	PLC Memory bits 1 - 80		R
8900-8909	PLC Memory bits, eight at a time.		R
9000-9199	Parameter values 0 – 199	See Chapter 14.	R

Index	Description	Returns	R/W
10000	Mill: Height offset amount, active H	Floating point value	R/W
10001-10200	Mill: Height offset amount, H001 – H200	Floating point value	R/W
11000	Mill: Diameter offset amount, active D	Floating point value	R/W
11001-11200	Mill: Diameter offset amount, D001 – D200	Floating point value	R/W
12000	Mill: Tool H number, active tool (T)	0 - 200	R/W
12001-12200	Mill: Tool H number, tools 1 - 200	0 - 200	R/W
13000	Mill: Tool D number, active tool (T)	0 - 200	R/W
13001-13200	Mill: Tool D number, tools 1 - 200	0 - 200	R/W
14000	Mill: Tool coolant, active tool (T)	7, 8, 9	R/W
14001-14200	Mill: Tool coolant, tools 1 – 200	7, 8, 9	R/W
15000	Mill: Tool spindle direction, active tool (T)	3, 4, 5	R/W
15001-15200	Mill: Tool spindle direction, tools 1 - 200	3, 4, 5	R/W
16000	Mill: Tool spindle speed, active tool (T)	Floating point value	R/W
16001-16200	Mill: Tool spindle speed, tools 1 - 200	Floating point value	R/W
17000	Mill: Tool bin number, active tool (T)	Floating point value	R/W
17001-17200	Mill: Tool bin number, tools 1 - 200	Floating point value	R/W
18000	Mill: Tool putback, active tool (T)	Floating point value	R/W
18001-18200	Mill: Tool putback, tools 1 – 200	Floating point value	R/W
24103	Z axis reference (machine position)	Floating point value	R
25010	Calculated spindle speed if P78 = 1.0 and a spindle encoder is mounted.	Integer	R
29000-29999	User variables. These variables retain their values until the CNC software is exited.	Floating point value	R/W

Examples:

```
#100 = #5041        ; set user variable #100 to the X axis current position
G90 X[#5041+1+7/32] ; move the X axis 1 7/32 units (1.2188) incrementally
#2501 = #5021       ; set WCS#1 X value to the current X position
#2703=[#2703+1/8]   ; add 1/8 units (.125) to the WCS#3 Z value

; Subroutine parameter and local variable access.
G1 X#A Y#B Z#C F#F  ; move to the coordinates passed as parameters
#[Q] = #F * .10     ; Assign local variable #Q to 10% of #F
#17  = #7 * .10     ; Same statement as previous using number references.
#[C] = 0.05         ; Reassign #C. (Value passed as parameter is lost.)
```

Advanced Macro Statements (Optional)

Warning: Branching and conditional execution are extremely powerful tools that, combined with access to system variables, allow you to do many things that would otherwise be impossible. Nevertheless, using branching and conditional execution can introduce undesirable and even unpredictable behavior into your programs. Undesirable effects can occur simply by graphing a program. The least of these undesirable effects could be entering an endless loop, failing to draw anything, or wiping out all the information in your tool library or WCS settings. It is your responsibility to make sure that undesirable things do not happen in your programs. You must monitor the job processing and search modes in your program, if necessary, and take appropriate action. Until you are confident of the actions of your program, you should step through it one block at a time to confirm your program logic.

GOTO - Branch Execution

To branch to another line within the same program or subprogram, use the statement:

GOTO <expression>

where <expression> is any expression that evaluates to a valid block number in the program. GOTO causes an immediate branch to the specified destination. Program codes preceding a GOTO on the same line will be executed normally. Any program codes following GOTO on the same line will cause an error.

If fast branching is disabled (parameter 145 = 0) then CNC10 searches forward in the program for the first matching block number and resumes searching, if necessary from the top of the program. For this reason when fast branching is disabled, backward branches take longer than forward branches and backward branch times depend on the total program size. If the program is sufficiently large, use of the GOTO statement could introduce temporary pauses.

When fast branching is enabled (parameter 145 = 1) then CNC10 remembers the locations of block numbers as it finds them during program execution. Backward branches always take place immediately. The first forward branch to a block not yet encountered will take additional time as CNC10 searches forward for the block number; however, subsequent forward branches to that block number will take place immediately. The trade-off for using fast branching is that all line numbers at a given level of program or subprogram must be unique and programs will use more memory (approximately 16kilobytes of memory for every 1000 block numbers in the program.)

IF THEN ELSE - Conditional Execution

Program symbols, G codes, M codes and GOTO commands may be executed conditionally using the IF statement. The general form of the IF statement is:

IF <expression> THEN <execute if true> ELSE <execute if false>

where <expression> is any valid expression, <execute if true> is one or more program codes to execute if <expression> evaluates to “true” (non-zero) and <execute if false> is one or more program codes to execute if <expression> evaluates to “false” (zero). All parts of the IF statement must appear on the same line. The “ELSE <execute if false>” part of the statement is optional and may be omitted. The “THEN” may be omitted; however, <expression> must be enclosed in brackets ([]). The IF statement may follow other program codes on the same line. Compound conditionals are possible but they cannot be nested. The first THEN always pairs with the first IF. ELSE always pairs with the first <expression> that evaluates to “false”. All program codes executed are executed as part of the same block.

Examples:

```
; Branch to N200 if machine position is okay, otherwise go to N300
N100 IF #5041 LE 5.0 THEN GOTO 200 ELSE GOTO 300
; Force subprogram parameter #D to be within range.
IF [#D LE 0.005] #[D] = 0.005
; Compound conditionals
IF [#A LE 0.0] GOTO 100 ELSE IF [#A LE 2.5] GOTO 200 ELSE GOTO 300
IF [#A GT 0.0] IF [#D/#A GE 0.0] #[C] = SQRT[#D/#A]
```

INPUT – Prompt Operator for Input

The INPUT macro prompts the operator for numeric input. The general form of the INPUT statement is:

INPUT "<prompt>" <variable>

Where <prompt> is the message prompt for the operator and <variable> is the variable in which to store the input. CNC10 will display a dialog with the given prompt and space for the operator response. The operator may enter any numeric expression (see above) including variables as a response. The operator must press **CYCLE START** or **Alt-S** to dismiss the dialog. Pressing **ESC** will cancel the job.

CNC10 parses well ahead of the current execution to maximize throughput and efficiency. For this reason, an INPUT macro may prompt the operator for input immediately even though the INPUT macro is located in the middle or near the end of the job. Parsing pauses while the dialog is displayed. Any statements parsed prior to the INPUT macro will have been queued and will continue to execute in the background while the prompt is displayed. Job processing will pause only if all queued statements have been executed before the operator supplies a response.

INPUT macros will not graph. If you must graph the job, first set the input variable to a default value and use a conditional to execute the INPUT only if the job is being run normally.

Use search mode cautiously with INPUT macros. To have search work properly, you may have to supply exactly the same input during the search as you did during the last actual run.

Examples:

```
; Ask operator for pocket depth.  Store result in #101
; Note: this will not graph.
INPUT "Enter pocket depth" #101

; Allow job with INPUT statements to be graphed.
#101 = 0.5 ; Supply a default value for graphing
; Ask for operator input only if not graphing.
IF NOT #4201 THEN INPUT "Enter pocket depth" #101
```

Chapter 12
CNC Program Codes: G-codes

G-code		Group	Description
G00	*	A	Rapid Positioning
G01		A	Linear Interpolation
G02		A	Circular or Helical Interpolation CW
G03		A	Circular or Helical Interpolation CCW
G04		B	Dwell
G09		B	Exact Stop
G10		B	Parameter Setting
G17	*	C	Circular Interpolation Plane Selection XY
G18		C	Circular Interpolation Plane Selection ZX
G19		C	Circular Interpolation Plane Selection YZ
G20	*	K	Select Inch Units
G21		L	Select Metric Units
G22		O	Work envelope on
G23	*	O	Work envelope off
G28		B	Return to Reference Point
G29		B	Return from Reference Point
G30		B	Return to Secondary Reference Point
G40	*	D	Cutter Compensation Cancel
G41		D	Cutter Compensation Left
G42		D	Cutter Compensation Right
G43		E	Tool Length Compensation (+)
G44		E	Tool Length Compensation (-)
G49	*	E	Tool Length Compensation Cancel
G50	*	M	Scaling/Mirroring Off (Optional)
G51		M	Scaling/Mirroring On (Optional)
G52		B	Offset Local Coordinate System Origin (Optional)
G53		B	Rapid Position in Machine Coordinates (Optional)
G54		L	Select Work Coordinate System #1 (Optional)
G55		L	Select Work Coordinate System #2 (Optional)
G56		L	Select Work Coordinate System #3 (Optional)
G57		L	Select Work Coordinate System #4 (Optional)
G58		L	Select Work Coordinate System #5 (Optional)
G59		L	Select Work Coordinate System #6 (Optional)
G61		F	Exact Stop Mode
G64	*	F	Cutting Mode
G65		J	Call Macro (Optional)
G68		N	Coordinate Rotation on
G69	*	N	Coordinate Rotation off
G73		G	High Speed Peck Drilling
G74		G	Counter Tapping (Optional)
G76		G	Fine Bore Cycle
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 " ***** ".

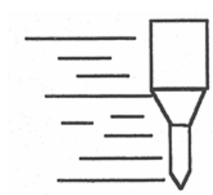
- A given line of a program may contain more than one G-code.

- 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.

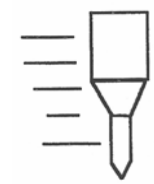
When the Z axis is commanded to move in the - direction, all axes but the Z axis will move to their new position along a straight line, then the Z axis will move down to its new position.

Example:

```
G0 X0.0 Y0.0 Z0.0      ;Rapid move to X0, Y0, Z0
```

* NOTE: G0 moves are only affected by the feedrate override knob if rapid override is ON.

G01 - Linear Interpolation

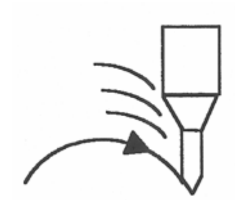


G1 moves to the specified position at the programmed feedrate. The coordinates may be either absolute positions (G90) or incremental positions (G91). The movement will be along a straight line. G1 is modal and remains in effect until another positioning mode (G0, G2, G3 etc.) is commanded.

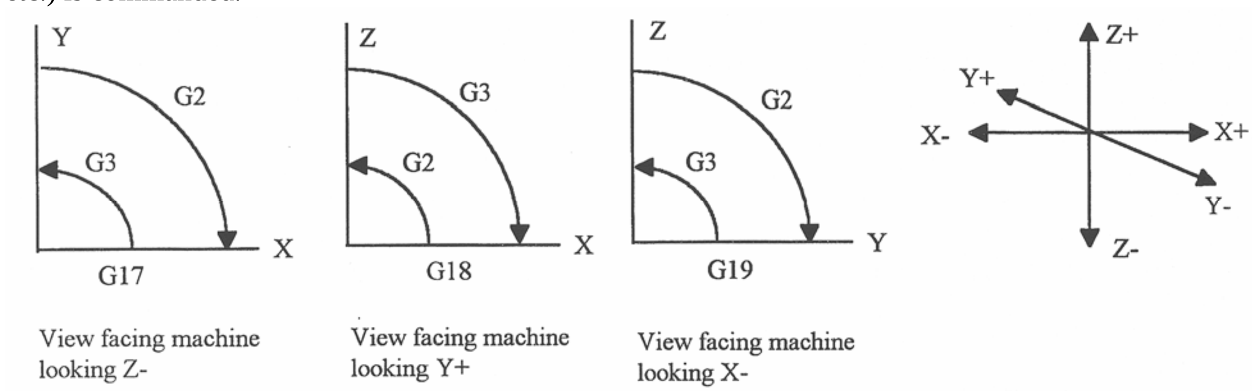
Example:

```
G01 X2 Y3 Z4 W5 F10     ;Linear move to X2, Y3, Z4, W5 at a 10in/min
G91 X6 Y7               ;Linear move to X8, Y10
Z3 W4 F20               ;Linear move to Z7, W9 at 20in/min (G91 is modal)
```

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.



* **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.

<u>METHOD 1: USING FINAL POINT AND RADIUS</u>

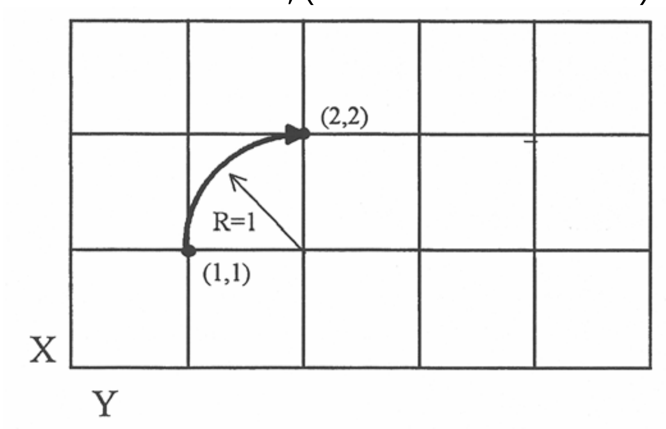
The commands G2 and G3 will have the following structure:

```
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.

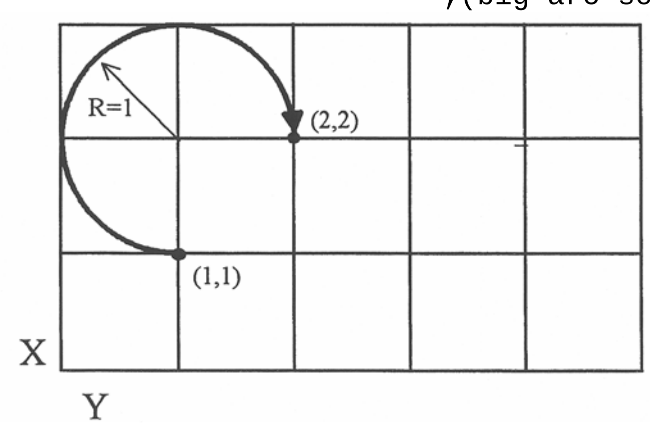
Example 1 (small arc solution: positive radius):

```
G17 G90 F25             ;selects 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             ;selects 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)
```



<u>METHOD 2: USING FINAL POINT AND PARAMETERS I, J, K</u>

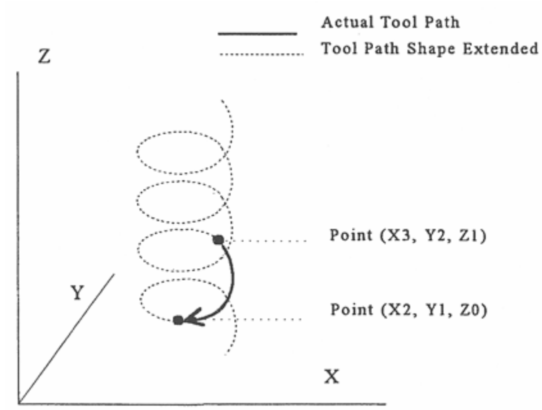
Another way to specify a helical or circular operation is using the parameters I, J, K instead of the radius R. The parameters I, J, and K are the **incremental** distances from the start point to the center of the arc. For absolute positioning on I, J, and K, parameter 2 bit 0 will need set. See the parameter section in Chapter 14.

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:

<u>Circular motion</u> (See graph in method 1, example 2)

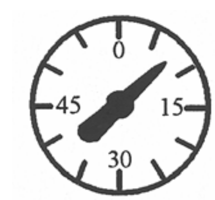
```
G17 G90 F25           ;selects XY plane and absolute positioning
G00 X1.0 Y1.0 Z0      ;rapid to start position X1, Y1, Z0
G02  X2 Y2 Z0 J1      ;arc to X2 Y2 Z0 with radius of 1
```



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



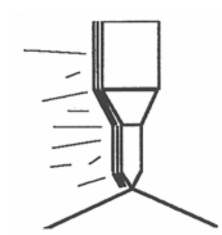
G4 causes motion to stop for the specified time. The P parameter is used to specify the time in seconds to delay. G4 causes the block to decelerate to a full stop.

The minimum delay is 0.01 seconds and the maximum is 327.67 seconds. The dwell time is performed after all motion is stopped and M functions on the line are completed. 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
G1 X2 Y2         ;Linear move to 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 lib.
G10 D3 R.25        ;Sets tool diameter offset #3 to .25 in the offset lib.
```

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.

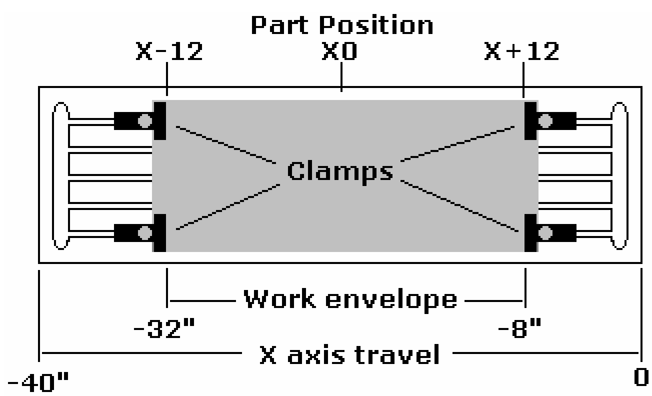
G21 - Select Metric Units

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.

G22/G23 – Work Envelope On/Off

G22 turns on programmable work envelope in machine coordinates. When the machine tries to move into the forbidden area, let's say the x-axis, an "x-axis work envelope exceeded" message is displayed, letting you know which line of the program is at fault. The work envelope is set with the X, Y, Z for the '+' limit and I, J, K for the '–' limit. G22 is modal and remains on until turned off by G23 or the end of the job. The limits entered in the X, Y, Z and I, J, K parameters are stored in the WCS menu under **F3 - Work Envel**. For more information see chapter 4.

Example: (Machine homes to the X + switch and has 0 to –40 inches of travel in the X axis,)




```
G22 X-8 I-32       ;Keeps programs from moving into the outside 8 inches of
                   ;X axis of travel
G1 X-13 F20        ;Would generate a "X axis work envelope exceeded, line 3"
                   ;message
G23                ;Allows travel into G22 forbidden area.
M25                ;Z home
G0 X-13            ;Ok to move X here now
```

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 -0.5 (from current position), then move all
                       ;three axes to reference point
G28 G90 X2 Y4 Z.1      ;move all axes to (X2, Y4, Z0.1), 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.

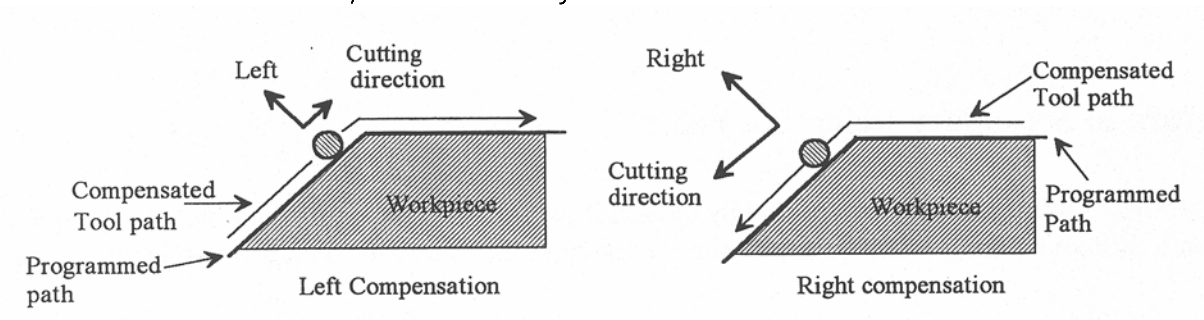
G41 offsets the cutter tool one half of the tool diameter selected with a D code, to the left of the workpiece, relative to the direction of travel.

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:

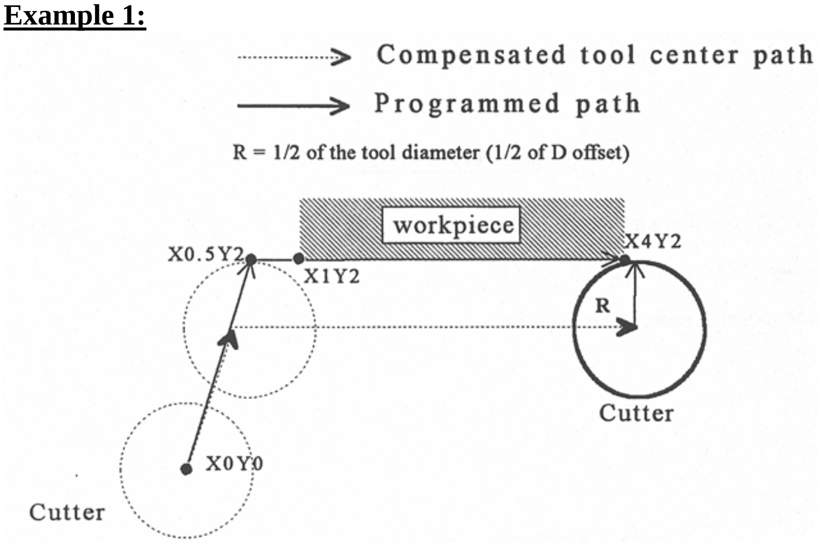
```
G41 D03           ;Tells the machine to compensate left half of the
                  ;diameter of the amount that corresponds to D03 in the
                  ;Tool Library
```



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:



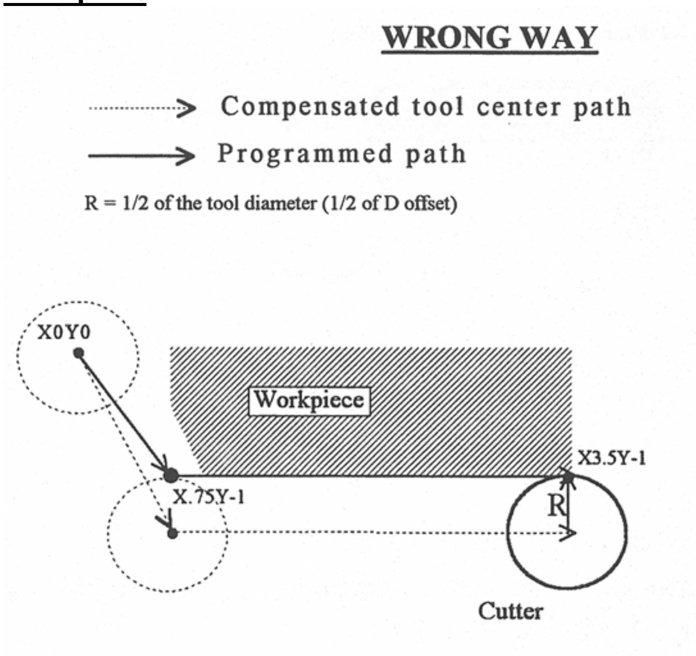

```
G0X0Y0      ;Rapid tool to X0, Y0
G42 D3      ;Turn cutter compensation on, with a diameter of D3
G0X.5Y2     ;Rapid to X0.5, Y2
G1x4.1Y2    ;Linear cut to X4.1, Y2.
            ;Cut to X4.1 to clear material.
G40         ;Turn cutter compensation off.
G0X5Y0      ;Rapid to X5, Y0.
```

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.

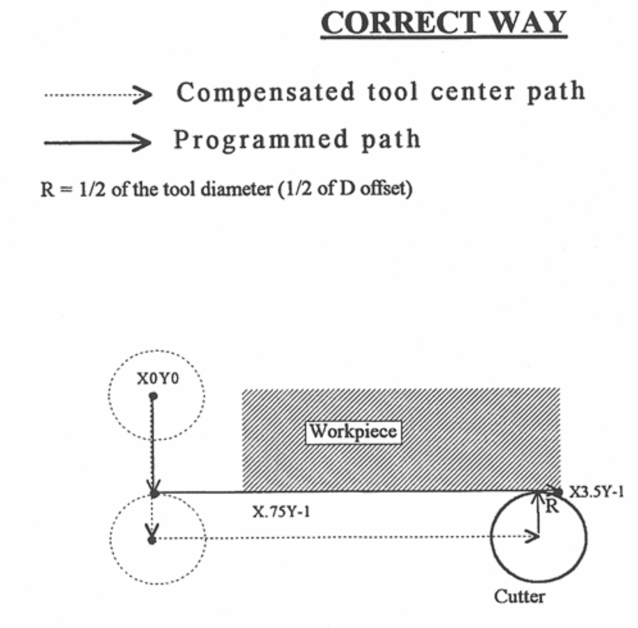
Example 2:




```
G0 X0Y0          ;Rapid tool to X0, Y0
G42 D5           ;Turn cutter compensation on, with a diameter of D5
G1 X.75Y-1 F5    ;Linear move to X0.75, Y-1. (notice this damages the
                 ;corner of the work piece)
X3.6             ;move X to 3.6
G40              ;Turn cutter compensation off.
G0 X4Y-2         ;Rapid to X4, Y-2
```

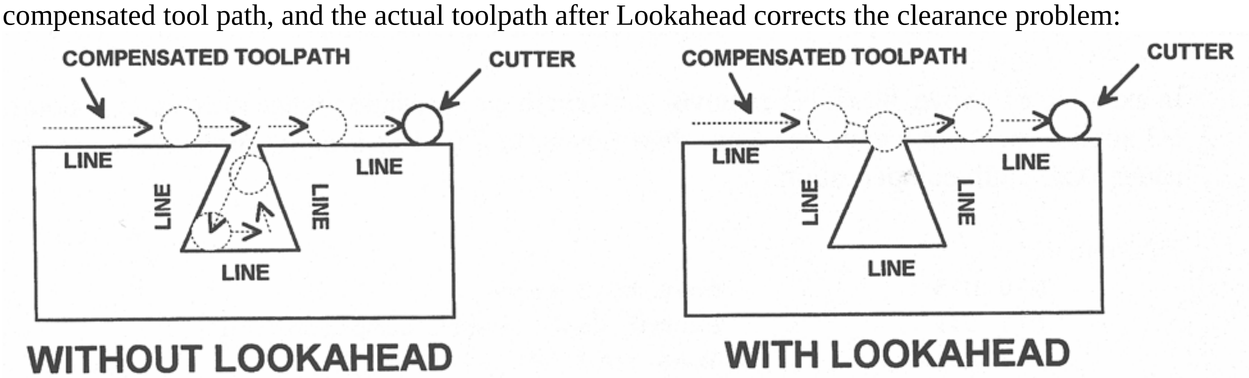
* NOTE: This problem could have been avoided by selecting a transitional point between X0 Y0 and X.75 Y-1. A transitional point such as X-1 Y-1 would properly modify the lead-in path, keeping the cutter from damaging the corner of the workpiece. Example 3 shows the correct way of performing this operation.

Example 3:




```
G0X0Y0           ;Rapid tool to X0, Y0
G42D5            ;Turn cutter compensation on, with a diameter of D5
G0X0Y-1          ;Rapid tool to X0, Y-1
G1X.75Y-1        ;Linear cut to X0.75, Y-1.
X3.6             ;move X to 3.6
G40              ;Turn cutter compensation off.
```

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 14 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:

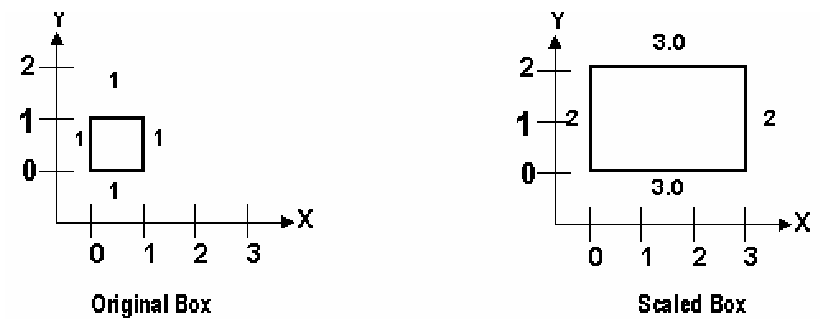
```
G43 H01     ;tells the machine to offset the amount that
            ;corresponds to H01 in the Offset Library
```

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:

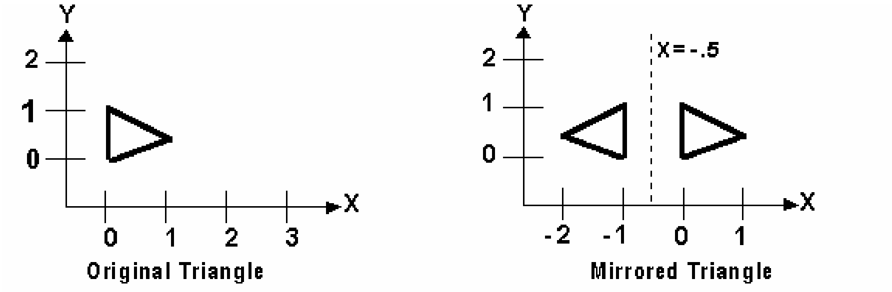
```
G51 X0.0 Y0.0 Z0.0 I3.0 J2 K1     ;turn scaling on
G00 X0.0 Y0.0 Z1.0                ;rapid to X0, Y0, Z1
G01 X1.0 Y0.0 Z1.0                ;line to X1, Y0, Z1
G01 X1.0 Y1.0 Z1.0                ;line to X1, Y1, Z1
G01 X0.0 Y1.0 Z1.0                ;line to X0, Y1, Z1
G01 X0.0 Y0.0 Z1.0                ;line to X0, Y0, Z1
G01 X0.0 Y0.0 Z0.0                ;line to X0, Y0, Z0
G50                               ;cancel scale
```



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 (x axis -0.5 mirror line)
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 mirror
```



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)

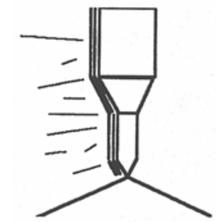
G54 through G59 select among the six work coordinate systems. Subsequent absolute positions will be interpreted in the new coordinate system.

Example:

```
G54 G0 X0 Y0 Z0         ;select first WCS, move to origin
G2 X1 I.5 Z-.5          ;mill something...
G0 Z.1                  ;Rapid to position Z0.1
G55 X1 Y1               ;select second WCS, move to X1, Y1
```

Using Extended Work Coordinate Systems: There are actually a total of 18 work piece origins. They can be set using Set Part 0/Position or in the WCS Table. In G-codes, the 12 additional work piece origins may be selected with either "G54 P1" (WCS # 7) through "G54 P12" (WCS #18) or "E7" through "E18." The E parameter can be used for all work coordinates (E1 through E18).

G61 - Exact stop mode

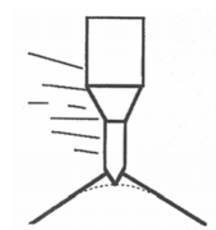


G61 invokes the exact stop mode. This forces deceleration to an exact stop at the end of each block (equivalent to G9 in each block). G61 is modal and remains in effect until canceled with G64 (cutting mode).

Example:

```
G0 X0 Y0                ;move to origin
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                ;move to origin
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" L*rrrr Arguments*

Where *xxxx* is the macro number (in file O*xxxx*.CNC, 0000-9999 allowed, leading zeros required in filename), *rrrr* is the repeat value, "program.CNC" is the name of the macro file, and *Arguments* is a list of variable identifiers and values.

Arguments to macro calls are specified by using letters A-Z, excluding G, L, N, O, and P.

Macros are written just like normal programs. However, macro programs may access their arguments by using #A, #B, etc., or by using numbers: #1 for A, #2 for B, etc. (exceptions: #4-6 for I-K, #7-11 for D-H). Arguments I, J, and K can be used more than once in a macro call, with the first set of values stored as #4-6, the second as #7-9, etc., to a maximum of 10 sets.

* See example at the end of this G65 section.

A macro can use the negative of an argument by placing a minus sign before the '#'. No other arithmetic operations are supported. Macros can call other macros (up to 4 levels of depth), Macro M-functions, and subprograms. Macro M-functions and subprograms can similarly call macros.

Macros 9100 - 9999 may be embedded into a main program, using 091*xx* to designate the beginning of the macro and M99 to end it. CNC10 will read the macro and generate a file 091*xx*.CNC, but will not execute the macro. It will be executed when G65 is issued.

Example 1:

Main program:
```
G65 "TEST.CNC" A5 B3 X4
```

Macro TEST.CNC:
```
G1 X#X Y#A Z-#B
```

This call will produce
```
G1 X4 Y5 Z-3
```

Example 2:

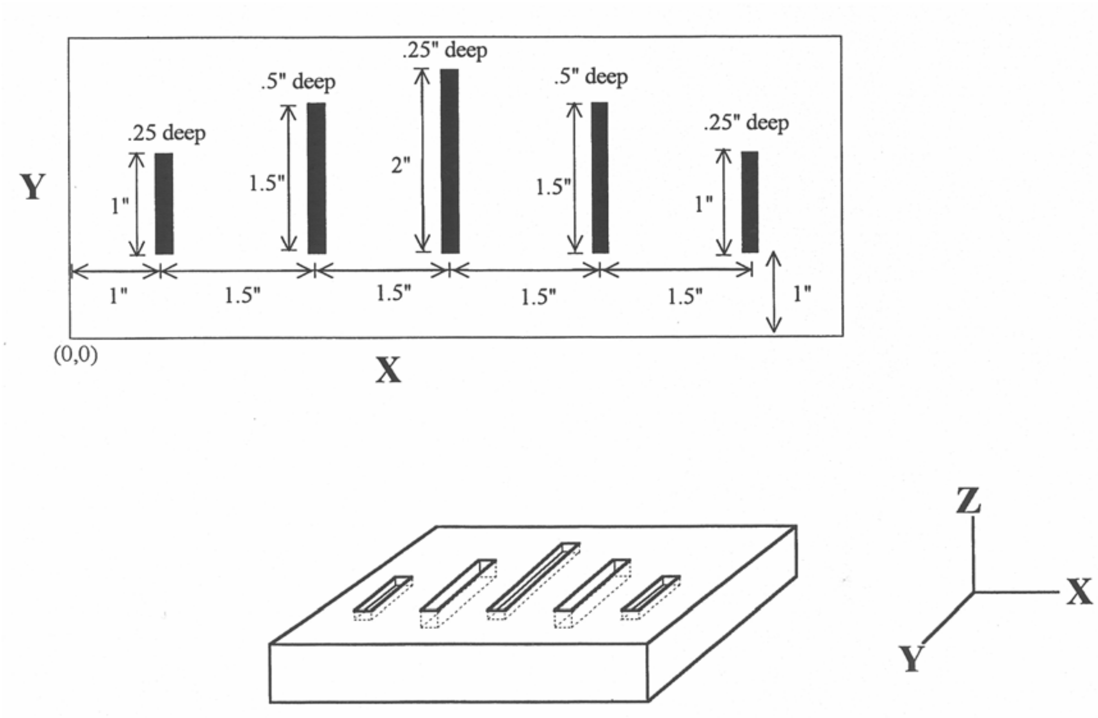
Main program:
```
G65 "TEST2.CNC" I5 J3 K40 I-1 J2 I0 J0
```

Macro TEST2.CNC:
```
G1 X#4 Y#5 F#6
G1 X#7 Y#8 F#9
G1 X#10 Y#11 F#12
```

This call will produce
```
G1 X5 Y3 F40
G1 X-1 Y2
G1 X0 Y0
```

Example 3:

Suppose a piece is to have notches of different lengths and depths along the x-axis:



The macro variables would handle the length in the Y direction and depth in the Z direction:

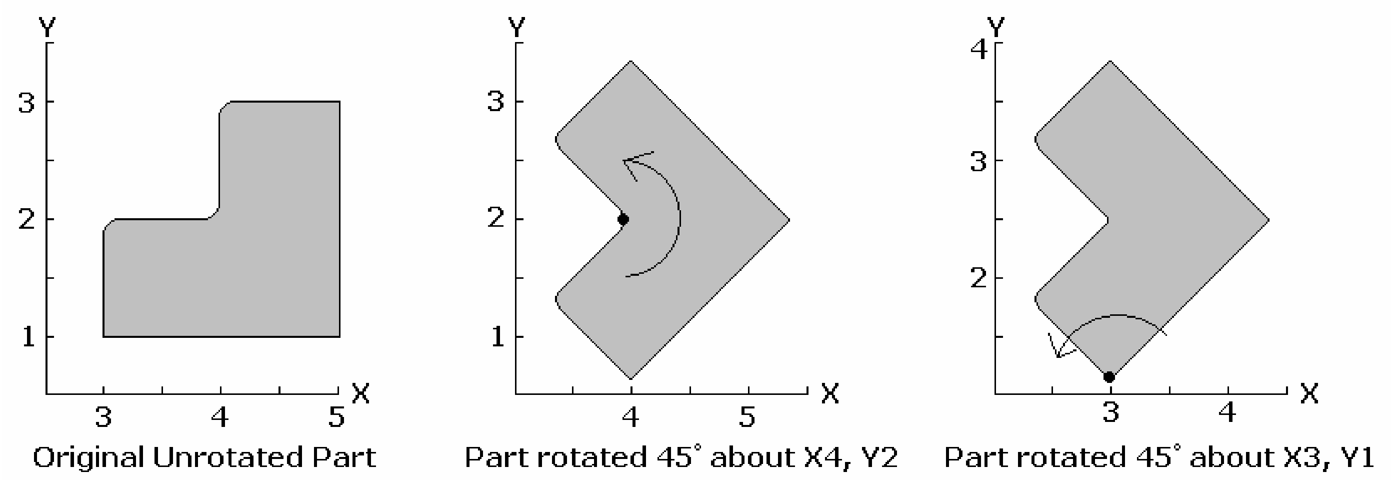
```
O0002
G90 G1 Z0 F30          ;Linear move to Z0
Z#Z F5                 ;Cut to variable depth
G91Y#Y F10             ;Cut variable length
G90 G0 Z0.1            ;Retract
```

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=.25"
G90 G0 X2.5 Y1             ;Move to second notch
G65 P0002 L1 Y1.5 Z.5      ;Call macro and assign Y=1.5" and Z=.5"
G90 G0 X4 Y1               ;Move to third notch
G65 P0002 L1 Y2 Z.25       ;Call macro again
G90 G0 X5.5 Y1             ;Move to fourth notch
G65 P0002 L1 Y1.5 Z.5      ;Call macro again
G90 G0 X7 Y1               ;Move to fifth notch
G65 P0002 L1 Y1 Z.25       ;Call macro again
:End program
```

G68, G69 - Coordinate Rotation on/off

G68 rotates program G-codes a specified angle R. G68 rotates all positions, lines, and arcs until a G69 is entered. The center of rotation can be specified by X, Y and Z values (X, Y for G17 plane). If the center is not specified then a default center of rotation is used as determined by machine parameter #2 (see Chapter 14 for parameter #2). The default plane of rotation is G17 (X, Y).



Example:

```
G68 R45 X4 Y2              ;Rotate 45 degrees centered on X4 Y2
G0 X3.0 Y1.0               ;Rapid to position
G1 X5.0 Y1.0 F20           ;Start part profile
X5.0 Y3.0
X4.125 Y3.0
G3 X4.0 Y2.875 J-0.125
G1 X4.0 Y2.125
G2 X3.875 Y2.0 I-0.125
G1 X3.125 Y2.0
G3 X3.0 Y1.875 J-0.125
G1 X3.0 Y1.0               ;End part profile
G69                        ;Rotate Off
```

G73, G80, G81, G82, G83, G85, G89 - Canned Drilling Cycles

G74, G84 - Canned Tapping Cycles

G code	-Z direction (machine hole)	Operation at bottom of hole	+Z direction	Use
G73	Intermittent Feed (Set with the Q parameter)	----------	Rapid traverse	High speed peck drilling cycle
G74	Feed	Spindle CW, then Dwell (Set with the P parameter) parameter)	Feed	Counter tapping (Left-hand thread)
G80	----------	----------	----------	Cancels 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, spot facing
G83	Intermittent Feed (Set with the Q parameter)	----------	Rapid traverse	Peck and deep hole drilling cycles
G84	Feed	Spindle CCW, then Dwell (Set with the P parameter)	Feed	Tapping (Right-hand thread)
G85	Feed	----------	Feed	Boring cycle
G89	Feed	Dwell (Set with the P parameter)	Feed	Boring cycle

Table 1. Canned drilling and tapping cycles

Canned Cycle Operation

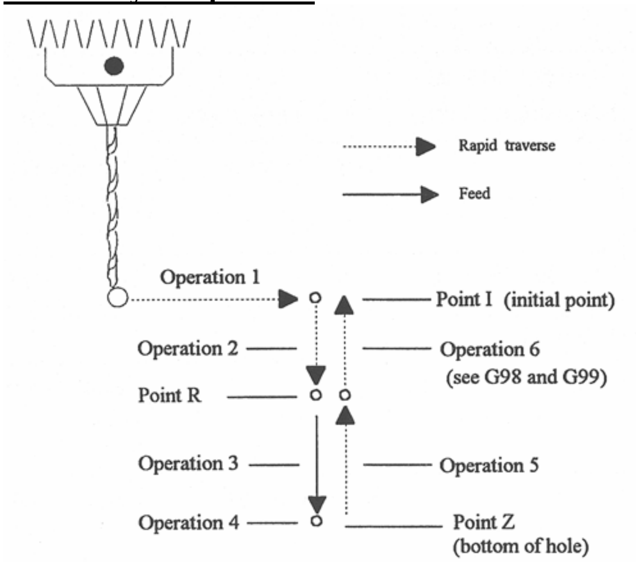
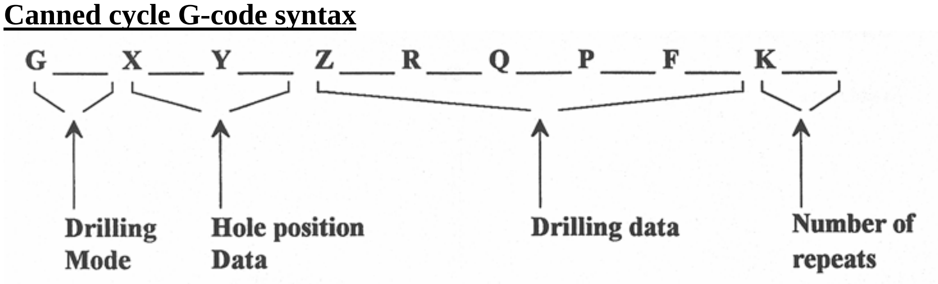


Figure 1. Drilling cycle operation

Operation 1:	Position the X, Y axes.
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 <u>not</u> 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 <u>even after</u> 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.

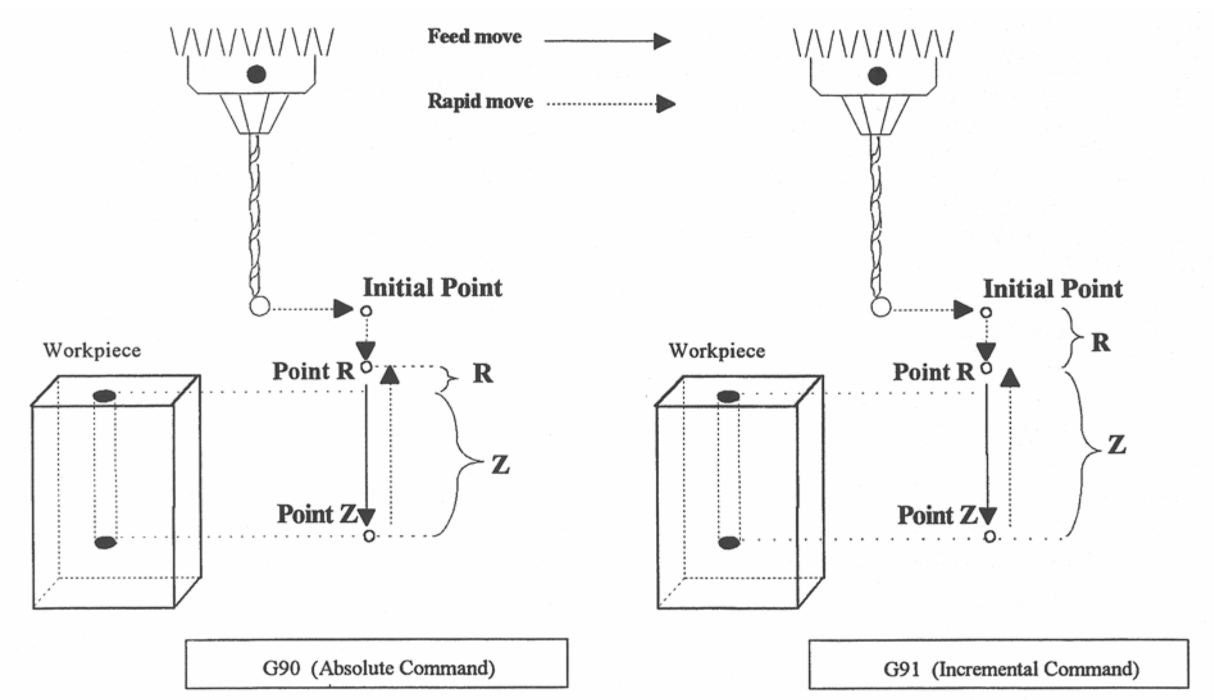


Figure 2: Canned Cycle Absolute and Incremental modes

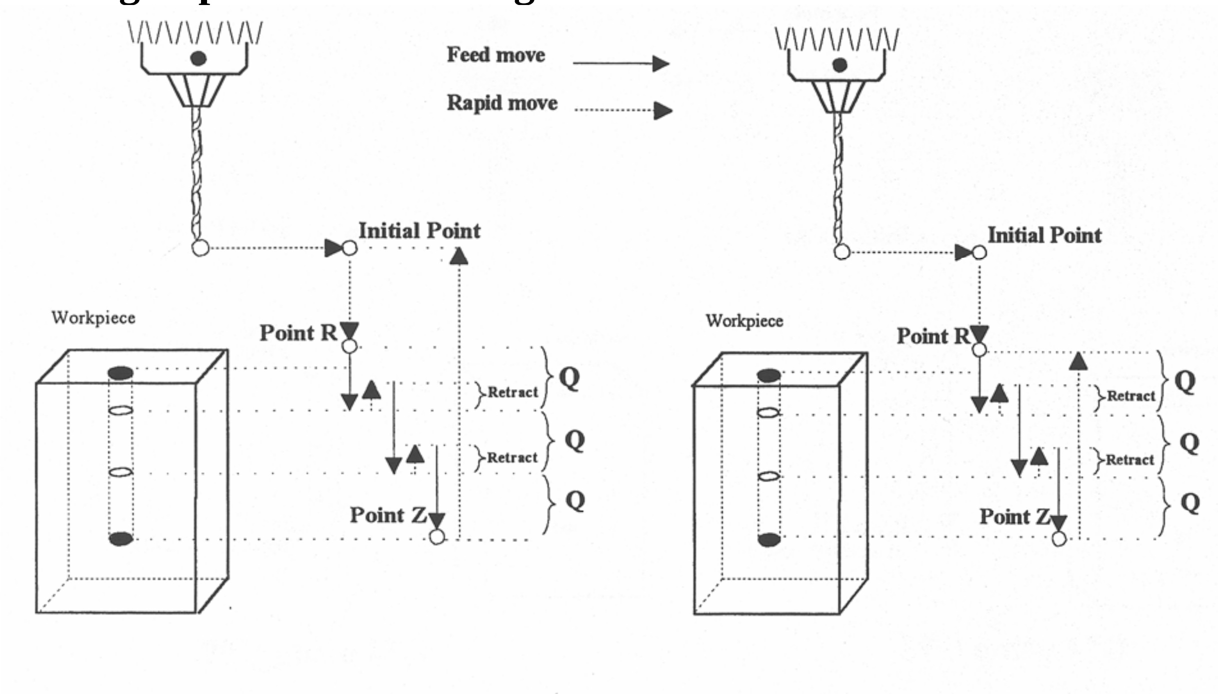
* NOTE: In incremental mode the Z depth of the hole is measured from R, and R is measured from the initial tool position.

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.)

<u>Absolute</u>	<u>Incremental</u>
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

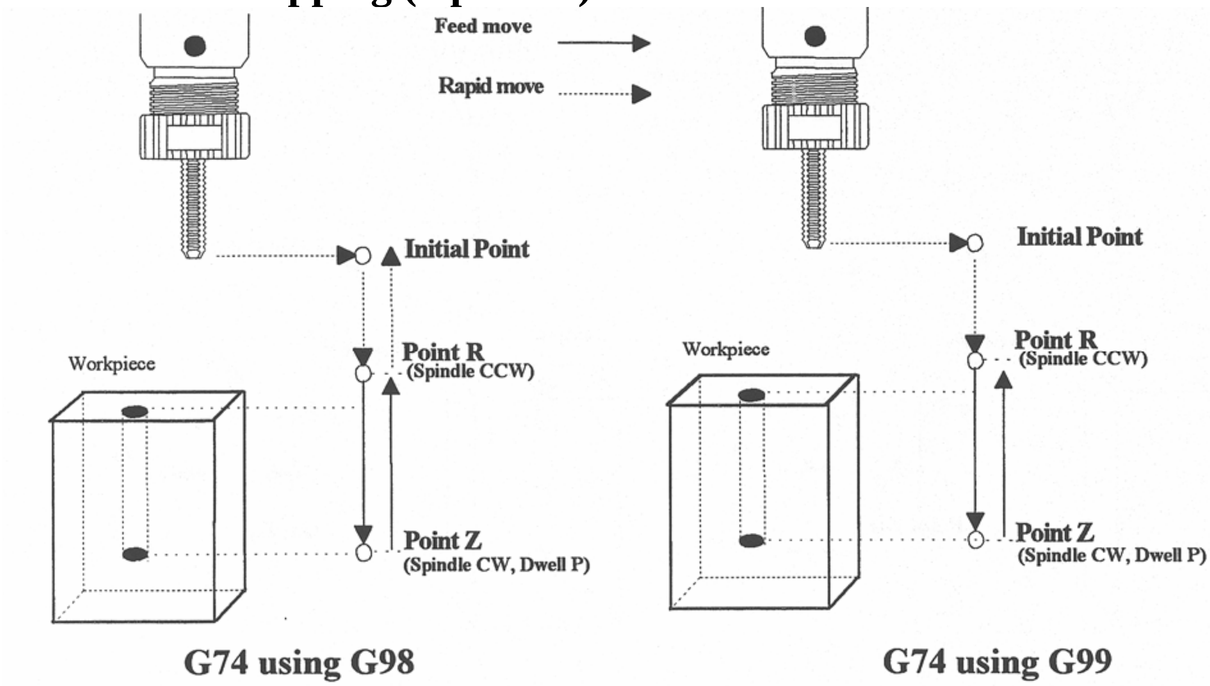
G73 using G99

G73 is the peck drilling cycle. The hole is drilled in a series of moves: down a distance Q at a given feedrate, up the retract distance at the rapid rate, and then down again at the given 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.

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
```

G76 – Fine Bore Cycle

WARNING!!! G76 requires that the machine be capable of orienting the spindle and that a custom M19 macro is present in order to command the inverter to orient the spindle. Please contact your dealer to confirm that your machine meets these requirements before attempting to use this cycle.

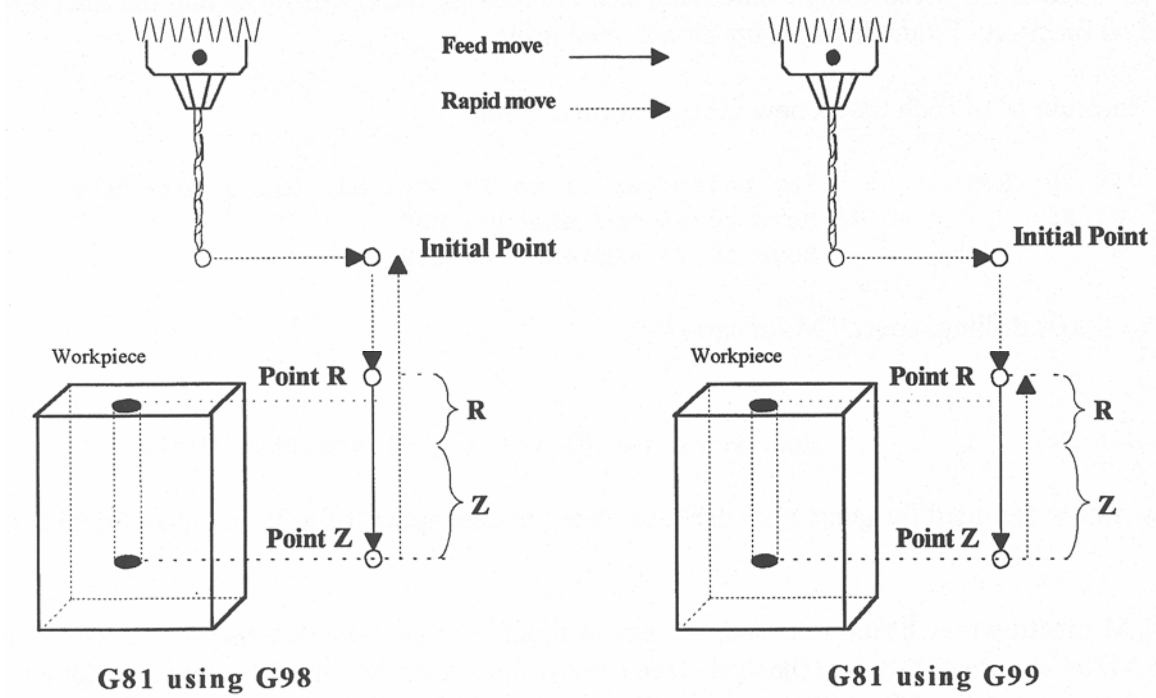
Format G76 X__ Y__ Z__ R__ Q__

R = Point R Q = Distance to pull away from wall in Y+ direction at bottom of hole.

Example:

```
G76 X1 Y1 Z-3 R.1 Q.2        ;Bore hole at X1 Y1 retract .2 in Y+ direction
Y10                           ; ... and another one at X1 Y10
G80                           ;cancel canned cycles
```

G81 - Drilling and Spot Drilling



G81 is a general purpose drilling cycle. The hole is drilled in a single feedrate move, and then the tool is retracted at the rapid rate.

Example:

```
G90                              ;Absolute positioning
G01 X3.00 Y1.50 Z.5              ;G01 mode before canned cycle
G99                              ;Set for R point return
G81 X3.250 Y1.75 Z-.650 R.1 F3   ;Drill at X3.25 Y1.75
X4.5 Y3.5                        ;Drill at X4.5 Y3.5
G80                              ;Cancel canned cycle, return to G1
```

G81 - Drill Cycle Transformation to G81 Air Drill Cycle

G81 may be modified to execute an M function instead of moving the Z-axis by setting parameter #81 to the desired M function. Example use is for air-actuated drills.

Example: Execute M39 each time a new G81 position is given:

```
G10 P81 R39           ;Set parameter 81 to 39 (G81 air drill with M39)
G81 X5                ;Move to X5 and execute M39
Y3                    ;Move to Y3 and execute M39
```

To revert to Z-axis drilling, specify M function #-1.

Example:

```
G10 P81 R-1           ;Set parameter 81 to -1 (G81 drilling cycle)
```

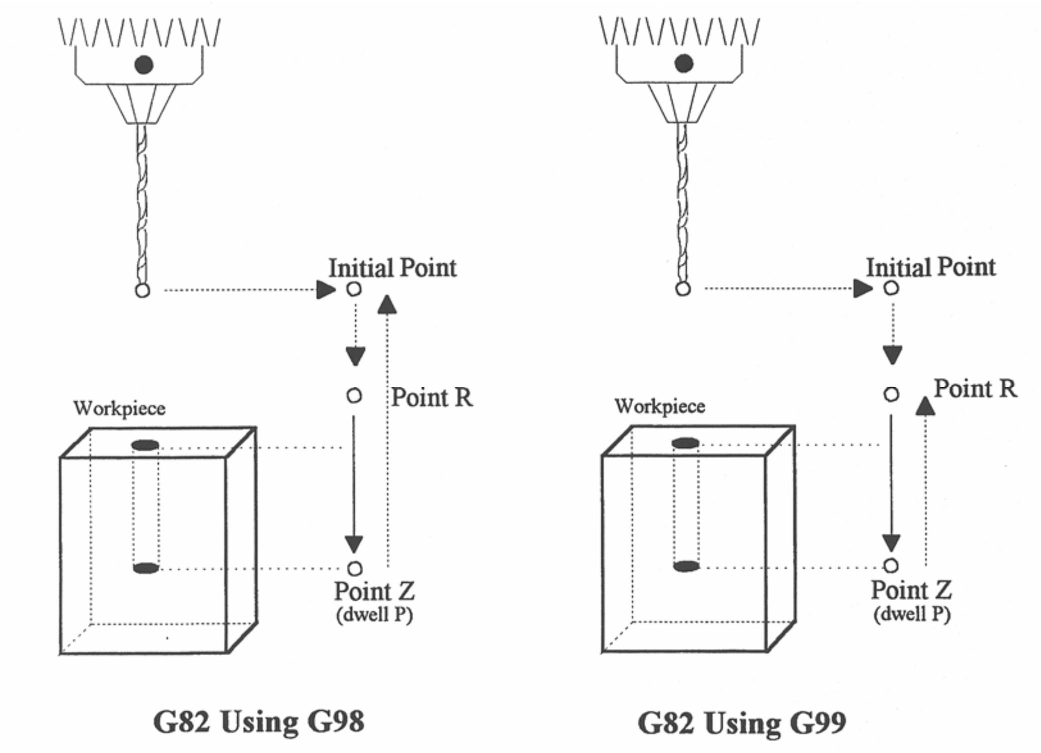
M function #39 is designed for general air drill use. See the description of M39 in the M functions section.

A different M function may be used instead, but any M function used must be a macro file that uses the M103 and M104 commands to time the cycle (see the example in the M function section under M103). If the macro file does not use M103, the control will automatically cancel the job 1/2 second after starting G81. For information on creating customized M functions, review Macro M functions in Chapter 12.

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.

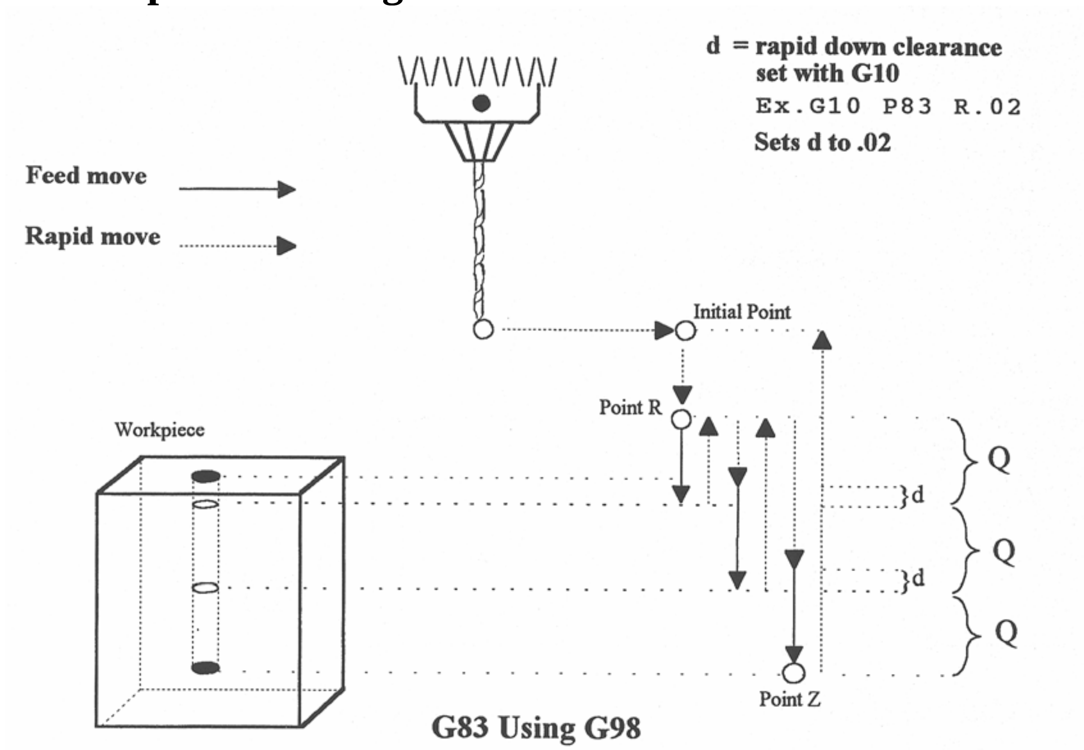
G82 - Drill with dwell

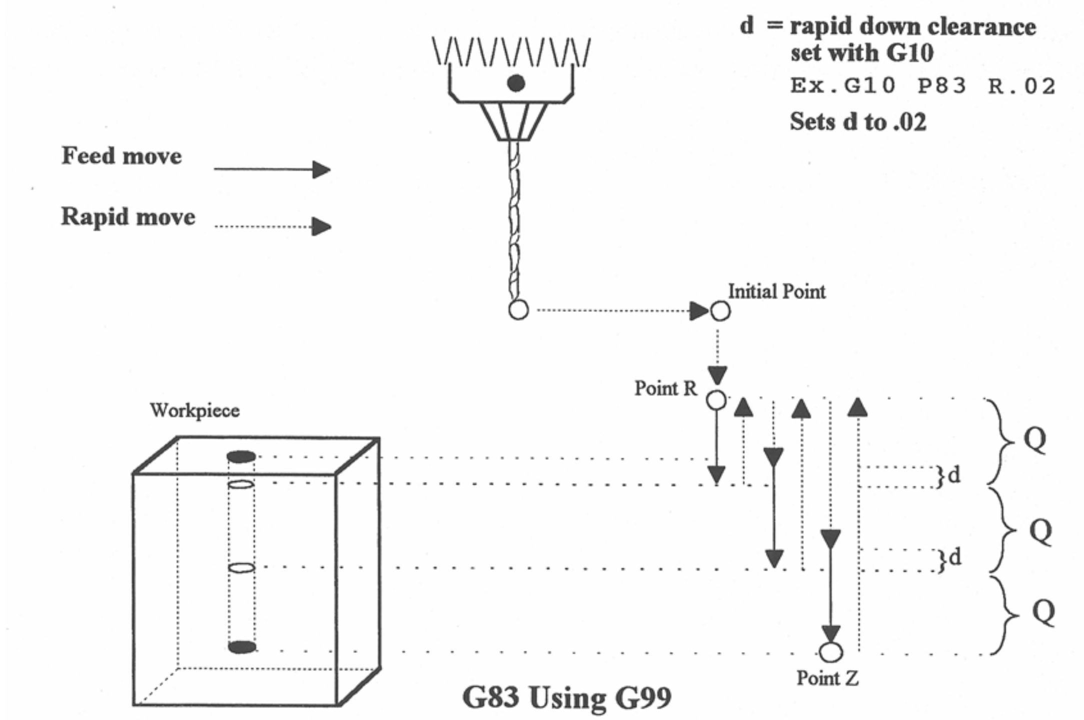


G82 is a general purpose drilling cycle similar to G81. However, G82 includes an optional dwell at the bottom of the hole before retracting the tool. This can make the depth of blind holes more accurate.

Example:
```
G82 X1 Y1 R.1 Z-.5 P.5           ;drill to Z-.5, dwell .5 seconds
```

G83 - Deep hole drilling



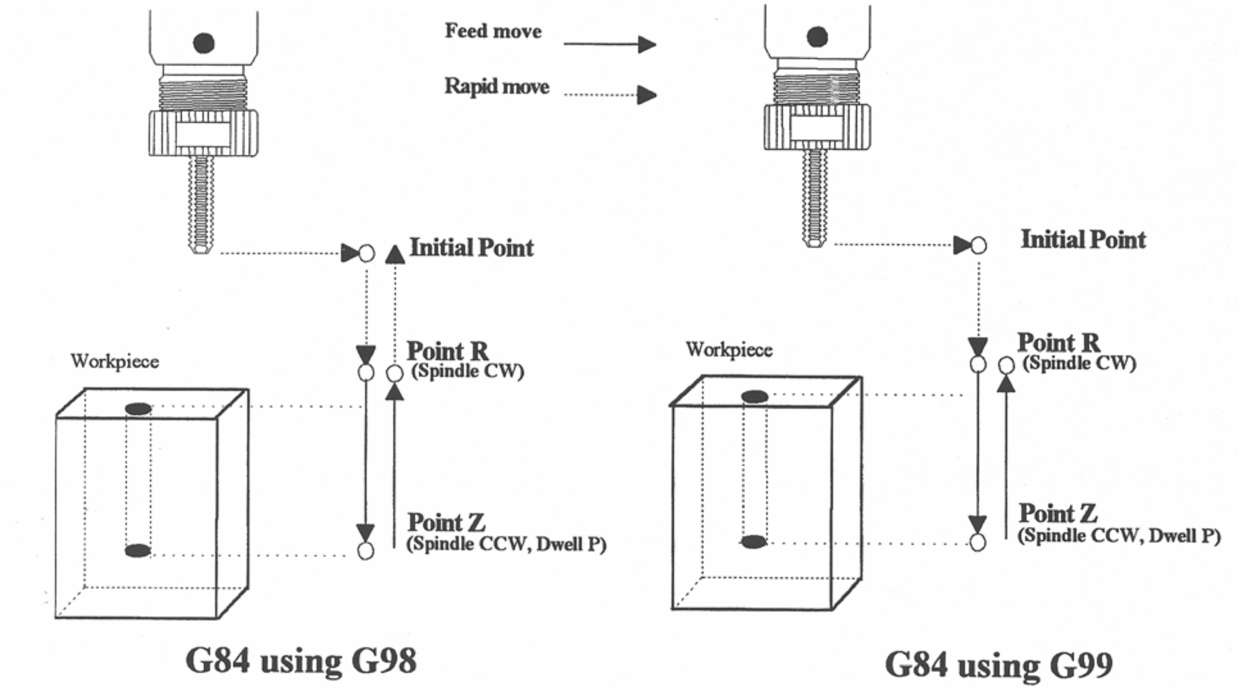


G83 is a deep hole drilling cycle. It periodically retracts the tool to the surface to clear accumulated chips, then returns to resume drilling where it left off. The retract and return are performed at the rapid rate. Because there may be chips in the bottom of the hole, the tool does not return all 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 <u>not</u> 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:

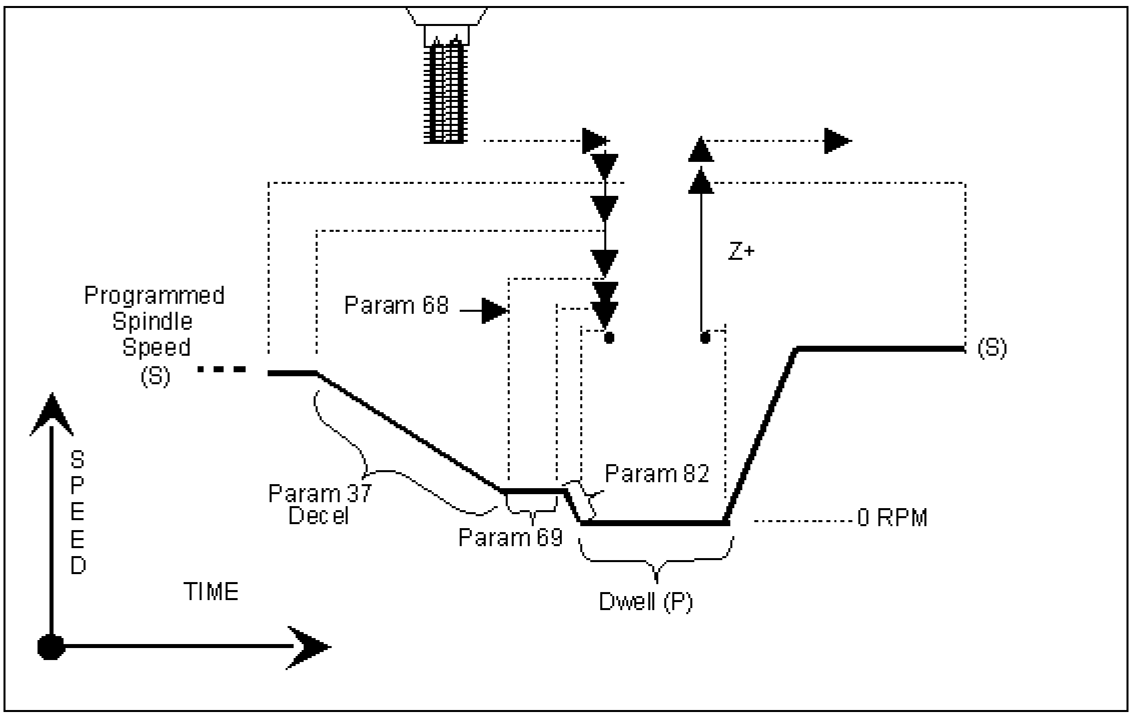
```
M3 S500 F27.78              ;start spindle CW, set up for 18 pitch tap
G84 X1 Y1 R.1 Z-.5          ;tap a 0.5 deep hole at X1 Y1
Y1.5                        ;... and another one at X1 Y1.5
G80                         ;cancel canned cycle
…
; Using Tool 15 Rigid Tap 6-32
G84 X0.337 Y0.925 Q0.03125 Z-0.35 R0.1      ;tap first hole
G84 X3.312 Y0.925 Q0.03125 Z-0.35 R0.1      ;tap second hole, must use Q
G80                                         ;cancel canned cycle
…
; Using Tool 22 Rigid Tap 4-40
G84 X1.862 Y1.627 Q0.025 Z-0.19 R0.1        ;tap first hole
G84 X2.862 Y1.627 Q0.025 Z-0.19 R0.1        ;tap second hole, must use Q
G84 X4.262 Y1.627 Q0.025 Z-0.19 R0.1        ;tap third hole, must use Q
G80                                         ;cancel canned cycle
```

Tech Tip – How to Setup Rigid Tapping

Overview

This section describes the theory of rigid tapping parameters, to control accuracy of depth of cut and quality of threads, in various working materials.
There is no set specification on over-travel or under-travel associated with rigid tapping; therefore rigid tapping must be considered an "art" of machining. Centroid makes no warranty or guarantee regarding rigid tapping depth of over travel.
If rigid tapping results are not satisfactory, the user should consider purchasing an inexpensive and very forgiving floating tap attachment or using the very versatile thread milling technique, both are supported by Centroid.

Graphic representation of parameter controls



Rigid tapping parameters

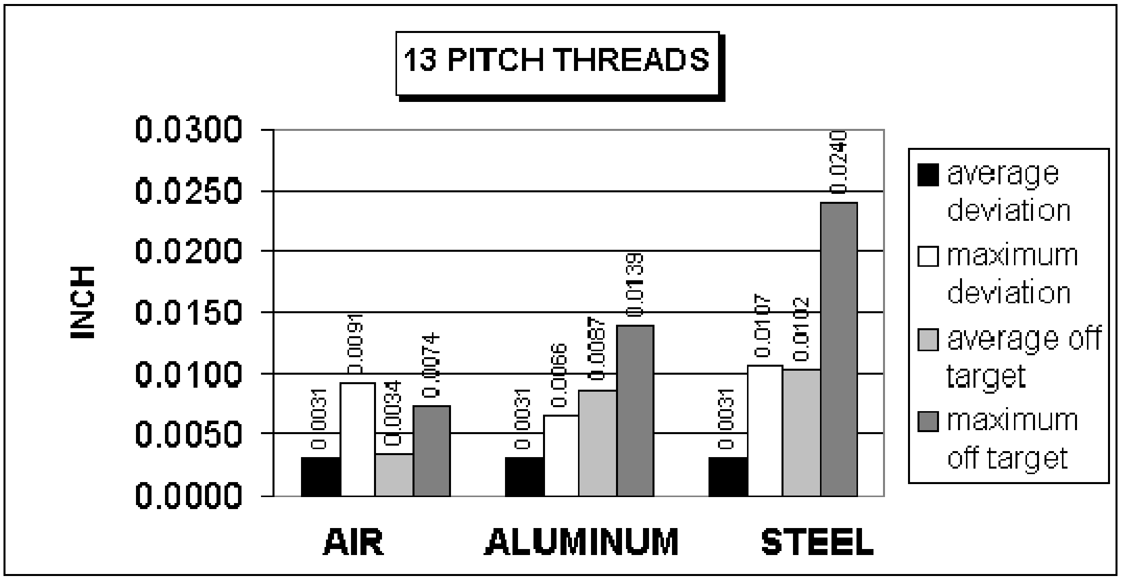
Parameter	Value	Function
34	4,096	Sets the counts per revolution (cpr) of the encoder. The encoders so far have been 1024 lines, providing 4096 cpr. If the encoder counts up when the spindle turns CW then the value should be positive. If the encouder counts up when the spindle turns CCW then the value should be negative.
35	5	Spindle encoder input (See chart below)
36	Bitwise value	Bit 0: 0-Disable rigid tapping, 1-Enable rigid tapping Bit 1: 0-Wait for index pulse during rigid tapping, 2-Do not wait for index pulse Bit 2: 0-Do not allow spindle override, 4-Allow spindle override. *Example:* A value of 3 will enable rigid tapping (bit 0 = 1) and during execution will not wait for the index pusle to start (bit 1 = 2) and the spindle override will not change the spindle speed (bit 2 = 0).

37	3.0	Spindle deceleration time. This value is used for setting the spindle deceleration rate from the programmed spindle speed (S) down to the spindle speed of Parameter 68. Check the value of your inverter setting and enter it. (i.e. if the inverter is set to 3 seconds deceleration, enter 3).
68	400 RPM	Minimum Rigid Tapping Spindle Speed (in RPM). This parameter hold the value that the spindle slows down to from the programmed spindle speed towards the end of the tapping cycle. The lower the value, the more accurately the Z axis will land on target, but at the expense of possibly stalling the motor which in turn will cause Z to stop short. If this value is too large, the off-target error increases.
69	1.25 sec	Duration for Minimum Spindle Speed mode (in seconds). This is a buffer time value to allow the spindle time to decelerate to S=Parameter 68. If the number is too small, overshoot may occur. If too large, the user waits longer for hole to be tapped at the slow speed specified by Parameter 68.
74	4	Spindle M function to be run at the ***bottom*** of the hole for **G84** tapping. Spindle M function to be run at the ***top*** of the hole to for **G74** counter tapping.
84	3	Spindle M function to be run at the ***bottom*** of the hole for **G74** tapping. Spindle M function to be run at the ***top*** of the hole to for **G84** counter tapping.
82	720 (2 revs)	Spindle Drift Adjustment (in degrees of rotation, i.e. 360 = 1 full rotation, 90 = ¼ rotation). This value is the number of turns that the spindle makes to coast to a stop when it is shut off at the rpm specified by parameter 68. This value is proportional to the distance above the Z target at which the spindle motor must be shut off in order for Z to land on target. (Remember that Z is slaved to the spindle speed during rigid tapping.)

Spindle Encoder input Chart

<table>
<tr><td colspan="2">CPU7/CPU9</td><td colspan="4">CPU10</td></tr>
<tr><td>Encoder Input</td><td>Parameter 35</td><td colspan="2">DC</td><td colspan="2">AC</td></tr>
<tr><td>3</td><td>2</td><td rowspan="2">Encoder Input</td><td rowspan="2">P35</td><td rowspan="2">Drive Type</td><td rowspan="2">P35</td></tr>
<tr><td>4</td><td>3</td></tr>
<tr><td>5</td><td>4</td><td rowspan="2">5</td><td rowspan="2">4</td><td>SD3</td><td>6</td></tr>
<tr><td>SD3 drive 6</td><td>5</td><td>SD1</td><td>21</td></tr>
</table>

Graphic representation of test results for precision



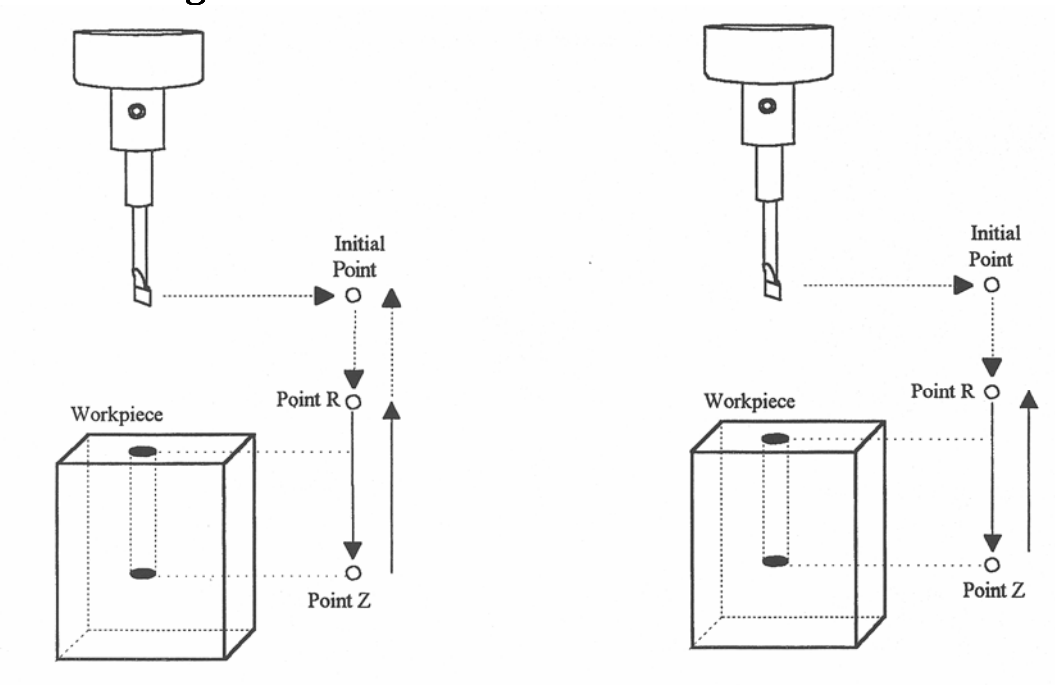
The above charts show test results of rigid tapping, utilizing version 7.14 software. The tool used in the testing was a ½-13 spiral fluted tap with TiN coating. Coolant used was water base soluble oil. Hole size was .4218. Tapping depth was .800. Also note that the parameters were adjusted to cut air, and not changed for aluminum or cold rolled steel for these tests. It can be seen, as the material changes, so does the off target values. This is due in part to the amount of torque required from the spindle to cut the various types of material. For testing purposes, the parameter settings for the above results were as follows.

Parameter 36 = 1, Parameter 37 = 3, Parameter 68 = 100, Parameter 69 = 1.25, Parameter 82 = 108

Summary

Rigid tapping parameters will vary from machine to machine. Not all machines are built the same (i.e. Spindle hp, inverter type, rigidity, etc.), and tooling will play a roll in performance also. It was found through our testing, if we changed one physical parameter, (i.e. using a tapping oil instead of water base coolant), it improved the off target values by 1.5%. This is due to the fact that less friction is present when using special cutting oil, therefore requiring less hp by the spindle to drive the tap. In most cases, rigid tapping depths should be able to be held within +/- .008 inch or less by adjusting parameter 82 for specific cases.

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.

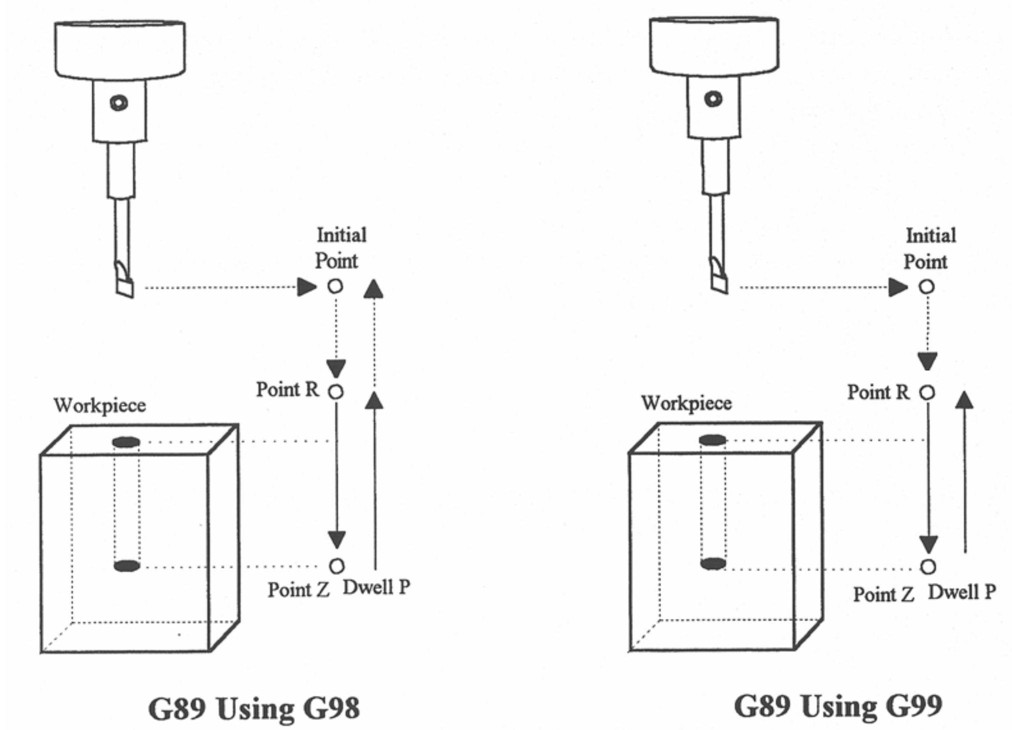
Example 1:

```
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                      ;enable feedrate and spindle overrides
G80                           ;cancel canned cycle
```

G89 - Boring cycle with dwell



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 (0,0,0,0). 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
                         ;X2, Y3.
G91 X1 Y0                ;moves the X axis 1 inch referenced from the last X
                         ;position, the Y axis does not move.
```

G92 - Set Absolute Position

G92 sets the current absolute position to the coordinates specified. If you are using multiple work coordinate systems, positioning in all coordinate systems will be changed by the same amount (-4 in X, -3 in Y, +2 in Z, and -4 in W in the example below).

Example:

```
G0 X5 Y3 Z-2 W5          ;Moves to the specified location
G92 X1 Y0 Z0 W1          ;Sets the current position to the absolute
                         ;position specified.
```

G98 - Initial Point Return

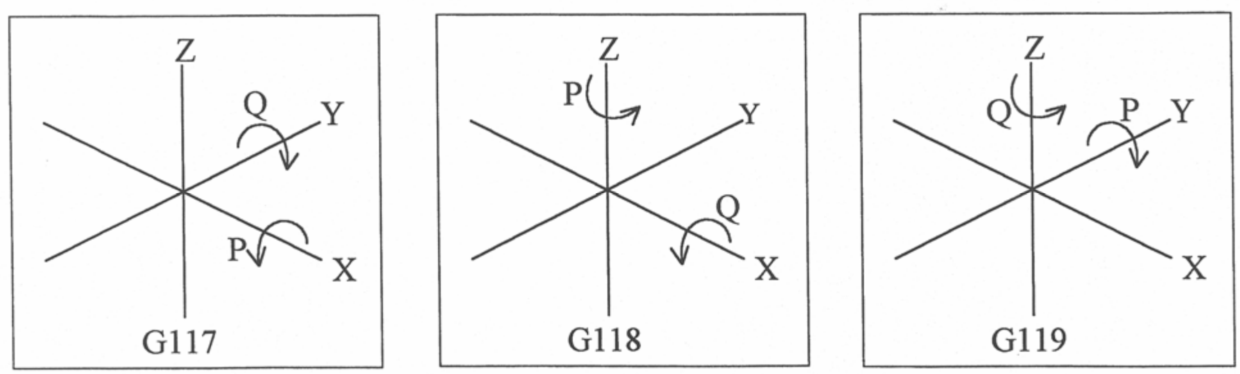
G98 sets the +Z return level to point I as pictured in Figure 1 in the Canned Cycle Section.
(G98 is the default setting)

G99 - R Point Return

G99 sets the +Z return level to point R as pictured in Figure 1 in the Canned Cycle Section.

G117, G118, G119 - Rotation of Pre-set Arc Planes

G117, G118 and G119 have the same functionality as G17, G18 and G19, respectively, except that they include 2 optional parameters P and Q to specify the arc plane rotation away from the pre-set arc plane: P specifies the arc plane angle of rotation (in degrees) around the first axis and Q specifies the arc plane angle of rotation around the second axis.



For the G117 plane, the "first axis" is X and the "second axis" is Y.
For the G118 plane, the "first axis" is Z and the "second axis" is X.
For the G119 plane, the "first axis" is Y and the "second axis" is Z. If P and/or Q is not specified, the angles are assumed to be 0 degrees. If both P and Q parameters are 0, then the plane is assumed to be an orthogonal (pre-set) arc plane. The center of the arc can be specified by the user in a 3D form both in G17-G19 and in G117-G119 (all I, J, K values are allowed at the same time with G2 and G3). Any arc center component outside the circular plane is ignored.

Example:

```
G00 X0 Y0 Z1                                ;rapid move
G03 G18 X1 Y0 Z0 K-1 F20                    ;arc mill
G00 X0 Y0 Z1.1                              ;retract move
G01 Z1                                      ;move to start of contour
G03 G118 P1.000000 X0.9998 Y0.0175 Z0 K-1   ;arc mill rotated about Z
```

NOTE: G117-G119 will not be permitted while cutter compensation is turned on. Also, scaling is not allowed while G117-G119 is specified and G117-G119 is not allowed while scaling is active.

Chapter 13

CNC Program Codes: M functions

M functions are used to perform specialized actions in CNC programs. Most of the M-series Control M functions have default actions, but can be customized with the use of macro files.

Macro M functions

Most M-Series CNC M functions from 0 through 90 can be fully customized. Exceptions are M2, M6, and M25 that can be customized, but will always move the 3rd (Z) axis to the home position before executing the macro M function commands. No M functions above 90 may be customized with macros. The default action listed will be performed unless that M function has been customized.

To create a macro for an M function, a file must be created in the C:\CNC10 directory. The file's name must be CNC10.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:\CNC10\CNC10.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 12); 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 14).

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

Note: adding 256 to parameter 178 can switch M10 and M11.

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
```

M30 - Custom M Code

Intercon posts an M30 at the end of every G code program. By default M30 performs no operation. If you wish to perform certain operations at the end of every program, this M code can be customized to meet you needs. For more information on customizing M codes see the beginning of this chapter.

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 tripped, the axis reverses until the home switch clears, and stops when it detects the index pulse.

Example:

```
M91/X        ; moves 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 tripped, the axis reverses until the home switch clears, and stops when it detects the index pulse.

Example:

M92/X	; moves the X-axis to the plus home switch.
G92 X+10	; Sets X plus home switch at +10

M93 – Release/Restore Motor Power

M93 releases or restores motor power for the axis specified. If no axis is specified, then all axes are released.

Example:
To release motor power:

M93/X	; releases the X axis.
M93	; releases the motors on all axes.

Example:
To restore power:

M93/X P1	; restore power to the X axis motor.
M93 P1	; restore power to the motors on all axes.

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 input 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 doesn't need to 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 a certain operation (e.g. a drilling pattern, contour, etc.) many times throughout a main program.

Calling methods:

M98 P*xxxx* L*rrrr*

Or

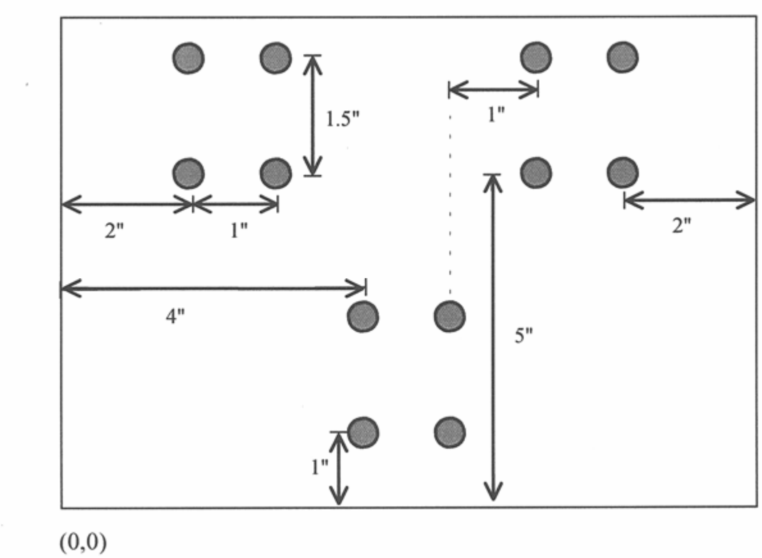
M98 "program.cnc" L*rrrr*

Where *xxxx* is the subprogram number (in file O*xxxx*.CNC, 9100-9999 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. CNC10 will read the subprogram and generate a file O9*xxx*.cnc. CNC10 will not execute the subprogram until it encounters M98 P9*xxx*.
NOTE: An embedded subprogram definition must be placed before any calls to the subprogram.



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:

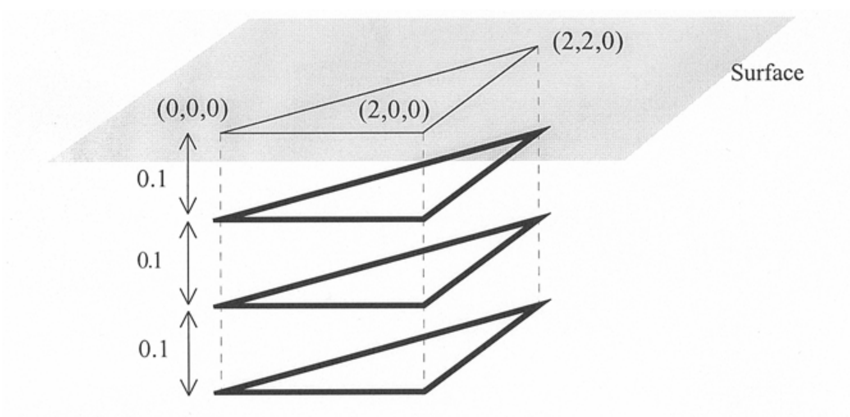
```
O9101                    ;Program O9101.cnc
G91 F10                  ;Incremental positioning
G81 X0 Y0 R -.4 Z-.6     ;Drill lower left hole
Y1.5 R -.4 Z-.6          ;Drill upper left hole
X1 R -.4 Z-.6            ;Drill upper right hole
Y-1.5 R-.4 Z-.6          ;Drill lower right hole
G80                      ;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 P9101 L1               ;Call subprogram O9101.cnc
G90 G0 X4Y1 Z0.5           ;Move to second hole pattern
M98 P9101 L1               ;Call subprogram
G90 G0 X6 Y5 Z0.5          ;Move to third hole pattern
M98 P9101 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 input request 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 input request 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 input request 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 in minus direction 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 in the plus direction at 30"/min, until switch #3 open
G92 X10                ; Sets X position to 10
```

M107 - Output BCD (Binary Coded Decimal) Tool Number

M107 sends the current tool number to the automatic tool changer, via the PLC. The number is sent as BCD. M107 does not set the tool changer strobe or look for an acknowledgement from the changer (see M6).

Example:

```
M107                   ; send request for tool to change
M94/16                 ; 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 were disabled with M109. A parameter of “1” indicates the feedrate override; “2” indicates the spindle speed override.

Example:

```
M109/1/2               ; disables feedrate and spindle speed overrides
M108/1                 ; re-enables feedrate override
M108/2                 ; re-enables 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 in clockwise direction, at 500 rpm
F27.78                 ; set feedrate for 18 pitch tap
M109/1/2               ; disable feedrate and spindle speed overrides
G85 X0 Y0 R.1 Z-.5     ; tap a hole
M108/1/2               ; re-enable overrides
```

M115/M116/M125/M126 – Protected Move Probing Functions (Optional)

The protected move probing functions provide the capability to program customized probing routines.

The structure for these commands is:

Mnnn /*Axis pos* P*p* F*f* *L1*

Where:

nnn	is either 115, 116, 125, or 126.
Axis	is a valid axis label, i.e., X, Y, Z, etc.
pos	is an optional position
p	is a PLC bit number, which can be negative.
f	is a feedrate (in units per minute.)
L1	is an option for the M115/M116 commands that prevents an error if the probe does not detect a surface

For M115 and M116 functions, the indicated *axis* will move to *pos* (if specified) until the corresponding PLC bit *p* state is 1, unless *p* is negative, in which case movement is until the PLC bit state is 0. A "*p* value" of 1 to 80 (or -1 to -80) specifies PLC bits INP1-INP80, 81 to 160 (or -80 to -160) specifies PLC bits OUT1-OUT80, and 161 to 240 (or -161 to -240) specifies PLC bits MEM1-MEM80. Warnings are generated in the CNC10 message window for "*Missing P value*" and "*Invalid P value*."

If "*pos*" is not specified, M115 will move the *axis* in the negative direction, and M116 will move the *axis* in the positive direction. Note if "*pos*" is specified, then if does not matter whether M115 or M116 is used.

If "*pos*" is not specified, the movement is bound by the settings in the software travel limits. In the absence of software travel limits, movement is bound by the maximum probing distance (Machine Parameter 16). In cases where "*pos*" is specified, it is still bound by the software travel limits.

If the bounded position is reached before the awaited PLC bit state is found, a "*Probe unable to detect surface*" error will be generated unless the *L1* option is specified.

For M125 and M126 protected move functions, the behavior is identical to that of the M115 and M116 commands, except in regards to the PLC bit state. M125 and M126 will generate an "*Unexpected probe contact*" error message if the specified PLC bit state is triggered, again stopping any running job.

In summary, the M115 and M116 commands are to be used when one expects contact to be made, and M125 and M126 commands are to be used when one does not expect any contact to be made.

Example:

Finding the center of a vertical slot. In this example, it is assumed that there is a probe connected to INP15 and that the probe tip is positioned somewhere in the slot, such that movement along the X-axis will cause a probe trigger.

```
G20                 ; Set mode to English
M115/X P-15 F20     ; Move X minus at 20 ipm until probe trip
M116/X P15 F5       ; Move X plus at 5 ipm until probe clears
#100 = #5041        ; Record the point in user variable #100
M116/X P-15 F20     ; Move X plus at 20 ipm until probe trip
M115/X P15 F5       ; Move X minus at 5 ipm until probe clears
X[[#100+#5041]/2]   ; Move X to center of slot
```

M120 - Open data file (overwrite existing file)

This M function will open the requested data file for writing. If no drive or directory is specified with the file name, then the file will be opened in the same directory as the CNC program. If the file cannot be successfully opened, then an error will be returned, ultimately terminating the job. If a data file is already open when M120 is called, that file will first be closed, then the new file opened.

Example:

M120 "probetst.dat"	; Opens probetst.dat file to write data too

M121 - Open data file (append to existing file)

This M function will open the requested file for writing at the end of the file. If no drive or directory is specified with the file name, then the file will be opened in the same directory as the CNC program. If the file does not already exist, it will be created. This is not an error. If the file cannot be successfully opened, then an error will be returned, ultimately terminating the job. If a data file is already open when M121 is called, that file will first be closed, then the new file opened.

Example:

M121 "c:\probetst.dat"	; Opens probetst.dat file to add data to it

M122 - Record position(s) and optional comment in data file

This M function will write the current expected position value to the data file, in the usual format (i.e. axis label before number, 4 decimal places in inch mode, 3 decimal places in millimeter mode. Any comment that appeared on the line with M122 will be outputted after the position(s). With no axis arguments, M122 will write the positions of all installed axes. With axis arguments, it will write the positions only of the requested axes. Positions will be written in local (not machine) coordinates, in native machine units. If no data file has been opened with M120 or M121 before M122 is called, then M122 will return an error and terminate the job. The parameter L1 may be used to suppress the new line character normally outputted after the last position.

Examples (M function and sample output):

M122	->	X1.2345 Y-3.2109 Z-0.5678
M122 /Z ; at 10 ipm	->	Z-.4321 ; at 10 ipm
M122 /X/Y	->	X-1.0000 Y0.8732
M122 /X L1	->	X-1.5000
M122 /X	->	X-1.5000 X-2.0000

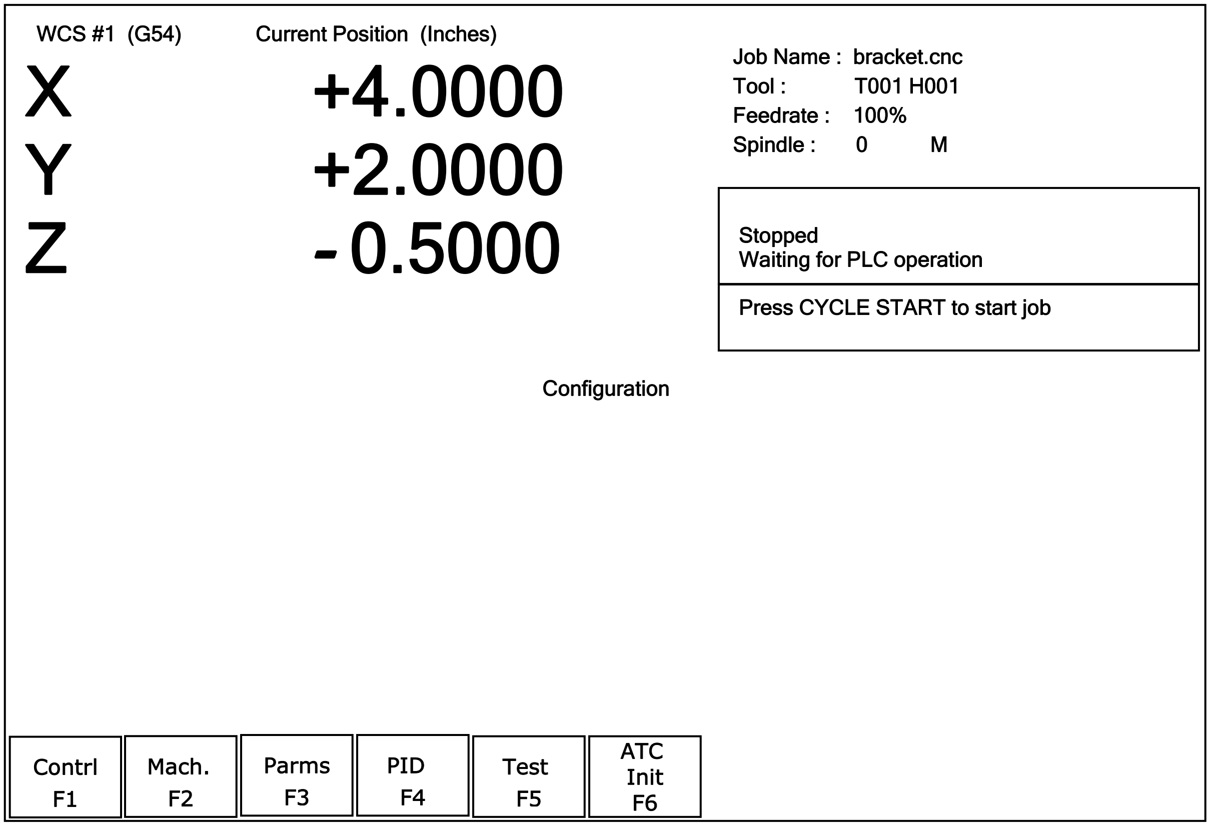
M123 - Record value and/or comment in data file

This M function will write the specified parameter value (if any) to the data file, followed by any comment that appeared on the line with M123. If a P value is specified, M123 will record the numeric value (4 decimal places in inches, 3 in millimeters). If neither a P value nor a comment was specified, M123 does nothing. This is not an error. If no data file has been opened with M120 or M121 before M123 is called, then M123 will return an error and terminate the job. The parameter L1 may be used to suppress the new line character normally outputted after the last value. The R and Q parameters can be used to specify the field width and precision, respectively.

Examples (M function and sample output):

M123 P1.2345	->1.2345
M123 P#A ; first macro argument	->1.2345 first macro argument
M123 ; Probing X+ to surface	->Probing X+ to surface
M123	->*<nothing>*
M123 ;	->*<nothing>*
M123 ;; my comment	->; my comment
M123 Q0 P1.23	->1
M123 Q1 P1.23	->1.2
M123 R7 Q2 L1 P1.234	
M123 R7 Q2 P98.765	->1.23 98.77

Chapter 14
Configuration



General

The first four options, **F1** through **F4**, will display a set of parameters. Each option is explained in detail below. The **ESC** key will return you to the previous screen (Setup).

The **F5 - Test** key should only be used by qualified Centroid technicians to perform automated system tests.

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 - Config** 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.

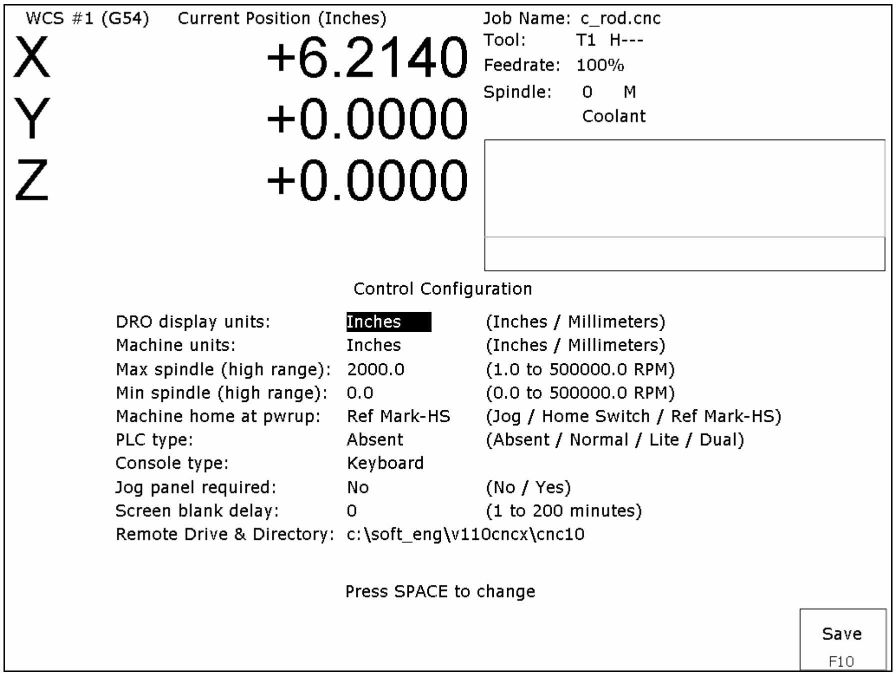
If you know the password, type it and press **ENTER**. If the password you enter is incorrect, a message will appear telling you the password was incorrect and the password prompt will reappear. Pressing **ESC** will remove the prompt.

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 - Contrl** from the configuration screen will display the Control Configuration screen. 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 the **SPACE** bar to toggle. When you are done editing, press **F10 – Save** to save any changes you have made. If you wish to discard your changes and restore the previous values, press **ESC**.



DRO Display Units
This field controls the units of measure that 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** bar 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 the **SPACE** bar 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 sent to the PLC as percentages of this maximum value.

If your machine is equipped with a dual range drive and VFD, the controller will not exceed the spindle speed set by this field while in high gear. See the Machine Parameters section below for information on setting the gear ratios for medium and low gear ranges. If your machine has a VFD but is not equipped with a dual range drive, this field determines the maximum spindle speed.

Minimum Spindle Speed (High Range)
This parameter is used to adjust the minimum spindle speed for the high range. This parameter allows the operator to set the minimum value for spindle speed to a value other than 0. All changes in spindle speed are made in relationship to this value, with this parameter as the minimum value. The values stored can range from 0 to 500000.0 RPM.

Machine Home At Powerup
This field controls how the machine will home at powerup. Set Machine Home at Powerup to Home Switch if you have limit/home switches or safe hard stops for all axes, and wish to use the switches or stops for homing. Set Machine Home at Powerup to Ref Mark – HS if you have fixed reference marks for any axis. In Ref Mark homing, axes that contain a zero (0) for the plus or minus home switch in the Machine Configuration designate that axis to have a Ref Mark home, while non-zero values specify Limit Switch homing. Set Machine Home at Powerup to Jog if you need to manually move or jog the machine to its home position. See Chapter 1 for more information about machine home.

PLC Type
This field tells the controller which PLC type is installed. The possible values are 'Absent,' 'Lite,' 'Normal,' and 'Dual.' The value should not be changed unless a different PLC type is installed. Use the **SPACE** key to select among the four options.

The standard PLC types installed are dependent on your M-series number and the options that may have been purchased. Check the information sheet on page *ix* for which type of PLC is installed on your machine, or check with your dealer for more information.

Console Type
Set for type of console installed. "T-" for lathe, "M-" for mill. Current console type is “Uniconsole-2”.

Jog Panel Required
This field tells the controller whether a Jog Panel must be installed in order to run jobs. Also if set to “Yes” the control requires you to press cycle start twice to start a program but if it’s set to “No” programs will start with one press of cycle start.

Screen Blank Delay
This field determines the delay used for the screen blank function. Entering a value will specify the number of minutes of idle time until the screen will go blank. The blanking function only works if no jobs are running. For example, 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.
A value of zero will disable the blank function. However, if the display is kept on for long periods of time without the blank function 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 - Remote** key in the Load Job screen. This allows you to conveniently load files from an attached computer via LAN network (via RJ-45 Ethernet connection). The Control will usually remap the attached computer's C: hard drive as drive E:, depending on the way it was set up.

User Specified Paths

Operators can now specify paths for INTERCON files, posted INTERCON files, Digitize files and CAD files. These paths are specified in *pathm.ini*. This file is automatically generated by CNC10 if it does not exist. The default *pathm.ini* file is:

```
INTERCON_PATH=c:\intercon\
ICN_POST_PATH=c:\cnc10\ncfiles\
DIGITIZE_PATH=c:\cnc10\ncfiles\
CAD_PATH=c:\cnc10\ncfiles\
```

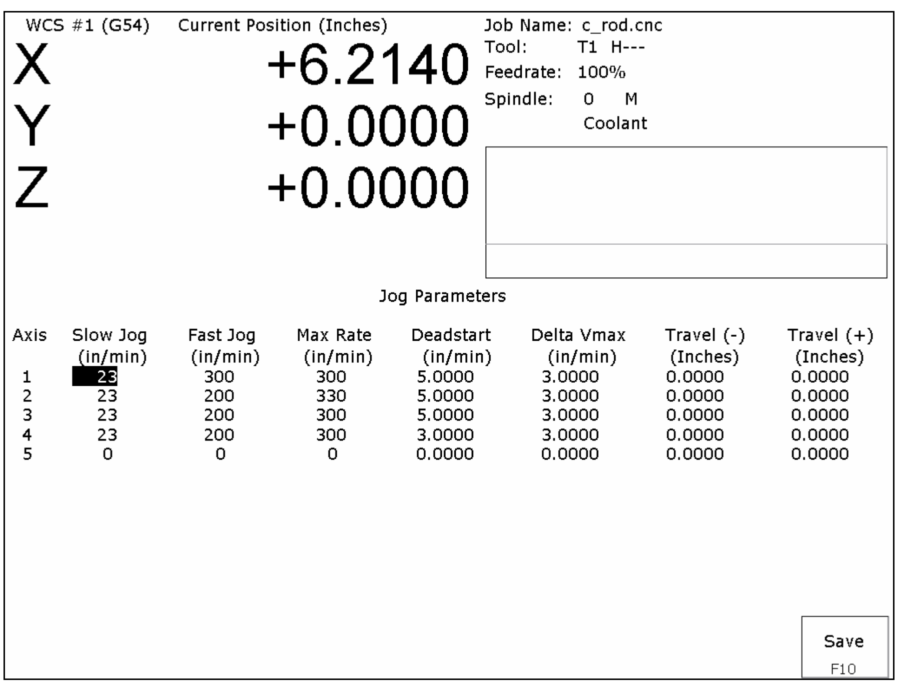
Path tag	Purpose of path
INTERCON_PATH	Main directory containing *.icn files
ICN_POST_PATH	Directory INTERCON places *.cnc files created when posting *.icn files.
DIGITIZE_PATH	Directory digitize files are saved to. Directory used by F4 key in Load Job menu when parameter 4 is set to 2.
CAD_PATH	Default directory used by the Import DXF file menu in Intercon. This directory is also used to store CAD files generated with the DIG->CAD option in the Utility menu.

Machine Configuration

Pressing **F2 - Machine** from the configuration screen will display the machine configuration screen. The machine configuration screen provides you with a method of changing machine dependent data.

If you wish to change the Jog or Motor parameters, press **F1 - Jog** or **F2 - Motor** to select the Jog or Motor screens. Use the arrow keys to move the cursor and select the desired field. Type the new value and press **ENTER** or press the **SPACE** bar to toggle. When you are done editing, press **F10 - Save** 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 (Configuration).

F1 - Jog Parameters (Values should be recorded on the Control Parameters page at beginning of manual.)
This screen contains jog and feedrate information. See the figure below.



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).

<u>Slow Jog</u>*:* 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.

<u>Fast Jog</u>*:* 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.

<u>Max Rate</u>: 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.

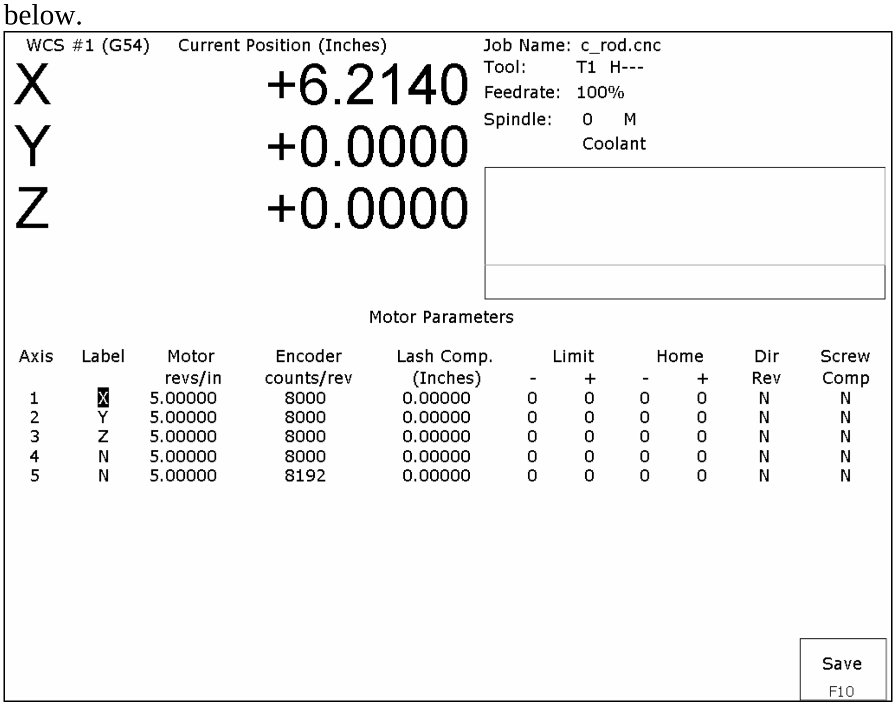
<u>Deadstart:</u> Determines the speed an axis will decelerate to before stopping or reversing direction. A low setting will cause a large slowdown before a reverse in direction, causing your machine to be more accurate. A high setting will cause less slowdown before reversals, but this may cause your machine to "bang" which may cause a decrease in accuracy. This parameter should not be changed.

<u>Delta Vmax:</u> The maximum instantaneous velocity change that will be commanded on a vector transition. This parameter should not be changed.

<u>Travel (-):</u> The maximum distance the axis can travel in the minus direction from the home position. Set this parameter to create a software limit that stops the axis before the fixture or tool collides with the limit switches or hard stops.

<u>Travel (+):</u> The maximum distance the axis can travel in the plus direction from the home position. This parameter is especially useful when using a part or fixture larger than the table. Set this parameter to create a software limit that stops the axis before the fixture or part collides with the machine or the limit switch/hard stop.

F2 - Motor Parameters (Values should be recorded on the Control Parameters page at beginning of manual.)
This screen contains information about the motors, ballscrews, and switches installed on your machine. See the figure below.



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.

<u>Special function indicators</u>: These appear, if present, between the axis number and the label. 's' – indicates the axis is the spindle, 'p$' – axis is paired with axis '$', 'h$' – axis is a handwheel paired with axis '$', '*' – pairing conflict. See Machine Parameters for more information on setting up special functions.

<u>Label:</u> The letter you want to use to identify the axis. The first three axes should normally be X, Y, and Z. If a fourth axis is installed, it is usually named W or B. If you change a label, for example from X to A, the controller will then accept G-codes for axis A instead of X.

If fewer than four axes are present, the unused entries should be labeled N. If an axis is manually operated (it has an encoder but no motor), it should be labeled M.
For a manual Z-axis, the 3^{rd} axis label should be set to @ symbol. This setting allows for two axes posting in Intercon.
* WARNING: Intercon does NOT post two axis programs if the 3^{rd} axis is labeled M.

* NOTE: Tool length compensation (G43-G44) and canned drilling cycles (G73-G89) always affect the third axis, regardless of its axis label. Tool diameter compensation (G41-G42) always affects the first and second axes, regardless of their axis labels.

<u>Motor revs/unit:</u> 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. <u>Handwheel note</u>: For handwheels, this number is the number of clicks per revolution of the handwheel. If your handwheel has no detents (click positions) use "100".

<u>Encoder counts/rev:</u> The counts per revolution of the encoders on your servomotors.

<u>Lash compensation:</u> 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.

<u>Limits:</u> 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.

<u>Home:</u> 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.

<u>Direction reversed:</u> Used to match the +/- reference of your machine to the control electronics. Toggle this value if you actually move in the X minus direction (reverse) when you jog X+.

<u>Screw Compensation*:</u> 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-Find Home** to move an axis to its plus or minus home switch.

F4 - Set Home Press **F4-Set Home** 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.

* WARNING: The ballscrew compensation tables **should not** be changed without contacting your dealer. Corrupt or incorrect values could adversely affect the accuracy of your machine.

Machine Parameters

(**F3 - Parms** from Configuration)

Machine Parameters

0	0.0000	20	72.0000	40	0.0001	60	0.0000	80	0.0000
1	0.0000	21	0.0400	41	0.2500	61	0.5000	81	-1.0000
2	0.0000	22	0.0400	42	54.0000	62	115.0000	82	1000.0000
3	0.0000	23	0.0400	43	0.0010	63	1.5000	83	0.0500
4	1.0000	24	0.0400	44	5.0000	64	0.0000	84	3.0000
5	0.0000	25	0.6800	45	0.0000	65	1.0000	85	1.0000
6	0.0000	26	0.6800	46	0.0000	66	1.0000	86	0.0000
7	1.0000	27	0.6800	47	1.0000	67	1.0000	87	36.0000
8	3.0000	28	0.6800	48	0.1000	68	640.0000	88	36.0000
9	0.0000	29	150.0000	49	0.0000	69	1.7500	89	36.0000
10	0.0000	30	180.0000	50	1.0000	70	0.0010	90	36.0000
11	15.0000	31	0.0000	51	0.0000	71	0.0000	91	0.0000
12	10.0000	32	19200.0000	52	0.0010	72	0.0000	92	0.0000
13	0.0500	33	1.0000	53	0.0100	73	0.1000	93	0.0000
14	30.0000	34	-8000.0000	54	0.0000	74	4.0000	94	0.0000
15	6.0000	35	5.0000	55	0.0000	75	0.0000	95	4.0000
16	10.0000	36	1.0000	56	32000.0000	76	0.0000	96	4.0000
17	3.0000	37	3.0000	57	0.0000	77	0.0000	97	4.0000
18	10.0000	38	0.0000	58	179.9500	78	0.0000	98	2.0000
19	0.0000	39	200.0000	59	0.0100	79	49.0000	99	2.0000

E-Stop PLC Bit Number

Next Table
F3

Save
F10

Pressing **F3 - Parms** 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 – Save** to save any changes you have made. Press **ESC** to return to the previous screen (Setup). A short description of the parameter will appear below the table. In the screen example above, parameter 6 determines whether an Auto Tool Changer is installed.

F3 - Next Table will toggle the display parameters between parameters 0-99 and parameters 100-199.

Bit-mapped parameters

Certain control parameters are defined by bit-mapped values. In order to change these parameters you must understand how bit mapping works. A bit-mapped parameter is stored as a number, representing a 16-bit value in the control. If a certain bit needs to be turned on, that bit's binary value must be added to the parameter value, if the bit needs turned off, its binary value must be subtracted from the parameter value. The values for each of the 16 bits' can be seen in the table below.

Bit-Mapped Parameter Bit's																
Bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
Value	32768	16384	8192	4096	2048	1024	512	256	128	64	32	16	8	4	2	1

To set bit-mapped parameters simply add together the bit values that you need to have enabled.

Examples:

Parameter value	**Bit number and settings**															
	15	**14**	**13**	**12**	**11**	**10**	**9**	**8**	**7**	**6**	**5**	**4**	**3**	**2**	**1**	**0**
0	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X
1	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X	ON
11 = 8+2+1	X	X	X	X	X	X	X	X	X	X	X	X	ON	X	ON	ON
24 = 16+8	X	X	X	X	X	X	X	X	X	X	X	ON	ON	X	X	X

The following parameters are currently defined:

Parameter	Definition	Default setting
0	E-Stop PLC Bit	0
1	Y jog key orientation	0
2	G-Code Interpretation Control and Slaving Rotary axis feedrate	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	10
15	Slow Probing Rate	1
16	Probing Search Distance	10
17	Tool Detector Reference Number	0
18	PLC Input Spindle Inhibitor	0
19	MPG modes	0

20	Ambient Temperature	72
21-24	Motor Heating Coefficients	Refer to text
25-28	Motor Cooling Coefficients	Refer to text
29	Warning Temperature	150
30	Limit Temperature	180
31	Spindle Speed Output Port	0
32	Spindle Serial Port Baud Rate	19,200
33	Spindle Motor Gear Ratio	1
34	Spindle Encoder Counts/Rev	8,000
35	Spindle Encoder Input	4
36	Rigid Tapping Enable/Disable	0
37	Spindle Deceleration Time	10
38	Multi-Axis Max Feedrate	0
39	Feedrate Override Knob Limit	120
40	Basic Jog Increment	0.0001”
41	Handwheel x100 Speed, User Jog Increment	0.25”
42	Password for Configuration Menus	0
43	Automatic tool measurement options	0
44	TT1 PLC input #	0
45	Z axis MPG enable PLC input	0
46	Z axis MPG handwheel Turns Ratio	0
47	Z axis MPG handwheel encoder counts	0
56	Feedrate Override Display Properties	0
60	Digital Filter Size	1
61	High Power Stall Timeout	0.5
62	High Power Stall PID Limit	115
63	High Power Idle PID Multiplier	1.5
64	Fourth Axis Pairing	0
65-67	Spindle Gear Ratios	1
68	Minimum Rigid Tapping Spindle Speed	100
69	Duration For Minimum Spindle Speed	1.25
70	Offset Library Inc/Decrement Amount	.001"/.02mm
71	Part Setup Detector Height	0
72	Data M-Function Options	0
73	Peck Drill Retract Amount	0.05
74	M-Function executed at bottom of tapping cycle	4
75	Axis Summing Display Control	0
76	Manual Input Unrestricted Distance	0
77	Manual Input Movement Tolerance	0
78	Spindle Speed Display and Operations	0
79	Auto Brake Mode PLC Bit for Uniconsole-2	70
80	Voltage Brake Applied Message Frequency	1
81	Air Drill M-Function (executed instead of Z movement in drilling)	-1
82	Spindle Drift Adjustment	108
83	Deep Hole Clearance Amount	0.05
84	M-Function executed at return to initial point of tapping cycle	3
87-90	Autotune Accel Time and ka.	48
91-94	Axis Properties	0

95-98	Autotune Move Distance	2
99	Cutter Compensation Look-ahead	2
100	Intercon comment generation	0
101	Intercon clearance amount	0.1
102	Intercon spindle coolant delay	3.0
103	Intercon corner federate override	50.0
104	Intercon modal line parameters	0
105	Intercon modal arc parameters	0
106	Intercon modal drilling cycle parameters	0
110	Intercon Ramp Angle Parameter	0
115	Intercon Help	0
116	A Axis Y Coordinate	0
117	A Axis Z Coordinate	0
118	B Axis X Coordinate	0
119	B Axis Z Coordinate	0
120	Probe stuck clearance amount	0.10”
121	Grid digitize prediction minimum Z pullback	0.002”
122	Grid digitizing deadband move distance	0.0002”
123	Radial digitizing clearance move	0
124-127	PLC inputs for jogging	0
128	Handwheel MPG mapping	0
129	Handwheel MPG display control	0
130	Z axis on/off selection	0
131	4th axis on/off selection	0
132	5th axis heating coefficient	Refer to text
136	5th axis cooling coefficient	Refer to text
140	Message log priority level	1
141	Maximum message log lines	1000
142	Message log trim amount	1000
143	DRO properties (load meters, 4/5 digits, DTG)	0
144	Comparison rounding	0
145	Advanced macro properties (fast branching)	0
146	Feed hold threshold for feed rate override	0
148	Fast Jog Before Home is Set	0
149	Spindle Speed/Surface Footage Threshold	0
150	Run-Time Graphics	0
151	Repeatability tolerance for probing	0
152	5th axis Autotune accel time and Ka	48
156	5th axis Autotune move distance	2
160	Enhanced ATC	0
161	ATC Maximum Tool Bins	0
162	Intercon M6 Initial M-Code	0
164	ATC Feature Bit	0
166	5th axis properties	0
170-177	XPLC parameters	0
178	PLC I/O configuration (PLC program specific)	0
179	Lube Pump Operation	0
180	File Transfer COM Port	0

181	File Transfer Baud Rate	19.2
182	File Transfer Data, Parity and Stop bit settings	801
183	File Transfer Flow Control Setting	0
184	File Transfer COM timeout	10
185	File Transfer Serial Port Option	0
188-199	Aux key functions	0

Parameter 0 – E-Stop PLC Bit
This parameter specifies the PLC bit to which the physical Emergency Stop switch is connected. It is mainly used for ATC applications that use custom PLC messages. See table below for examples.

PLC Type	ESTOP Input on PLC	Parameter Value
RTK3	Input 11	11
PLCIO2	Input 11	11
DC3IO	Input 11	11
Servo3IO	Input 1	1

Parameter 1 – Y jog key orientation
This parameter is a 3-bit field where bit 0 is not used in the mill software. Bit 1 sets the direction of movement for the Y+ and Y- jog keys and bit 2 will swap the X and Y jog keys. This should always be set to 0 except for very special applications.

Bit	Function Description	Parameter Value
0	Not Used	
1	Flip direction of Y jog keys?	Yes=2, No=0
2	Exchange X axis and Y-axis jog keys?	Yes=4, No=0

Parameter 2 - G-code Interpretation Control and Slaving Rotary axis feedrate
This parameter is a 3-bit field that controls optional interpretation of several G-codes. The following table shows the functions performed by the value entered in this parameter:

Bit	Function Description	Parameter Value
0	Arc centers I, J, K are absolute in G90 mode?	Yes = 1 No = 0
1	Allow Z being specified alone to be sufficient to trigger execution of a canned tapping or drilling cycle to be executed?	Yes = 2 No = 0
2	Interpret dwell times associated with G4, G74, G82, G84, and G89 as milliseconds rather than seconds?	Yes = 4 No = 0
3	Slaving rotary axis feedrate to non-rotary axis feedrate	Yes = 8 No = 0
4	Selects the center for scale, mirror and rotate. By default the center will be 0,0,0. Add 16 to this parameter to make the center of scale, mirror and rotate the current position.	Yes = 16 No = 0

Parameter 3 - Modal Tool and Height Offset Control
This parameter is a 2-bit field. Bit 1 controls whether or not the last tool and height offset activated during a job run will remain active after the job is complete as well as controlling the Tool status display in the Status Window.

Bit	Meaning	Parameter Value
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.	Yes = 0 No = 1
1	Reference tool position set to Z home	Yes = 1 No =0

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 or the **F3 – Remote** option. These drives (i.e. drives D, E, F, etc.) are presumed to be network drives, Interlink drives or extra hard drives.

Value	Meaning
0	Job files are not copied or cached. They are run from whichever drives they reside on.
1	Job files are copied to the C drive (c:\cnc10\ncfiles) when they are loaded. The local copy is used when the job runs.
2	Turn on file caching. Job files are temporarily cached on the C drive. The cached copy is used while the job is running. The cached copy is deleted when the next job is loaded or when Parameter 4 changes to a 0 or 1. Digitize files are cached as the machine is digitizing. When digitizing is complete, the resulting file is copied to the digitize directory specified in *pathm.ini*.

File caching is useful for machines with both a flash card and a hard drive. By caching job files from the hard drive on the flash card, the hard drive is not used while the job is running. As a result, the life of the hard drive is extended and the flash card does not fill up with job files.

Parameter 5 - Suppress Machine Home Setup
This parameter controls machine homing upon startup of the control. The following table details the functions controlled by this parameter:

Bit	Function Description	Parameter Value
0	Suppress the requirement to set machine home before running jobs?	Yes = 1 No = 0
1	Display router bit map at homing screen	Yes = 2, No = 0
2	Disable stall detection when CNC10 first starts.	Yes = 4, No = 0

Bit 0 suppresses the requirement to set machine home before running. If bit 0 of Parameter 5 is 0, machine home must be set before jobs may be run. If bit 0 of Parameter 5 is 1, machine home is not requested or required, but Graphing and running of jobs will not work until the Z-axis is homed.

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 an automatic tool changer installed on your machine. This field affects the action of M6 in your CNC programs. See M6 under M functions in Chapter 13. It also affects whether the ATC key is present in the Tool Offset Setup and whether to save the last tool change number in the job files.

Value	Meaning
0	Auto Tool Changer NOT Installed
1	Auto Tool Changer Installed

Parameter 7 - Display Colors
This parameter determines what combination of colors will be used for display. If you have a color display, set this parameter to 0. If you have a monochrome display (especially a monochrome LCD panel) set this parameter to 1.

Parameter 8 - Available Coolant Systems
This parameter is used by Intercon to determine what coolant systems are available on the machine. It should be set as follows:

Value	Meaning
1	Mist Coolant (M7) only
2	Both coolant systems
3	Flood Coolant (M8) only

Parameter 9 - Display Language
This parameter determines what language will be used for menus, prompts and error messages.

Value	Meaning
0	English
1	Spanish
2	French
3	Traditional Chinese
4	Simplified Chinese

Parameter 10 - Macro M function handling
This parameter is a 4-bit field that controls various aspects of M functions. The following table shows the functions performed by the value entered in this parameter. The default value is 0.

Bit	Function Description	Parameter Value
0	Display M & G codes in M function macros?	Yes = 1, No = 0
1	Step through M function macros in Block Mode?	Yes = 2, No = 0
2	Brushless motor option: Decelerate smoothly to stop (pause) on M105 and M106? (Digitizing and Probing moves.) “Yes” means decelerate smoothly. Choosing “yes” takes more time on each probing move and is slightly less accurate. “No” means hard stop. “No” is faster and slightly more accurate but can cause excessive vibration on brushless systems.	Yes = 4, No = 0
3	Move to Z home on M6?	No = 8, Yes = 0

Parameter 11 – DP4 PLC Input Number and Contact State
This parameter is used for the PLC input number that is used by the DP4 Touch Probe. Allowable range is a single value, +/- 1 through 240. A Positive number indicates Closed on contact and a negative number indicates Open on contact. A list of default settings for different console types are listed below. **Warning:** Changing this parameter can cause damage to your probe. You should contact your Dealer or Local Tech Rep before any modifications are made.

Console Type/Model	Input Number	PLC Type/Model	Input Number
M39	15	DC3IO	14
M39s	14	RTK3	14
M400	15	Servo3IO	2
M400s	14	PLCIO2	15
M15-10	2	15/15	15
M400 ATC (RTK3)	14	RTK2	15
M400 ATC (PLCIO2)	15	Koyo ATC	1

Parameter 12 – Touch Probe Tool Number
This parameter is the tool number of the DP4 probe. Allowable range is 0 through 200. By default the value is 10. This is used to look up the length offset and tip diameter of the probe in the Tool Offset Library.

Parameter 13 – Recovery Distance
This parameter is the distance, in inches, the probe moves off a surface after initial contact (only during probing cycles), before returning to the surface to take a recorded reading. The default setting is 0.05.

Parameter 14 – Fast Probing Rate
The fast probing rate is used for positioning moves and initial surface detection, and is determined by the machines response time and the permitted probe deflection. The default is 10 in/min. This is a very conservative feedrate, in actual use 20 to 30 in/min is a good value and will not have any detrimental impact on accuracy in most cases.

Parameter 15 – Slow Probing Rate
The slow probing rate is used for the final measurement moves. The default setting is 1 in/min. The following are some typical accuracy tolerances with the corresponding value set in parameter 15:

Probing Rate	Accuracy
0.5 in/min	0.0001”
1 in/min	0.0002”
1.5 in/min	0.0005”
3.5 in/min	0.001”
18 in/min	0.005”

Parameter 16 – Maximum Probing Distance
This is the maximum distance that the Boss and Web probing cycles “search” for a surface in a given direction if no travel limits have been entered. The default setting is 10 inches. A larger value should be entered for the boss and web cycles if you are measuring very large features. These settings are conservative measurements and can be used initially for startup purposes. However, they can be changed to accommodate your work.

Parameter 17 – Detector Location Return Point
A non-zero value specifies the number of the reference return point (entered into the WCS menu) directly above a permanently mounted TT-1 tool detector. When the Auto function is called up in the tool offset library, the control will position the table to the return point specified by this parameter, and touch the tool off the TT-1 Tool detector. In controls with software version 7.07 or earlier, there are two separate return points available: enter 1 or 2. In controls with software version 7.08 or later, there are four separate return points available: enter 1, 2, 3, or 4.

Entry	Return Point
0	None
1	G28
2	G30
3	G30P1
4	G30P2

A zero indicates that the tool detector is no permanently mounted; automatic tool measurement will be performed without X/Y axis movement.

Parameter 18 - PLC Input Inhibit Parameter
This parameter stores the input 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.

Console Type/Model	Input Number	PLC Type/Model	Input Number
M39	10	DC3IO	15
M39s	15	RTK3	15
M400	10	Servo3IO	5
M400s	15	PLCIO2	10
M15-10	5	15/15	10
M400 ATC (RTK3)	15	RTK2	10
M400 ATC (PLCIO2)	10	Koyo ATC	2

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 0 will enable the MPG on power up. Bit 1 will enable the x100 speed limit. Bit 2 will allow the z-axis to be moved with the MPG while running a job independent of the x and y axes, it also enables parameters 45, 46 and 47 which define Z axis MPG operation.

Bit	Function Description	Parameter Value
0	MPG always active	Yes = 1, No = 0
1	MPG speed limit	x100 = 2, x10 = 0
2	Enable Z axis MPG*	Yes = 4, No = 0

***Z-axis MPG operation is not available with all controls. In addition a custom handwheel, custom wiring is required to connect the handwheel to the 6th axis encoder input.**

Parameters 20-30 - Motor Temperature Estimation-DC

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.

DC Brush Motors and Drives						
Parameter	Axis	Values	Values	Values	Values	Values
Servo Drive		8A Drive, 15 in/lb motors	12A Drive, 29 in/lb motors	15A Drive, 29 in/lb motors	15A Drive, 40 in/lb motors	25A Drive, 40 in/lb motors
20	N/A	72	72	72	72	72
21	X	0.028	0.02	0.027	0.03	0.04
22	Y	0.028	0.02	0.027	0.03	0.04
23	Z	0.028	0.02	0.027	0.03	0.04
24	4TH	0.028	0.02	0.027	0.03	0.04
25	X	0.68	0.68	0.68	0.68	0.68
26	Y	0.68	0.68	0.68	0.68	0.68
27	Z	0.68	0.68	0.68	0.68	0.68
28	4TH	0.68	0.68	0.68	0.68	0.68
29	ALL	150	150	150	150	150
30	ALL	180	180	180	180	180

Motor Temperature Estimation-AC

AC Brushless Motors and Drives						
Parameter	Axis	Values	Values	Values	Values	Values
SD Drive		SD3, SD1 750 W motors	SD3, SD1 1,2 KW motors	SD3, SD1 (finned heatsink) 1,2 KW motors	SD1 45A (finned heatsink) 3 KW motors	SD1 45A (finned heatsink) 4 KW motors
20	N/A	72	72	72	72	72
21	X	0.05	0.5	0.23	0.23	0.23
22	Y	0.05	0.5	0.23	0.23	0.23
23	Z	0.05	0.5	0.23	0.23	0.23
24	4th	0.05	0.5	0.23	0.23	0.23
25	X	0.68	9.0	12.0	12.0	14.5
26	Y	0.68	9.0	12.0	12.0	14.5
27	Z	0.68	9.0	12.0	12.0	14.5
28	4th	0.68	9.0	12.0	12.0	14.5
29	ALL	150	150	150	150	150
30	ALL	180	180	180	180	180
132	5th	0.05	0.5	0.23	0.23	0.23
136	5th	0.68	9.0	12.0	12.0	14.5

Parameter 31 – Spindle Speed Output Port

Parameter 31 determines the destination for the raw spindle speeds generated and outputted by the Control. Below are the possible values for this parameter. Note; if your machine uses a serial type spindle controller, you should not set this parameter to 0.

Value	Meaning
-1	DC3IO/RTK3/PLCIO2/Koyo PLC Direct (12-bit resolution)
0	RTK2 or 15/15 PLC (8-bit resolution)
1	COM1 - SPIN232, SERVO3IO, or to 3rd-party serial interface (12-bit resolution)
2	COM2 - SPIN232, SERVO3IO, or to 3rd-party serial interface (12-bit resolution)

Parameter 32 - Spindle Vector Drive Serial Port Baud Rate
This parameter is for the baud rate (e.g. 9600, 19200, etc.) of the serial port at which the control should communicate with the SPIN232 board. This parameter is only used if Parameter 31 is set to 1 or 2, for COM1 or COM2 spindle speed output.

Parameter 33 - Spindle Motor Gear Ratio (Baldor Vector Drive Only)
Warning - The default value for this parameter is 1 and should not be changed unless you have consulted your dealer or local Technical representative!!!
Parameter 33 is used for the gear or belt ratio between the spindle motor and the chuck in high gear range. It 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. 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. A value of 5 is used for the 6^{th} axis encoder input; this is used on SD3 based systems.

Spindle Encoder Plugged into?	DC System Value	AC System Value
CPU10 Encoder input 1	N/A	N/A
CPU10 Encoder input 2	1	17
CPU10 Encoder input 3	2	18
CPU10 Encoder input 4	3	19
CPU10 Encoder input 5	4	20
CPU10 Encoder input 6	N/A	21
SD3 spindle encoder input	N/A	5

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. AC systems cannot have a maximum feedrate above 100%.

Parameter 40 - Basic Jog Increment
This parameter holds the basic jog increment (0.0001" or 0.002 mm by default). This value is used by the x1, x10, and x100 jog keys (0.0001, 0.001 and 0.01 on older consoles); it also specifies the distance per click for handwheels (MPG).

Parameter 41 - Handwheel x100 Speed, User Jog Increment
On newer consoles, this parameter holds the actual handwheel speed in x100 mode. For normal x100 operation it should be 100. On some systems x100 is way too fast and this value is set to a more reasonable value such as 20 or 30.

On older consoles, this parameter holds the user jog increment (0.250" or 1.0 mm by default). The 0.250 jog key on older consoles uses this value.

Parameter 42 – Password for Configuration Menus
This parameter determines the password that the user must enter in order to gain full 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" Example: for the password to be 1234, set to 1234.1234
Any other number	Password is "137"

Parameter 43 – Automatic tool measurement options
This parameter is a 2-bit field that is used to configure properties of the TT1.

Bit	Function	Parameter Value
0	The height of the tool detector (parameter 71) will be subtracted from the measured height of the tool.	Yes = 1; No = 0
1	Which PLC input to use for the Tool Z reference measurement	Use Parameter 11 = 0 Use Parameter 44 = 2

Parameter 44 – TT1 PLC input number
This parameter is the input number that the TT1 is wired into on the PLC. If a shared PLC input is used for the TT1 and the DP4 probe, then the value can be left at zero or set to the same value as parameter #11.
Warning: If using a different PLC input for the TT1 and DP4, when setting the Z reference in the tool library with the DP4 don't use a ruby probe tip. The TT1 is continuity based and the ruby tip is not conductive!

Parameter 45 – Z-axis MPG enable PLC input
PLC input number that is used by the aux key or switch that you want to use to toggle between normal Z-axis operation and Z-axis MPG operation*. This parameter can be configured to use either a normally open or normally closed input by specifying a negative or positive value for the PLC input number. Also, this input must be held in a constant state in order for the function to remain active. For this reason, it is recommended that a non-physical input (M-functions INP 33-48) be entered for this parameter, which can then be programmed to latch/toggle and mapped in the PLC program to an Aux key if desired.

*Z-axis MPG operation is not available with all controls. In addition a custom handwheel, custom wiring is required.

Parameter 46 – Z-axis MPG handwheel Turns Ratio
This value is the number of clicks (detents) per revolution, if the MPG does not have detents use a value of 100.

Parameter 47 – Z-axis MPG handwheel encoder counts
This value is the number of counts generated per rotation of the handwheel. The higher the encoder counts the smoother the operation.

Parameter 56 – Feedrate Override Display Properties
This parameter is a 3-bit field that is used to define how the federate override is displayed in the status window.

Bit	Function	Parameter Value
0	Not used	
1	Display programmed rate not actual	Yes = 2; No = 0
2	Display a bar meter of percentage	Yes = 4; No = 0

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 for early detection of an axis if it’s stopped against some abnormal resistance, such that it will probably overheat later.

Parameter 64 – Fourth/Fifth Axis Pairing
This feature enables the 4th and 5th axes to be paired together or individually be run in a slaved state with any of the other axes. 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 or 5th axis. In order to pair both the 4th and 5th axes on the same system add the 4th axis value with the 5th axis value. Example: 4th axis paired with the X-axis and 5th axis paired with the Z-axis a value of 65 would be entered into parameter 64. The axes are slaved upon power up but it is still possible to move the paired (4th or 5th) axis independently if the axis is labeled. * NOTE: You cannot run Autotune on paired axes.

Value	Meaning
0	No Pairing (Default)
1	Pair 4th axis with X Axis
2	Pair 4th axis with Y Axis
3	Pair 4th axis with Z Axis
16	Pair 5th axis with X Axis
32	Pair 5th axis with Y Axis
48	Pair 5th axis with Z Axis
64	Pair 5th axis with 4th Axis

Parameters 65-67 - Spindle Gear Ratios
These parameters tell the control the gear ratios for a multi-range spindle drive. Up to four speed ranges are supported; high range is the default. Parameters 65-67 specify the gear ratio for each lower range, relative to high range. For example, if the machine is a mill with a dual range spindle, and the spindle in low range turns 1/10 the speed it turns in high range, then parameter 65 should be set to 0.1.

Parameter 65 is the low range gear ratio.
Parameter 66 is the medium-low range gear ratio.
Parameter 67 is the medium-high range gear ratio**.**

These parameters work in conjunction with the PLC program, which uses the states of INP63 and INP64 to signal to the CNC10 software which range is in effect, according to the table below.

PLC INPUT	Spindle Range			
	High Range	Medium High Range	Medium Low Range	Low Range
INP63	0	1	1	0
INP64	0	0	1	1

Parameter 68 – Minimum Rigid Tapping Spindle Speed
This parameter holds the value that the spindle slows down to from the programmed spindle speed towards the end of the tapping cycle. The lower the value, the more accurately the Z-axis will land on target, but at the expense of possibly stalling the spindle motor which in turn will cause Z-axis to fall short. If this value is too large, the off target error will increase. The suggested starting value is 100 RPM.

Parameter 69 – Duration for Minimum Spindle Speed Mode
This is the duration of time, in seconds, that the control will stay at minimum spindle speed. If the number is too small, overshoot will occur. If the number is too large, the user waits longer for the hole to be tapped at the slow speed specified by parameter 68. The suggested starting value is 1.25 seconds.

Parameter 70 - Offset Library Inc/Decrement Amount
Sets the increment and decrement amount used in the offset library.

Parameter 71 – Part Setup Detector Height
If this Parameter is set to a non-zero value, it indicates that the **F3 - Auto** feature in part setup should be available using the tool detector (TT1) instead of the probe. The value in this parameter is the height of the detector. A value of 0 disables this feature.

When this feature is enabled:
a) Probe detection (Parameter 18) is not checked
b) The tool number and/or edge finder diameter entered by the operator are used; Parameter 12 is ignored.
c) The value from Parameter 71 is added to (or subtracted from, depending on approach direction) the part position.

Parameter 72 – Data M Function Options
The setting of this parameter affects the operation of the data M functions M122 and M123.

Bit	Function Description	Parameter Value
0	Suppress output of axis labels by M122?	Yes = 1, No = 0
1	Insert commas between positions/values with M122 and M123?	Yes = 2, No = 0
2	Suppress spaces between positions/values outputted by M122 and M123?	Yes = 4, No = 0

Parameters 73, 74 - Canned Cycle Parameters
P74 specifies the number of the M-function that is executed at the bottom of the G74 or G84 tapping cycle.
P73 specifies the retract amount used during a G73 peck drilling cycle.

Parameter 75 – Summing Display Control
This parameter indicates which axes are to be summed and how the results are to be displayed on the DRO. The parameter can contain up to four digits. The position and value of each digit has special significance as indicated in the tables below:

Parameter Digit Position	Axis Display Controlled
1's Column	Axis 1
10's Column	Axis 2
100's Column	Axis 3
1,000's Column	Axis 4

Digit Value	Meaning
0	Summing off
1 – 4	Axis to Sum
5	(Reserved)
6	Disable display
7	Display if moved
8	Display if other moves
9	(Reserved)

Here are some examples using the axis summing display parameter:

Desired Display	Parameter
Sum Z-axis (3) with M (4), display sum in Z DRO position	400
Sum Z-axis (3) with M (4), display sum in M DRO position.	3000
Sum Z-axis (3) with M (4), display sum in Z DRO position, and suppress M display.	6400
Sum Z-axis (3) with M (4), display sum in M DRO position, and suppress Z display.	3600
Sum Z-axis (3) with M (4), display sum in Z DRO position, and show M only if M moves.	6300
Sum Z-axis (3) with M (4), display sum in Z DRO position, and show M if either Z moves.	7400

The DRO will display both labels when displaying a summed axis. For example, "ZM" or "MZ" depending on where the sum is displayed.

Parameter 76 – Manual Input Unrestricted Distance
This parameter is intended to be used with Z-axis summing. It defines the maximum distance from the summed axis start of travel in which manual movements can occur without causing a fault. Use a negative value to specify a distance from the minus travel limit, a positive value for a distance from the plus travel limit.

When used with manual drilling, for example, setting this parameter will allow the operator to keep a hand on the quill at all times and even begin pulling on the quill in anticipation of a programmed stop.

Setting this value to zero will cause a fault if there is any manual movement.

To completely disable manual movement restrictions, set this parameter to a value exceeding the total travel of the summed axis.

Minimum = -99999.9999, maximum = 9999.9999, default = 0, typical = +/- 1.0 inch or +/- 20.0 mm

Parameter 77 – Manual Input Movement Tolerance
This parameter specifies the manual movement tolerance while a job is running. It is intended for use with a quill locking mechanism. It allows the lock to distort and/or slip a small amount when under stress. If the quill moves more than the given tolerance, the job will stop with a fault. A typical setting for Parameter 77 is 0.005 inches.

Parameter 78 – Spindle Speed Display and Operations
Bit 0 specifies how the spindle speed is determined and displayed in the CNC10 status window. When set to 1.0, the spindle speed is determined by reading the encoder feedback from the axis specified according to parameter 35. Which has the number of encoder counts/revolution specified in parameter 34. When set to 0.0, the displayed speed is not measured; the speed is calculated based upon the set speed, spindle override adjustment, and gear range. Bit 1 allows the control to slow the programmed feed rate if the spindle speed slows down. Bit 2 will make the control wait until spindle at speed is at least the set percentage that is set in parameter 149.

Bit	Function	Value
0	Display actual spindle speed	Yes = 1, No = 0
1	Slave feed rate to programmed spindle speed	Yes = 2, No = 0
2	Wait for spindle at speed	Yes = 4, No = 0

Parameter 79 – Auto Brake Mode PLC Bit for Uniconsole-2
This parameter specifies which PLC bit signals the state of automatic brake mode when using the Uniconsole-2 console type. For other console types, it has no effect. This parameter can be changed to allow the Auto Brake mode key to be located in different positions on the Uniconsole-2 jog panel. The PLC program must be updated to reflect any change in this parameter.

Parameter 80 – Voltage Brake Message Frequency
This parameter specifies the number of times the “450 Voltage brake applied” message has to occur before we show it in the message window and message log. A value of 0 or 1 will display the message for every instance that it occurs.

Parameter 81 – Canned Cycle Parameter
P81 (when not equal to –1.0) specifies the M-function to be called in place of Z-axis movement during a G81 drilling cycle.

Parameter 82 – Spindle Drift Adjustment
This value is the number of degrees that the spindle will take to coast to a stop, when the spindle is turned off at the minimum spindle speed specified by parameter 68.

Parameter 83 – Canned Cycle Parameter
Parameter 83 specifies the clearance amount used during a G83 deep hole drilling cycle.

Parameter 84 – Canned Cycle Parameter
Parameter 84 specifies the number of the M-function that is executed after the return to the initial point of a G74 or G84 tapping cycle.

Parameters 87-90 - Autotune Accel Time and Ka
These parameters are used by autotune. Increasing the value will lengthen acceleration time and reduce the ka value given by autotune. Lowering the value will decrease the acceleration time and increase Ka. First set the parameters and then run autotune. The default value is 48 for DC systems and 36 for AC systems. The maximum value is 64 and the minimum value is 1. For the fifth axis, use parameter 152.

Parameters 91-94 – Axis Properties
These parameters may be used to set various axis properties, specifically the X, Y, Z and fourth axis.

Bit	Function Description	Parameter Value
0	Rotary/Linear Axis Selection	Rotary Axis = 1, Linear Axis = 0
1	Rotary Display Mode	Wrap Around = 2, Show Rotations = 0
2	Suppress direction check when doing Tool Check?	Don’t Check = 4, Check = 0
3	Suppress park function?	Don’t Park = 8, Park = 0
4	NOT USED ON MILL	
5	Linear Display of Rotary Axis	Linear Display = 32, Default Rotary = 0
6	4^{th} Axis works like Z axis	Yes = 64, No = 0
7	NOT USED ON MILL	
8	Tilt rotary axis	Yes = 256, No = 0
9	Hide an axis display from DRO	Yes = 512, No = 0

Bit 0: Turning this bit on will cause the DRO display for the affected axis to be displayed in degrees. Also this information is used by Intercon to make rotary axis support available (by setting parameter 94 to 1, indicating that the fourth axis is rotary). This bit is also used when performing inch/mm conversions: values for a rotary axis will not be converted since they are assumed to be in degrees regardless of the system of linear units.
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.
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).
Bit 5: This setting overrides only the DRO display options for an axis that has bit 0 set (including the Rotary Display Mode – bit 1) so that the display does not reflect a degree symbol or any indication of the number of rotations, but appears as a linear axis.
Bit 6: This bit only works for Parameter 94 (4^{th} axis). Setting this bit will cause the 4^{th} axis to respond to Z-axis only commands just like the Z-axis, for example issuing an M25 with this bit set will cause the Z and 4^{th} axes to go the home (G28) position.

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. The fifth axis Autotune distance is set in parameter 156.

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 99 are allowed for this parameter and default is 1.

Parameters 100–115 - Intercon parameters
These parameters are some of the Intercon setup parameters. See Chapter 10 for more information about these parameters. Changing values will change Intercon settings and may affect the output of the G-code program if it is re-posted.

Parameter 116 – A Axis Y Coordinate
This parameter is used in conjunction with a tilt table setup. It is used to set the Y coordinate of the centerline of the A axis (rotary)

Parameter 117– A Axis Z Coordinate
This parameter is used in conjunction with a tilt table setup. It is used to set the Z height of the centerline of the A axis (rotary).

Parameter 118 – B Axis X Coordinate
This parameter is used in conjunction with a tilt table setup. It is used to set the X coordinate of the pivot line of the B axis (tilt).

Parameter 119 – B Axis Z Coordinate
This parameter is used in conjunction with a tilt table setup. It is used to set the Z height of the pivot line of the B axis (tilt).

Parameter 120 – Probe Stuck Clearance Amount
This parameter specifies the distance that digitizing or probing functions will move to try to clear a stuck probe condition. A stuck probe condition exists when the probe detects a point and then moves away but the probe input has not changed. It is recommended that this parameter should not be changed from its default value without consulting a qualified technician.

Parameter 121 – Grid digitize prediction minimum Z pullback
This parameter specifies the minimum distance the Z-axis will move upward when pulling back from a surface. The digitizing function attempts to predict the slope of a part surface because time is saved when the Z-axis does not have to travel upward to the starting Z depth for every digitized point. When probe contact is made traversing in the XY plane, this parameter specifies the minimum distance the Z-axis moves upward before attempting another XY plane move. Smaller values are better when the surface being digitized has smooth curves. Larger values are better for surfaces that have steep walls. It is recommended that this parameter should not be changed from its default value without consulting a qualified technician.

Parameter 122 – Grid digitizing deadband move distance
This parameter specifies a deadband distance used for internal calculations when doing a clearance move. It is recommended that this parameter should not be changed from its default value.

Parameter 123 – Radial Clearance Move

This parameter only applies to radial digitizing and determines what type of positioning move the digitizing probe will make should it encounter an unexpected probe contact with the surface of the part during Radial Digitizing. Unexpected probe contact is defined as probe contact occurring while the probe is traversing towards the user defined center point.

With Parameter 123 set to 0: When the probe encounters an unexpected probe contact, the digitizing program stops data collection. The control then prompts the operator to jog the probe to a clear position. This can be any place inside the digitizing radius and above the part, that the probe stylus has a clear path to the defined center position. To restart data collection press **Cycle Start**. The probe moves in the XY plane from the position the operator placed it at, to the center position defined in the radial setup menu. After reaching the center position, the probe will feed down to the Z-axis position it was at when the data collection was interrupted. The digitizing run will resume with the probe approaching from the defined center position.

With Parameter 123 set to 1: When the probe encounters an unexpected probe contact, it will automatically move (with probe detection turned off) to the maximum Z height, then moves the X and Y-axis to the defined center position. The probe will then move to the Z position it was at when the unexpected contact occurred. It will then move from the defined center position, towards the measurement position it was trying to approach when the unexpected probe contact occurred and continue digitizing.

With Parameter 123 set to 2: When the probe encounters an unexpected probe contact, it will automatically move back to the defined center position (with probe detection turned off), at its present Z height. It will then move from the defined center position, towards the measurement position it was trying to approach when the unexpected probe contact occurred and continue digitizing.

Warning! Settings 1 and 2 should only be used with extreme caution because probe detection during some positioning moves is turned off, and damage to the probe or work piece could occur!

Parameters 124-127 PLC Inputs for Jogging

Parameters 124 – 127 allow up to 4 PLC inputs to be used for jogging of the first 2 axes on the control. The first 2 digits (1's and 10's) of the parameter specify the axis and direction; the 3rd and 4th digits (100's and 1000's) specify the PLC input being used.

1's and 10's digit	Function
40	Jog first axis minus
41	Jog first axis plus
42	Jog second axis minus
43	Jog second axis plus

For example: A value of 840 in parameter 124 will cause the first axis to jog minus when the PLC input 8 is closed and stop jogging when the PLC input 8 is opened, A value of 1243 in parameter 127 will cause the second axis to jog minus when the PLC input 12 is closed and stop jogging when the PLC input 12 is opened.

Parameter 128 – Handwheel (MPG) Mapping

This parameter selects how the axes are paired for handwheel operation. Each digit in the displayed number represents an axis. The first axis is at the far right. The value of each digit represents the companion axis, 1 to 5. A zero digit means no pairing. The table below shows how the digits are mapped to axes:

Axis:	5	.	4	3	2	1
Parameter value	0	.	0	0	0	0

Example Value	Axis/Companion 5	4	3	2	1	Comments
0.0000						No pairing.
0.1000		1			4	Axes 1 & 4 paired.
0.2100		2	1			Axes 1 & 3, 2 & 4 paired.
0.0021				2	1	Invalid – does nothing. Axes are paired with themselves.

Only manual axes that are paired with powered axes will produce a valid configuration. Manual axes specified by Parameter 128 must be properly configured as handwheel axes in the Motor Parameters screen of the Machine Configuration. See the Machine Configuration section earlier in this chapter.

Parameter 129 – Handwheel (MPG) Display
By default, manual axes paired by Parameter 128 are not displayed in the DRO. This parameter enables the display of the manual axis in the DRO, if desired. The parameter has the same axis mapping for each digit as shown in Parameter 128. To display an otherwise hidden manual axis, set the digit corresponding to the axis number to a "1". For example, "0.1000" would display axis 4, if it is a manual axis that is paired with some other powered axis.

Parameter 130 – Z-axis on/off selection (see Parameter 131)

Parameter 131 – 4th axis on/off selection (only uses 1's and 10's digit)
These parameters control the display of the 3rd and 4th axes, respectively. The tens digit of the parameter value specifies the label of the affected axis when it is enabled, with values 1-9 corresponding to axis labels ABCUVWXYZ. The ones digit specifies the label of the axis when it is disabled, with 0.0 meaning the axis is not switchable, 1.0 meaning it turns off (N), a 2.0 meaning manual (M), and a 3.0 meaning 2-axis with manual Z (@). P130 also supports additional modes depending upon the value of the hundreds digit. When P130/P131 is configured for axis switching, the Setup menu displays function keys **F5**/**F6** to switch the axes.

Example 1:
A value of 192 in parameter 130 will toggle the 3rd axis between Z and M and power off all axes. The 1 sets bit one to power all axes off, the 9 enables the 3rd axis to "Z", and the 2 changes the axis label to "M" once toggled with the F5 key in the Setup menu.

Example 2:
A value of 392 will toggle the 3rd axis label between Z and M and power off all axes and receive its positions from the 4th axis encoder input. The 3 sets bits two and one to power off all axes and use the 4th encoder input as a scale input, the 9 enables the 3rd axis to "Z", and the 2 changes the axis label to "M" once toggled with the F5 key in the Setup menu.

Example 3:
A value of 61 in parameter 131 will toggle the 4th axis between W and N. The 6 enables the 4th axis to "W" and the 2 changes the axis label to "N" when toggled with the F6 key in the Setup menu. This will turn the 4th axis on and off.

Parameter 130 Options:

Hundred's Digit	**Function Description**	**Parameter Value (add**
Bit1	Axis motor power when switching to two-axis mode.	1 = power all axes off, 0 = power 3rd off only
Bit2	Use 4th encoder input for scale input	2 = Use 4th encoder input, 0 = no manual input
Bit3	Use 5th encoder input for scale input	4 = Use 5th encoder input, 0 = no manual input

Enabled Axis:	**A**	**B**	**C**	**U**	**V**	**W**	**X**	**Y**	**Z**
Ten's Digit	1	2	3	4	4	6	7	8	9

Disabled Axis:	**N**	**M**	**@**
One's Digit	1	2	3

Parameters 132 – 5th Axis Heating Coefficient
This parameter sets the heating coefficient for the 5th axis. See parameters 20-30 for more information.

Parameters 136 – 5th Axis Cooling Coefficient
This parameter sets the cooling coefficient for the 5th axis. See parameters 20-30 for more information.

Parameter 140 – Message log priority level
This parameter controls the messages that are written to the message log, which can be accessed through the **F9 - Logs** function in the Utilities menu. With the priority Level set to 1, CNC10 logs numbered error messages and most other messages except "Moving...", "Jogging...", "Stopped", etc. At priority Level = 9, all messages are logged including user prompts. Message logging can be disabled be setting this parameter to –1.

Parameter 141 – Maximum message log lines
This parameter is the number of lines that will be kept in the message log. If this parameter is set to 10,000, for example, the newest 10,000 messages will be retained. CNC10 will delete the oldest messages, trimming the log file to the given number of lines at startup and periodically while CNC10 is in an idle state. Parameter 142 controls the frequency of the log cleanup.

Parameter 142 – Message log trim amount
This parameter is the number of additional lines above the minimum that can be added to the log before it is reduced to the minimum size. Setting this parameter to a lower value will cause the log file to be trimmed to its minimum size more often. The higher the value, the less often the log will be trimmed. The speed of the disk drive and total size of the log file at the time it is trimmed will determine how long the log cleanup takes. Under most circumstances, using 10,000 and 1,000 for parameters 141 and 142 will provide a reasonable and useful log size with no noticeable effects on performance. If parameters 141 and 142 are set to excessively high values, the message "Trimming excess lines from log file" will be presented. This message will appear at startup and very infrequently when CNC10 is idle. Normal operation can proceed after the message disappears. If the delay is unacceptable, reduce the values of parameters 141 and 142.

Parameter 143 – DRO Properties (load meters, 4/5 digits, DTG)
This parameter controls the display of the axis load meters and 4/5 digit DRO precision.

Bit	Function Description	Parameter Value
0	Enable Load Meters	Enable = 1, Disable = 0
1	Load Meter Outline	Enable = 2, Disable = 0
2	DRO 4/5 Digit Precision	5 digits = 4, 4 digits = 0
3	Mini DRO (Distance to Go)	Enable = 8, Disable = 0

Use a value of 3 to display load meters with outlines. The axis load meters will be colored green for values that are up to 70% of maximum power output, yellow for values between 70% and 90%, and red for values between 90% and 100%. The axis load meters appear below the DRO for each axis (see Chapter 1).

Parameter 144 – Comparison Rounding
This parameter determines the built in rounding for the comparison operators (‘EQ’, ‘NE’, ‘LT’, ‘GT’, etc.) in expressions. Rounding of comparison arguments is necessary due to extremely small errors that are part of every floating-point calculation. The result of such errors is that two floating-point values are rarely exactly equal. The value of parameter 144 represents the precision of comparison in places after the decimal point. If the parameter is set to 9.0, for example, then comparison operators will declare two numbers that differ in value by less than 0.0000000005 as being equal. The value 0.0 is a special value that turns comparison rounding off. When comparison rounding is off, it is up to the G code programmer to build the precision into conditional statements, for example “IF ABS[#A - #B] LT 0.00005 THEN GOTO 100”. When comparison rounding is off, the “EQ” usually returns “false”. If parameter 144 is set to 9, the programmer can shorten the previous example to “IF #A EQ #B THEN GOTO 100”.

Parameter 145 – Advanced Macro Properties (Fast Branching)
This parameter turns fast branching on (1) and off (0). The other bits of this parameter are reserved for future use. If fast branching is disabled, CNC10 searches forward in the program for the first matching block number and resumes searching, if necessary, from the top of the program. For this reason, backward branches take longer than forward branches and backward branch times depend on the total program size. If the program is significantly large, use of the GOTO statement could introduce temporary pauses.

When fast branching is enabled, CNC10 remembers the locations of block numbers as it finds them during program execution. Backward branches always take place immediately. The first forward branch to a block not yet encountered will take additional time as CNC10 searches forward for the block number; however, subsequent forward branches to that block number will take place immediately. The trade-off for using fast branching is that all line numbers at a given level of program or subprogram must be unique and programs will use more memory (approximately 16kilobytes of memory for every 1000 block numbers in the program.)

Parameter 146 – Feed Hold Threshold for Feed Rate Override
This parameter sets the lowest value permitted as the feed rate override percentage before feed hold is engaged. Feed hold will be released when the override percentage is greater than this value.

Parameter 148 – Fast Jog before Home Set
This parameter allows you to fast jog before home is set. A value of 1 will enable it.

Parameter 149 – Spindle Speed/Surface Footage Threshold
This parameter defines the threshold at which linear motion will be permitted. It is specified as a percentage of the programmed spindle speed. For example a value of 0.8 would inhibit linear motion until 80 percent of the programmed spindle speed was reached. To enable this parameter a value of 4 must be added to parameter 78.

Parameter 150 – Run-Time Graphics
This parameter controls the default value of the Run-Time Graphics option in the Run Menu. It can also allow you to display CSR positions when graphing. If this parameter is set to 0.0, the RTG option in the Run Menu defaults to OFF when CNC10 is started.

Bit	Function Description	Parameter Value
0	Sets Run Time Graphics option default to ON	Enable = 1, Disable = 0
1	Displays CSR positions in graphing	Enable = 2, Disable = 0

Parameter 151 – Repeatability tolerance for probing and radial digitizing.
Default is 0, repeatability check disabled. When disabled, only one measurement per point is taken in the probing cycles and radial digitizing. When enabled, a minimum of two measurements are taken per point and the difference (if any) is then compared to the repeatability tolerance as set in Parameter 151. If the difference is less than or equal to parameter 151 the point is stored and probing continues. If the difference is greater than parameter 151, 2 more measurements are taken and the process repeated up to a maximum of 10 times. In probing cycles, if the repeatability tolerance cannot be met the cycle is cancelled and an error message generated. In radial digitizing, the point is discarded and digitizing continues without interruption.

Parameter 152 – 5th Axis Autotune Accel Time and Ka
This parameter sets the Autotune accel time and Ka for the 5th axis. See parameters 87-90 for more information.

Parameter 156 – 5th Axis Autotune Move Distance
This parameter sets the autotune move distance for the 5th axis. See parameters 95 – 98 for more information.

Parameter 160 – Enhanced ATC

This parameter controls enhanced automatic tool changer (ATC) options. A value of 1 indicates a nonrandom type of ATC (carousel ATC) and a value of 2 indicates a random type ATC. A value of 0 disables enhanced ATC features. A warning is displayed when attempting to enable enhanced ATC features as these features work in conjunction with specific PLC programs. The enhanced ATC option has the following characteristics:

The beginning of an M6, whether it be a customized CNC10.M6 routine or not, flags the job file, setting the ATC error flag field to 1.
The end of an M6, whether customized or not, performs the following:
(a) The ATC error flag is set to zero.
(b) The tool number displayed on the screen is updated and this value is saved in the CNC10.JOB file.
(c) The tool library bin fields are updated in this manner:
If there was a valid tool in the spindle at the start of the M6, then the tool library bin field for this tool will be updated with either the "putback" field for that tool (if nonrandom type) or the current ATC carousel position (for random type). For both random and nonrandom types, the "putback" field is set to 0. The "putback" field is an internal field for each tool in the tool library. It can be displayed by using the CNC10CONV utility with the –dt option to display the tool library.
For nonrandom types, the new tool now in the spindle will have its "putback" field updated to the current ATC carousel position.
For both random and nonrandom types, the new tool now in the spindle has the bin field set to 0.
The current ATC carousel position is constantly monitored. When there is a change, the ATC bin field in the CNC10.JOB file is updated and the file is saved. The ATC carousel position is read from PLC bits OUT41-OUT48, which should be written by the PLC program in a binary format (not BCD).
At the start of running a job, to include MDI mode, the ATC error field is checked. If this field is 1, then a warning message is displayed with a prompt to either clear the fault by entering a 'Y' or canceling the job by pressing some other key.
A tool change is not performed if the requested tool is already in the spindle.
An M107 command sends the bin number for the specified tool number, not the tool number.
For random types, tool changes in Intercon are posted as a tool change (Tnn M6) followed by a pre-fetch command for the next tool in the program (Tn2 M107). This allows the PLC program to rotate the tool carousel to the next tool while a job continues with the current tool.
For random types, a job search for a tool number will look for lines of the form Tnn M6, i.e., the search bypasses lines of the form Tnn M107, which are just pre-fetch commands.
The tool library allows editing of the bin fields to specify which carousel bin number the tools are stored in.

Parameter 161 –ATC Maximum Tool Bins

This parameter sets the number of tool changer bins (carousel positions) used with the enhanced ATC option described above. PLC programs are responsible for reading this value. The tool library interface uses this parameter to validate bin fields and perform initialization of the bin fields.

Parameters 162 – Intercon M6 Initial M-Code

This parameter affects how Intercon programs post M&G codes for a tool change operation. When set to a non-zero value, Intercon will post out an M-code at the start of a tool change and an M5 command after the Txx M6 command. This parameter should be set to 17 so that Intercon will post an M17 code at the start of a tool change. The M17 command turns off spindle and coolant and starts the spindle orientation process.

Parameters 164 – ATC Feature Bit

This parameter controls specific features of the ATC system. It should be set to 1.0 to enable the ATC Reset feature, which will appear as the **F6 - ATC** Reset function key in the tool library menu. This parameter only works with ATC3 plc programs.

Parameters 166 – 5th Axis Properties

This parameter sets the axis properties for the 5th axis. See parameters 91-94 for more information.

Parameters 170-179 – XPLC Parameters
These parameters are accessed by the XPLC through LP0 - LP9 commands. Parameters 177, 178, 179 have been standardized for specific applications. Parameter 177 is used for trouble shooting purposes only.

Parameter 178 – PLC I/O configuration
This parameter can be use to set switch types from NC to NO and some other options. Each Bit corresponds to a different function. All values are to be added to the current setting. For example, if you need to reverse M10 and M11 and parameter 178 currently has a value of 17. (AC Drive and a Lube pump that closes a contact on fault) Change this parameter to 273 (current value - 17 + 256 = 273). **NOTE:** This parameter works only with specific PLC programs. The PLC program installed in the control MAY NOT be mapped as indicated below. These parameters should only be changed by a qualified Centroid technician. The example given below is intended for reference only:

Bit	Function	Default state	Opposite State
0	Lube Fault	Closed = OK	Add 1
1	Spindle Fault	Closed = Fault	Add 2
2	Air Fault	Closed = OK	Add 4
3	Tool Counter Sensor	Closed = Count	Add 8
4	Servo Fault	Closed = OK	AC Drive – add 16
5	Zero Speed Signal	Closed = Zero Spd	Add 32
6	Orient Complete	Closed = Oriented	Add 64
7	Low gear Reverse Spindle	N/A (0)	Add 128 No reverse spindle
8	Reverse Clamp M10/M11	N/A (0)	Add 256
9	Spin Range Input	Closed = Low gear	Add 512
10	Chiller Fault	Closed = OK	Add 1024

179 – Lube Pump Operation
This parameter can be configured to control a variety of lube pumps. The value is formatted as MMMSS, MMM for minutes and SS for seconds. Below is a table of some examples. For more information on setting this parameter please refer to TB171 or contact your Dealer.

Type of Pump	MMM	SS	Operation
Mechanical/CAM	0	0	179=0 Power is on when machine is running a job or in MDI Mode
Electronic “lube first”	16	00	179=1600 Holds power on to the pump for 16 minutes of job or MDI time
Electronic “lube last”	16	00	179=1600 Holds power on to the pump for 16 minutes of job or MDI time
Direct Controlled Pump	30	15	179=3015 Waits for 30 min of job or MDI time, then applies power for 15 seconds.

Parameters 180 – File Transfer COM Port
This parameter specifies which COM port will be used for file transfer. Accepted values are 0 disabled and 1-4 for COM1 – COM4. Setting this parameter to an accepted value other than 0 will provide an **F6 - Download** and an **F7 - Upload** key in the Load Job Menu.

Parameters 181 – File Transfer Baud Rate
This parameter sets the maximum file transfer rate for serial communication. The value of this parameter is in KBaud and has a range of 1.2 to 115.2. The default is 19.2Kbaud. The longer the serial cable the lower the baud rate that can be used for file transfer.

Parameters 182 – File Transfer Bit Parameters
This parameter sets the number of data bits, type of parity and the number of stop bits for the serial communication file transfer. The default value is 801 for 8 data bits, no parity and 1 stop bit.

Digit	Function	Value
1's	Stop bits	1 or 2 stop bits accepted
10's	Parity	0 = No Parity; 1 = Even Parity; 2 = Odd Parity
100's	Data bits	5 – 8 data bits accepted

Parameters 183 – File Transfer Flow Control
The setting of this parameter determines the COM port file transfer flow control.

Value	Meaning
0	No Flow Control
1	Software (XON/XOFF) Flow Control
2	Hardware (CTS/RTS) Flow Control

Parameters 184 – File Transfer Timeout
This parameter is used for the timeout for downloads. So the value set in this parameter is the amount of time you have to start the file transfer after the **F6 - Download** button is pressed or the download will time out. The default value of this parameter is 10 sec, but can be set from 6 to 600 seconds (10 mins).

Parameters 185 – File Transfer Options
This is a 2 bit parameter to set file transfer options.

Bit	Function	Value
0	Ignore Carriage Return on downloads	1= Yes, 0 = No
1	Translate NL (new line) to CR on upload.	2= Yes, 0 = No

Parameters 187 – Hard Stop Homing
This parameter is used when homing off hard stops. The value set in this parameter determines the amount of current sent to the motor while homing. Value range is 0-32000; typical value for a DC system is 16000.

Parameters 188-199 – Aux Key Functions
These parameters are used to assign a function to aux keys 1-12 (i.e. P188 = Aux1 … P199 = Aux12). The following is the list of possible functions that can be executed when an aux key is pressed.

Function	Parameter Value
No Function	0
Input X Axis Position	1
Input Y Axis Position	2
Input Z Axis Position	3
Set Absolute Zero	4
Set Incremental Zero	5
One Shot - Drill	6
One Shot - Circular Pocket	7
One Shot - Rectangular Pocket	8
One Shot - Frame	9
One Shot - Face	10
Execute M Code file	m11*
Free Axes	14
Power Axes	15

Function	Parameter Value
XYZ Set Absolute Zero	16
One Shot - Drill Bolt Hole Circle	17
One Shot - Drill Array	18
Jog Axis 1 (+)	21
Jog Axis 2 (+)	22
Jog Axis 3 (+)	23
Jog Axis 4 (+)	24
Jog Axis 5 (+)	25
Jog Axis 1 (-)	31
Jog Axis 2 (-)	32
Jog Axis 3 (-)	33
Jog Axis 4 (-)	34
Jog Axis 5 (-)	35

For example, if you wanted Aux4 to call up the Circular Pocket – One Shot. You would set parameter 191 to 7.

The Input Axis Position functions must be used with the Set ABS/INC Zero functions. After entering the desired value at the input field provided by the Input Axis Position function, press an aux key assigned either the function Set ABS Zero or Set INC Zero.

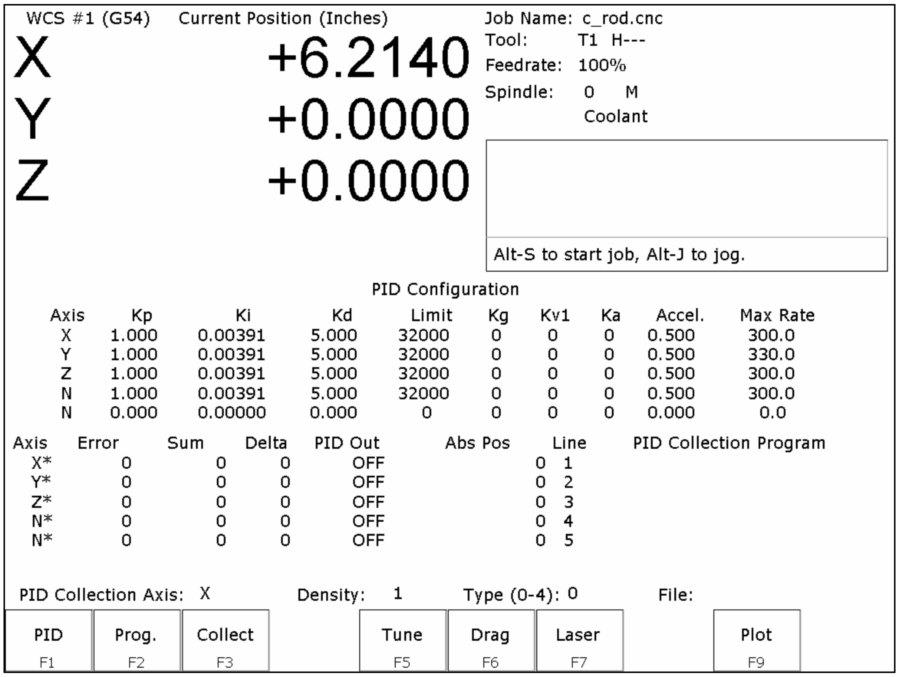
* *m* is the number of the M code to execute. For example, if the parameter value is set to 7211, the file CNC10.M72 will be loaded when the Aux key was pressed and executed after Cycle Start is pressed.

Custom overlays with the keys that represent these functions are available; contact your dealer for pricing.

All remaining parameters are reserved for further expansion.

PID Configuration

Pressing **F4 - PID** 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.



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 **<u>DRAMATIC</u>** 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 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.

<u>Accel</u>: (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.

<u>Max Rate</u>: 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.

WARNING: Improper PID values can ruin the machine, cause personal injury, and/or destroy the motor drives!!!

F2 - PID Collection Program

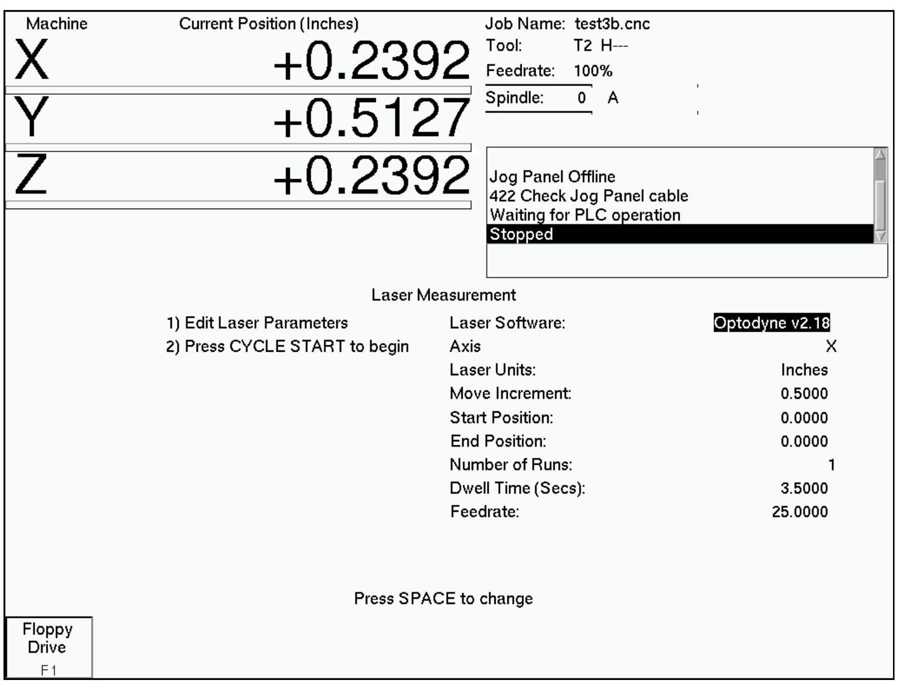
This option allows qualified technicians to test the PID parameters by entering up to 5 lines of G-codes to be executed with the Collect Data command below.

F3 - Collect Data
This option allows qualified technicians to collect data on the movement of one of the motors. The data is saved using the file name entered at the file prompt located at the lower right hand side of the screen.

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 and 156 to change the autotune distance) 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 parameters 95-98 and 156 for 5th axis.)
* NOTE: You cannot run Autotune on paired axes. 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. To run a drag test press **F6 - Drag** to begin the drag test. Press **F1 - Next Axis** to select the axis you wish to check and then 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:\CNC10 directory. If significant drag occurs, a message will be displayed on-screen. Contact your dealer to correct the problem as soon as possible.

F7 - Laser
This option is used by qualified technicians to take automated laser measurements and create or adjust the ballscrew compensation tables. Do not attempt to run automatic laser compensation without first contacting your dealer for details.



F8 - Drive
This menu will only appear on AC systems. It is not for general viewing and definitely not for modification by any unqualified individual. For more information about this menu option, refer to the SD installation manual.

F9 - Plot
This option is used by qualified technicians to plot data.

Handwheel Configuration

If you are using a manual input as a handwheel (MPG) input, be sure to configure all handwheel/MPG parameters. This list serves as a guide to configuration of the handwheels. Motor Parameters do not apply to MPG's that use the special MPG input. You may configure any unused encoder input as a handwheel input.

Screen	Parameter	Value	Comments
Jog Parameters	Travel (-), Travel (+) For an axis controlled by a handwheel.	Actual travel limits of the powered axis.	Axis controlled by a handwheel must have travel limits set.
Motor Parameters	Label	M	Handwheel input must be a manual axis.
Motor Parameters	Motor Revs/Unit	Number of "clicks" per rev.	If the wheel has no detents, use 100.
Motor Parameters	Encoder Counts/Rev	Actual number of counts generated per rotation of the handwheel.	Use higher resolution encoders for smoother operation.
Motor Parameters	Lash, Limits, Homes	0, 0, 0	Do not apply to handwheels.
Motor Parameters	Direction reversed, Screw Compensation	N, N	Do not apply to handwheels.
Machine Parameters	Parameter 19 – MPG Modes	As desired to select MPG on at power-up and MPG speed limit.	Be sure to enable or disable 100x operation here. See Machine Parameters for more information.
Machine Parameters	Parameter 40 – Basic Jog Increment	0.0001 in. or 0.002 mm by default.	This specifies the distance per "click" in x1 mode. Note: Also used for jogging.
Machine Parameters	Parameter 41 – Handwheel 100x Speed, User Jog Increment	Set to 100 for 100x movement. If this is too fast, choose a smaller value.	This speed will be used in 100x mode.
Machine Parameters	Parameter 128 – Handwheel Mapping	As needed to achieve the desired mapping.	See Machine Parameters for more information.
Machine Parameters	Parameter 129 – Handwheel Display	0 will work fine. Handwheel display will be suppressed.	See Machine Parameters for more information.
Machine Parameters	Parameters 130, 131 – Z and 4th axis on/off	0 will work fine. See Machine Parameters for more information.	Be aware that use of this feature may cause the handwheel to be turned on and off when the axis is switched.

The distance per turn of the handwheel in x1 mode is determined by the following equation:

Distance/Turn = Distance/Click * Clicks/Turn

Parameter 40 is the distance/click. Motor parameter Revs/Unit holds the Clicks/Turn value. You may adjust the Clicks/Turn value to achieve a different distance per turn. For example, if Parameter 40 is 0.0001 inches and Clicks/Turn is 100, the distance per turn is 0.01 inches. To get 0.05 inches per turn, use 500 clicks per turn. (This assumes that the encoder counts per rev is accurate.)

Be aware that Axis Summing parameter (75) may conflict with handwheel configuration. If you wish to use both handwheels and axis summing, be sure that the manual input for axis summing is the first manual axis. Axis summing cannot use a manual input that is used as a handwheel.

Test
This menu only appears if the system has not yet been configured and initialized or if a new solid state disk has been installed. For more information, please contact your dealer.

ATC Init.
This menu will only appear on enhanced ATC systems. Please see your ATC manual for further information.

Chapter 15

CNC10 Messages

CNC10 Startup errors

Error	Message	Cause	Effect	Removed
101	Error initializing graphics... cannot continue	Missing *.ggf files	Exit CNC10 with return code 63.	Re-install CNC10 software
102	Error initializing CPU7... cannot continue	Error while sending *cnc8.hex*. Other messages with more detail of error appear on screen before this message.	Exit CNC10 with return code 63.	Contact dealer
103	Error sending setup (windowed message).	ESC key pressed while sending setup	No setup command sent to CPU10. CPU10 probably not responding.	Timed message
104	Error sending PID setup (windowed message).	ESC key pressed while sending PID setup.	No PID setup command sent to CPU10. CPU10 probably not responding	Timed message
105	*cnc10.plc* file read error..cannot continue	Missing or error in *cnc10.plc*	Exit CNC10 with return code 63	Contact dealer
106	The PC clock appears to be wrong	The time on the PC internal clock is earlier than the time recorded in a previously stored file	None	Start of new job

Messages issued upon exit from CNC10

Error	Message	Cause	Effect	Removed
201	Return code 63	CPU10 not responding, or *cnc8.hex*, *cnc10.plc*, or font file is missing or damaged	Exit CNC10 with return code 63	Contact dealer
202	Return code 64 (start menu)	A floating-point math error occurred	Exit CNC10 with return code 64	Contact dealer. Re-start CNC10
203	Return code 65 (start menu)	*cnc10.cfg* file is missing or damaged	Exit CNC10 with return code 65	Contact dealer

Messages and Prompts in the Operator Status Window Status messages

Error	Message	Cause	Effect	Removed
301	Stopped	No operations in progress	None	
302	Moving...	Motors are moving while a CNC program is running	None	
303	Paused...	Motion is paused while a CNC program is running (FEED HOLD)	None	
304	MDI...	CPU7 running in MDI mode	None	
305	Processing...	CPU7 running in a mode other than MDI	None	
306	Job finished	Normal end of CNC program	None	
307	Operator abort: job canceled	ESC or CYCLE CANCEL pressed	Job canceled	Start of new job
308	Waiting for input #NN	M100 or M101 executing.	'Mnn' or 'M6: Insert Tool # NNN Tool library description' message displayed if M function macro executing	After input is received
309	Waiting for CYCLE START button	M0, M1, M100/75, or Block Mode	'Block Mode' message displayed if CNC program running in block mode	After CYCLE START pressed
310	Waiting for output #NN	M100 or M101 executing	'Mnn' or 'M6: Insert Tool # NNN Tool library description' message displayed if M function macro executing	After output is in correct state
311	Waiting for memory #NN	M100 or M101 executing	'Mnn' or 'M6: Insert Tool # NNN Tool library description' message displayed if M function macro executing	After memory bit is in correct state
312	Waiting for PLC operation (Mnn)	PLC program not clearing PLC operation in progress	'Mnn' or 'M6: Insert Tool # NNN Tool library description' message displayed if M function macro executing	After PLC program completes operation in progress
313	Waiting for dwell time	G4 executing	'Mnn' or 'M6: Insert Tool # NNN Tool library description' message displayed if M function macro executing	After specified time has elapsed
314	Input search data	Run/search key pressed	None	After search data input
315	Searching...	Run/search in progress	None	
316	Search complete. Processing...	Run/search mode. Search successful. Preprocessing job	None	
317	Waiting for automatic tool change	M6 executing with automatic tool changer	None	After changer signals that tool change is complete
318	_ axis too close to switch	Motor shaft is in the wrong position at the home switch release point	May result in unreliable homing	Re-align motor shaft

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.

Error	Message	Cause	Effect	Removed
401	PLC failure detected	CPU10 stopped with PLC failure bit set	Job canceled	When PLC failure bit removed. Typical implementation: correct PLC then press and release EMERGENCY STOP
402	PLC Online	PLC has returned on line	None	
404	Spindle drive fault detected	CPU10 stopped with spindle drive fault bit set	Job canceled	When spindle drive fault removed by PLC. Typical implementation: Check inverter for fault or reset spindle contactor OCR, then press and release EMERGENCY STOP
405	Lubricant level low	CPU10 stopped with low lube fault bit set	Current job continues to run, but a new job cannot be started	When low lube fault bit removed by PLC. Typical implementation: add lube then press and release EMERGENCY STOP
406	Emergency Stop detected	CPU10 stopped with no fault bits set	'Fault: Job canceled.' prompt	When Emergency Stop released
407	X+ limit (#1) tripped	CPU10 stopped with limit switch status	Job canceled	Start of new job, when limit cleared
408	Programmed action timer expired	M103 time expired before M104 encountered	Job canceled	Start of new job
409	_ axis lag	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	Job canceled	1.If the problem is occasional heavy cuts, slowing down the cutting feedrate can solve the problem. 2.If the problem only occurs on high speed moves then either the maximum speed or the acceleration is set too high. Lower the values in the Motor Setup screen or rerun Autotune again to determine new values. 3.If the problem is persistent lag errors in normal operations it indicates that the motors are too weak to handle the required loads. Increase the gear ratios or get more powerful motors
Note: If the Lag Distance (Allowable Following Error) is exceeded for more than .025 seconds, then no acceleration will occur on any axis. However, no error message is generated at this point because no fatal error exists.				

Error	Message	Cause	Effect	Removed
410	_ axis position error	A position error > .25 inches is detected on any axis. All axis motion is stopped, power to the motors is released (all servo drive commands cease) and the CNC program is aborted. The probable causes of this error are: 1.The motor is wired up backwards. 2.Noise is getting into the system via the motor cables (the line integrity has been violated). 3.An encoder error occurred.	Job canceled	Try a slow jog the motor and watch the DRO position. If the position on the DRO goes opposite the direction indicated on the jog button, then the motor is wired up backwards. Change the motor wiring. Check the motor cabling paying particular attention to the ground connections. Replace the cable if it is damaged or repair the motor connections. Jog the motor awhile, at the maximum rate, using the fast jog buttons. (Check the fast jog rate in the motor jog parameters screen to make sure it is set equal to the maximum motor rate.) If the motor seems to jump around rather than accelerate and decelerate smoothly then you are probably fighting an encoder error. Swap the motor with one from another axis and see if the error follows the motor. If it stays with the axis, replace the CPU. If it follows the motor, replace the motor cable. If the problem still persists, replace the motor and encoder. Start of new job.
411	_ axis full power without motion	90% Power (PID Output > 115) is applied to any axis and no motion >.0005 inches is detected, for more than the time specified in parameter 61 (usually .5 seconds). All axis motion is stopped and the CNC program is aborted. The probable causes of this error are: 1.) One of the axes is against a physical stop. 2.) The servo drive has shutdown due to a limit switch input. 3.) The Z home switch is the same as the Z + limit switch.	Job canceled	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 re-zero to a point that permits the required CNC moves to be made. If the axis is not on a physical stop, check the limit switch to see if it is tripped. If it is then the software is commanding a move into the switch but the hardware is shutting the move down. Go to the motor setup screen and enter the limit switch if this is applicable. Make sure the switch input is not unstable or noisy. If it is then replace the switch. If the problem persists it may be necessary to create separate home and limit switch inputs. Use slow jog to move opposite the direction causing the error and clear all limit switches. Jog toward the direction causing the error, if no motion occurs then a servo drive failure is indicated. Start of new job.

Error	Message	Cause	Effect	Removed
412	_ axis encoder connection is bad	Axis is enabled but a differential encoder signal is not detected. May indicate a loose or severed encoder cable or a bad encoder.	All axis motion is stopped and the CNC program is aborted.	Reconnect encoder or repair encoder and/or encoder cable.
413	CPU Failure #01: power down	CPU10 has experienced a problem with the PC reset line. Z80 Failure. Problem with the ZiLOG chip.	Power down, and then power up the system. The error should disappear.	Never - system must be powered down.
414	CPU Failure #02: power down	CPU10 detected CPU failure. DSP failure.	Power down, and then power up the system. The error should disappear.	Never - system must be powered down.
415	CPU fault #XX detected	Invalid stop reason from CPU10.	Power down, and then power up the system. The error should disappear.	Never - system must be powered down.
416	Motion fault #XX detected	Invalid motion status from CPU10.	Power down, and then power up the system. The error should disappear.	Never - system must be powered down.
417	Abnormal end of job	Job ended without reason.	Job canceled	Start of new job.
418	Search data not found	Requested search input data not found in loaded CNC file. Removed: Jogging, start of new job, other error.	Job canceled	Start of new job
419	Search line in embedded subprogram	Requested search line is part of an embedded subprogram; Search can only be used to start in the main program.	Job canceled	Start of new job
420	_ axis motor overheating	CNC10 estimates that a motor has reached the warning temperature (set in Parameter 29).	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.	When next message appears
421	Motor(s) too hot: job canceled	CNC10 estimates that one or more motors have reached the limit temperature (set in Parameter 30).	The current job is canceled and power is released.	Start of new job (after motors have cooled below warning temperature).

Error	Message	Cause	Effect	Removed
422	Jog Panel Offline	Jog panel failure or loose cable.	All buttons on jog panel are inoperative.	By reconnecting jog panel cable and appearance of next message.
423	Jog Panel Online	Loose jog panel cable has been reconnected.	All buttons on jog panel are operative.	
424	Feedrate Override Offline	Jog panel failure or loose cable.	Feedrate knob and some jog panel keys are inoperative.	By appearance of next message.
425	Feedrate Override Online	Loose jog panel cable has been reconnected.	Feedrate Override knob is operative.	
426	Spindle Override Offline	Jog panel failure or loose cable.	Spindle knob and some jog panel keys are inoperative.	By appearance of next message.
427	Spindle Override Online	Loose jog panel cable has been reconnected.	Spindle Override knob is operative.	
428	MPG Offline	MPG failure or loose cable.	MPG is inoperative.	By reconnecting MPG cable and appearance of next message.
429	MPG Online	Loose MPG cable has been reconnected.	MPG is operative.	
430	CPU7 PIC Offline	Power supply or hardware problem.	Power down, and then power up the system. The error should disappear.	
431	CPU7 PIC Online	CPU7 is back on line.	None	
432	External PLC Offline	Koyo PLC Direct failure or loose cable.	None	When PLC failure removed or cable reconnected.
433	External PLC Online	PLC failure corrected.	None	
434	_ idling too high: Releasing power	Axis is not moving and no job is running but axis has stopped against some abnormal resistance.	Power to motors is released.	Run autotune to adjust motor settings. Start of new job
435	_ axis runaway: Check motor wiring	Axis was moving more than 300 RPM while power was supposed to be off. 1.) Motor may be wired backwards. 2.) May be a shorted servo drive. 3.) Axis motion is canceled but motor continues to move due to inertia, which is probably caused by an unbalanced axis.	Power to motors is released.	Start of new job (after wiring or servo drive failure has been removed).

Error	Message	Cause	Effect	Removed
436	Servo drive shutdown	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 CNC10 software via the servo drive fault input to the PLC.	Can be resolved by observing the servo drive LEDs. Also check fibers 4&5. On AC systems check Parameter 178	This message is removed and the condition is reset only if the ESTOP is pressed and released. The PLC program that is responsible for latching this condition is also responsible for clearing this condition.
437	Servo power removed (Also see error 435)	Axis was moving more than 300 RPM while power was supposed to be off. 1.) Motor may be wired backwards. 2.) May be a shorted servo drive. 3.) Axis motion is canceled but motor continues to move due to inertia, which is probably caused by an unbalanced axis.	Power to motors is released.	Start of new job (after wiring or servo drive failure has been removed).
438	Axis cannot keep up with spindle	During a slaved move the axis can not keep up to the spindle speed (i.e. rigid tapping) 1.) Check parameter 34 for wrong sign in front of encoder counts. 2.) Need to slow down spindle RPM's.	Job canceled	Start of new job (after parameter 34 is corrected or the programmed spindle RPM is lowered).
439	_ axis servo drive data output error	Logic power failure or lost of communication from the drive to the CPU10.	No motor power	Is logic LED on? Check fiber optic cables to drive. For SD1 drives, make sure bus cables are shielded and are as short as possible. Power unit down and check drive connections.
441	_ axis overvoltage	Input power has gone higher than 340VDC and will shutdown the drive and remove power. The motor brake will engage for 5 seconds in this condition.	No motor power.	Check input voltage is below 340VDC. If not, incoming VAC needs lowered.
442	_ axis undervoltage	Drive input power is less than 80 VDC.	No motor power.	Check supply voltage.
443	_ axis commutation encoder bad	Control detected invalid commutation zone value.	No motor power.	Perform a motor Move Sync in the Drive menu. A Zero (0) or Seven (7) is an invalid zone. Check for: a.) Wiring problem in the encoder cable or motor end cap (broken encoder wires). b.) Encoder cable shield connected at motor end, when it shouldn't be. c.) Bad encoder. d.) Motor power cable shields not connected. e.) Drive not grounded properly.

Error	Message	Cause	Effect	Removed
444	_ axis overtemperature detected	Drive overtemp sensor tripped.	No motor power.	The drive is being run at over capacity or the cooling fan is either not functioning or its air flow is blocked.
445	_ axis overcurrent detected	Overcurrent detected on an axis.	No motor power.	Try to jog the axis. The drive will reset the current limit and try to move the motor. If the error comes back, check for a short in the motor output.
446	_ axis servo drive data input failure	Communication CheckSum error.	No motor power.	Check fiber optic cables. Verify continuity between drive chassis, ground strip and Earth ground.
447	_ axis (#) bad index pulse detected	Noise picked up by encoder cable or misaligned encoder.	No motor power.	Remove noise or align the encoder.

CNC syntax errors

Error	Message	Cause	Effect	Removed
501	Invalid character on line NNNNN	Invalid character on CNC line.	Job canceled	Start of new job.
502	Invalid G code on line NNNNN	Invalid G code encountered on CNC line.	Job canceled	Start of new job.
503	Invalid M function on line NNNNN	Invalid M function encountered on CNC line.	Job canceled	Start of new job.
504	Invalid parameter on line NNNNN	Invalid or missing number after letter.	Job canceled	Start of new job.
505	Invalid value on line NNNNN	Value out of range (T, H, D).	Job canceled	Start of new job.
506	Only 1 M code per line	More than one M code appears on the line.	Job canceled	Start of new job.
507	No closing quote	The closing quotation mark (“) is missing.	Job canceled	Start of new job.
508	Macro nesting too deep	Macro nesting limit exceeded on attempt to invoke a subroutine.	Job canceled	Start of new job.
509	Option not available	Attempt to access a locked software option.	Job canceled	Start of new job.
510	Too many macro arg’s	Too many arguments were given in a G65 macro.	Job canceled	Start of new job.
511	Missing parameter	A parameter is required or expected but not found.	Job canceled	Start of new job.
513	Expected **“=”**	Error in expression to left of “=”, missing “=”, or orphaned parameter.	Job canceled	Start of new job.
514	Empty expression	The expression contains no operands.	Job canceled	Start of new job.
515	Syntax error in expression	Illegal character in number, variable or function.	Job canceled	Start of new job.
516	Unmatched bracket (parenthesis)	Brackets or parentheses are paired improperly or misplaced.	Job canceled	Start of new job.
517	Evaluation stack overflow	Brackets or parentheses are nested too deeply.	Job canceled	Start of new job.
518	Undefined variable	The variable name does not exist.	Job canceled	Start of new job.
519	Too many variables	The space allotted for user-defined variables has been exceeded.	Job canceled	Start of new job.
520	Invalid variable name	The variable name contains an illegal character.	Job canceled	Start of new job.
521	Divide by zero	Attempt to divide by zero.	Job canceled	Start of new job.
522	Domain error	Imaginary number would result (square root of a negative number).	Job canceled	Start of new job.
523	Invalid value in assignment	Attempt to assign an illegal value to a system variable.	Job canceled	Start of new job.
524	Variable is read-only	Attempt to assign a value to a read-only system variable	Job canceled	Start of new job.

Cutter compensation errors

Error	Message	Cause	Effect	Removed
601	Error: no compensation in MDI	G41 or G42 entered in MDI.	MDI is not canceled, but cutter compensation does NOT go into effect. Remainder of line processed.	
602	Arc as first comp. move on line NNNNN	Cutter compensation started with arc as first move.	Job canceled	Start of new job.
603	Arc as first uncomp. move on line NNNNN	Arc specified as first move after end of compensation (G40).	Job canceled	Start of new job.
604	Plane must be XY on line NNNNN	Cutter compensation started with YZ or ZX plane selected.	Job canceled	Start of new job.
605	Canned cycle not allowed on line NNNNN	Canned cycle attempted during compensation.	Job canceled	Start of new job.
606	G53 not allowed on line NNNNN	G53 attempted during compensation.	Job canceled	Start of new job.
607	Set home not allowed on line NNNNN	M26 attempted during compensation.	Job canceled	Start of new job.
608	Ref. point move not allowed on line NNNNN	G28, G29, or G30 attempted during compensation.	Job canceled	Start of new job.
609	File read error on look ahead	Error reading file used for cutter comp look ahead.	Job canceled	Start of new job.

Parameter setting errors

Error	Message	Cause	Effect	Removed
701	G10 error: no R-value on line NNNNN	G10 used with no R-value.	Job canceled	Start of new job.
702	G10 error: invalid D on line NNNNN	G10 D0 Rxx specified.	Job canceled (D0 cannot be set; it is always zero).	Start of new job.
703	G10 error: invalid H on line NNNNN	G10 H0 Rxx specified.	Job canceled (H0 cannot be set; it is always zero).	Start of new job.
704	G10 error: invalid P on line NNNNN	G10 used with unknown P value.	Job canceled	Start of new job.
705	G10 error: No D, H, or P on line NNNNN	G10 used without D, H, or P to assign value.	Job canceled	Start of new job.

Canned cycle errors

Error	Message	Cause	Effect	Removed
801	Error: No R point on line NNNNN	No R-value specified.	Job canceled	Start of new job.
802	Error: Q = 0 on line NNNNN	Q value of 0 specified (Q used for G73 and G83 only).	Job canceled	Start of new job.
803	Error: No Z point on line NNNNN	No Z value specified for canned cycle.	Job canceled	Start of new job.
804	Error: Ggg invalid on line NNNNN (gg = 76, 86, 87, 88)	Unimplemented canned cycle requested.	Job canceled	Start of new job.
805	Error: No Q value on line NNNNN	Q value not specified for G73 or G83.	Job canceled	Start of new job.
806	Error: No P value on line NNNNN	P value (dwell time) not specified for G82 or G89.	Job canceled	Start of new job.

Miscellaneous errors

Error	Message	Cause	Effect	Removed
901	Ref. point invalid on line NNNNN	G30 with invalid P value (must be 1 or 2).	Job canceled	Start of new job.
902	No prior G28 or G30 on line NNNNN	G29 with no preceding G28 or G30.		
903	Warning: No coordinates for G92 on line NNNNN	G92 with no axis coordinates to set.	Remainder of line processed; job continues.	When next message appears.
904	Invalid plane for arc on line NNNNN	I, J, or K specified with wrong plane (e.g. K with G17, or I with G19).	Job canceled	Start of new job.
905	Warning: 0 radius arc on line NNNNN	Arc move was specified with a zero radius.	Move is done as a linear move; job continues.	When next message appears.
906	Warning: unknown arc on line NNNNN	Position of arc move could not be determined from parameters (e.g. G91 G2 X0 Y0 R1).	Move is done as a linear move; job continues.	When next message appears.
907	_ axis travel exceeded on line NNNNN	Software travel limit would be exceeded by the requested move.	Job canceled	Start of new job.
908	Option not available on line NNNNN	A code for an extra-cost option was specified, but the option has not been licensed.	Job canceled	Start of new job.
909	Program too long: job canceled	Attempt to run a job over 1MB in length, without the unlimited program size option.	Job canceled	Start of new job.
910	No subroutines in MDI	Specified O9100 - O9999 in MDI, which would begin an embedded subprogram.	MDI canceled	Start of new job.
911	Illegal recursion	Attempt to execute a subprogram or macro that calls itself, either directly or indirectly.	Job canceled	Start of new job.
912	Too many subprogram calls	Attempt to run a job with 20 or more levels of subprogram nesting.	Job canceled	Start of new job.
913	Could not open file *filename.ext*	Attempt to call a subprogram or macro, but the subprogram file does not exist.	Job canceled	Start of new job.
914	Tool library invalid for T*nn*	Enhanced ATC is enabled and the tool library does not have a valid bin number assigned.	Job canceled	Start of new job.

Scaling/Mirroring errors

Error	Message	Cause	Effect	Removed
1001	Invalid scaling parameter on line NNNNN	Invalid parameter specified (I, J, K, P).	Job canceled	Start of new job.
1002	Invalid scaling center on line NNNNN	Invalid parameter specified (X, Y, Z).	Job canceled	Start of new job.
1003	G code not allowed when scaling on line NNNNN	G28/G29/G30/G92 is not allowed when scaling or mirroring is turned on.	Job canceled	Start of new job.
1004	Turn scaling off before rescaling	Tried to rescale while scaling is turned on.	Job canceled	Start of new job.
1005	Cannot scale arcs with different scale factors	Scaling factors of the arc axes are different.	Job canceled	Start of new job.
1100-1199	Custom messages defined in *cnc10xmsg.txt*. Please contact your dealer if you have any questions regarding a particular message.			