CENTROID

CNC4

Operator's Manual

Version ___

REV 920512

Table of Contents

INTRODUCTION

The CENTROID CNC4 motion controller provides 2 axis circular and 3 axis linear moves using industry standard CNC codes. The CNC4 is capable of supplying up to 8 amps per motor phase at 50 VDC to large frame stepper motors. Motors of this size are suitable for moving virtually any milling machine table under full CNC control.

Full micro-stepping (1600 step/rev) is used to eliminate the resonances and vibration normally associated with stepper motors. The result is a low cost CNC controller which provides excellent positioning and repeatability characteristics.

The CNC4 is suitable for use with almost any milling machine, however it is highly recommended that you install a set of re-circulating ball screws on all three axes. If the machine is worn appreciably, it is recommended that you have the mill over-hauled. Positioning accuracy and repeatability is almost entirely dependent upon the mechanical performance of your mill. On a typical knee mill with ball screws (Bridgeport Series 1 or equivalent), repeatability to plus or minus .001 to .0015" is typical. On a larger and more rigid mill (bed type mill with square ways and good ball screws) repeatability to plus or minus .0005" or less is practical.

SPECIFICATIONS

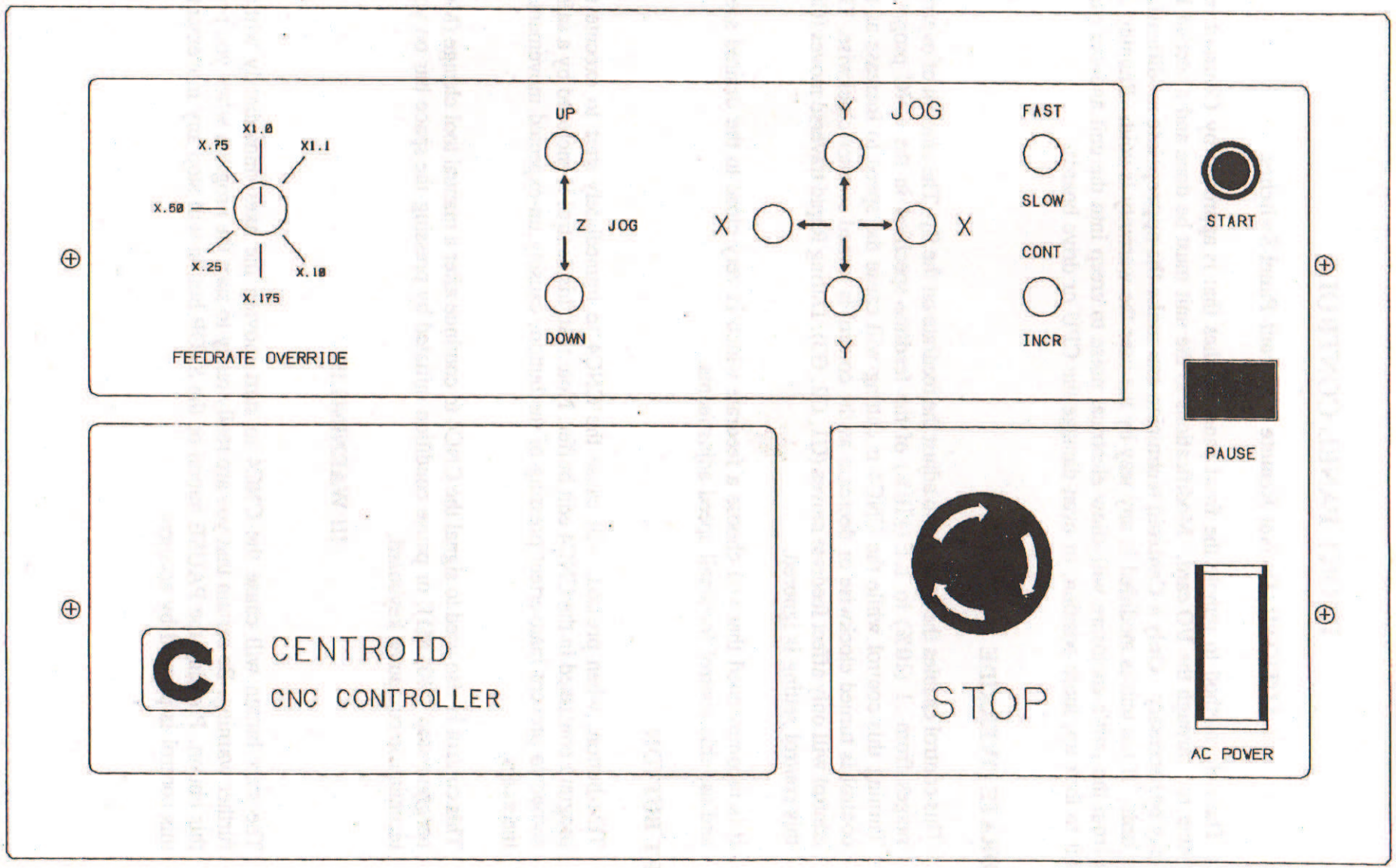
Drive motor	-DC stepping motor 4, 6, or 8 lead
Drive type	-open loop, or closed loop 50 VDC bi-polar micro-stepping chopper-drives, 8 amps/phase max, 8 micro steps per full step (1600 microsteps per revolution on a 1.8 degree stepping motor)
Interpolation	-2 axis circular (XY, XZ, YZ), 3 axis linear (XYZ)
Positioning mode	-incremental (G91) or absolute (G90)
Units of measure	-inch or metric (G70, G71)
Programming Standard	- EIA RS-274 (G codes)
Programming macros	-up to 99 macros; loops

Canned cycles	-13 canned cycles (G77-G89) drilling, boring, pockets, facing, bolt-hole circles
Communications port	-RS-232 at up to 19,200 baud
Programming input	-external ASCII terminal (dumb terminal) or PC emulating a dumb terminal
Internal program storage capability	-24,575 ASCII characters (equivalent to 200' of punched tape)
Program and set-up data back-up	-lithium cell, 5-year life
Front panel user fast controls	-Jog buttons X, Y, and Z in the plus or minus direction, slow or and incremental or continuous jog, pause/resume, feed-rate override, program start/resume, Emergency kill button
I/O capability	-8 M-function relay driver outputs, 8 programmable switch inputs (limits, sense inputs) standard. Optional 2nd I/O board with #0-24 BCD with strobe, outputs, 8 additional switch inputs, suitable for driving a tool-changer or other device with a BCD interface.
AC power requirements	-120 VAC, 800 watts
Enclosure dimensions	-17"W, 10 3/8"H, 15"D, 40 lbs

The following performance specifications are based on a .200" pitch (5 turn per inch) ball screw with zero machine table lash or flex, CENTROID 1350 oz/in or 2130 oz/in motors:

Minimum programmable increment	.000125"
* Repeatability	+/-.000250" to .000500"
Rapid traverse speed single axis	120 IPM
Maximum linear milling rate (G1)	120 IPM
Maximum Circular milling rate (G2, G3) radius > 3"	120 IPM
Maximum Circular milling rate (G2, G3) radius < .125	24 IPM
Average circular milling rate (G2, G3)	48 IPM

* Actual results will vary according to machine rigidity and amount of machine lash.


X1.0
X.75
X1.1
X.50
X.25
X.10
X.175
FEEDRATE OVERRIDE
UP
Z JOG
DOWN
Y JOG
X
X
Y
FAST
SLOW
CONT
INCR
START
PAUSE
STOP
AC POWER
CENTROID
CNC CONTROLLER

FRONT PANEL CONTROLS

CAUTION!! Do Not Remote the Front Panel Switches.

The only method to remote the front panel swithes that is approved by Centroid is to have them run through the I/O card. Modifications to the unit must be done and a second I/O card may be necessary. **Only a Centroid technician can make the appropriate modifications to the unit.** If the unit is modified in any way by the user the warranty is void. Running any wires from the unit's enclosure will allow electrical noise to creep into the unit and can cause the unit to lock up, loose position, or even damage the CPU or drive boards.

FEEDRATE OVERRIDE

This control enables the operator to adjust the feedrate on the fly. The amount of override ranges from .1 (10%) to 1.1 (110%) of the feedrate specified in the CNC program. Turning this control while the CNC4 is cutting will cause the speed to increase as the control is turned clockwise or decrease as the control is turned counter-clockwise. This control will only affect feedrate moves (G1, G2, G3). During Rapid traverse moves (G0), this control setting is ignored.

It is recommended that you choose a feedrate which is very close to the desired speed and use this control for small speed adjustments.

START BUTTON

This button, when pressed, will cause the CNC4 to immediately start to execute the program contained in the CNC4 edit buffer. Note that this control is enclosed by a safety shroud to prevent inadvertent pressing of the button, causing un-expected movement of the axes.

This control is also used to signal the CNC4 to continue after a manual tool change (M6), program stop (M0, M1), or pause condition initiated by pressing the space bar on your terminal or computer keyboard.

!!! WARNING !!!

The start button will cause the CNC4 to start moving the axes immediately without further warning. Be certain that you are really ready to start the program when you press this button. Pressing the PAUSE button or the STOP button will stop any movement if this control is pressed by accident.

PAUSE

When the Pause button is pressed, all axis movement will decelerate to a controlled stop and the following prompt will appear on the terminal or computer screen indicating that the CNC4 is paused:

PAUSE Pushed, Release PAUSE to Continue, ESCape to Cancel.

The axes will remain paused until the control is pressed again. When the Pause button is released the axes will accelerate and normal program execution will resume. If for any reason you wish to stop program execution altogether, press the PAUSE button. When the axes have stopped and the screen prompt appears, pressing <CONTROL C> or <CONTROL X> will cause the Control Menu to appear on the screen.

Note that only pressing the PAUSE button will not turn off devices connected to the M-function relay outputs. For example, if the Spindle motor is connected to the relay outputs and was turned on with an M3 in the CNC program, it will continue to run (while the program is paused) until:

1) The Space Bar or the Start Button is pushed. The program then resumes and the spindle will turn off when a M5 is encountered in the CNC code.

2) The Control C keystroke sequence is entered on the keyboard. If the unit is Paused, and Control C is pressed, all relays will be forced open and the spindle will stop.

3) M5 is issued from the Run Immediate Mode.

In addition to the front panel pause control, the spacebar on your terminal or computer keyboard will also cause the CNC4 to pause. Pressing the space bar again (or the front panel START button) will cause the CNC4 to resume program execution.

JOG BUTTONS

There are six momentary contact push buttons on the CNC4 front panel which enable the operator to move the axes by simply pressing one of the buttons.

There are two buttons for each axis. The labeling on the CNC4 front panel shows the axis, and direction of movement for each jog button. Only one axis can be jogged at a time. If + and - jog buttons move the axis in the wrong directions, change the direction inversion in the Setup Menu choice 7.

In addition to the six jog buttons, There are two toggle switches marked SLOW/FAST and CONT/INCR. These switches determine whether the jog speed will be slow (4 IPM) or fast (120 IPM), and how far the axis will move when a button is pushed.

SLOW/FAST

When this switch is in the FAST position and a jog button is pressed, the axis will move at the rapid traverse (G0) rate. If the switch is in the slow position, the axis will move at the slow jog feedrate.

It is important to note here that during a SLOW jog, all limit switches will be ignored. This means that if you happen to trip a limit switch, you can use SLOW jog to move the axis off the limit switch. Be sure you are jogging in the correct direction to get off of the switch. Remember to reestablish axes zeros before continuing.

CONT/INCR

When this switch is in the CONT position and a jog button is pressed, the axis will move continuously until the button is released. When the switch is in the INCR. position, the axis will move a pre-defined distance and then stop. The jog button must be released and then depressed again before any further axis movement can occur.

NOTE: The jog buttons will not operate if the PAUSE button is actuated.

STOP

This control is used as an emergency stop or "PANIC" stop. If for any reason you want to immediately cause all axes motion to stop, pressing this mushroom button in will kill AC power to the CNC4. All axes motion will stop, and all M-Function relays will open. This will cause the spindle motor to stop provided that the spindle motor is interfaced to the M3 and M4 relay outputs. Remember to reestablish axes zeros before continuing.

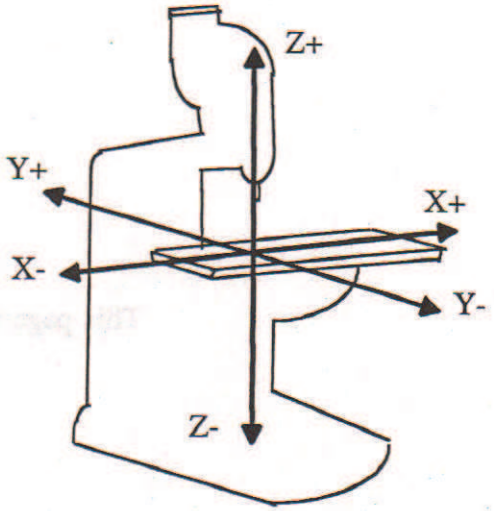
AC POWER

This control is a combination AC power switch and circuit breaker. When the power is turned on, this rocker switch will light up. If the breaker trips for any reason, turn the control to the OFF position and wait for a few minutes before turning the power on again. If the breaker trips again, then un-plug the CNC4 from its AC power source and contact CENTROID for further instructions.

AXIS CONVENTION

All CNC program code and software uses the standard cartesian coordinate system. If you are facing the mill the X axis is defined positive to your right; the Y axis is defined positive into the mill and the Z axis is defined positive upward, perpendicular to the XY plane.

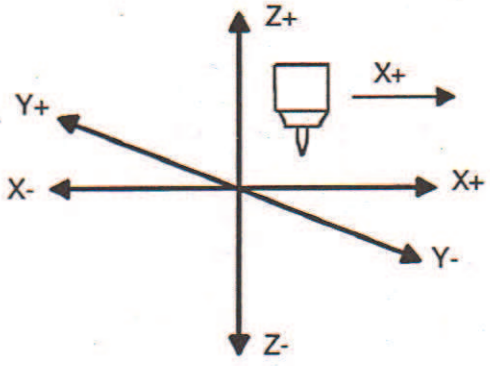
Please note, the direction of motion is defined by the CUTTER motion, not by the TABLE movement.



ATTENTION: Arrow direction shows cutter motion, not table movement!!!

G90
G0 X0 Y0
G1 X4.
Says to rapid traverse using absolute positioning to the point X = 0, Y = 0; then linear mill to the point X = 4.

The cutter will move 4 units to the right.



KEYSTROKE CONVENTIONS

The following responses to an "OPTION NUMBER?" prompt will allow the operator to choose between operating modes, or to choose between different functions within an operational mode:

CARRIAGE RETURN <ENTER>

Pressing the key marked <ENTER> or RETURN will re-display the present operational menu for operator inspection. This response is useful because the menus are not always automatically redisplayed after the completion of each processing option. The operator can always use this response like a help command to inspect all options available. If a letter or numerical value is typed in at a prompt, it must also be terminated by a carriage return. Throughout the rest of this manual, the symbol <ENTER> will indicate that the key marked ENTER or RETURN is to be pressed.

CONTROL C <CTRL C>

Control-C (pressing and holding the Control key depressed and then pressing the "C" key) will cancel the function you are currently utilizing. This is useful when you enter incorrect information at a prompt and have already pressed the <ENTER> key. Control C will enable you to quickly break out of a function sequence so that you can re-start that function and correct errors. Throughout the rest of this manual, the symbol <CTRL C> will indicate that the CONTROL key and C key are to be pressed simultaneously.

CONTROL X <CTRL X>

Control-X (pressing and holding the Control key depressed and then pressing "X" key) will cancel the function you are utilizing and cause the Control Menu to appear on the screen. This feature is a convenient way to quickly change a parameter and return to the Control Menu. Note that this response at any screen prompt (any text?_ followed by a question mark and blinking cursor) will cause the Control Menu to appear on the CRT screen. Throughout the rest of this manual, the symbol <CTRL X> will indicate that the CONTROL key and the X key are to be pressed simultaneously.

ESCAPE <ESC>

The ESCAPE code will cause the CNC4 to act in the same manner as a CONTROL C. Some communications software and/or CRT terminals will not allow you to transmit an ESCAPE. For those software - hardware combinations which do allow it, the ESCAPE allows a single keystroke in place of the CONTROL C sequence. Throughout the rest of this manual, the symbol <ESC> will indicate that the ESCAPE key is to be pressed.

CONTROL MENU OPTIONS

When the CNC4 is turned on and the terminal is operating correctly ("Dumb Terminal" or PC with commincations software) the following menu and operator prompt should be displayed on the terminal or computer screen:

CONTROL MENU

1 - Run/Auto
2 - Run/Block
3 - Run/Add
4 - Run/Immediate
5 - Run/Search
6 - Setup
7 - Edit
8 - Release
9 - Motor Test

X: 0.0000 Y: 0.0000 Z: 0.0000 OPTION NUMBER?_

It is from this menu that the three main operational modes of the CNC4 are chosen. A menu option is chosen by entering the menu choice number at the "OPTION NUMBER?" prompt and then depressing the <ENTER> key on the CRT or computer keyboard. The three main operational modes of the CNC4 are:

Run modes

Choosing one of the run options causes the CNC4 to execute the program contained in its edit Options buffer. The axes will move in a manner determined by the CNC codes and coordinates contained within the CNC program.

Setup mode

It is in this mode that the CNC4 is given information about the stepping motors and the milling machine to which they are attached. The CNC4 needs to know the details of your motors and milling machine so that it knows how far and how fast to move the stepper motors to achieve the programmed position and feedrate. Tool parameters (diameter, length, tool number), and certain operating characteristics of the CNC4 are also changed in this mode.

Edit mode

This is the operating mode of the CNC4 in which the CNC program is created and/or modified directly from the CRT terminal keyboard. This is also the operating mode chosen when a CNC program is to be uploaded to, or downloaded from a host computer.

RUN MODE OPTIONS

1 - RUN/AUTO

Run the entire CNC program contained in the on-board edit buffer. This option loads the initial conditions established in the setup mode, and outputs the following prompt to the CRT screen:

X: 0.0010 Y: 0.0010 Z: 0.0000 Hit START to Continue.

When the START button is depressed, program execution commences. As the program executes, status messages such as:

X: 2.1500 Y: 3.6250 Z: 1.0000 F: 14.0000 N:10 Line: 5

are output before each block (line). Other messages are also displayed as required. If a tool change is programmed or a designated switch must be depressed, a message relating this condition is output to the display when the event occurs. If operator visible comments are contained in the CNC program or block output is turned on from the setup menu, these outputs are displayed for operator inspection.

REMEMBER

Pressing the front panel START button immediately causes the CNC program in the edit buffer to execute. The axes may start to move immediately.

2 - RUN/BLOCK

Run the CNC program, one block (line) at a time. This option loads the initial conditions established in the setup mode. Before each block (line) is executed the following prompt is output:

X: 0.0010 Y: 0.0010 Z: 0.0000 Hit START to Continue.

When the Start button is depressed, the single block (line) is executed. As each block (line) is executed and after each block (line) is completed, status messages such as:

X: 0.0010 Y: 0.0010 Z: 0.0000 F: 4.0000 N: Line: 5

are output. Other messages are also displayed as required. If a tool change is programmed or a designated switch must be depressed, a message relating this condition appears on the display when the event occurs. If operator visible comments are contained in the CNC program or block output is turned on from the setup menu, these outputs are displayed for operator inspection.

3 - RUN/ADD

Runs the entire CNC program as in option 1 and, upon completion, prompts the operator for any additional CNC lines to be executed.

This option loads the initial conditions established in the setup mode and displays the prompt:

X: 0.0010 Y: 0.0010 Z: 0.0000 Hit START to Continue.

When the START button is depressed, program execution commences. As the program executes, status messages such as those described with option 1, Run/Auto, are output. When the program is completed, a message showing the final position is output.

Absolute Position X: 7.3420 Y: -56.0000 Z: 0.0000

The following prompt is then issued for additional CNC blocks (lines):

X: 0.0000 Y: 0.0000 Z: 0.0000 Block?

As the operator enters each line from the keyboard, it is executed immediately. At the completion of each block execution, the absolute position message is redisplayed. When no more CNC blocks are to be entered, the operator depresses carriage return with no additional input. This terminates option 3 and re-displays the Control Menu.

Note that this option is used for executing macros directly from the keyboard. The cutter compensation G codes (G41 and G42) and loop requests (e.g. =N100/5) are invalid responses to the Run/Add "Block?" prompt.

4 - RUN/IMMEDIATE (MDI)

This option loads the initial conditions established in the setup mode and outputs message showing the position of the X, Y and Z axes.

Absolute Position X: 7.3420 Y: -5.6123 Z: 0.0000

The following prompt is then issued for the entry of CNC blocks (lines):

X: 0.0000 Y: 0.0000 Z: 0.0000 Block?

As each line is typed in and terminated by pressing <ENTER>, it is executed immediately. At the completion of each block execution, the absolute position message is redisplayed, along with the BLOCK? prompt. When no more CNC blocks are to be entered, the operator depresses <ENTER> without other input. This terminates option 4 and re-displays the Control Menu.

Note that the Run Immediate mode is used for positioning of the axes and activating M-Functions directly from the keyboard. The cutter compensation G codes (G41 and G42), loop requests (e.g. =N100/5) and macro calls (e.g. =#3) are all invalid responses to the Run/Immediate "Block?" prompt.

5 - RUN/SEARCH

Search for a specific program block (line) number. This option loads the initial conditions established in the setup mode and outputs the prompt:

X: 0.0000 Y: 0.0000 Z: 0.0000 Number to Search for?

When the block (line) number (eg. 150 for block N150) is entered, a second prompt is output:

X: 0.0010 Y: 0.0010 Z: 0.0000 Hit START to Continue.

When the Start button is depressed, the program is searched block (line) by block for the specific number. As the CNC4 searches through the program line by line, each block (line) preceded by a "." is executed; all others are ignored. When the specified number is located, the following message is output:

X: 2.5000 Y: 4.5650 Z: 1.5000 F: 24.0000 N: 11 Line: 11

X: 0.0000 Y: 0.0000 Z: 0.0000 Search Done: Hit START to Continue.

When the START button is depressed, the program continues just as with the Run/Auto option.

This option can be extremely useful for continuing programs which use absolute positioning (G90) and were interrupted in the middle of executing; however, the use of the Run Search mode must be planned out so that you do not get unexpected results. Remember to always insert a "." before the necessary blocks to establish the proper operating modes (Rapid or feedrate traverse, feedrate, etc.) needed to continue milling the work-piece at the desired start point.

It may be necessary to insert some "dummy" blocks just before the desired starting point in your program. Not only will this method allow you to set the desired operating modes, it will also allow you to position the cutter in preparation for starting at the previous stop point.

The RUN/SEARCH mode can also be used to determine if there are any invalid characters contained in the CNC program. This method will not check for incorrect coordinates, but it will catch errors such as GO (G letter O) instead of G0 (G zero). If

the CNC4 cannot make sense of the line as it is typed, then it will display the line number and warn you of a program error.

To check a program, simply tell the CNC4 that you want to search for a line number that does not exist. The CNC4 will read through the complete program looking for the line number you entered. As it searches, it will display the line number on the screen of any invalid lines it encounters. This may save you a lot of frustration and scrapped parts. See page 42 for an explanation of CNC editor error messages.

6 - SETUP

Transfer to the setup mode. Establish initial program operating conditions, tool parameters, and axis motor specifications. When this option is chosen a Setup Menu is output. See page 13 of this manual for specific information concerning the setup mode.

7 - EDIT

Enter the EDIT mode. This mode is used for CNC program entry, alteration or inspection. When this option is chosen, an editor menu is output. See page 27 of this manual for specific information concerning the edit mode.

8 - RELEASE

Release axis motor power. (Note that this may also be accomplished under CNC program control using the M93 command. When this option is selected the prompt:

X: 0.0996 Y: 0.0996 Z: 0.0995 X, Y, Z Axis or All (X, Y, Z or A)?

is issued. Power is released from the axis motor(s) selected.

9 - MOTOR TEST

Test the motor parameter settings and motor hookups. This option starts all three axis motors moving back and forth immediately, for a distance of 1 inch, at a feedrate of 4 inches/minute. This back and forth motion continues until Hold or STOP are detected. The STOP switch and any limit switches must be closed for this test to run.

SETUP MODE - PARAMETER ENTRY AND AXIS POSITIONING

When the Setup mode is entered (option 6 from the Control Menu), the following menu and input prompt is output to the CRT display:

SETUP MENU

1 - Display Setup Parameters
2 - Display Tool/Motor Parameters
3 - Positioning/Units/Tool Changer
4 - Delete/Stop/Menu/Block/LF Settings
5 - Machine Specific Parameters
6 - Tool Parameters
7 - Motor Parameters
8 - Zero Axis Position
9 - Move Axis
10 - Go To Axis Zero
11 - Go To Axis Limit
12 - Set Z Axis Home
13 - Control Menu

X: 0.0000 Y: 0.0000 Z: 0.0000 OPTION NUMBER?

The menu choices in the setup mode allow the operator to configure the CNC4 for operation with the mill and the stepper motors.

DISPLAY SETUP PARAMETERS - OPTION 1

Display the current setup parameters. This option shows the current machine and controller setups. Option 1, in conjunction with Setup Menu options 3,4, and 5 allows the operator to see and change any of the setup parameters listed below:

SETUP PARAMETERS

Absolute Positioning	*Delete Mode*	*Off*	*Jog Increment*	*0.0500*
Inch Units of Measure	*Optional Stop*	*Off*	*Feedrate as 1.0 Units/Min*	
Manual Tool Changer	*Menu Displays*	*On*	*Transmission Delay*	*0*
	Block Output	*Off*	*Spindle BCD Divisor*	*0*
Z Axis Home 0.0000	*LF Suppression*	*Off*	*Spindle M3/M4 Delay*	*0*

X: 0.0000 Y: 0.0000 Z: 0.0000 OPTION NUMBER?_

DISPLAY TOOL/MOTOR PARAMETERS - OPTION 2

Setup Menu option 2 will display the current tool parameters for all 24 tools as well as the stepping motor and lead screw parameters. Setup Menu options 6 and 7 allows the operator to change these parameter values. A tool/motor parameter display, showing all of the default settings, follows:

TOOL/MOTOR PARAMETERS

Tool	*Length*	*Diameter*	*Tool*	*Length*	*Diameter*	*Tool*	*Length*	*Diameter*
1	*0.0000*	*0.0000*	*2*	*0.0000*	*0.0000*	*3*	*0.0000*	*0.0000*
4	*0.0000*	*0.0000*	*5*	*0.0000*	*0.0000*	*6*	*0.0000*	*0.0000*
7	*0.0000*	*0.0000*	*8*	*0.0000*	*0.0000*	*9*	*0.0000*	*0.0000*
10	*0.0000*	*0.0000*	*11*	*0.0000*	*0.0000*	*12*	*0.0000*	*0.0000*
13	*0.0000*	*0.0000*	*14*	*0.0000*	*0.0000*	*15*	*0.0000*	*0.0000*
16	*0.0000*	*0.0000*	*17*	*0.0000*	*0.0000*	*18*	*0.0000*	*0.0000*
19	*0.0000*	*0.0000*	*20*	*0.0000*	*0.0000*	*21*	*0.0000*	*0.0000*
22	*0.0000*	*0.0000*	*23*	*0.0000*	*0.0000*	*24*	*0.0000*	*0.0000*

Axis	*Turns /Unit*	*Steps /Turn*	*Slow Jog*	*Max Rate*	*Lash Comp*	*Limits - +*	*Dir Rev*	*Mtr Rev*	*Closed Loop*	*Pulse /Turn*
X	*5.000*	*1600*	*800*	*16000*	*0.0000*		*On*	*Off*	*Off*	
Y	*5.000*	*1600*	*800*	*16000*	*0.0000*		*On*	*Off*	*Off*	
Z	*20.000*	*1600*	*800*	*6400*	*0.0000*		*Off*	*Off*	*Off*	

X: 0.0000 Y: 0.0000 Z: 0.0000 OPTION NUMBER?

The values shown above are the factory default settings in the CNC4 when shipped. The operator must type in the correct parameters so that the CNC4 can store them in memory. Once the correct parameters are entered, the CNC4 will retain the values in memory while the CNC4 power switch is turned off.

These parameters will vary depending upon the lead screw pitch, type of stepper motor used, and size of the mill. Once the correct tool and motor parameters have been established and entered, use the sheet at the beginning of the CNC4 manual to record these values.

POSITIONING/UNITS OF MEASURE - OPTION 3

Choosing Setup Menu option 3 allows the operator to change the parameters in the left hand column of the Machine setup parameter display shown above. When Setup Menu option 3 is chosen, the following prompts will be output to the CRT display. If you do not wish to change the parameter that you are prompted for, simply press the <ENTER> key to continue on to the next prompt.

"Absolute or Incremental (A or I)?"
"Inch or Metric Units (I or M)?"
"Manual or Auto Tool Change (M or A)?"

Option	Description
Positioning	Absolute or Incremental positioning method used for axis motor moves. (default = absolute)
Units of measure	Inch or metric (mm) units of measure used for axis motor moves. (default = inches)
Tool Changer	Used with the optional BCD tool changer I/O board. When set to manual, operator must manually change tools and signal CNC4 by pressing <ENTER> or START button when tool is changed. (default = manual)
	When set to Automatic, tool number is output to BCD I/O board. Tool changer will signal CNC4 that tool change is complete and CNC4 will continue. Refer to pages 127 through 132 for specific details covering BCD tool-changer hookup.

DELETE/STOP/MENU/BLOCK SETTINGS - OPTION 4

Setup Menu option 4 allows the operator to change the parameters shown in the middle column of the machine Setup Parameters shown above. If you do not wish to change a parameter value when prompted to do so, simply press <ENTER> to continue on to the next prompt.

"Delete Mode (OFF or ON)?"
"Optional Stop (OFF or ON)?"
"Menu Displays (OFF or ON)?"
"Block Output (OFF or ON)?"
"Line Feed Suppression (OFF or ON)?"

Option	Description
Delete Mode	Delete mode affects blocks (lines) preceded by a "/" when a CNC program is run. If delete mode is turned on, blocks preceded by a "/" are ignored. If delete mode is turned off, blocks preceded by a "/" are executed. (default = off)
Optional Stop	Optional stop affects the M1 function when a CNC program is run. If optional stop is turned on when M1 is encountered, the program stops for operator inspection. If optional stop is turned off when an M1 is encountered, then the M1 is ignored. (default = off)

Menu Displays Menu display affects whether or not menus are output. If menu displays are turned on then the complete control, edit and setup menus are output as each menu is selected. If menu displays are turned off, only the headings "CONTROL MENU", "EDIT MENU" and "SETUP MENU" are output when a menu is selected. Note that in any case, carriage return in response to an "OPTION?" prompt always causes the complete menu to be displayed. (default = on)

Block Output Block output affects whether or not CNC program blocks (lines) are output when a CNC program is run. If block output is turned off then the blocks are not displayed. If block output is turned on then the CNC blocks are output to the display, as they are executed, during a run. (default = off)

Line Feed Suppression Line feed suppression causes the CNC4 to output only an ASCII carriage return at the end of each line instead of a carriage return/linefeed. This feature allows the CNC4 to communicate with CRT displays or computers which do not require the end of line Linefeed character. This parameter value should normally be off. (default = off)

MACHINE SPECIFIC PARAMETERS - OPTION 5

Establish the jog increment, feedrate multiplier, transmission delay and spindle parameters. After selecting this option, the following prompts (shown with typical replies) are output:

"Jog Increment?"
"Feedrate as 1 or .1 Units/Min?"
"Transmission Delay Count?"
"Spindle Speed to BCD Output Divisor?"
"Spindle M3/M4 Delay Count?"

Option	Description
Jog increment	The distance an axis will travel when the jog button is pushed once and released in the incremental mode. (default = .050")
Feedrate multiplier	The multiplying factor that feedrates are scaled by. There are two choices: 1 or .1. A F100 in the CNC program will either be run at 100 IPM (100 x 1) or 10 IPM (100 x.1). (default = 1)
Transmission Delay Count	Allows terminals or host computers, with internal delays, to communicate with the Centroid at the higher baud rates. The typical response is 3. Other common responses are 0 (no delay) and 7 (long delay). (default = 0)

Spindle Speed BCD Output Divisor

This parameter is for use with a BCD Divisor spindle speed changer. A value of 0 (zero) shuts off all spindle speed BCD output. This value must be 0 unless you have a BCD spindle speed changer connected to the CNC4. Refer to the manual section "Optional I/O Board used to Control BCD Interface" for more information. (default = 0)

Spindle M3/M4 Delay Count

The spindle M3/M4 delay count is a time delay that occurs before M3 or M4 is executed. The delay can be specified from 0 to 32000 where 0 corresponds to no delay and 32000 corresponds to a delay of 320 seconds.

For example, if the Spindle M3/M4 Delay count is entered as 200, then an M4 issued in the CNC program results in the output of an implicit M5 and a pause of 2 seconds before the M4 is issued.

TOOL PARAMETERS - OPTION 6

Establish the lengths and diameters of up to 24 tools. When this option is selected, the following prompts are issued:

"Tool Number?"
"Length?"
"Diameter?"

Tool information input may also be entered or altered from within a CNC program using the tool select, T symbol and the tool change, M6 function. See page 34 for information concerning this method of tool parameter entry.

All current tool parameters can be viewed by choosing option 2 from the Setup Menu. If tool parameters are entered using option 6 from the Setup Menu, use option 2 from the Setup Menu to verify correct parameter entry. In addition, If the tool parameters are entered within the CNC program or from the Run/Immediate mode with the T command, the entered values will override any previous values and can be viewed by choosing option 2 from the Setup Menu.

Option	Description
Tool Number	Tool number. From 1 to 24.
Tool length	This length is used while a CNC program is running to adjust the Z axis position, based on the tool inserted. For example, if the Z axis position is at 0.0 and a -1 inch tool is inserted, the Z axis position display is adjusted -1.0 inch so that the position of the tip of the tool will then position properly. This action will not cause the Z axis to offset immediately. (default = 0.0)

The offset will occur during the first Z axis move after the tool has been loaded with an M6.

It is recommended that all tool lengths be determined in the following manner: The longest tool being equal to a length of zero, all shorter tools be entered as a minus tool length. See explanation below.

Tool diameter This diameter is used while a CNC program is running to adjust the X and Y axis positions, when cutter compensation is turned on. If cutter compensation is turned on, then the milling head is moved half of the tool diameter (the tool radius) away from the milling surface. This parameter also allows the operator to compensate for tool wear. Do not use a tool diameter of zero when using cutter compensation. (default = 0.0)

DETERMINING TOOL LENGTHS

The following procedure can be used to determine tool lengths when using the CNC4 tool length compensation. This procedure will prevent the possibility of running a tool into the workpiece after a power failure, or after a program run.

General Approach

Z zero (Z=0.00) is at the top surface of the workpiece.

Z home is far enough above the workpiece so that the longest tool can be loaded.

Tool #1 (T1) is the longest tool. Use tapered tool holders or your favorite method to ensure that once a tool length is determined, all tools are reloaded to their exact previously determined length.

Setup and Tool Measuring Procedure

1. Select the Setup option (Option 6) from the Control menu

2. Jog the Z axis clear of the table and any fixture or vice to a point where you can load the longest tool. Insert the longest tool. This will be tool #1 (T1).

3. Jog the Z axis down so that the tip of the tool just touches the surface of the workpiece. Make this Z zero by selecting Setup Menu option 8. Reply Z to the X, Y, or Z AXIS? prompt. Alternatively, this Z position can be zeroed by issuing the command G92 Z0 during Run Immediate Mode.

4. Jog the Z axis back up to a point where you can get the longest tool out. Make this the Z axis Home position by selecting option 12 from the Setup Menu or issuing M26 when in the Run Immediate Mode.

5. Remove the longest tool and insert the next longest tool. This will be tool #2 (T2). Jog the Z axis down so that the tip of the tool just touches the surface of the workpiece.

6. Because this tool is shorter than tool #1, the Z axis position displayed on the screen will be negative. Record and use this negative (minus) position as the tool length for tool #2. Note that we have measured the difference in length between tool #1 (longest tool) and tool #2 (next longest).

7. Jog the Z axis back up so that you can remove the tool, and follow steps 5, and 6 to determine the length for all remaining tools (up to a maximum of 24), working down from the longest to the shortest tool.

8. The recorded negative (minus) tool lengths for each tool should be entered either at the Setup Menu option 6 (Tool parameters) or within the CNC program using the T command.

9. Remember to specify tool #1, the longest tool, as having zero length. This will prevent the possibility of damage to the tool and/or workpiece on a G0 Z0 command.

General Comments for Advanced Users

If it bothers you to touch the tool to the upper surface of the workpiece, you can touch off on a block of known thickness above the workpiece to prevent tool marks. You can also use an appropriate tool length measuring device to determine the differences in tool length between the longest tool and all subsequent shorter tools.

REMEMBER

Tool parameters for a given tool will not be recognized by the CNC4 until that tool is selected by a T and M6.

MOTOR PARAMETERS AND LIMIT SWITCHES - OPTION 7

Alter the characteristics of the Centroid CNC controller to reflect the characteristics of the axis motors. Allows for the selection of a user defined limit switch configuration and to select the direction of rotation of the motors. When this option is chosen the following prompts are output:

Prompt	Response, Description
X, Y, or Z axis?	Select the proper axis X, Y or Z.
Lead Screw Turns/Unit? (default = 5- X,Y 20- Z)	Enter the turns/unit (inch or mm) length of the axis lead screw, typically 5 or 10. Press <ENTER> to continue on without changing the default setting.

Motor Steps/Turn? (default = 1600)	Enter the number of micro-steps/revolution of the stepping motor, typically 1600 steps. Press <ENTER> to continue on without changing the existing setting.
Slow Jog (Steps/Sec)? (default = 800)	Enter the slow rate speed in microsteps/second, of the motor. This is the speed at which the motor will turn during a SLOW jog.
Max. rate (Steps/Sec)? (default = 16000)	Enter the maximum rate, in micro-steps/second, at which the motor turns. This is also the rapid traverse rate used for G0. Press <ENTER> to continue on without changing the default rate.
Lash Comp. Distance? (default = 0.0)	Enter the amount of lash present on the specified axis. Press <ENTER> to continue on without changing the lash default distance.
Limit Switch (- Direction)?	Enter the input switch number (1-8) hooked up for the minus direction of the axis. Enter 0 for none (or to ignore the switch). (default = 0) (See "Hookup Limit Switches" for more information.)
Limit Switch (+ Direction)?	Enter the limit switch number (1-8) (default = 0) hooked up for the plus direction of the axis. Enter 0 for none (or to ignore the switch). (See "Hookup Limit Switches" for more information.)
Direction Reversal (OFF or ON)?	Turning the Direction Inversion ON causes the motor to run in the opposite direction. This eliminates re-wiring a motor when one or more of the axes is not moving in the proper direction. (default = OFF)
Motor Reversal (OFF or ON)?	Turning the Motor Reversal ON causes the count direction of the encoder inputs to be changed. This allows the CNC4 to be used with a variety of user supplied Linear scales and encoders. (default = OFF)
Closed Loop (OFF or ON)?	Closed loop operation is enabled by setting this menu selection to ON. (default = OFF)
Closed Loop Pulses/Turn?	This is the number of pulses that the encoder or scale outputs per revolution of the motor. (default = 1600)

Further descriptions and notes on the motor parameters follow:

LEAD SCREW TURNS/UNIT

The entry of the lead screw turns/unit is very straight forward for normal use. This parameter allows the CNC4 to compute the number of motor steps required for a

programmed distance. Entry of un-even turns ratios such as 4.77931 are valid. The CNC4 may round off the value when displaying it however, calculations will be made based upon the actual input value, for a total of 6 digits.

MOTOR STEPS/TURN

Motor steps/turn indicate the number of micro-steps a motor takes to go through a full revolution. Most stepping motors are 1.8 degree/step motors. This means that it takes 200 full steps to go through 360 degrees, a full revolution (360 / 1.8 = 200). A microstep as utilized by the CNC4 is 1/8 of a full step. Therefore (200 full steps) * (8 micro steps) =(1600 micro-steps) per turn.

SLOW JOG

The slow jog is the speed at which the motor is stepped during a SLOW jog. This value can be anything you wish, up to the default value of 800. Entering a lower value will cause the CNC4 to move slower during and CONT SLOW jog. DO NOT enter a value greater than 800. Doing so could cause mis-positioning of the axis.

MAXIMUM RATE

This is the ultimate top speed of the motor in micro-steps per second. During a rapid traverse (G0) the motor immediately starts moving at 200 micro-steps per second and accelerates up to the maximum rate. This speed will typically be 16000 micro-steps per second. If your motors stall during a rapid traverse (G0), then you must lower this value. This value is determined by the stepper motor characteristics. The CNC4 is capable of running at 100,000 steps per second however, stepper motors will not run at this speed. A value of 16000 is equal to 2000 full steps per second. If your motors are of the medium or high inductance type, you may have to lower this value to 8000 or less.

As a general rule, stepper motors with low inductance will run faster. Also, 8 lead motors wired in the parallel configuration will run faster than those wired in the series configuration. 6 lead motors are series wired and cannot be changed. More information concerning motor hookup can be found starting on page 115.

Motor rates can be determined empirically. Basically, best guess estimates are established for the rates using setup option 7. The motors are then moved using the front panel jog buttons. Option 7 is then re-used to alter the rates according to the results observed. This is repeated until satisfactory rates are determined. Finally, a simple CNC program is run to validate the rate values.

The following steps outline this procedure in detail:

1. Have the motors mounted on the milling machine. Testing unmounted motors is not representative. It will not demonstrate how well the mill axis will perform.

2. Enter best guess estimate for the maximum rate.

3. Set the front panel jog speed switch to the FAST position and the INCR./CONT. switch to the INCR. position. Using option 5 from the set-up menu, set the jog increment to 5 inches. Bump the jog button for the motor you want to test. If the motor moves successfully and without trouble then use option 7 to raise the maximum rate. Use the jog buttons again until the point where the test fails is determined. Multiply the maximum rate of the failure point by .85 and use option 7 to establish this value as the maximum rate used.

4. Use the jog buttons a final time to make sure that the motor rates work and that the motor is on a valid step position (so that further testing gets a fair start).

5. Write a simple CNC program to test the rates. Have the program move the milling machine axes from one known position to another known position with a rapid traverse. A sample CNC test program follows:

```
G91              ;SETUP (INCREMENTAL POSITIONING)
G0 X10 Y8 Z-1    ;RAPID TRAVERSE OUT - MULTI AXIS
G0 X-10 Y-8 Z1   ;RAPID TRAVERSE BACK
G0 X10           ;RAPID TRAVERSE OUT - SINGLE AXIS
G0 Y8            ;RAPID TRAVERSE OUT - SINGLE AXIS
G0 Z-1           ;RAPID TRAVERSE - SINGLE AXIS
G0 X-10 Y-8 Z1   ;RAPID TRAVERSE BACK - MULTI AXIS
G1 X10 Y7 F120   ;LINEAR TRAVERSE OUT - 120 INCHES/MINUTE
G1 X-10 Y-7      ;LINEAR TRAVERSE BACK - 120 INCHES/MINUTE

M98 G4/100 M2    ;AUTOMATIC RESTART AFTER 1 SECOND DELAY
```

6. Run the program several times. At the end of the runs, check the position of the milling cutter to make sure that the position measured is the position expected. If it is not then go into the setup menu and use option 7 to decrease the maximum rate by 10%. Use the jog buttons to move the motors to a known position. Run the CNC program test again until it passes.

7. Run the program for an hour. At the end of the hour check to see that the milling head is in the correct position.

REMEMBER

DO NOT FORGET TO SET THE JOG INCREMENT BACK TO A REASONABLE DISTANCE AFTER YOU HAVE COMPLETED THE SETUP PROCEDURE LISTED ABOVE !!

LASH COMPENSATION

parameters cause these lash errors to be removed automatically. I.E. Each time an axis reverses direction, an extra move is inserted whose distance is equal to the lash specified. This lash compensation occurs both during jogs and CNC program runs.

During continuous operation the movement of the axes is monitored to remove lash at the proper times. However, after a manual axis move, power down/up, power release or system reset the monitored direction may be incorrect. Therefore it is a good idea to re-zero on the workpiece at the start of a run or after any power down condition.

WHEN USING SCALES WITH THE CNC4 LASH COMPENSATION MUST BE SET TO ZERO ON AXES WHERE SCALES ARE USED.

SETUP MENU OPTIONS 8 THROUGH 13

Setup Menu options 8 through 13 allow the user to perform the operations listed below. Note that these operations can also be accomplished directly from the RUN/IMMEDIATE mode (Control menu option 4) by typing the proper G or M codes and associated X,Y,Z coordinates.

ZERO AXIS POSITION - OPTION 8

Set the axis absolute position to 0. Following the selection of the option, the following prompt is output:

X: -12.4858 Y: 0.0996 Z: 0.0995 X, Y or Z Axis (X, Y or Z)?

The response determines which axis absolute position is set to zero.

The axes can also be zeroed by issuing a G92 from the Run/Immediate mode at the Control menu.

MOVE AXIS - OPTION 9

Jog the axes directly from the keyboard. Note that neither the front panel PAUSE control, nor the front panel Jog controls will work whenever this option is chosen. After this option is chosen, the following prompt is output:

X: 0.0000 Y: 0.0996 Z: 0.0995 X, Y or Z Axis (X, Y or Z)?X

Enter the axis letter which corresponds to the axis you want to jog, and press <ENTER>. The following prompt will appear:

X: 0.0000 Y: 0.0996 Z: 0.0995 Distance (ENTER for Jog)?

Enter the distance you want to move the axis. If you want to continuously jog the axis, just hit <ENTER>. After your response, a final message will be displayed:

X: 0.0000 Y: 0.0996 Z: 0.0995 B,N,Space Bar,ESC (-,+,Stop,Cancel).B

At this point, the operator must strike either the B key to move the axis in the minus direction or the N key to move the axis in the plus direction. The axis will travel the distance which was previously entered. If no distance was entered, then the axis will jog continuously until the operator strikes the SPACE BAR (or until the axis encounters an axis limit switch). Allow room for the axis to decelerate when jogging near an axis limit or with a cutting tool in the spindle.

This method of jogging is redundant considering the front panel jog buttons, however you may find it useful if your terminal is some distance away from the CNC4 itself.

GO TO AXIS ZERO - OPTION 10

Move an axis from the present position to absolute 0 position. Following the selection of the option, the following prompt is output:

X: 0.0000 Y: 0.0996 Z: 0.0995 X, Y or Z Axis (X, Y or Z)?

The response determines which axis is moved to the absolute zero position.

The axes can also be sent to their zero positions by issuing a G0 from the Run/Immediate mode at the Control menu.

GO TO AXIS LIMIT - OPTION 11

Move to an axis limit switch. Following the selection of this option, the following prompts are output:

X: 0.0000 Y: 0.0996 Z: 0.0995 X, Y or Z Axis (X, Y or Z)?X

X: 0.0000 Y: 0.0996 Z: 0.0995 Direction (- or +)?+

Once the response is received to the direction prompt, the movement begins. The axis motor drives at eight times the slow rate until the axis limit is detected and then backs off 160 micro-steps.

Note that options 9 and 10 travel at the maximum speed while option 11 causes the motor to traverse at 8 times the slow rate. If it is desired to drive an axis to a distant limit, it is wise to first use option 9 (or the front panel jog controls) to jog close to the limit and then use option 11 or M90 or M91 limit switch.

If, during the course of axis motion, a limit switch is detected the message "AXIS LIMIT DETECTED" is displayed, Setup option 11 can then be utilized since it is designed to drive to a limit and then back off until the limit switch changes state. You may also use the jog buttons to get off of the limit switch. Set the FAST/SLOW switch to the SLOW position. Press the jog button which will move the axis in the correct direction off of the limit switch. You may have to re-zero the axis because of the fact that the controller cuts power to the axis motor immediately without deceleration.

Driving to a limit switch can also be accomplished by issuing an M91 or M92 in the Run/Immediate mode from the Control Menu. See the M Code section of the manual for an explanation of M91 and M92.

SET Z AXIS HOME POSITION - OPTION 12

Establish the present Z axis position as the home position. This fixed position is used each time a Z axis traverse to home is required during a tool change (M6) or a command to go to the Z axis home position (M25). This position remains unchanged until either

this option is selected again or an M26 is encountered in a CNC program. Note that this is the Z axis home position, NOT NECESSARILY THE Z AXIS 0 POSITION.

Since the Z axis home position is really a tool change position, the Z axis home position should be far enough above the Z zero position to allow easy removal and replacement of all cutting tools.

The Z axis home position can also be set by issuing an M26 in the Run/Immediate mode from the Control menu. Page 82 describes the M26 command.

CONTROL MENU - OPTION 13

Exit the setup mode and return to the Control Menu. An <ESC>, <CTRL X>, or <CTRL C> will also return to the Control Menu.

EDIT MODE OPERATIONS

When the Edit mode is chosen (option 7 from the Control Menu), the following menu and operator prompt is output to the CRT display:

EDIT MENU

T - Top
B - Bottom
I - Insert
D - Delete
L - List
M - Move
C - Copy
F - Display Free Space
S - Squeeze Space
K - Kill Program
X - Exit to Control Menu

COMMAND?

The Edit Menu choices allow the operator to create or alter a CNC program directly from the CRT terminal keyboard. The operator can also upload CNC programs from the CNC4 to a computer or download CNC programs from a computer into the CNC4 edit buffer.

T - Top

Display the top (first) line of the CNC program.

B - Bottom

Display the bottom (last) line of the CNC program.

I - Insert

Insert CNC program lines. The full form of this command is I "number", where CNC program lines are to be inserted before line "number".

When this command is selected, the "number" followed by a colon (":") is displayed. The operator then enters the desired CNC program lines. As each line is entered the next "number" and ":" are output to the screen in anticipation of further input. When the CNC program entry is completed the operator depresses <ENTER>, without other input. This terminates insert mode.

If an existing CNC program is:

```
1: N100 G90 G0 X4.0 Y3.0
2: N120 X5.2
3: N140 X6.7
```

And more lines are to be added, an operator might enter:

```
"COMMAND?"I3 <ENTER>

N130 G91 Y4.2 <ENTER>
N135 G90 <ENTER>
<ENTER> without other input
```

The resultant program would look like this:

```
1: N100 G90 G0 X4.0 Y3.0
2: N120 X5.2
3: N130 G91 Y4.2
4: N135 G90
5: N140 X6.7
```

Note that the editor assigns a "one-up" line number to each CNC program line. These line numbers are in addition to any line numbers which are supplied by the user (i.e. N100, N7010, N5,...). These numbers assure that each CNC program line is uniquely identifiable to both the editor and the run time processor.

If "number" is omitted from the I command, then the editor assumes that a CNC program is being downloaded to the controller from a host computer. Inputs received across the RS232 interface are stored as CNC program lines. Each new line must be terminated by a carriage return. In this mode, line "numbers" followed by colons (":") are not displayed. The input is terminated when a carriage return is received without any other input. I.E. to the editor, downloading looks like a fast typist is entering a CNC program. To download a CNC program from a host computer to the CNC4, an operator enters the following at the "COMMAND?" prompt:

```
K <ENTER>
```

The CNC4 will then issue the following prompt:

X: 0.0000 Y: 0.0996 Z: 0.0995 Are You Sure (Y or N)?

At this point, you must decide whether the CNC program is to be erased from the CNC4 edit buffer or not. If you want to keep this program, enter N at the prompt and proceed to the List - Upload function to save the program. Once you have saved the program to disk, you must then Kill it before you can download a new program from the computer into the CNC4 edit buffer.

After the existing CNC program in the CNC4 edit buffer has been erased (or KILLed), enter the following at the "COMMAND"? prompt:

I <ENTER>

The operator then instructs the host computer to transmit the CNC program file. The resultant display looks like:

```
N100 G90 G0 X4.0 Y3.0
N120 X5.2
N130 G91 Y4.2
N135 G90
N140 X6.7
```

Note also that it is possible to overrun the editor's input buffer if the host computer downloads at a high rate of speed. The solution to this problem is simple. Have the host computer verify that each character transmitted to the Centroid is echoed back before transmitting the next character. This makes sure that each character is processed by the Centroid before any more input is received.

D - Delete

Delete CNC program lines. The full form of the delete command is D"initial number","ending number". The range of program lines to be deleted goes from "initial number" to "ending number".

If an operator wished to delete lines 1 and 2 from the above example, the entry of:

"COMMAND?"D1,2

would produce:

```
1: N130 G91 Y4.2
2: N135 G90
3: N140 X6.7
```

Note that if no "ending number" is input, only "initial number" is deleted.

Note also that in order to change a line you must first delete it and then reinsert it using the I command.

L - List

List CNC program lines. The full form of the list command is L"initial number","ending number". Each program line from initial number to ending number is displayed preceded by a line "number" and a colon (":").
The entry of:

"COMMAND?"L1,5

for the preceding example produces the following output:

1: N100 G90 G0 X4.0 Y3.0
2: N120 X5.2
3: N130 G91 Y4.2
4: N135 G90
5: N140 X6.7

If "initial number" and "ending number" are omitted from the L command then the editor assumes that the CNC program is being uploaded to a host computer from the Centroid. All of the CNC program lines are output across the RS232 interface. The host computer must be ready to receive the CNC lines and store them in a file. In this mode, no line "numbers" or colons (":") are output.

To upload a CNC program to a host computer from the CNC4, an operator first instructs the host computer to receive a file. The operator then enters the following at the "COMMAND" prompt:

L <ENTER>

The resultant CNC4 output is:

N100 G90 G0 X4.0 Y3.0
N120 X5.2
N130 G91 Y4.2
N135 G90
N140 X6.7

Note that the List command supports XON and XOFF handshaking. This allows host computer receive packages to successfully upload CNC programs from the CNC4 without loss of data.

Note also that the uploaded program contains several unwanted lines (The first few and the last few). These lines may be easily removed using any available editor on the host computer to read the uploaded program, remove the unwanted lines and then re-save the program to disk. Using Cencomm's Rec'v (F3) command will keep the unwanted lines from being uploaded.

M - Move

Move CNC program lines. The full form of the move command is M "initial number", "ending number", "insertion number". The command moves the program command lines from "initial number" to "ending number", to the place before "insertion number".

If an operator wished to move lines 2 and 3 from the preceding example just before line 5 (after line 4), then the entry of the following at the "COMMAND"? prompt:

M2,3,5 <ENTER>

would produce:

1: N100 G90 G0 X4.0 Y3.0
2: N135 G90
3: N120 X5.2
4: N130 G91 Y4.2
5: N140 X6.7

C - Copy

Copy CNC program lines. The full form of the copy command is C"initial number","ending number","insertion number". The command copies the program command lines from "initial number" to "ending number", to the place before "insertion number".

If an operator wished to copy lines 2 and 3 from the preceding example to the end of the program, then the entry of the following at the "COMMAND"? prompt:

C2,3,6 <ENTER>

would produce:

1: N100 G90 G0 X4.0 Y3.0
2: N120 X5.2
3: N130 G91 Y4.2
4: N135 G90
5: N140 X6.7
6: N120 X5.2
7: N130 G91 Y4.2

F - Free

Display program free (unused buffer) space. The free command is useful for two reasons. It displays the total program lines entered and the total space left for further program entry.

The entry of the following at the "COMMAND"? prompt:

F <ENTER>

produces a message like the following one:

7 PROGRAM LINES USED, 24383 CHARACTERS OF FREE SPACE AVAILABLE

This means that 7 program lines referred to by the first "F" command used 182 (24575 - 24383) characters of buffer space.

Note that if no CNC program is entered, then the following message is output:

0 PROGRAM LINES USED, 24575 CHARACTERS OF FREE SPACE AVAILABLE

S - Squeeze

Squeeze program free (unused buffer) space. The squeeze command reclaims any unavailable program buffer space for CNC program entry. It is useful only if numerous deletes or moves have been performed. If the S command was selected after the F command in the preceding example, the following result might be attained:

OPERATION COMPLETE

7 PROGRAM LINES USED, 24575 CHARACTERS OF FREE SPACE AVAILABLE

This indicates that the squeeze command regained 87 characters from a fragmented memory.

Note that the squeeze is made necessary because the editor requires rapid storage of program lines to keep up with host computer transmissions. To facilitate these rapid operations the editor places all newly inserted program lines at the bottom of all previously existing program lines, in the unused buffer space. If numerous program updates are performed the free (unused buffer) space becomes fragmented. The S command regains this fragmented buffer space.

K - Kill Program

Kill (erase) all CNC program lines. This command is used to regain all CNC program buffer space for new program entry. When this command is chosen the following prompt is output:

X: 0.0000 Y: 0.0996 Z: 0.0995 Are You Sure (Y or N)?

If the response is Yes, then the CNC program is killed. The program buffer space is erased. If the response is No, then the CNC program is left untouched.

WARNING!

THIS COMMAND DELETES ANY ENTERED CNC PROGRAM

X - Exit to Control Menu

Exit to the Control Menu. This command exits the edit mode and returns to the Control Menu. <CTRL X> or <ESC> will also return to the Control menu.

MISCELLANEOUS CNC PROGRAM SYMBOLS

F - Feedrate

The feedrate is expressed in inches/minute. It is the rate of travel used for milling or drilling operations. F100 expresses a feedrate of 100 inches/minute F10 expresses a feedrate of 10 inches/minute

N - Line Numbers

Line numbers identify CNC program lines. These line numbers are optional, except when a program line is to be used as a loop end point or as the object of a search (control menu option 3). If line numbers are used, they should be placed in ascending order. This improves readability and helps avoid confusion.

```
N100 G0 X5.2 Y7.6
N110 G1 Z-2.7
N120 M2
```

-or-

```
=N300/4
G0 G91 X2 Y3 Z-.2
G1 Z.2 F40
N300
```

S - Spindle Speed - Normal operation

The spindle speed is expressed in revolutions/minute. Each time a tool select is performed, the spindle speed associated with the tool is output to the CRT display. This value is for operator inspection only (unless the spindle is controlled via a BCD speed changer connected to the optional second I/O card) and produces no direct movement action.

T14/1.7/.5 S500 M6

Produces the following message output:

Tool:14 Length: 1.7000 Diameter: 0.5000 Spindle Speed: 500.000

X: 1.0000 Y: 0.0996 Z: 0.0000 Insert Tool: Hit START.

S - Spindle Speed - Automatic Spindle Speed Changer

It is possible to change the spindle speed automatically from within a CNC program under the following conditions:

1. The spindle speed change control on the mill must respond to BCD (Binary Coded Decimal) output numbers 0 through 39.

2. The spindle speed change control on the mill must output an acknowledge signal when it has changed spindle speed.

3. The spindle speed change control on the mill must be connected to the optional spare I/O board on the CNC4 (page 127-133).

4. The proper spindle speed divisor is programmed into the CNC4 in the Setup Menu, option 5 (page 15).

When the conditions listed above are met, the spindle speed can be changed during a tool change. The following CNC program line:

T1/0/.625 S1000 M6

will cause the following sequence to occur:

1. The Z axis will be retracted to the Home position and an M5 (spindle off) will be issued. If the Spindle motor is interfaced to the M3 M4 (R1 R2) relays, the spindle will stop rotating.

2. The tool change will occur first. If the tool changer type is manual (Setup Menu option 5), then the following line will appear on the CRT display:

Tool: 1 Length: 0.0000 Diameter: 0.6250 Spindle Speed: 1000.000

X: 1.0000 Y: 0.0996 Z: 0.0000 Insert Tool: Hit START.

When the Start button is depressed, the following prompt will appear:

Spindle Speed Change, Awaiting Switch Input to Continue.

A BCD number will be output to the spindle speed control connected to the optional second I/O board. One second later, the SPD strobe output on the second I/O board (See pp. 128,131) will activate. When the spindle speed control sends back an acknowledge (connected to I16 on second I/O board) program execution will continue.

T - Tool Select - Manual Operation

The tool select, and its associated values, can be used to define the length and diameter of 24 tools. Note that these values can also be established at the Setup Menu (option 6).

When associated with a tool change M function, T defines the tool number to be inserted. If T0 is the tool inserted then the length and diameter are automatically set to zero.

The most current tool parameters can be viewed by choosing option 2 at the Setup Menu.

T12/-2.3/1.5

Define tool 12 as having a length of -2.3 units and a diameter of 1.5 units. No message is output.

T22/-1.0

Define tool 22 as having a length of -1.0 units and a diameter left unchanged from any previously defined value.

T15//.6

Define tool 15 as having a length left unchanged from any previously defined value and a diameter of .6 units.

T5/-2.6/.4 M6

Produces the following message output:

Tool: 5 Length: -2.6000 Diameter: 0.4000 Spindle Speed: 1000.000

X: 1.0000 Y: 0.0996 Z: 0.0000 Insert Tool: Hit START.

T0 M6

Produces the following message output:

Tool: 0 Length: 0.0000 Diameter: 0.0000 Spindle Speed: 1000.000

X: 1.0000 Y: 0.0996 Z: 0.0000 Insert Tool: Hit START.

This is the cue to remove the tool.

T - Tool Select - Automatic Operation

The CNC4 is capable of driving an automatic tool changer provided the following conditions are met:

1. The tool changer will respond to BCD (Binary Coded Decimal) numbers 1 through 24, strobe, and will send an acknowledge signal when the tool change is complete.

2. The Tool changer is connected to the optional second I/O board on the CNC4 (page 127 - 133).

3. Automatic tool changer is programmed into the CNC4 at Setup Menu option 5 (page 15).

When the above conditions are met, the CNC4 will drive an automatic tool changer. The following CNC program line:

T1/0/.625 S1000 M6

will cause the following sequence of events to occur:

1. The Z axis will be retracted to the Home position and an M5 (spindle off) will be issued. If the spindle motor is interfaced to the M3 M4 (R2 R2) relays, the spindle will stop.

2. The following prompts will be output to the CRT screen:

Tool: 1 Length: 0.0000 Diameter: 0.6250 Spindle Speed: 1000.000

X: 1.0000 Y: 0.0996 Z: 0.0000 Insert Tool: Hit START.

3. BCD number 1 will be output to the tool changer. 1 second later, the TCH output on the second I/O board (See pp. 128,131) will strobe the tool changer, causing it to recognize the programmed tool number (1). When the tool changer has completed its work, it must send an acknowledge signal back to the CNC4. When the CNC4 receives this acknowledge signal, the spindle speed message (Spindle speed divisor=0) or Spindle speed BCD number (Spindle speed Divisor = 1 to 32000) will be output. When the Spindle speed changer (if used) sends acknowledge, CNC program execution will continue. If no automatic spindle speed changer is connected, CNC program execution will continue as soon as the Tool change is completed.

. - Line Identifier For Search Mode

A period (.) preceding CNC program lines has no effect during normal program runs (control menu options 1 and 2). However, during a program search (control menu option 3) only CNC program lines preceded by a period (.) are executed. CNC program lines which establish modes of operation should be preceded by a period (.). CNC program lines which move the axes should never be preceded by a period (.).

Search mode (control menu option 3) is used to resume programs, shut down in the middle of processing, at the point of interruption. The search mode line identifier, ".", is used to reestablish all program defined setup conditions. No X, Y, or Z motion should be produced while a search for a point of interruption is underway. Once a search is completed, search mode gives way to normal run procedures and all program lines, with or without a preceding period (.), are executed from that point on.

```
.G0 G90 G75        ;Rapid, Absolute, multi-quadrant
N100 X2 Y2         ;Axis movement
N110 Y-2           ;Axis movement
.G1                ;Linear
N120 X4 Y0 Z-1     ;Axis movement
N130 X5 Z0         ;Axis movement
```

If a search is run for lines N100 or N110 then the movement is set as a rapid traverse (G0) using absolute positioning (G90). Multiple quadrant circles (G75) are also setup. If a search is run for lines N120 or N130, rapid traverse (G0) is replaced by linear traverse (G1).

/ - Line Identifier For Delete Mode

Delete mode (setup menu option 4) affects blocks (lines) preceded by a "/" when a CNC program is run. If delete mode is turned on, blocks preceded by a "/" are ignored. If delete mode is turned off, blocks preceded by a "/" are executed. This feature allows a CNC program to act one way if delete mode is turned on and another way if delete mode is turned off.

```
/M25                ;Z to home
G0 G90 X2.5 Y4 Z0   ;Go to X = 2.5, Y = 4, Z = 0
G1 Z-.5 F80         ;Drop down to Z = -.5
X0 Y0 Z0            ;3 axis linear to X = 0, Y = 0, Z = 0
/M2                    ;Z to home, restart
```

If delete mode is turned off the program looks like:

```
M25                 ;Z to home
G0 G90 X2.5 Y4 Z0   ;Go to X = 2.5, Y = 4, Z = 0
G1 Z-.5 F80         ;Drop down to Z = -.5
X0 Y0 Z0               ;3 axis linear to X = 0, Y = 0, Z = 0
M2                     ;Z to home, restart
```

If delete mode is turned on the program looks like:

```
G0 G90 X2.5 Y4 Z0   ;Go to X = 2.5, Y = 4, Z = 0
G1 Z-.5 F80         ;Drop down to Z = -.5
X0 Y0 Z0            ;3 axis linear to X = 0, Y = 0, Z = 0
```

: - Visible Comment Identifier

CNC program lines preceded by a colon (:), are displayed to the operator when the CNC program is run. This feature allows a CNC programmer to display messages or reminders to a CNC operator at run time.

```
G90 G75 F80
G1 Z-.25
:Move from 0 to position 1
X4.7 Y3
:Move from position 1 to 2
X6 Y2
:Move from position 2 to 3
X7.2 Y1
```

```
:Return from position 3 to 0
X0 Y0
:    **** CHANGE THE WORKPIECE ****
M2
```

generates the following output when run:

```
Line: 2 N:  X: 0.0000 Y: 0.0000 Z: 0.0000 F: 80.0000
:Move from 0 to position 1
Line: 4 N:  X: 0.0000 Y: 0.0000 Z: -0.2500 F: 80.0000
:Move from position 1 to 2
Line: 6 N:  X: 4.7000 Y: 3.0000 Z: -0.2500 F: 80.0000
:Move from position 2 to 3
Line: 8 N:  X: 6.0000 Y: 2.0000 Z: -0.2500 F: 80.0000
:Return from position 3 to 0
Line: 10 N:  X: 7.2000 Y: 1.0000 Z: -0.2500 F: 80.0000
:    **** CHANGE THE WORKPIECE ****
Line: 12 N:  X: 0.0000 Y: 0.0000 Z: 0.0000 F: 80.0000

Program Stop, Press START button or ENTER key for program restart.
```

; - Internal Comment Identifier

The semi-colon (;) is used to indicate the start of a comment within a CNC program line. Any characters entered after a semi-colon (;) in a program line are ignored when the CNC program is run. The comment field is used to document the purpose of the CNC program to anyone reading the CNC program listing.

```
N100 X4.0 Y4.5          ;GO TO THE START OF A WORKPIECE

=#10                    ;Drill pattern macro

                        ;START THE INTERNAL MILL

N200 M2                 ;PROGRAM STOP
```

Note that the semi-colon (;) may appear anywhere in a CNC program line. The colon (:), used to display comments to the operator, must be the first character of a line.

MACRO SUBROUTINES AND PROGRAM LOOPS

MACRO SUBROUTINES

Macros are a series of program lines. They have imbedded in them a mixture of fixed and variable movement commands. Macros may be defined once, but called many times during a CNC program. Macros condense program lines, which are repeated several times, into one central location.

A program to drill 3 sets of 4 holes each may be written

```
N100 G90 G81 X5.0 Y4.0 Z1.0 F100      ;Drill hole set 1
N110 G91 X1.0
N120 X1.0
N130 X1.5
N140 G90 G81 X-3.2 Y-2.0 Z2.0 F120    ;Drill hole set 2
N150 G91 X1.0
N160 X1.0
N170 X0.2
N180 G90 G81 X7.63 Y-10.25 Z3.7 F80   ;Drill hole set 3
N190 G91 X1.0
N200 X1.0
N210 X3.6
```

This same program can be condensed and rewritten using a macro and three macro calls. The macro contains all of the program lines which contain repeated information. The macro calls modify the macro, each time it is used, with the variable information.

```
#1                                    ;Macro to drill a 4 hole pattern
N100 G90 G81 X* Y* Z* F*
N110 G91 X1.0
N120 X1.0
N130 X*
$
=#1 X*5.0 Y*4.0  Z*1.0 F*100 X*1.5    ;Drill hole set 1
=#1 X*-3.2 Y*-2.0 Z*2.0 F*120 X*0.2   ;Drill hole set 2
=#1 X*7.63 Y*-10.25 Z*3.7 F*80 X*3.6  ;Drill hole set 3
```

Up to 99 macros, numbered 1-99, may be defined in a CNC program. A macro number may be redefined an unlimited number of times. The macro calls always refer to the most recent macro definition.

Macros may be nested up to four levels deep. However, a maximum of twelve arguments are allowed for all macros called in the nesting.

When variable macro arguments are used, they should be defined in the order in which they are used. This practice increases CNC program understandability and readability while reducing programming confusion and errors. WITHIN EACH PROGRAM LINE, macro variable replacements are performed in a very strict order. This order is X, Y, Z, I, J, K and F.

Macro subroutines are defined with the # symbol and terminated with the $ symbol. Macro calls are made with the =# construct. Within a macro the variable arguments are defined by a letter followed by *. In a macro call, variable arguments are defined by a letter, followed by *, followed by the value to be associated with the variable.

```
#3                          ;Macro to drill a 5 hole pattern
G0 G90 X* Y* Z0             ;Go to the initial absolute position
G81 G91 Z-.5 F70            ;Drill hole 1, establish incremental
X1.5                        ;Drill hole 2
X1                          ;Drill hole 3
X-.8 Y-1.35                 ;Drill hole 4
Y-.673                      ;Drill hole 5
G80 G90                     ;Cancel drill cycle, reestablish absolute
$
=#3 X*4 Y*3                 ;Drill 5 hole pattern at X = 4, Y = 3
=#3 X*6 Y*2                 ;Drill 5 hole pattern at X = 6, Y = 2
=#3 X*8 Y*7                 ;Drill 5 hole pattern at X = 8, Y = 7
#5                          ;Macro to engrave '5'
G0 G91 Y.1
G1 Z-.03
G3 X.1 Y.1 I-.1
G1 X-.1
Y.1
X.15
G0 X.1 Y-.3 Z.03
$
#7                          ;Macro to engrave '7'
G1 G91 Z-.03
X.2 Y.3
X-.2
G0 X.25 Y.3 Z.03
$
=#7                         ;Engrave serial number 77557
=#7
=#5
=#5
=#7
```

```
#11                                   ;Macro to engrave 'A'
G1 G91 Z-.03
X.1 Y.3
X.1 Y-.3
G0 X-.15 Y.15 Z.03
G1 Z-.03
X.1
G0 X.1 Y-.15 Z.03
$
#14                                   ;Macro to engrave 'D'
G1 G91 Z-.03
Y.3
X.1
G2 X.1 Y-.1 J.1
G1 Y-.1
G2 X-.1 Y-.1 I.1
G1 X-.1
G0 X.25 Z.03
$
=#14                                  ;Engrave 'D A D'
X.4
=#11
X.4
=#14
```

CNC PROGRAM LOOPS

=N - Loop

A loop is placed in a CNC program when one or several program lines are to be repeated more than once. Loops may be nested up to 4 levels. Each loop definition requires an associated loop count.

```
G0 G90 X0 Y0 Z0
=N300/3
G0 G91 X2 Y3 Z-.2
G1 Z.2 F40
N300
```

moves the axes to the following absolute positions:

X = 0.0 Y = 0.0 Z = 0.0
X = 2.0 Y = 3.0 Z = -0.2

Z = 0.0
X = 4.0 Y = 6.0 Z = -0.2
Z = 0.0
X = 6.0 Y = 9.0 Z = -0.2
Z = 0.0

N400 G91
=N500/3
=N450/4
N440 X1.0 Y2.5
N450 X1.5
N490 Y1.0
N500 X-2.0 Y-7.3

executes the following program lines:

N400, N440, N450, N440, N450, N440, N450, N440, N450, N490, N500, N440, N450, N440, N450, N440, N450, N440, N450, N490, N500, N440, N450, N440, N450, N440, N450, N490, N500

CNC4 PROGRAM ERROR MESSAGES

If an error is encountered in a CNC program line while the CNC4 is in the Run mode, a message of the following format is output:

****** ERROR *** Line 12/ "Error Description"***

The line number tells where the error occurred. The "Error Description" tells what error occurred.

"Error Descriptions" follow:

"Invalid Parameter"

The program line contains an unrecognizable command or character.

GO (gee ohh) instead of G0 (gee zero)

M23 is an invalid M function number

T1/1%2 instead of T1/1/2

"Value Out of Range"

Number, specified with a CNC code, is too big or too small (usually 0).

G1 X3 F00 contains an invalid feedrate of zero

G81 Z0 contains an invalid drilling depth increment of zero

G91 G79 J0 contains a circle radius of zero

"Undefined Number"

A call was issued for a macro (=# macro number) which was never used in a macro definition (# macro number).

"Nesting too Deep"

The nesting level (number of imbedded) for macros or loops exceeds 4.

"Too Many Arguments"

The number of macro arguments used in a single macro call or in nested macro calls exceeds 12.

```
#1
=#2 X*13
$
=#1 X*1 Y*2 Z*3 F*4 X*5 Y*6 F*7 Y*8 X*9 F*10 F*11 X*12
```

"Unmatched Macro Limits"

The end of a macro ($) was encountered before a corresponding macro was defined (# macro number).

```
#1
G0 X0 Y0
$
$ ;This macro end has no corresponding macro definition
```

"No Argument Supplied"

A macro variable was not supplied with a value when the macro was called.

```
#10
G0 X*
$
=#10 ;The macro call is not supplying the needed argument for X
```

"Prohibited During Cutter Compensation"

A prohibited operation was encountered while cutter compensation was turned on (G41 or G42).

Prohibited operations during cutter compensation are G18, G19, G77, G78, G79, G81-G89, switching between G41 and G42 without an intervening G40, and program termination before G40.

This error is easily fixed by inserting a G40 in the CNC program before proceeding with the invalid operation.

GØ - Rapid Traverse (default)

A GØ says to move at the maximum motor speeds to the defined end point. Ignore the feedrate, traverse at the maximum rate defined by setup menu option 7.

X, Y and Z, used with GØ, define the destination of the move. For absolute positioning the X,Y and Z values define the absolute positions of the X, Y and Z axes at the completion of the move. For incremental positioning the X, Y and Z values define the movement distances along the X, Y and Z axes.

If the tool is being picked up (+Z direction) then the movement is +Z followed by X and Y. If the tool is being placed down (-Z direction) then the movement is X and Y followed by -Z.

GØ G9Ø X4.5 Y-6.25 Z-1.2 says to rapid traverse using absolute positioning (G9Ø). Move the X and Y axes to X = 4.5, Y = -6.25. At the completion of this move rapid traverse down to Z = -1.2.

GØ G9Ø Y2.5 says to rapid traverse using absolute positioning (G9Ø). Move only the Y axis to Y = 2.5.

GØ G91 X2 Y3 Z1 says to rapid traverse using incremental positioning (G91). Move the Z axis up 1 unit. At the completion of this move rapid traverse the X axis 2 units and the Y axis 3 units.

GØ G91 X2
Y3

Says to rapid traverse using incremental positioning (G91). Move the X axis 2 units. At the completion of the first move, rapid traverse the Y axis 3 units.

G1 - Linear Traverse

Move at the programmed feedrate to the defined end point. Traverse in a linear (straight line) motion.

X, Y and Z, used with G1, define the destination of the move. For absolute positioning the X, Y and Z values define the absolute positions of the X, Y and Z axes at the completion of the move. For incremental positioning the X, Y and Z values define the movement distances along the X, Y and Z axes.

G1 can be a 1,2 or 3 axis simultaneous move. It is intended to move, in a straight line, from one point to another during a milling operation.

G1 G90 X4.5 Y-6.25 Z-1.2 F10 says to linear traverse using absolute positioning (G90). Move the X, Y and Z axes simultaneously to X = 4.5, Y = -6.25 and Z = -1.2 at a feedrate of 10 inches/minute.

G1 G90 X.75 F40 says to linear traverse using absolute positioning (G90). Move only the X axis to X = 0.75 at a feedrate of 4 inches/minute.

G1 G91 X2 Y3 Z1 F80 says to linear traverse using incremental positioning (G91). Move the X, Y and Z axes simultaneously 2 units, 3 units and 1 unit respectively at a feedrate of 8 inches/minute.

G1 G91 X2 F120
X-4 Y3
Says to linear traverse using incremental positioning (G91). Move the X axis 2 units at a feedrate of 12 inches/minute. At the completion of the first move, linear traverse the X and Y axes -4 units and 3 units respectively at the same feedrate.

G2 - Circular Move, Clockwise

Move at the programmed feedrate to the defined end point. Traverse in a clockwise circular motion.

X, Y and Z, used with G2, define the destination of the move. For absolute positioning the X, Y and Z values define the absolute positions of the X, Y and Z axes at the completion of the move. For incremental positioning the X, Y and Z values define the movement distances along the X, Y and Z axes from the start position.

For full circles, no X, Y or Z axis movement values are required.

I, J and K used with G2, define the circular center point of the move.

- For absolute positioning the I, J and K values define the X, Y and Z axis centers.

- For incremental positioning the I, J and K values define the distances from the X, Y and Z axis **start** to the X, Y and Z axis **center** points. That is, for incremental positioning:

I = X center - X start,
J = Y center - Y start and
K = Z center - Z start.

G2 is a 2 axis, simultaneous movement. It is intended to move, in a clockwise and circular motion, from one point to another during a milling operation.

Circular traverses will go slower as the circle radius decreases. The maximum circular traverse rate will be limited by the maximum motor speed as defined in the motor setups and the circle or arc radius.

G17 G75 G9Ø
G2 X3.4 Y6.7 I1.3 J2.9 F4Ø
Says setup for XY plane circles (G17) and use absolute positioning (G9Ø). Construct a clockwise circle. Circular traverse from the present X,Y position to circle ending point X = 3.4, Y = 6.7 at a feedrate of 4Ø inches/minute. Use circle center point X = 1.3, Y = 2.9.

G2 G17 G75 G9Ø I4.3 J-4.6 F6Ø Says setup for XY plane circles (G17) and use absolute positioning (G9Ø). Construct a clockwise circle. Circular traverse from the present X,Y position to the present X,Y position (full circle) at a feedrate of 6Ø inches/minute. Use circle center point X = 4.3, Y = -4.6.

G18 G75 G9Ø F1ØØ hello
GØ X4 Z-2
G2 G91 X-2 Z-1 I-2 K-2.5
Says setup for XZ plane circles (G18) and start out with absolute positioning (G9Ø). Rapid traverse (GØ) to the starting point of the circle X = 4, Z = -2. Construct a clockwise (G2) circular arc using incremental positioning (G91). Traverse from the present X,Z position to circle ending point X =-2, Z = -1. I=-2 (X center - X start), and K=-2.5 (Z center - Z start). Use the center point X = 2 Z = -.5.

IMPORTANT

Circular arcs less than .ØØ2 radians (.11 degrees) must have an arc length of greater than .Ø1 inches. Otherwise the CNC4 will mill a complete circle.

G3 - Circular Move, Counterclockwise

Move at the programmed feedrate to the defined end point. Traverse in a counterclockwise circular motion.

X, Y and Z, used with G3, define the destination of the move. For absolute positioning the X, Y and Z values define the absolute positions of the X, Y and Z axes at the completion of the move. For incremental positioning the X, Y and Z values define the movement distances along the X, Y and Z axes.

For full circles,no X,Y or Z axis movement values are required.

I, J and K used with G3, define the circular center point of the move.

-For absolute positioning the I, J and K values define the X, Y and Z axis centers.

-For incremental positioning the I, J and K values define the distances from the X, Y and Z axis **start** to the X, Y and Z axis **center** points. That is, for incremental positioning:

I = X center - X start
J = Y center - Y start, and
K = Z center - Z start.

G3 is a 2 axis, simultaneous movement. It is intended to move, in a counterclockwise and circular motion, from one point to another during a milling operation. Circular traverses will be limited by the maximum motor rate as defined in the motor setups, and the circle or arc radius.

G17 G75 G90
G3 X-2.5 Y7.1 I-1.25 J3.55 F4
Says setup for XY plane circles (G17) and use absolute positioning (G90). Construct a counterclockwise circle. Circular traverse from the present X,Y position to circle ending point X = -2.5, Y = 7.1 at a feedrate of 4 inches/minute. Use center point X = -1.25, Y = 3.55.

G3 G17 G75 G91 XØ YØ IØ J-1 Says setup for XY plane circles (G17) and use incremental positioning (G91). Construct a counterclockwise circle. Circular traverse from the present X,Y position to the present X,Y position (full circle) at a feedrate of 6 inches/minute. The center point is XØ Y-1.

G19 G75 G9Ø F1ØØ
G3 Y3 Z-1.2 JØ K-.2
arc. Circular traverse from the present Y,Z position to circle ending point Y = 3, Z = -1.2 at a feedrate of 1ØØ inches/minute. Use the circle center point Y = Ø, Z = -Ø.2.

IMPORTANT

Circular arcs less than .ØØ2 radians (.11 degrees) must have an arc length of greater than .Ø1 inches. Otherwise the CNC4 will mill a complete circle.

G4 - Dwell Time Setting

Used in conjunction with M98 to create time delays in various operations

Time in .Ø1 seconds (Ø default)

Ranges from 1 to 65535 (.Ø1 seconds to Ø.9 minutes)

G4/2ØØ Says Establish a dwell time 2 seconds (2ØØ * .Ø1 seconds).

G4/5 G9Ø
G82 X4.7 Y-3 Z1 F12Ø
Says establish a dwell time of .Ø5 seconds. Use this dwell to cause a .Ø5 second delay with the spotfacing cycle (G82).

M98 G4/25Ø M2 Says establish the program continuation method as time based (M98). Establish a dwell time of 2.5 seconds (25Ø * .Ø1 seconds). When this program line is encountered at the end of a CNC program the following message is output:

Program Stop, Awaiting Timeout for Program Restart.

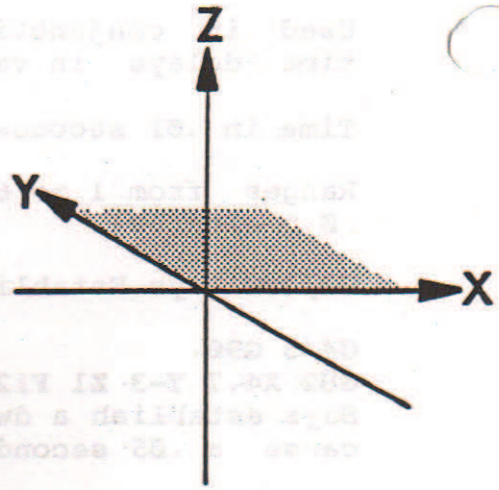
After 2.5 seconds the program automatically restarts (M2) at line 1. All tool (T) values are set to tool zero.

NOTE:

G4 can be used with M98, G82, G88, and G89, to create programmed time delays.

G17 - XY Circular Plane (default)

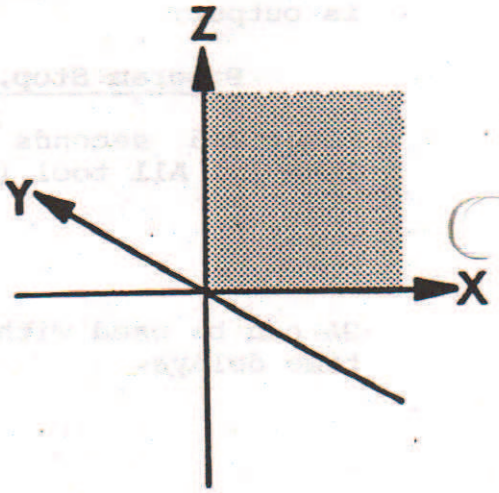
Used in conjunction with the G2 and G3 functions.



G2 G17 G75 G90 I4.3 J-4.6 F6 Says setup for XY plane circles (G17) and use absolute positioning (G90). Construct a clockwise circle (G2). Circular traverse from the present X,Y position to the present X,Y position (full circle) at a feedrate of 6 inches/minute. Use as the circle center point X = 4.3, Y = -4.6.

G18 - ZX Circular Plane

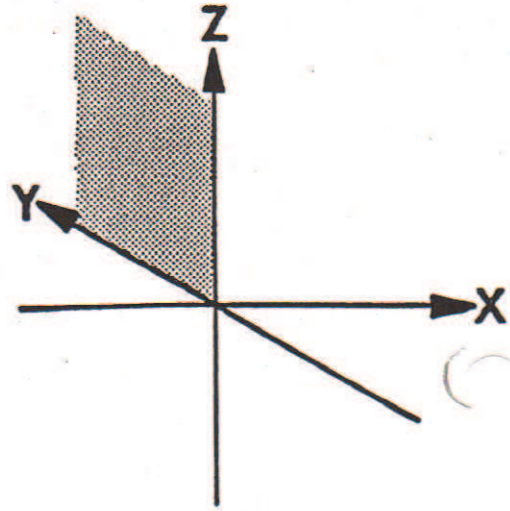
Used in conjunction with the G2 and G3 functions.



G18 G75 G90 F10
G0 X4 Z-2
G2 G91 X-2 Z-1 I2 K-1.5
Says setup for XZ plane circles (G18) and start out with absolute positioning (G90). Rapid traverse (G0) to the starting point of the circle X = 4, Z = -2. Construct a clockwise (G2) circular arc using incremental positioning (G91). Traverse from the present X,Z position to the circle ending point X = 2, Z = -3 at a feedrate of 10 inches/minute. Use center point X = 2, Z = -.5.

G19 - YZ Circular Plane

Used in conjunction with the G2 and G3 functions.



G19 G75 G90 F10
G3 Y3 Z-1.2 J0 K-.2
Says setup for YZ plane circles (G19) and use absolute positioning (G90). Construct a counterclockwise (G3) circular arc. Circular traverse from the present Y,Z position to circle ending point Y = 3, Z = -1.2 at a feedrate of 10 inches/minute. Use the circle center point Y = 0, Z = -0.2.

G30 - Cancel the Axes of Symmetry

G30 cancels G31 and G32

G31 - Reverse the image over an X axis (backwards)

The axis of symmetry is taken from the X, Y position when the G31 is issued.

After a G31 is encountered, all X positions are flipped over the axis of symmetry.

G31 may be combined with G32 to produce an image which is reversed over both the X and Y axes (backwards and upside down).

G31 stays in effect until G30 is issued.

G32 - Reverse an image over the Y axis (Upside down)

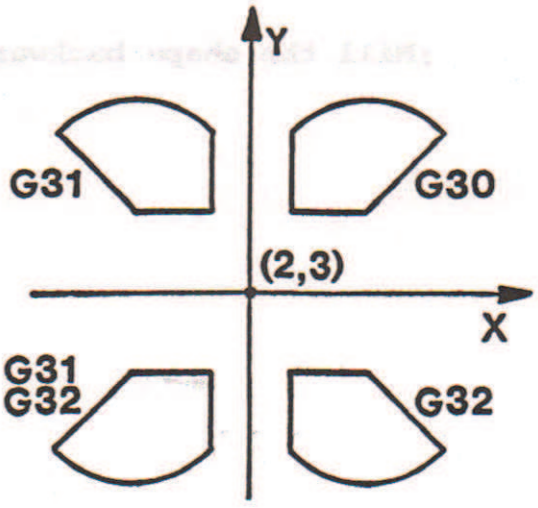
The axis of symmetry is taken from the X,Y position when the G32 is issued.

After a G32 is encountered, all Y position values are flipped over the axis of symmetry.

G32 nay be combined with G32 to produce an image which is reversed over both the X and Y axes (upside down and backwards).

G31 stays in effect until G30 is issued.

The following picture depicts the image reversals produced by using G31 and G32 alone and in combination. The symmetry points used for reversals are X=2 and Y=3. Following the picture are two sample programs, each of which creates the pictured reversals. The first using absolute positioning, the second using incremental positioning.



```
#1
G1                          ;Macro to mill a simple shape
G0 G90 X2.25 Y4 Z-.1        ;Traverse to the first absolute position
G1 Y3.5                     ;Mill the shape
X2.75
X3.25 Y4
G3 X2.25 I2.75 J3.5
G0 X2 Y3 Z0                 ;Traverse back to the symmetry point
$
G17 G75 G90                 ;XY, Multi-quadrant circles, Absolute
G0 X2 Y3 Z0 F80             ;Traverse to the symmetry point
=#1                         ;Mill the shape
G31
=#1                         ;Mill the shape backwards
G30
G32
=#1                         ;Mill the shape upside down
G31
G32
=#1                         ;Mill the shape backwards and upside down
```

Create the pictured image reversals using incremental positioning

```
#1
G1                         ;Macro to mill a simple shape
G0 G91 X.25 Y1 Z-.1        ;Traverse to the first incremental position
G1 Y-.5                    ;Mill the shape
X.5
X.5 Y.5
G3 X-1 I.5 J.5
G0 X-.25 Y-1 Z1
$
G17 G75 G90                ;XY, Multi-quadrant circles, Absolute
G0 X2 Y3 Z0 F80            ;Traverse to the symmetry point
=#1                        ;Mill the shape
G31
=#1                        ;Mill the shape backwards
G30
G32
=#1                        ;Mill the shape upside down
G31
G32
=#1                        ;Mill the shape backwards and upside down
```

CUTTER COMPENSATION

Cutter compensation used with tool select, T, and tool change, M6, automatically offsets the cutter traversal path away from the workpiece path defined by a CNC program. This automatic adjustment feature eliminates the need for manual computation of cutter tool offsets when writing CNC part program descriptions.

Cutter compensation offsets the cutter tool by one half of the cutter diameter (the cutter radius).

Cutter compensation offsets the cutter only in the XY plane. Cutter compensation is not valid for circular arcs in the XZ and YZ planes.

Cutter compensation is not valid for the G77, G78 and G79 special milling cycles.

Cutter compensation is not valid for scaling (G72,G73) or for preset absolute position (G92).

Cutter compensation is also referred to as tool offset compensation or tool path milling.

IMPORTANT !

YOU MUST FIRST SELECT A TOOL USING THE T COMMAND AND THEN ISSUE A TOOL CHANGE COMMAND (M6) BEFORE USING G41 OR G42!

G40 - Cutter Compensation Off

G40 cancels G41 and G42.

G40 causes no movement.

G41 - Cutter Compensation Left

G41 offsets the cutter tool one half of the cutter diameter, to the left of the workpiece, relative to the direction of travel. This offset will occur on the way to the first coordinate associated with the next move. The cutter can be forced to offset by commanding the CNC4 to go to the present position.

```
G0 X0 Y0
G40
T1/0/.375 M6
G41
G1 X1 Y1 F10
```

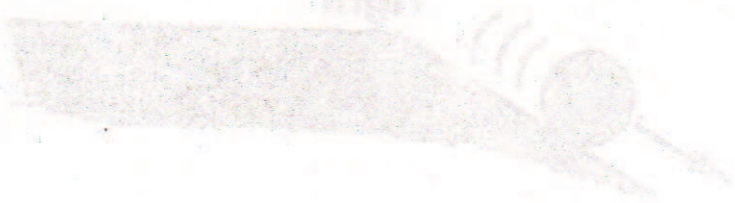
Causes the CNC4 to rapid traverse to the X0 Y0 position. Tool 1 which has a length of zero and a diameter of .375" is selected. Left cutter compensation is turned on. The cutter will then offset on its way to the X1 Y1 position.

```
G0 X0 Y0
G40
T1/0/.375 M6
G41
G1 X0 Y0
G1 X1 Y1 F10
```

Causes the CNC4 to rapid traverse to the X0 Y0 position. Tool 1 which has a length of zero and a diameter of .375" is selected. Left cutter compensation is turned on. The cutter will then offset before it responds to the G1 X1 Y1 command.

G41 may not be issued when G42 is in effect. G40 must be used to cancel G42 before a G41 may be used.

Programming with G41 is the same as programming without G41. In other words, you still enter coordinates as though you were referring to the center of the tool. G41 will do the rest.



G42 - Cutter Compensation Right

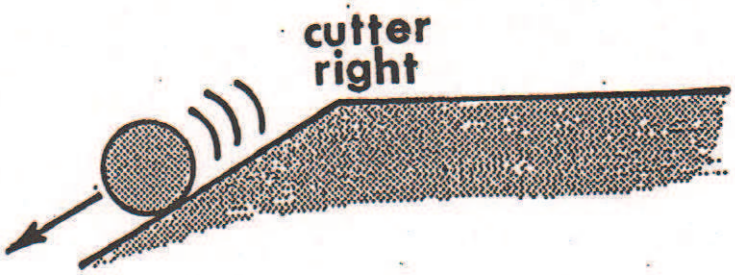
G42 offsets the cutter tool one half of the cutter diameter, to the right of the workpiece, relative to the direction of travel. This offset will occur on the way to the first coordinate associated with the next move. The cutter can be forced to offset by commanding the CNC4 to go to the present position.

```
GØ XØ YØ
G4Ø
T1/Ø/.375 M6
G42
G1 X1 Y1 F1Ø
```

Causes the CNC4 to rapid traverse to the XØ YØ position. Tool 1 which has a length of zero and a diameter of .375" is selected. Right cutter compensation is turned on. The cutter will then offset on its way to the X1 Y1 position.

```
GØ XØ YØ
G4Ø
T1/Ø/.375 M6
G42
G1 XØ YØ
G1 X1 Y1 F1Ø
```

Causes the CNC4 to rapid traverse to the XØ YØ position. Tool 1 which has a length of zero and a diameter of .375" is selected. Right cutter compensation is turned on. The cutter will then offset before it responds to the G1 X1 Y1 command.



G42 may not be issued when G41 is in effect. G40 must be used to cancel G41 before a G42 may be used.

Programming with G42 is the same as programming without G42. In other words, you still enter coordinates as though you were referring to the center of the tool. G42 will do the rest.

CUTTER COMPENSATION TIPS

Cutter compensation is easy to use. Just place a G41 or G42 in a CNC program block (line)after a T and M6 command and cutter compensation takes over. Position the cutter away from the workpiece edge to be milled when turning cutter compensation on or off.

Note that cutter compensation has no trouble traveling around the edges of intersecting circles. If turning on cutter compensation creates some non-intersecting circular end points, then an additional circular move is automatically inserted to connect the points.

The following pictures demonstrate two cases of cutter compensation around intersecting circles. Note that the second picture demonstrates the case where compensation creates two non-intersecting circles. An additional circular move is automatically inserted to connect the points, while maintaining the compensation offset.

The following program is provided to illustrate the usage of cutter compensation. The program is shown without cutter compensation, with the alterations required for an outside edge compensation (compensation left in this case) and then with the alterations required for an inside edge compensation (compensation right in this case). Notice that cutter compensation changes only a few program blocks (lines). Some when compensation is turned on, and some when it is turned off.

No Compensation	Compensation Left	Compensation Right	Comment
G17 G75 G90 F80	G17 G75 G90 F80	G17 G75 G90 F80	;Setup
T14/1.2/.5 M6	T14/1.2/.5 M6	T14/1.2/.5 M6	;Tool select
G0 X0 Y0 Z-.1	G0 X-1 Y-1 Z-.1	G0 X2 Y0 Z-.1	;Initial point
	G41	G42	;Compensation on
		G1 X1 Y-1	;(Second initial)
		X0 Y0	
G1 X2 Y2	G1 X2 Y2	G1 X2 Y2	;Normal path
G2 X4 Y2.05 I3 J2	G2 X4 Y2.05 I3 J2	G2 X4 Y2.05 I3 J2	
X6 Y2.05 I5 J2.05	X6 Y2.05 I5 J2.05	X6 Y2.05 I5 J2.05	
G1 X2 Y-2	G1 X2 Y-2	G1 X2 Y-2	
X0 Y0	X-1 Y1	X0 Y0	;Final point
		X2	;(Second final)
	G40	G40	;Compensation off


cutter
left


cutter
right

G7Ø - Inch Units of Measure (default)

Establish inches as the units of measure.

G71 - Metric Units of Measure (millimeters)

Establish millimeters as the units of measure.

G72 - Cancel Scale Changes (Resume Normal X,Y Positioning)

G72 cancels any scaling changes created by G73.

G73 - Change Scale on the X and/or Y Axes

G73 changes the scale on the X and/or Y axes to the value specified with the command.

```
G73 X2. Y2.          ;Double (multiply by 2) all X and Y moves
G73 X.5              ;Half (multiply by .5) all X moves
G73 Y1.25            ;Multiply by 1.25 all Y moves
```

G73 may not be on a block (line) with any other G codes.

G73 multiplies all following absolute and incremental moves by the specified scaling factor.

```
G73 X2 Y2            ;Double (multiply by 2) all X and Y moves
GØ G9Ø X1.5 Y-.5     ;Move to position X = 3.Ø, Y = -1.Ø.
G91 X.75 Y1          ;Move to position X = 4.5 (3.Ø + 1.5),
                         Y = 1.Ø (-1.Ø + 2.Ø)
```

Additional G73 commands multiply the scaling factor by the specified values (they do not define a new scaling factor).

```
G73 X.8 Y.8          ;Multiply by .8 all X and Y moves
G73 X.625 Y.625      ;Multiply by .5 all X and Y moves
                        (.8 multiplied by .625 = .5)
```

Engrave differing letter sizes

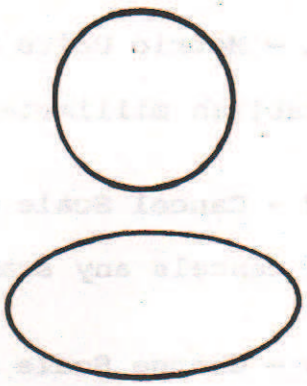
```
#24 ;Macro to engrave "N"
G1 G91 Z-.Ø3
Y.3
X.2 Y-.3
Y.3
GØ X.Ø5 Y-.3 Z.Ø3
$
=#24 ;Engrave full sized "N"
G73 X.8 Y.8
=#24 ;Engrave .8 sized "N"
G73 X.625 Y.625
=#24 ;Engrave half sized "N"
G72
```

```
G73 X.5
=#24 ;Engrave a skinny "N"
```

N

Mill a circle and an ellipse

```
G91 G2 XØ YØ I-.5 JØ ;Circle
```



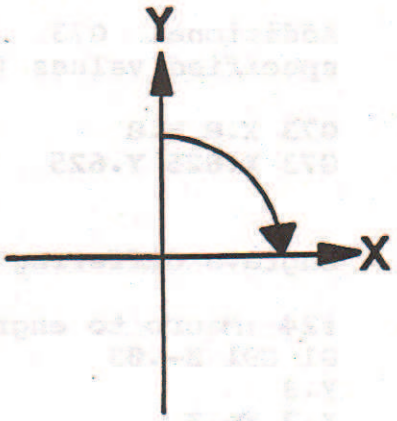
```
G73 X1.5 Y.75
G91 G2 XØ YØ I-.5 JØ ;Ellipse
```

Note the circle has a diameter of 1 unit (radius .5) in both the X and Y directions. The ellipse has an X dimension of 1.5 units (1 multiplied by 1.5) and a Y dimension of .75 units (1 multiplied by .75).

G74 - Single Quadrant Circular Interpolation

This function is provided for compatibility reasons only. **G75 IS RECOMMENDED.**

G74 alters the G2 and G3 functions to construct circular arcs less than or equal to 9Ø degrees. If G74 is used, circular arcs greater than 9Ø degrees must be created using multiple program lines.



When G74 is in force, X, Y and Z define the end points of the circular arc (see G2 and G3).

When G74 is in force, I, J and K are the UNSIGNED incremental distances from the starting point of the circle to the X, Y and Z axis center points. The actual position of the center point is computed by the CNC processor so that the resultant circular arc milled is less than or equal to 9Ø degrees.

```
G17 G74 G9Ø F1Ø
GØ X4 Y-2 Z-.1
G2 G91 X-2 Y-1 I-2 J-1.5
```

Says setup for XY plane (G17) single quadrant (G74) circles and start out with absolute positioning (G9Ø). Rapid traverse (GØ) to the starting point of the circle X = 4, Y = -2. Construct

a clockwise (G2) circular arc using incremental positioning (G91). Traverse from the present X,Y position to the circle ending point X = 2, Y = -3 at a feedrate of 10 inches/minute. Choose the center point so that the arc is less than or equal to 90 degrees. Use center point X = 2, Y = -.5.

G17 G74 G90 F8
G0 X1 Y1.5 Z-.16
G2 X3 Y-.5 J2
Says setup for XY plane (G17) single quadrant (G74) circles using absolute positioning (G90). Rapid traverse (G0) to the starting point of the circle X = 1, Y = 1.5. Construct a clockwise (G2) circular arc. Circular traverse from the present X,Y position to the circle ending point X = 3, Y = -.5 at a feedrate of 8 inches/minute. Choose the center point so that the arc is less than or equal to 90 degrees. Use center point X = 1, Y = -.5. Note that in this case the arc is exactly 90 degrees. Note also that although the X and Y circle end points are defined absolutely, the center point is still described by an UNSIGNED INCREMENTAL VALUE (J2).

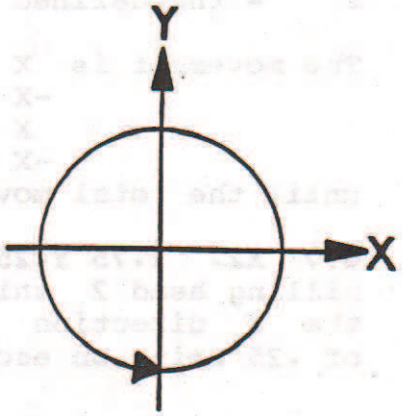
G75 - Multiple Quadrant Circular Interpolation (default)

G75 IS RECOMMENDED for circular arc construction, even for arcs less than 90 degrees. G75 is much less restricted than G74.

G75 allows the G2 and G3 functions to construct circular arcs of any size, including full circles, in a single CNC program line.

When G75 is in force, X, Y and Z define the end points of the circular arc (see G2 and G3).

When G75 is in force, I, J and K define the center points of the circular arc for the X, Y and Z axes respectively.



G17 G75 G90
G2 X3.4 Y6.7 I1.3 J2.9 F4
Says setup for XY plane (G17) multi-quadrant circles (G75) and use absolute positioning (G90). Construct a clockwise circle. Circular traverse from the present X,Y position to circle ending point X = 3.4, Y = 6.7 at a feedrate of 4 inches/minute. Use as circle center point X = 1.3, Y = 2.9.

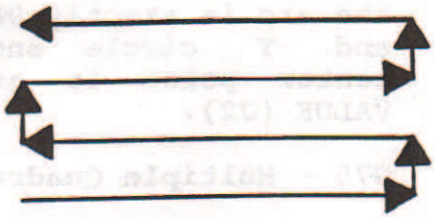
G2 G17 G75 G90 I4.3 J-4.6 F6
Says setup for XY plane (G17) multi-quadrant circles (G75) and use absolute positioning (G90). Circular traverse clockwise from the present X,Y position to the present X,Y position (full circle) at a feedrate of 6 inches/minute. Use circle center point X = 4.3, Y = -4.6.

G3 G17 G75 G91 XØ YØ J1 F6
Says setup for XY plane (G17) multi-quadrant circles (G75) and use incremental positioning (G91). Construct a counterclockwise circle. Circular traverse from the present X,Y position to the present X,Y position (full circle) at a feedrate of 6 inches/minute. The center point is X = present X position, Y = 1 unit less than the present Y position (1 unit radius).

G77 - Facing

Special mill used to face a workpiece at a defined feedrate.

G77 requires the following parameters:



X - the signed incremental move in the X direction

Y(1) - the signed incremental total move in the Y direction

Y(2) - the UNSIGNED incremental step in the Y direction

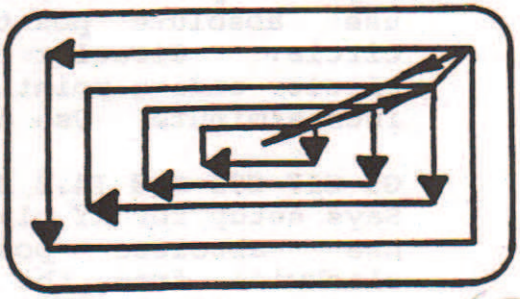
F - the defined feedrate

The movement is X then Y(2)
-X then Y(2)
X then Y(2)
-X then Y(2)
until the total move of Y(1) is reached

G77 X2. Y.75 Y.25 F1Ø Says face a workpiece by moving the milling head 2 units in the X direction and .75 units in the Y direction at 1Ø units/minute. Use a Y step increment of .25 units on each pass.

G78 - Pocket Milling

Special mill used to mill out a pocket at a defined feedrate.



G78 requires the following parameters:

X(1) - the incremental UNSIGNED distance from the center of the pocket to the pocket wall, minus the cutter radius

X(2) - the incremental UNSIGNED step distance in the X direction

X(3) - the incremental UNSIGNED distance

for the last step towards the pocket wall (defaults to .Ø1")

Y(1) - the incremental UNSIGNED distance from the center of the pocket to the pocket wall, minus the cutter radius

Y(2) - the incremental UNSIGNED step distance in the Y direction

F(1) - the feedrate of the initial pocket milling cycle. I.E. the feedrate of the roughing cycle. The mill from the center of the pocket until a distance X(3) away from the pocket wall.

F(2) the feedrate of the final pocket milling cycle. I.E. the feedrate of the finish mill. The mill along the outer pocket wall (defaults to 1.5 times F(1)).

The movement around the pocket is diagonal then rectangular for each mill. The X(2) and Y(2) incremental values are used to determine the X and Y moves on each milling pass. These incremental values are used until the mill is a distance X(3) from the pocket wall. The milling head is then moved the finish milling distance, X(3), and the final milling pass is performed at feedrate F(2). When the pocket mill is completed, the milling head is moved back to the original point.

The roughing passes of the pocket mill are performed in a conventional milling direction. The finish pass of the pocket mill is performed in a climb milling direction.

G78 X.8Ø X.25 X.Ø5 Y.5Ø Y.15 F4Ø Says internally mill a pocket by moving the milling cutter a total of .8 units in the X direction and .5 units in the Y direction. Use an initial X step increment of .25 units, an initial Y step increment of Ø.15 units and a feedrate of 4Ø units/minute. Continue until the mill has proceeded .75 units in the X direction and .45 units in the Y direction. Move an additional .15 units in each direction and perform a final milling pass at 6Ø units/minute in the opposite direction. When this pass is completed, return to the initial position.

G79 - Internal Hole Milling

Special mill used to create holes larger than the milling cutter.

G79 requires the following parameters:

J - the SIGNED incremental radius of the circle, minus the cutter radius The sign (+ or -) determines whether the cut will be a conventional or climb cut.

A positive J value means that the path around the internal hole will be in the clock-wise direction (conventional cut). A negative J value means that the path around the internal hole will be in the counter clock-wise direction (climb cut).

F - the feedrate of the mill

The first move is a linear move to a point half the distance of the radius in both the X and Y directions. The next move is a circular interpolation to the top of the internal circle mill. From this point the complete circle is milled. This circular mill is followed by two moves which mirror the first two moves and position the milling head back at the original position.

G79 J1.Ø F1Ø Says to internally mill a hole which has a radius of 1 unit (plus the cutter radius) at a feedrate of 1Ø units/minute. The positive J value means to do a conventional cut (CW around the hole).

G79 J-.625 F5Ø Says to internally mill a hole which has a radius of .625 units (plus the cutter radius) at a feedrate of 5Ø units/ minute. THe negative value of J means that the cut will be a climb cut (CCW around hole).

CANNED Z CYCLES

The canned Z cycles describe G codes which simplify the operations of hole drilling, spotfacing, tapping, boring, chip breaking, punching and pressing.

Any canned Z cycle can be used to make bolt hole circles with the addition of the A (starting angle), H (number of holes) and R (radius) parameters.

A description of bolt hole circles follows the description of the canned Z cycles.

G80 - Cancel Canned Z Cycles

G80 cancels the present canned Z cycle, G81-G89, and causes the traversal method to revert to the last traversal method used (G0, G1, G2 or G3).

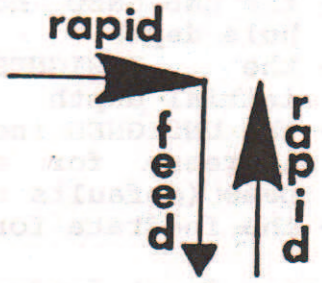
G0 also cancels all canned Z cycles

G81 - Drilling Cycle

This command provides a feedrate down, rapid traverse up cycle, suitable for drilling a series of holes at the same depth

G81 requires an X,Y position, a Z depth and a feedrate.

The Z depth is an incremental UNSIGNED distance from the starting point

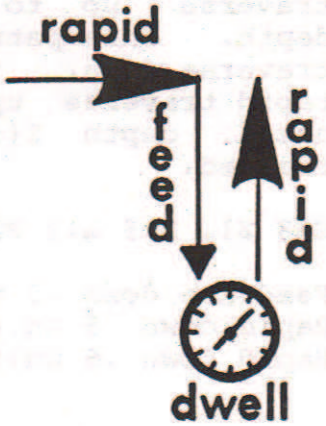



```
G90 Z0. F100        ;position Z and set the feedrate
G81 X4.0 Y5.0 Z1.2  ;drill a hole 1.2 units deep at X=4.0, Y=5.0
X5.0                ;drill a hole 1.2 units deep at X=5.0, Y=5.0
X6.0                ;drill a hole 1.2 units deep at X=6.0, Y=5.0
Y6.0                ;drill a hole 1.2 units deep at X=6.0, Y=6.0
G80                 ;cancel the drilling cycle
```

G82 - Spotfacing Cycle

This command provides a feedrate down, dwell, rapid traverse up cycle, suitable for spotfacing a series of points.

G82 requires an X,Y position, an UNSIGNED incremental axis depth, a feederate, and a dwell time.

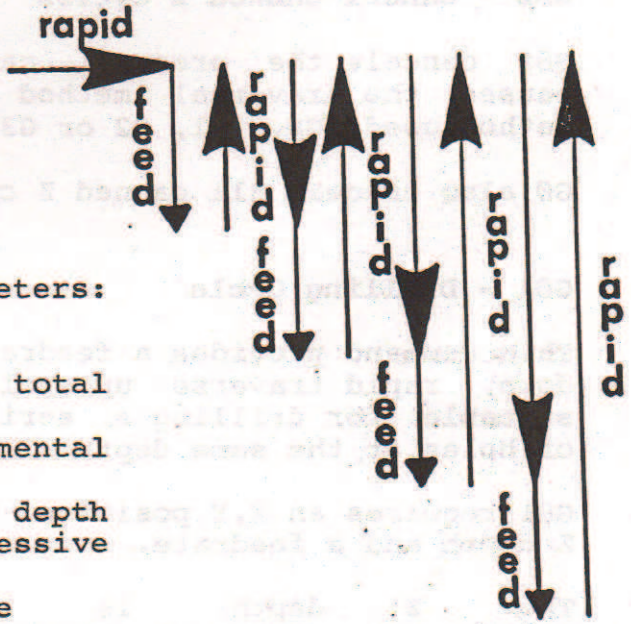


Except for the dwell time, this cycle is identical to G81.

The dwell time must be established with the G4 function.

```
G4/5 G9Ø Z0. F1ØØ  ;set the dwell, position Z, set the feedrate
G82 X4.Ø Y5.Ø Z1.2 ;spotface at X=4.Ø, Y=5.Ø, dwell .Ø5 seconds
X5.Ø               ;spotface at X=5.Ø, Y=5.Ø, dwell .Ø5 seconds
X6.Ø               ;spotface at X=6.Ø, Y=5.Ø, dwell .Ø5 seconds
Y6.Ø               ;spotface at X=6.Ø, Y=6.Ø, dwell .Ø5 seconds
G8Ø                ;cancel the drilling cycle
```

G83 - Deep Hole Drilling Cycle



This command provides a series of rapid traverse down, feedrate down, rapid traverse up cycles suitable for drilling deep holes

G83 requires the following parameters:

X - the X axis move
Y - the Y axis move
Z(1) - the UNSIGNED incremental total hole depth
Z(2) - the UNSIGNED incremental initial depth
Z(3) - the UNSIGNED incremental depth increase for each successive pass (defaults to Z(2))
F - the feedrate for the cycle

The Z axis first feedrates down to a depth equal to Z(2), then rapid traverses out to the initial depth. The Z axis then rapid traverses back down to the last depth followed by a feedrate down an incremental depth of Z(3). This is followed by a rapid traverse up to the initial depth. This pattern of rapid traverse down, feedrate down, rapid traverse up, continues until depth Z(1) has been reached.

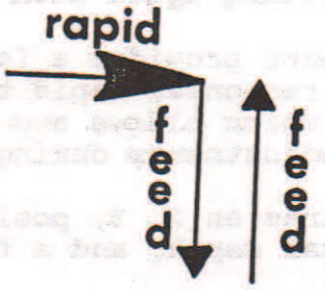
G83 Z1. Z.5 Z.1 F1ØØ Says to:

Feedrate down .5 units, rapid traverse out
Rapid down .5 units, feedrate down .1 units, rapid traverse out
Rapid down .6 units, feedrate down .1 units, rapid traverse out

Rapid down .7 units, feedrate down .1 units, rapid traverse out
Rapid down .8 units, feedrate down .1 units, rapid traverse out
Rapid down .9 units, feedrate down .1 units, rapid traverse out

G84 - Tapping Cycle

This command provides a feedrate down, feedrate out sequence suitable for tapping operations.



G84 requires an X,Y position, an UNSIGNED incremental Z axis depth, and a feedrate.

For tapping operations the feedrate and spindle speed are dependent on the pitch of the thread. These factors must be synchronized to provide for a proper push in and draw out of the tap. The formula needed to synchronize the operations follows:

$$\text{Feedrate units/minute} = \frac{\text{Spindle revolutions/minute}}{\text{Tap threads/unit}}$$

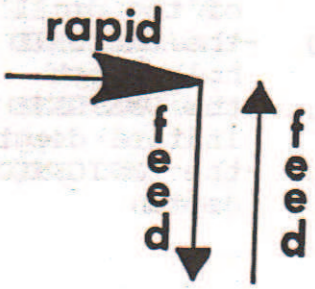
The inputs required for G84 are identical to those required for G81. The only functional difference is that the Z axis pull up movement is performed at the feedrate, not in rapid traverse.

Suppose a 10-32 tap is to be used. If the spindle speed is 160 revolutions/minute then:

$$\text{Feedrate} = \frac{160}{32} = 5 \text{ inches/minute}$$

G85 - Boring Cycle

This command provides a feedrate down, feedrate up cycle, suitable for boring operations.



G85 requires an X,Y position, a Z depth and a feedrate.

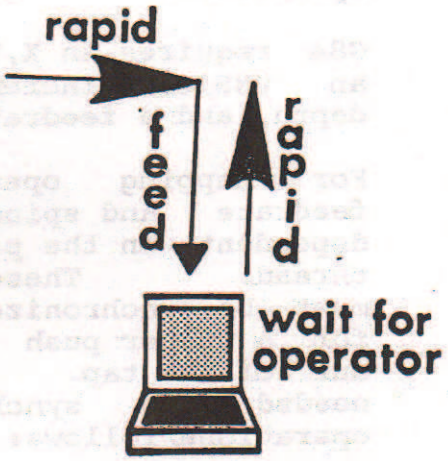
The Z depth is an incremental UNSIGNED distance from the starting point

G85 is identical to G81, except that

the Z axis pull up operation is at the feedrate, not in rapid traverse.

```
G90 Z.1 F120            ;position Z and set the feedrate
G85 X4.0 Y5.0 Z.5       ;bore .5 units deep at X=4 Y=5
X5.0                    ;bore .5 units deep at X=5 Y=5
X6.0                    ;bore .5 units deep at X=6 Y=5
Y6.0                    ;bore .5 units deep at X=6 Y=6
G80                     ;cancel the bore cycle
```

G86 - Drilling Cycle With Operator Interaction



This command provides a feedrate down, operator response, rapid traverse out sequence which allows and operator to perform adjustments during a drill cycle.

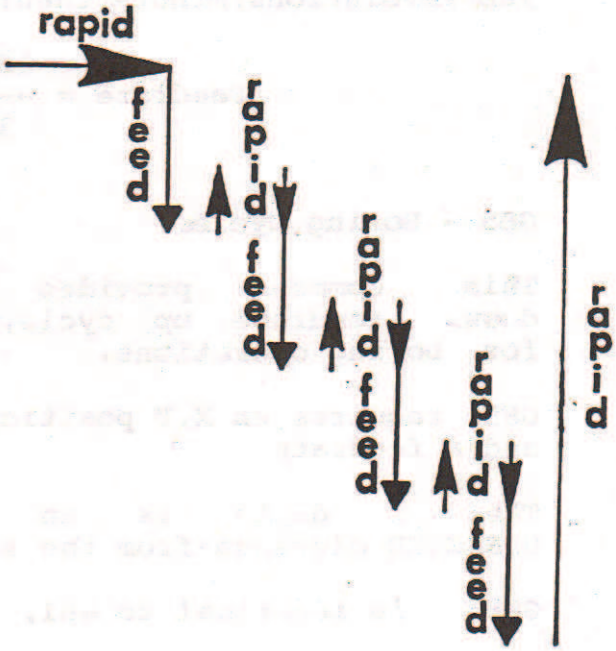
G86 requires an X, Y, position, a Z incremental depth, and a feedrate.

The Z depth is an incremental UNSIGNED distance from the Z axis starting point.

G86 is identical to G81, except for the operator interaction in the middle of the cycle.

```
G4/5 G90 Z0. F100  ;set the dwell, position Z, set feedrate
G86 X4.0 Y5.0 Z1.2 ;drill at X=4 Y=5, wait, rapid out
X5.0               ;drill at X=5 Y=5, wait, rapid out
X6.0               ;drill at X=6 Y=5, wait, rapid out
Y6.0               ;drill at X=6 Y=6, wait, rapid out
G80                ;cancel drill cycle
```

G87 - Chip Breaking Cycle



This command provides a series of feed down, rapid up, rapid down, feed down cycles suitable for chip breaking.

G87 requires the following parameters:

X	-the X axis position of the drill
Y	-the Y axis position of the drill
Z(1)	-the UNSIGNED total hole depth
Z(2)	-the UNSIGNED incremental initial depth
Z(3)	-the UNSIGNED incremental depth

increase for each successive pass
(defaults to Z(2)).
F - the feedrate of the cycle

The Z axis first feeds down to a depth equal to Z(2). The Z axis then rapids up .1 inches, rapids down .05 inches and feeds down .05 inches and feeds down an incremental depth equal to Z(3). This pattern of repeated until the total depth Z(1) is reached.

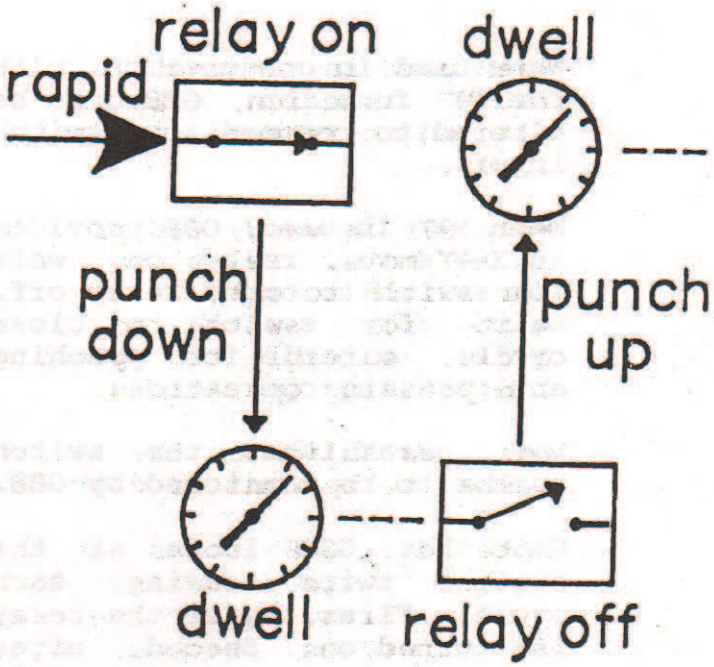
G87 is identical to G83, except that the intermediate rapid traverse moves up .1 inches and then back down .05 inches. The tool does not retract from the hole entirely.

G87 Z1 Z.5 Z.1 F100 Says to:

Feed down .5 units, rapid up .1 units
Feed down .05 units, feed down .1 units, rapid up .1 units
Feed down .05 units, feed down .1 units, rapid up .1 units
Feed down .05 units, feed down .1 units, rapid up .1 units
Feed down .05 units, feed down .1 units, rapid up .1 units
Feed down .05 units, feed down .1 units, rapid up .1 units

G88 - Punch/Press Cycle

This command provides an X-Y move, relay on, delay, relay off, delay cycle suitable for punching and pressing operations.



G88 requires only and X, Y position and a dwell time.

G88 turns the relay connected to the CNC4 R4 output on and off. This is the same output used by M10 and M11. Therefore M10 and M11 must not be used in a program with G88.

The dwell time must be established with the G4 function.

Note the G88 delays twice during each cycle. First,

after the relay is turned on. Second, after the relay is turned off. These delays allow time for the punch or press to be both lowered and raised.

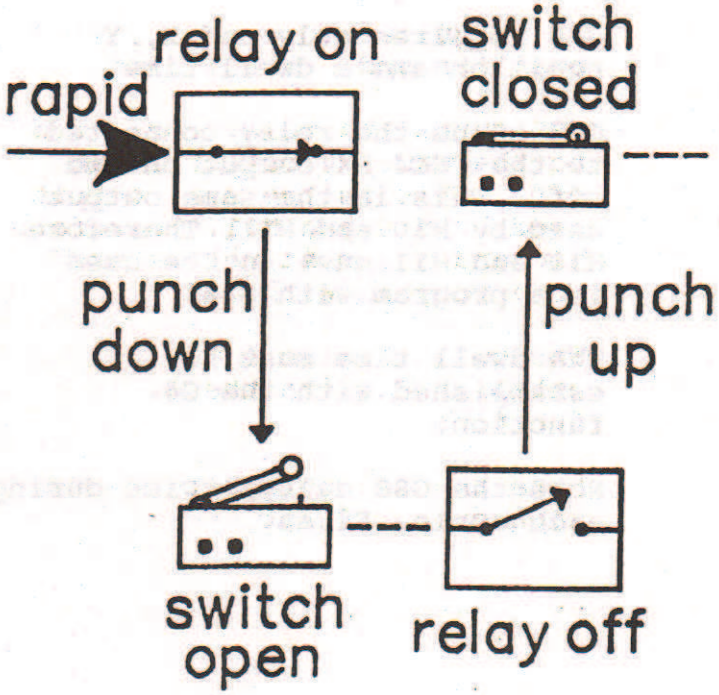
```
G4/5Ø G9Ø          ;set the dwell to .5 seconds
G88 X4.Ø Y5.Ø      ;punch at X=4.Ø, Y=5.Ø
;                  (Rapid traverse to X=4.Ø, Y=5.Ø.       )
;                  (Relay on.  Dwell .5 seconds.          )
;                  (Relay off. Dwell .5 seconds. Continue.)
X5.Ø               ;punch at X=5.Ø, Y=5.Ø
;                  (Rapid traverse to X=5.Ø, Y=5.Ø.       )
;                  (Relay on.  Dwell .5 seconds.          )
;                  (Relay off. Dwell .5 seconds. Continue.)
X6.Ø               ;punch at X=6.Ø, Y=5.Ø
;                  (Rapid traverse to X=6.Ø, Y=5.Ø.       )
;                  (Relay on.  Dwell .5 seconds.          )
;                  (Relay off. Dwell .5 seconds. Continue.)
Y6.Ø               ;punch at X=6.Ø, Y=6.Ø
;                  (Rapid traverse to X=6.Ø, Y=5.Ø.       )
;                  (Relay on.  Dwell .5 seconds.          )
;                  (Relay off. Dwell .5 seconds. Continue.)
G8Ø                ;cancel the punch/press cycle
```

When used in conjunction with the M97 function, G88 can be altered to respond to a switch input.

When M97 is used, G88 provides an X-Y move, relay on, wait for switch to open, relay off, wait for switch to close cycle, suitable for punching and pressing operations.

M97 establishes the switch number to be monitored by G88.

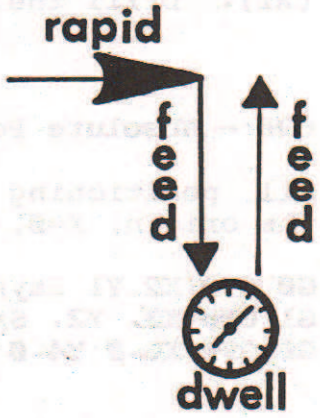
Note that G88 looks at the switch twice during each cycle. First, after the relay is turned on. Second, after the relay is turned off. This ensures that the punch/press is lowered when the relay is turned on and is raised when the relay is turned off, before any X-Y motion occurs.



```
M97 /2 G9Ø          ;use switch input number 2
 G88 X4.Ø Y5.Ø      ;punch at X=4.Ø, Y=5.Ø
 ;                  (Rapid traverse to X=4.Ø, Y=5.Ø.          )
 ;                  (Relay on.  Wait for switch to open.      )
 ;                  (Relay off. Wait for switch closure. Continue.)
 X5.Ø               ;punch at X=5.Ø, Y=5.Ø
 ;                  (Rapid traverse to X=5.Ø, Y=5.Ø.          )
 ;                  (Relay on.  Wait for switch to open.      )
 ;                  (Relay off. Wait for switch closure. Continue.)
 X6.Ø               ;punch at X=6.Ø, Y=5.Ø
 ;                  (Rapid traverse to X=6.Ø, Y=5.Ø.          )
 ;                  (Relay on.  Wait for switch to open.      )
 ;                  (Relay off. Wait for switch closure. Continue.)
 Y6.Ø               ;punch at X=6.Ø, Y=6.Ø
 ;                  (Rapid traverse to X=6.Ø, Y=5.Ø.          )
 ;                  (Relay on.  Wait for switch to open.      )
 ;                  (Relay off. Wait for switch closure. Continue.)
 G8Ø                ;cancel the punch/press cycle
 M96                ;restore the default continuation method
```

G89 - Boring Cycle with Dwell Time Delay

This command provides a feedrate down, dwell, feedrate up cycle, suitable for boring operations.



G89 requires an X,Y position, a Z depth, a feedrate and a dwell time.

The Z depth is an incremental UNSIGNED distance from the starting point

Except for the dwell time, this cycle is identical to G85

The dwell time must be established with the G4 function

```
G4/5 G9Ø Z.1 F12Ø   ;set the dwell, position Z, set the feedrate
G89 X4.Ø Y5.Ø Z.5   ;bore at X=4.Ø, Y=5.Ø, dwell .Ø5 seconds
 X5.Ø               ;bore at X=5.Ø, Y=5.Ø, dwell .Ø5 seconds
 X6.Ø               ;bore at X=6.Ø, Y=5.Ø, dwell .Ø5 seconds
 Y6.Ø               ;bore at X=6.Ø, Y=6.Ø, dwell .Ø5 seconds
 G8Ø                ;cancel the boring cycle
```

BOLT HOLE CIRCLES

Any canned Z cycle (G81-G89) can be used to make a bolt hole circle with the addition of the A (starting angle), H (number of holes) and R (radius) parameters.

The bolt hole circle starting angle (A) and the bolt hole circle radius (R) define the position of the first hole in the bolt hole circle, with respect to the bolt hole center point (X,Y).

The number of holes (H) and the bolt hole circle radius (R) define the positions of the remaining holes.

The depth of the bolt holes are defined by the Z parameter(s).

The feedrate(s) are defined by the F parameter(s).

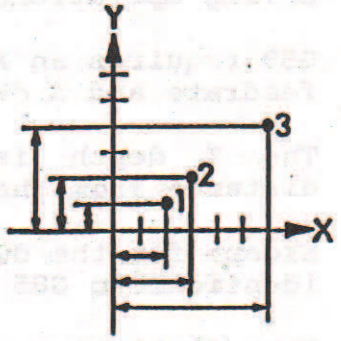
All holes in a bolt hole circle are equally spaced around the center point (X,Y).

GØ G9Ø ZØ F4
G81 X-2 Y3 Z.7 H4 A6Ø R1
Says to setup using absolute positioning (G9Ø). Rapid traverse to the Z axis Ø point. Define the feedrate as 4 inches/minute (F4). Establish a drilling cycle with the center at X = -2, Y = 3 and a drilling depth of .7 inches. Define a bolt hole circle with a total of 4 holes (H4), a starting angle of 6Ø degrees (A6Ø) and a radius of 1 inch (R1). Drill the 4 holes in the bolt hole circle.

G9Ø - Absolute Positioning (default)

All positioning is performed relative to the origin, X=Ø, Y=Ø and Z=Ø.

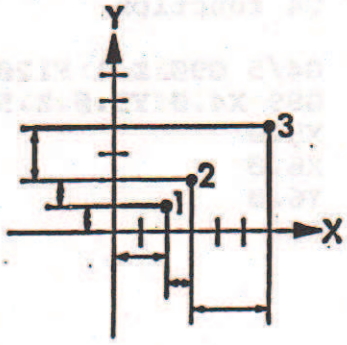
GØ G9Ø X2 Y1 Says move to X=2.Ø, Y=1.Ø
G1 G9Ø X3. Y2. Says move to X=3.Ø, Y=2.Ø
GØ G9Ø X6.Ø Y4.Ø Says move to X=6.Ø, Y=4.Ø



G91 - Incremental Positioning

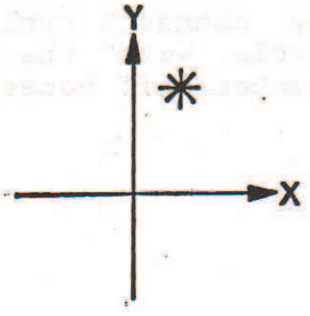
All positioning is performed relative to the last point.

GØ G9Ø XØ YØ Says move to the origin
GØ G91 X2. Y1. Says move to X=2.Ø, Y=1.Ø
G1 G91 X1.Ø Y1.Ø Says move to X=3.Ø, Y=2.Ø
G1 G91 X3 Y2 Says move to X=6.Ø, Y=4.Ø



G92 - Preset Absolute Position

The absolute position of the specified axis is set equal to the value defined by G92.



If a tool is loaded and a Z axis preset is requested, the tool length is subtracted from the preset value. This feature allows the Z axis preset to be performed without regard to the present tool.

G92 XØ YØ ZØ Says the present position the X, Y, and Z axis is now X, Y, and Z zero.

G92 X1 Y1 Z.25 Says that the present position of X is now 1, the present position of Y is now 1, and the present position of Z is now .25.

A G92 typed in on the keyboard while in the Run Immediate mode is a very convenient way of zeroing the axes.

Miscellaneous Functions

M functions perform specialized actions to enhance the CNC capabilities.

Some M function actions take place before other operations in CNC program lines. Some M function actions take place after other operations in CNC program lines. It depends on the individual M function definitions.

With the exception of M97, M98, and M99, each M function must appear in a separate CNC program line from all other M functions.

M3, M4, M5, M8, M9, M10, and M11 activate external relays. External devices connected to the switch contacts of these relays can be turned on or off under program control.

M0 - Program Stop for Inspection

If an M0 is contained in a line which also has X, Y, or Z movement, the X, Y, or Z axis movement is completed first. At the completion of the axis motion, the following message is output to the CRT screen:

"Program Stop for Inspection, Press START button to Continue"

When the message is displayed, the operator performs any inspections (and alterations) required to the present machine structure. When the inspection is finished, the operator presses the START button on the front panel to resume the program.

N500 X3. Y2.6 Z-.5 M0
X goes to 3.0, Y goes to 2.6 and Z goes to -0.5. The following message is output:

"Program Stop for Inspection, Press START button to Continue"

When the operator presses the START button, operations continue.

M0 can be altered using M97 or M98 to respond to a switch toggle or a time out.

M97/7 M0
Says continue on switch input 7 toggle. When this line is executed, the following message is output to the screen:

"Program Stop for Inspection, Awaiting Switch Input to Continue."

When the switch connected to input 7 is toggled, the program continues.

G4/100 M98 M0
Continue after 1 second. When this line is executed, the CNC4 pauses and outputs the following message:

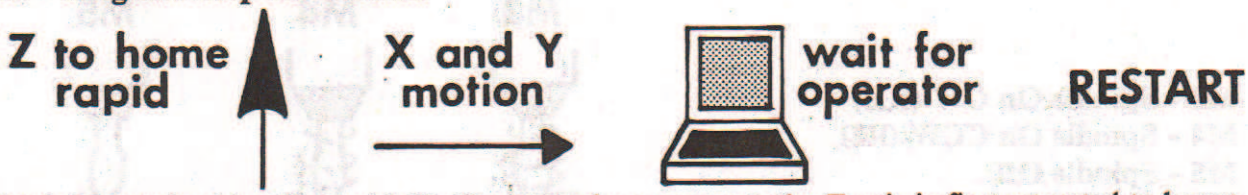
"Program Stop for Inspection, Awaiting Timeout to Continue."

When 1 second passes, the program continues.

M1 - Optional Program Stop for Inspection

M1 is identical to M0, except that the operation of M1 is dependent on the setting of the optional stop setup parameter. Optional stop is set with setup menu option 4. If optional stop is OFF, M1 is ignored. If optional stop is ON then the M0 and M1 functions are equivalent.

M2 - Program Stop for Restart



If M2 is contained in a line with X, Y, or Z axis movement, the Z axis is first retracted to home with a rapid traverse. Any required X and Y axis motions are performed. At the completion of the X and Y motion, the following message is output to the CRT screen:

"Program Stop, Press START button for Program Restart"

When the operator presses the START button, the program cycles back to the beginning.

N7000 X0. Y0. M2
Says that the Z axis will first be retracted to home. The X and Y axes are then set in motion to traverse to their 0. points. When the X and Y motion is completed, the following message is output:

"Program Stop, Press START button for Program Restart"

When the operator presses the START button, the CNC program restarts at the beginning.

M2 can be altered using M97 or M98 to respond to a switch toggle or a time out.

M97/5 M2
M96
Retract the Z axis to home and outputs the following message:

"Program Stop, Awaiting Switch Input for Program Restart."

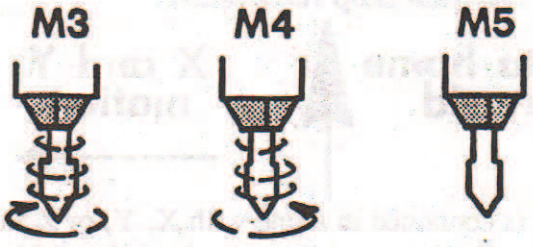
When the switch input number 5 is toggled, the program restarts.

G4/150 M98 M2
M96
Retract the Z axis to home and outputs the following message:

"Program Stop, Awaiting Timeout for Program Restart."

After 1.5 seconds has passed, the program restarts.

M3 - Spindle On CW (R1)
M4 - Spindle On CCW (R2)
M5 - Spindle Off



M3 and M4 are designed to turn spindle power on; M5 is designed to turn the spindle power off. M3 turns on the relay connected to auxiliary output pin R1 and turns off the relay connected to auxiliary output pin R2. If auxiliary output pin 1 is connected to the spindle clockwise motor power relay, then M3 turns the spindle clockwise.

M4 turns on the relay connected to auxiliary output pin R2, and turns off the relay connected to auxiliary output pin R1. If auxiliary output pin 2 is connected to the spindle counterclockwise motor power relay, then M4 turns the spindle counterclockwise.

M5 turns off the relay connected to both auxiliary output pins R1 and R2. If the spindle motor is connected to the M3 and M4 relays, then M5 will turn the spindle motor off. Relay connections are documented in the hookup instruction section of this manual. The spindle power relay(s) should be hooked up to auxiliary output pins R1 and/or R2 so that M3, M4 and M5 operate as designed.

M3
G0 X0 Y0 Z0
G1 Z-.1 F140
Turns the spindle in the clockwise direction and starts to mill.

G0 X0 Y0 Z0 M5
M6
Move X,Y and Z to their zero position and turns off the spindle, and then re-start the program from the beginning.

M3, M4, and M5 can be altered using M97, M98, or M99 to continue after a switch toggle, a time out, or an operator response.

M97/5 M3
M96
Turns the spindle on and outputs the following message:

"Program Stop for Inspection, Awaiting Switch Input to Continue."

When switch input number 5 is toggled, the program continues.

G4/100 M98 M4
M96
Turns the spindle on and outputs the following message:

"Program Stop for Inspection, Awaiting Timeout to Continue."

After 1 second passes, the program continues.

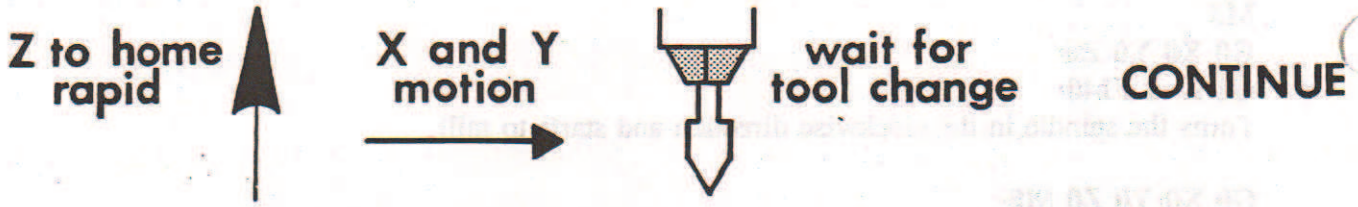
M99 M5
M96
Turns the spindle off and outputs the following message:

"Program Stop for Inspection, Press START button to Continue"

When the START button is pressed, the program continues.

!!! IMPORTANT !!!
ALWAYS CONNECT THE SPINDLE MOTOR TO THE CNC4 USING RELAYS. THIS WILL CAUSE THE SPINDLE TO STOP IF THE FRONT PANEL EMERGENCY STOP SWITCH IS DEPRESSED. IT ALSO ALLOWS FOR CONVENIENT OPERATION OF THE SPINDLE UNDER PROGRAM CONTROL.

M6 - Tool Change



M6 is used to select the tool being used. M6 is used with the tool select T, and the spindle speed select S commands.

A line with an M6 causes the Z axis to be retracted to the Z axis HOME position first with a rapid traverse. Any programmed X and Y moves are then performed. An automaatic M5 (spindle off) is issued. At the completion of any X or Y axis movement, the tool number and spindle speed parameters are referenced and a message of the following format is displayed on the screen:

"Tool:5 Length: 5.700 Diameter: 1.2500 Spindle Speed: 250.00"

"Insert Tool: Hit START to continue"

When the specified tool has been inserted, the operator presses the START button and the program continues.

When the START button is pressed, the Z axis position is adjusted by the length of the new tool and the program continues.

N275 T5/-5.75/1.25 S250 M6
Causes the Z axis to retract to Z axis HOME. No X or Y motion is required. The following messages are displayed on the screen:

"Tool: 5 Length: 5.700 Diameter: 1.2500 Spindle Speed: 250.00"

"Insert tool: Hit START to continue"

When the START button is pressed, the Z axis is adjusted to account for the new tool length and the program continues.

NOTE: the Z axis does not physically move to account for the new tool length. The X axis position is changed internally and displayed on the screen. When the X axis is moved again after the tool change, the tool length is then compensated for.

M6 can be altered by using M97 to respond to a switch toggle:

N275 T5/5.75/1.25 S250 M97/6 M6
M96
The Z axis is first retracted to the Z HOME position. No X or Y motion occurs. The following messages are displayed on the screen:

"Tool: 5 Length: 5.7500 Diameter: 1.2500 Spindle Speed: 250.00"

"Program Stop for Inspection, Awaiting Switch Input to Continue."

When switch input number 6 is toggled, the program continues.

REMEMBER !

Since an automatic M5 (spindle off) and M9 (coolant off) are issued when an M6 is used, you must issue an M3 or M4 (spindle on CW, CCW) and an M8 (coolant on) immediately after an M6 if the spindle and coolant pump are connected to M - function relays, and you want the spindle and coolant pump to turn back on !

M8 - Coolant On
M9 - Coolant Off

M8 is designed to turn coolant spray on; M9 is designed to turn the coolant spray off.

M8 and M9 turn on and off the relay connected to auxiliary output pin R3. If relay output pin R3 is connected to the coolant spray power relay, then M8 and M9 turn the coolant spray on and off.

Relay connections are documented in the hookup instruction section of this manual. The coolant spray relay should be hooked up to I/O board 1, between terminal R3 and ground (G).

M3
X0 Y0 Z0 M8
G1 Z-.1 F140
Turn the spindle on, Position to 0, turn the coolant spray on, and start to mill.

G0 X0 Y0 Z0 M9
M5
M2
Move to the final position and turn off the coolant spray. Turn off the spindle and restart.

M8 and M9 can be altered using M97, M98, or M99 to continue after a switch toggle, a time out or an operator response:

M97/1 M8
M96
Turn coolant spray on, continue on switch input 1 toggle.

M98 G4/300 M8
M96
Turn the coolant spray on and output the following message:

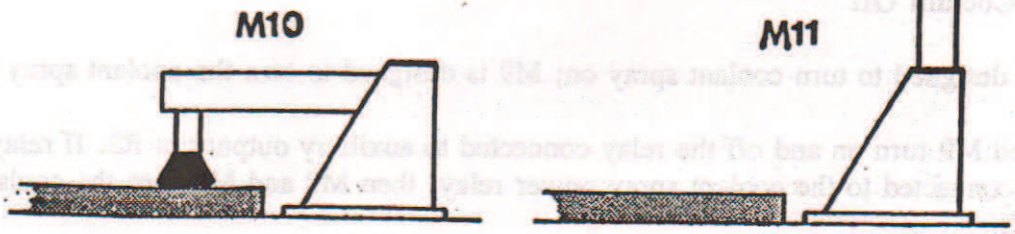
"Program Stop for Inspection, Awaiting Timeout to Continue."

After 3 seconds have passed, the program continues.

M99 M8
M96
Turn coolant spray on and output the following message:

"Program Stop for Inspection, Press START button to Continue"

M10 - Clamp
M11 - Unclamp



M10 is designed to turn the clamp relay on; M11 is designed to turn the clamp relay off.

M10 and M11 turn on and off the relay connected to auxiliary output pin R4. If auxiliary output pin R4 is connected to the clamp power relay, then M10 and M11 perform the clamp and unclamp functions.

Note that since G88 also utilizes auxiliary output pin R4 for punching and pressing cycles, M10 and M11 cannot be used in conjunction with G88.

Relay connections are documented in the hookup instruction section of this manual. The clamp relay should be hooked up to the first I/O board between terminal R4 and ground (G).

: * LAY THE WORKPIECE DOWN *****
M0
M10
G0 X0 Y0 Z0
First outputs the following messages:

<u>* LAY THE WORKPIECE DOWN ***</u>**

"Program Stop for Inspection, Press START button to Continue"

When the START button is depressed, clamp the workpiece. Move to the initial position and start to mill.

G0 X0 Y0 M25
M11
M2
Move to the X,Y zero position and retract the Z axis to home. Unclamp the workpiece and restart the program from the beginning.

M10 and M11 can be altered using M97, M98, or M99 to continue after a switch toggle, a time out or an operator response:

M97/4 M10
M96
Clamps the workpiece and outputs the following message:

"Program Stop for Inspection, Awaiting Switch Input to Continue."

When switch input 4 is toggled, the program continues.

G4/250 M98 M11
M96
Unclamps the workpiece and outputs the following message:

"Program Stop for Inspection, Awaiting Timeout to Continue."

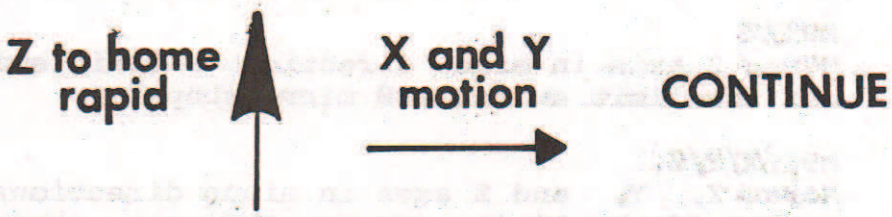
After 2.5 seconds pass, the program continues.

M99 M10
M96
Clamps the workpiece and outputs the following message:

"Program Stop for Inspection, Press START button to continue"

When the START button is pressed, the program continues.

M25 - Z Axis to Home



M25 retracts the Z axis to the home position. The Z axis home position is defined by setup menu option 12 or M26.

If a line contains an M25, the Z axis is first retracted to home with a rapid traverse. Any required X and Y axis motions are then performed.

G0 X0 Y0 M25
Retracts the Z axis to the home position and then rapid traverses (G0) the X and Y axes to the 0 positions.

M26 - Set Z Axis Home position

M26 sets the defined z axis position as the z axis home position.

M26 Z5
Moves Z axis to the absolute position Z 5" and makes that position the Z axis home position.

M26
Makes the present Z axis position the Z axis home position.

M91 - Go to minus limit switch
M92 - Go to plus limit switch
M93 - Release power to axis

M91, M92, and M93 must be on separate CNC program lines.

M91, M92 and M93 are most useful for traveling to limits and backing off under program control, or releasing power from one or all of the axis motors.

M91/X
Moves X axis in minus direction to limit switch and then backs off limit switch 160 micro steps.

M91/Y
Moves Y axis in minus direction to limit switch and then backs off limit switch 160 micro steps.

M91/Z
Moves Z axis in minus direction to limit switch and then backs off the limit switch 160 micro steps.

M91/X/Y/Z
Moves X, Y, and Z axes in minus directions to limits and then backs off the limit switches 160 micro steps.

M92/X
Moves X axis in plus direction to limit switch and then backs off limit switch 160 micro steps.

M92/Y
Moves Y axis in plus direction to limit switch and then backs off limit switch 160 micro steps.

M92/Z
Moves Z axis in plus direction to limit switch and then backs off the limit switch 160 micro steps.

M92/X/Y/Z
Moves X, Y, and Z axes in plus direction to limits and then backs off the limit switches 160 micro steps.

M93/X
Release power from X axis

M93/Y
Release power from Y axis

M93/Z
Release power from Z axis

M93/X/Y/Z
Release power from X, Y, and Z axes

M94 - Direct Output to Auxiliary Outputs - Power on
M95 - Direct Output to Auxiliary Outputs - Power off

M94 - M95 are most useful for directly controlling relays connected to the auxiliary outputs.

M94 turns the specified auxiliary outputs on; M95 turns the specified auxiliary outputs off

Relay connections are documented in the hookup instruction section of this manual.

M94	Turn on all relays connected to the auxiliary outputs,
M94/8	Turn on relay number 8.
M94/1/2/4/8	Turn on relays 1,2,4,and 8.
M95	Turn off all relay outputs.
M95/8	Turn off relay number 8.
M95/1/2/4/8	Turn off relays 1,2,4 and 8.

M94 and M95 can be used with M97, M98, and M99 functions to directly control devices external to normal X, Y and Z operations.

The normal operation of the M94 and M95 function is to output the specified power settings to the auxiliary outputs and then continue to the next program line. The "normal" operation can be augmented or altered in a number of ways.

For example:

If tool or spindle speed information is contained in the same line as the M94 function, a tool and spindle speed information message is issued before the power settings are output.

T7/3.25/1 S400 M94/2

Outputs the following message:

"Tool: 7 Length: 3.2500 Diameter: 1.000 Spindle Speed: 400.00"

After the message is output, the power setting is sent to the auxiliary outputs and the program continues.

M94/1
M98 G4/150 M0
M95/1
M96
Turns relay number 1 on and outputs yhe following message:

"Program Stop for Inspection. Awaiting Timeout to Continue"

After 1.5 seconds, Relay 1 is turned off.

M96 - Cancels M97, M98, or M99

Notice that in all of the previous examples using M97, M98, or M99 that an M96 was issued after the desired result was obtained. This prevents confusion when more than one of the three methods of interruption and continuation are being utilized in the same CNC program.

M96
Restores "normal" continuation conditions.

M97/1 G82 Z1 F40 ;Spotface using switch 1
M96 M2
Lowers the Z axis and outputs the message:

"Z Axis Lowered, Awaiting Switch Input to Continue."

When switch 1 toggles, the Z axis raises. "Normal" continuations are restored and the following message is output:

"Program Stop, Press START button for Program Restart."

When the START button is pressed, the program restarts. This is the "normal" operating procedure for M2.

M97/3 M0
M96 G4/2 G82 Z.5 F80
Stops the program and outputs the message:

"Program Stop for Inspection, Awaiting Switch Input to Continue."

When switch 3 is toggled, the program continues. "Normal" continuations are restored:

The Z axis is lowered .5 units and then raised after a dwell time of .02 seconds. This is the "normal" operating procedure for G82.

M97 - Continuation on switch input (1-16)
M97 allows CNC programs to be driven by switch inputs. M97, once issued, alters the defined continuation method for ALL program interruptions until an M96 is encountered.

M97 can alter the continuation mode for canned Z cycles G82, G86, G88, and G89 as well as miscellaneous functions M0, M1, M2, M3, M4, M5, M6, M8, M9, M10, M11, M91, M92, M93, and M94.

The number specified with an M97 refers to the switch input number.

M97/1 through M97/16 say continue on input from switches 1 through 16.

M97/1 G82 Z1 F40 ;Spotface using switch 1
M96
Lowers the Z axis and outputs the message:

"Z Axis Lowered, Awaiting Switch Input to Continue."

When switch 1 toggles, the Z axis raises. This is different from the "normal" G82 continuation after a dwell time.

N750 M97/7 M2 ;Restart on switch 7 input
M96
Outputs the message:

"Program Stop, Awaiting Switch Input for Program Restart."

When switch 7 toggles, restart the program. This is different from the "normal" M2 restart on operator response.

M97/3 M0 ;Program stop, continue on switch 3 toggle
M96
Stops the program and outputs the message:

"Program Stop for Inspection, Awaiting Switch Input to Continue."

When switch 3 is toggled, the program continues.

M98 - Continuation on dwell time expiration

M98 allows CNC programs to continue after a specific dwell time. M98, once issued, alters the defined continuation method for ALL program interruptions until an M96 is encountered.

M98 can alter the continuation mode for canned Z cycles G82, G86, G88, and G89 as well as miscellaneous functions M0, M1, M2, M3, M4, M5, M6, M8, M9, M10, M11, M91, M92, M93, and M94.

M98 is used with G4 to specify the length of the timed continuation.

G4/150 M98 M2
M96
Retract the Z axis to home and output the following message:

"Program Stop, Awaiting Timeout for Program Restart."

When 1.5 seconds pass, the program restarts. This is different from the "normal" restart on operator response.

G4/100 M98 T7/3.0/1.5 S300. M94/10
M96
Outputs power to relay 10 and then issues the messages:

"Tool: 7 Length: 3.0000 Diameter: 1.5000 Spindle Speed: 300.0000"

"Program Stop for Inspection, Awaiting Timeout to Continue."

After a 1 second dwell time expires, program operations continue. This type of operation could be used to control an auxiliary device.

M99 - Continuation on operator response

M99 allows CNC programs to continue after operator response. In this case, the operator must either press the <ENTER> key or the START button. M99, once issued, alters the defined continuation method for ALL program interruptions until an M96 is encountered.

M99 can alter the continuation mode for canned Z cycles G82, G86, G88, and G89 as well as miscellaneous functions M0, M1, M2, M3, M4, M5, M6, M8, M9, M10, M11, M91, M92, M93, and M94.

M99 G82 Z1.0 F40
M96
Lowers the Z axis 1 unit and outputs the message:

"Z Axis Lowered, Press START button to continue"

When the START button is pressed, the Z axis raises. This is different from the "normal" G82 continuation after a dwell time.

M99 M3
M96
Turns the spindle on and outputs the following message:

"Program Stop for Inspection, Press START button to continue"

When the START button is pressed the program continues. This is different from the "normal" M3 continuation without intervention.

M99 T7/3.0/1.5 S300. M94/10
M96
Outputs power to relay 10 and then issues the messages:

"Tool: 7 Length: 3.0000 Diameter: 1.5000 Spindle Speed: 300.0000"

"Program Stop for Inspection, Press START button to continue"

When the START button is depressed, program operations continue. This type of operation could be used to control an external auxiliary device.

REMEMBER

The M97, M98, and M99 functions alter the defined continuation method for the program line in which it is located and all succeeding program lines. Therefore, in most cases, each time an M97, M98, or M99 is used, an M96 should be specified in the next program line to return all continuation conditions back to the default settings.

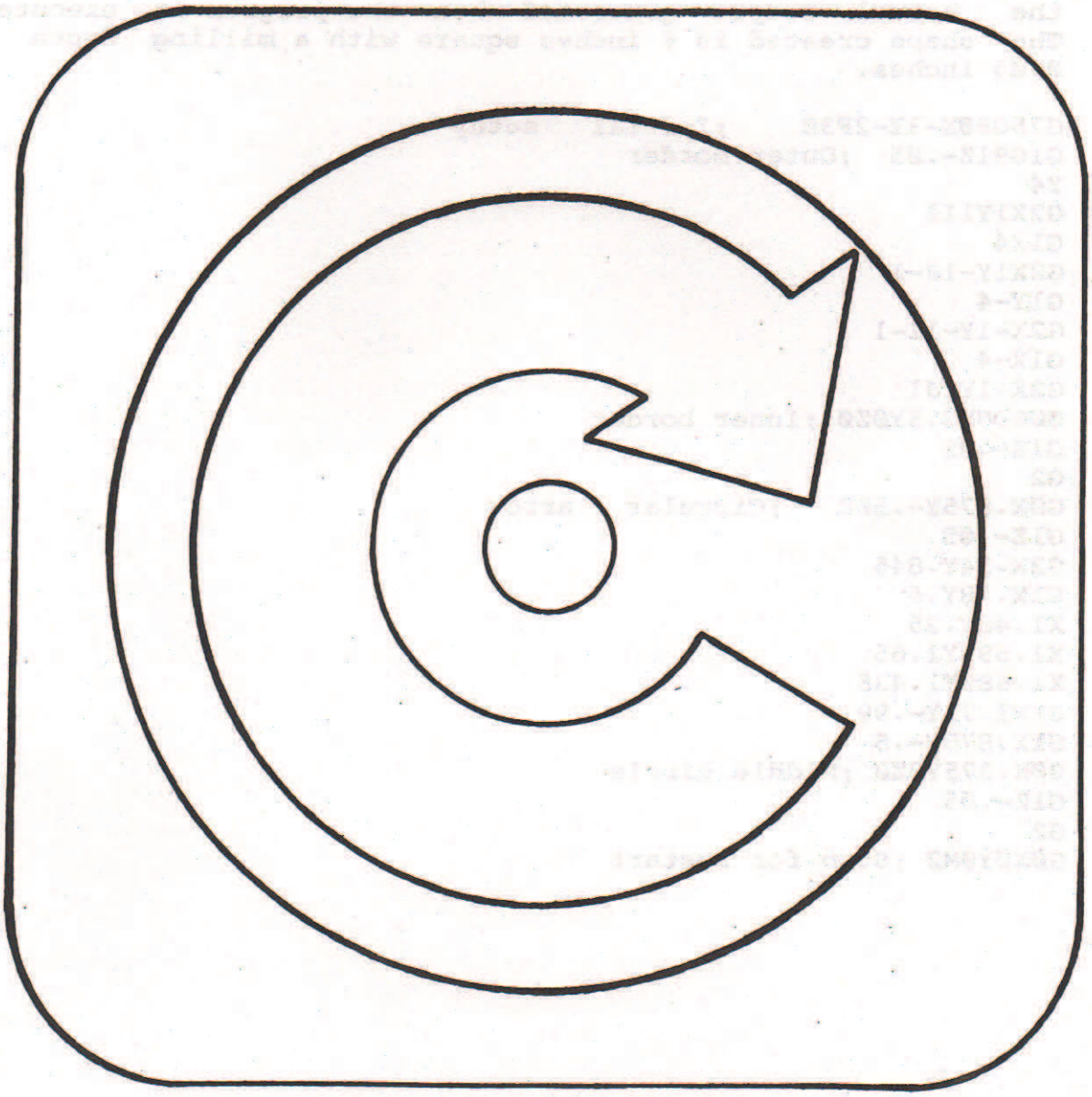
SAMPLE CNC PROGRAMS

The following examples show how the Centroid CNC4 is programmed to do various operations.

SIMPLE MILLING

The following sample program mills out a simple shape. Following the sample program is a representation of the actual output generated when the program is executed. The shape created is 6 inches square with a milling depth of Ø.Ø5 inches.

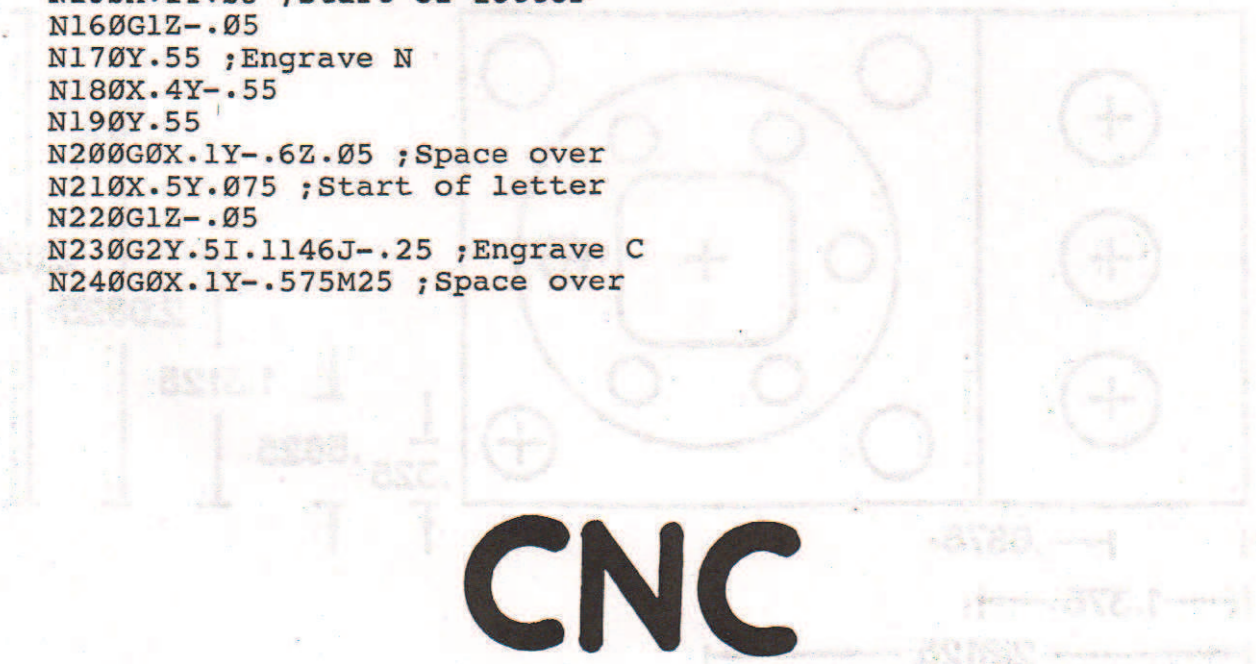
```
G75G90X-3Y-2F30    ;Initial   setup
G1G91Z-.Ø5  ;Outer border
Y4
G2X1Y1I1
G1X4
G2X1Y-1J-1
G1Y-4
G2X-1Y-1I-1
G1X-4
G2X-1Y1J1
GØG9ØX2.5YØZØ ;Inner border
G1Z-.Ø5
G2
GØX.875Y-.5ZØ   ;Circular   arrow
G1Z-.Ø5
G2X.54Y.845
G1X.18Y.6
X1.48Y.25
X1.695Y1.65
X1.385Y1.435
G3X1.73Y-.99
G1X.875Y-.5
GØX.375YØZØ ;Middle circle
G1Z-.Ø5
G2
GØXØYØM2 ;Stop for restart
```



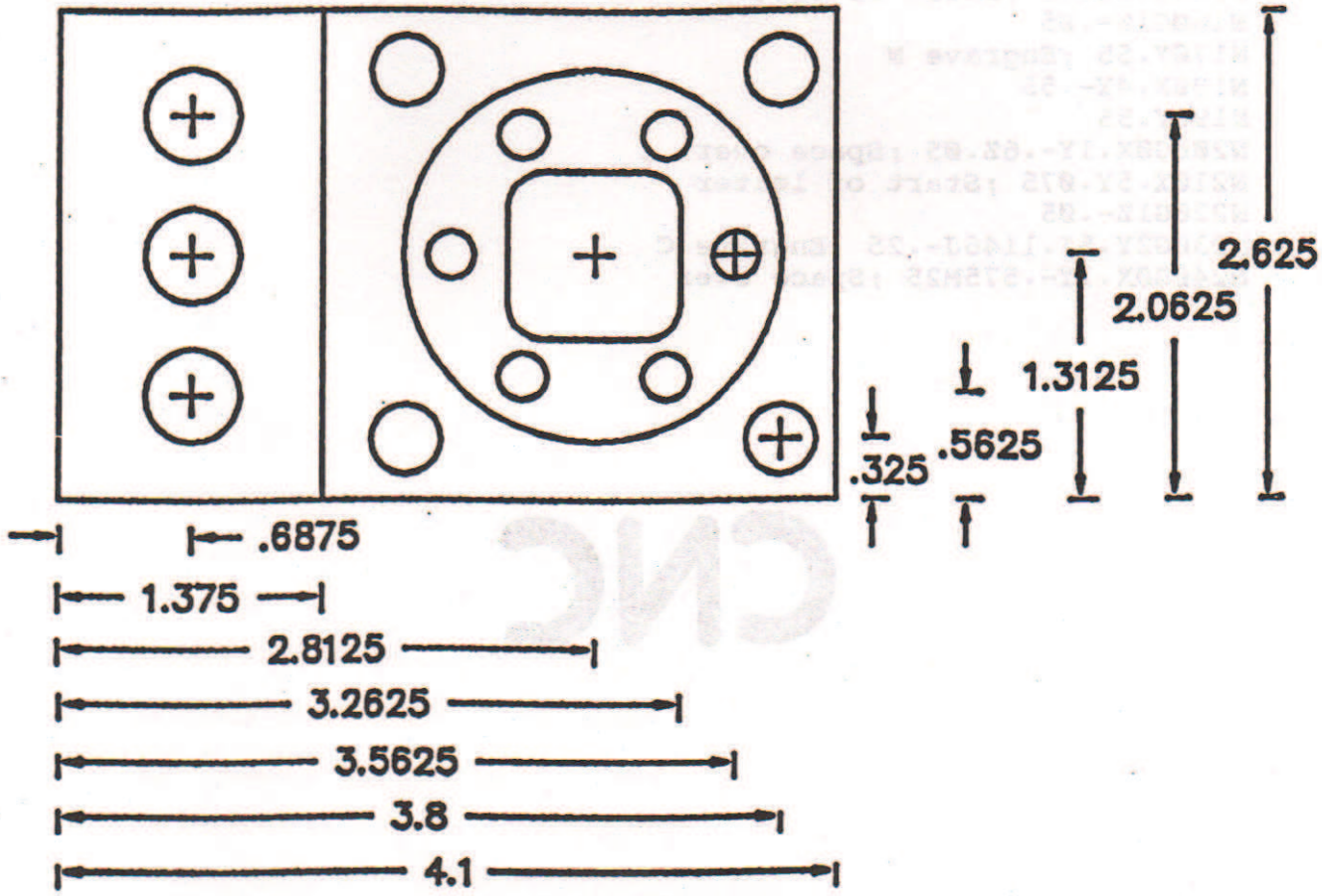
"CNC" ENGRAVING SAMPLE

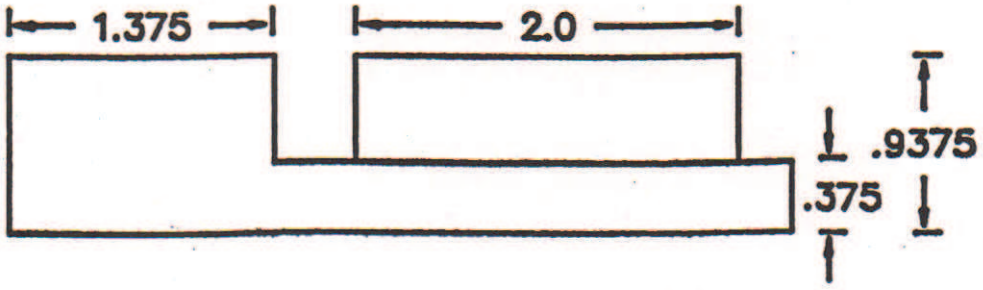
The following CNC program shows how to engrave the letters "CNC". The comments on the program lines indicate the actions in progress. Following the sample program is a representation of the actual output generated when this program is executed.

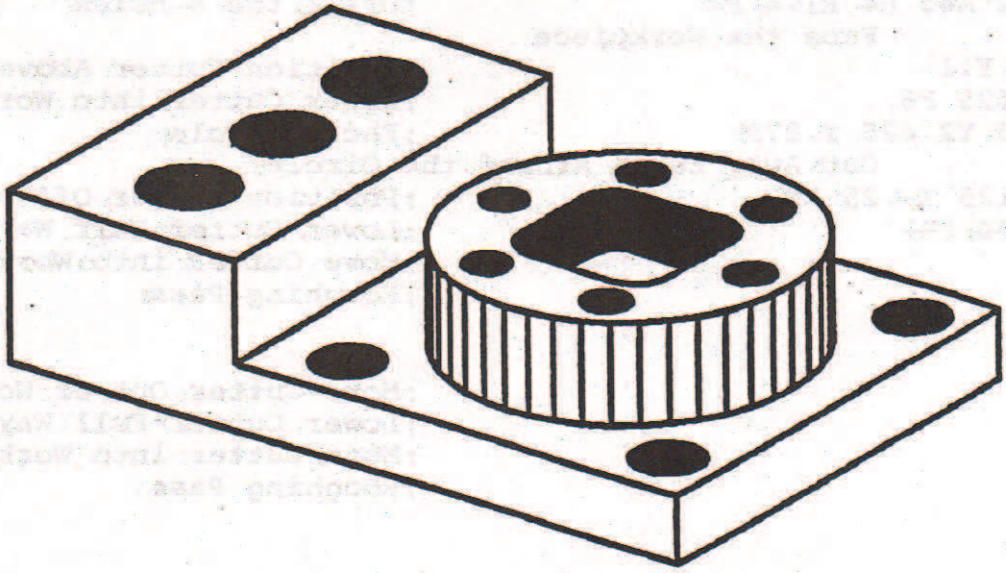
```
N100G0G91G75F30  ;Initial setup
N110X.5Y.075 ;Start of letter
N120G1Z-.05
N130G2Y.5I.1146J-.25 ;Engrave C
N140G0X.1Y-.575Z.05 ;Space over
N150X.1Y.05 ;Start of letter
N160G1Z-.05
N170Y.55 ;Engrave N
N180X.4Y-.55
N190Y.55
N200G0X.1Y-.6Z.05 ;Space over
N210X.5Y.075 ;Start of letter
N220G1Z-.05
N230G2Y.5I.1146J-.25 ;Engrave C
N240G0X.1Y-.575M25 ;Space over
```



The following sample demonstrates the milling of a more complex shape. The original piece of metal is cut to a size 4.1 inches long, 2.625 inches wide and 1 inch deep. The milling operation requires three tools. A 1/4" diameter drill, a 3/8" diameter milling cutter and a 1/2" diameter drill. The following illustration shows the shape to be milled along with the required dimensions. After the dimensioned illustration is a 3D representation of the completed workpiece. Finally, following the 3D representation is the program to mill the required piece.

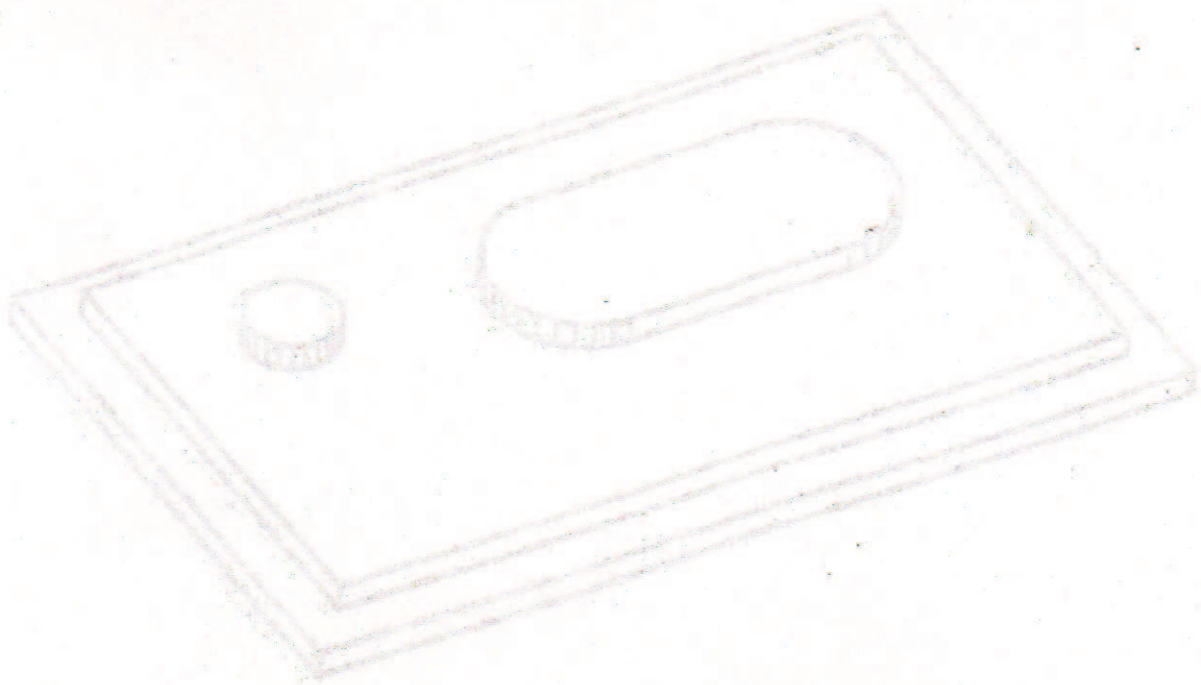






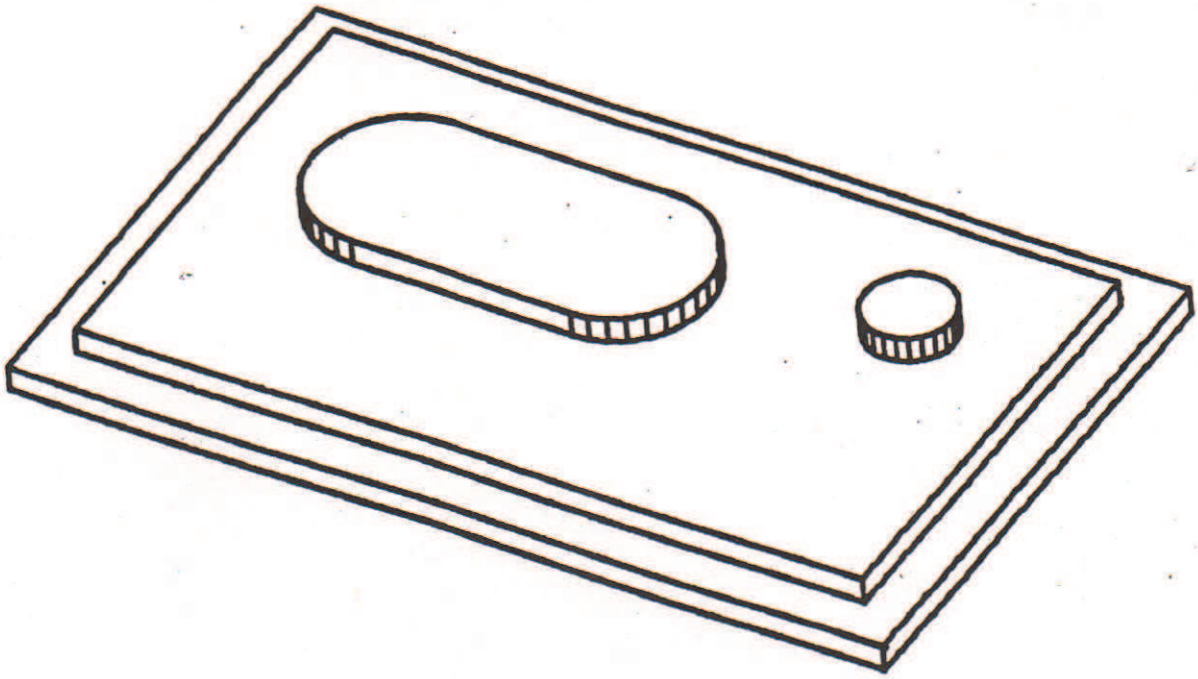
```
:              Initial Tool Setup
G9Ø  F6                                  ;Absolute Posit. 6 IPM
: Insert the 1/4" Diameter Drill
M5                                       ;Spindle Off
GØ X-2 YØ T1/-2.25/.25 M6                ;Tool Change
M3                                       ;Spindle On
:              Center Drill All Holes
GØ Z.Ø625
G81 X.6875 Y.5625 Z1.2 F4                ;3 Vertical 1/2" Holes
Y1.3125
Y2.Ø625
X2.8125 Y1.3125 Z.45                     ;Pocket Center
Z.5 AØ H6 R.75                           ;6 Hole Bolt Hole Circle
Z1.2 A45 H4 R1.4                         ;4 Hole Bolt Hole CIrcle
:              Tool Change to Milling Cutter
: Insert the 3/8" Diameter Milling Cutter
M5                                       ;Spindle Off
GØ X-2 YØ T2/-5.15/.375 M6               ;Tool Change
M3                                       ;Spindle On
:              Drill the 4 Hole Bolt Hole Circle
GØ X2.8125 Y1.3125 Z.Ø625                ;Position Cutter Above Workpiece
G81 Z1.2 A45 H4 R1.4 F4                  ;Drill the 4 Holes
:              Face the Workpiece
GØ X-.2 Y.1                              ;Position Cutter Above Workpiece
G1 Z-.Ø625 F6                            ;Lower Cutter into Workpiece
G77 X4.5 Y2.475 Y.275                    ;Facing Cycle
:              Cut Away Edges Around the Circle
GØ X1.6125 Y-.25 ZØ                      ;Position Cutter Off of Workpiece
G1 Z-.344 F4                             ;Lower Cutter Half Way
YØ                                       ;Move Cutter into Workpiece
X4.1                                     ;Roughing Pass
Y2.625
X1.6125
Y-.25                                    ;Move Cutter Out of Workpiece
Z-.625                                   ;Lower Cutter Full Way
YØ                                       ;Move Cutter into Workpiece
X4.1                                     ;Roughing Pass
Y2.625
X1.6125
YØ
X1.6125                                  ;Finish Pass on Straight Edge
Y2.625 F6
:              Mill Out the Circle
GØ Z-.344 X2.8125                        ;Raise Cutter Half Way
G1 Y2.55 F4                              ;Move Cutter into Workpiece
G3 I2.8125 J1.3125                       ;Roughing Pass - CCW
G1 Z-.625                                ;Lower Cutter Full Way
G3 I2.8125 J1.3125                       ;Roughing Pass CCW
G1 Y2.5 F6                               ;Move Cutter into Workpiece
G2 I2.8125 J1.3125                       ;Finish Pass (Clockwise)
GØ ZØ                                    ;Move cutter out of workpiece
GØ Y1.3125                               ;Position for pocket mill
:              Mill Out the Pocket
G1 Z-.4                                  ;Move Cutter into Workpiece
G78 X.2625 X.25 X.Ø5 Y.2625 Y.25 F4      ;Pocket Mill Cycle
: Insert the 1/2" Diameter Drill
M5                                       ;Spindle Off
```

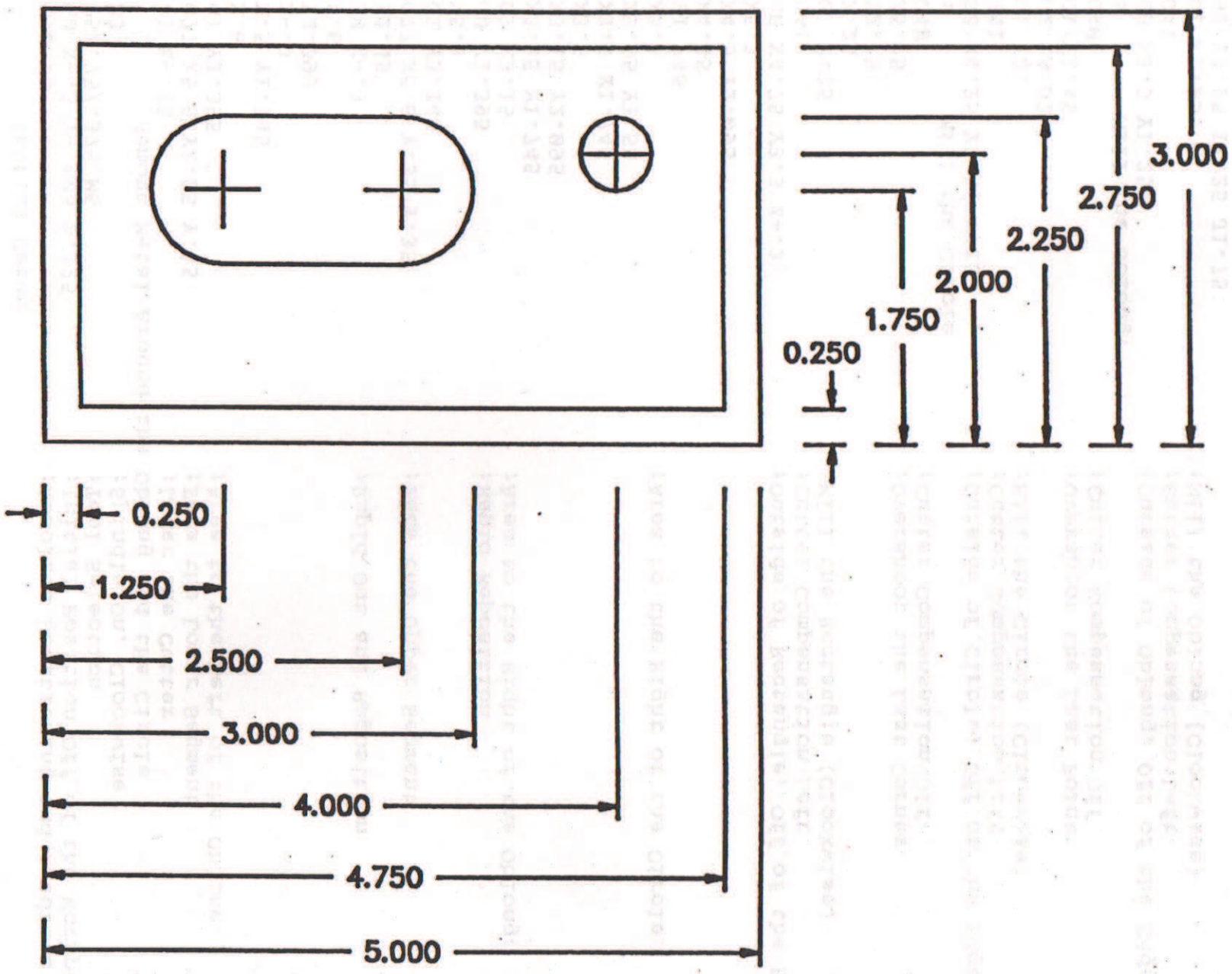
```
GØ X-2 YØ T3/Ø/.5 M6                    ;Tool Change
M3                                      ;Spindle On
GØ ZØ                                  ;Position Drill Above Workpiece
G81 X.6875 Y.5625 Z1.2 F4               ;Drill the 3 Holes
Y1.3125
Y2.Ø625
:           Restart
M5                              ;Spindle Off
GØ X-2 YØ M2                    ;Position Off Workpiece and Restart
```



CUTTER COMPENSATION LEFT

The following sample program mills raised surfaces out of a rectangular piece of metal. The program utilizes the cutter compensation left, G41, code. The original piece of metal is cut to a size 5 inches long, 3 inches wide and .5 inches deep. The depth of the cut for the top surfaces is .15 inches. The depth of the cut for the second surface is another .15 inches. Following is a 3D representation of the completed workpiece. After the 3D representation is an illustration which shows the shape to be milled along with the required dimensions. Finally, following the dimensioned illustration is the program to mill the required piece.

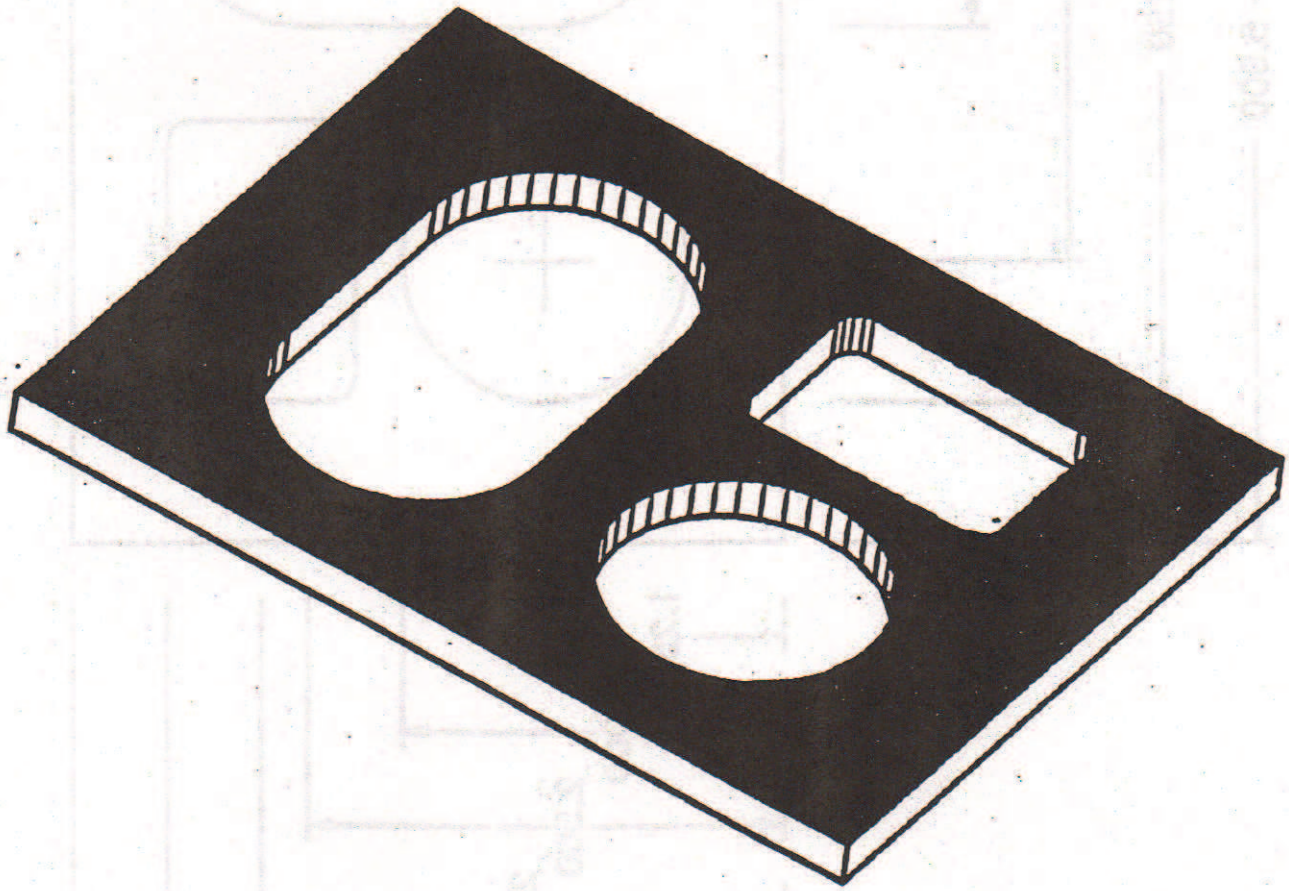


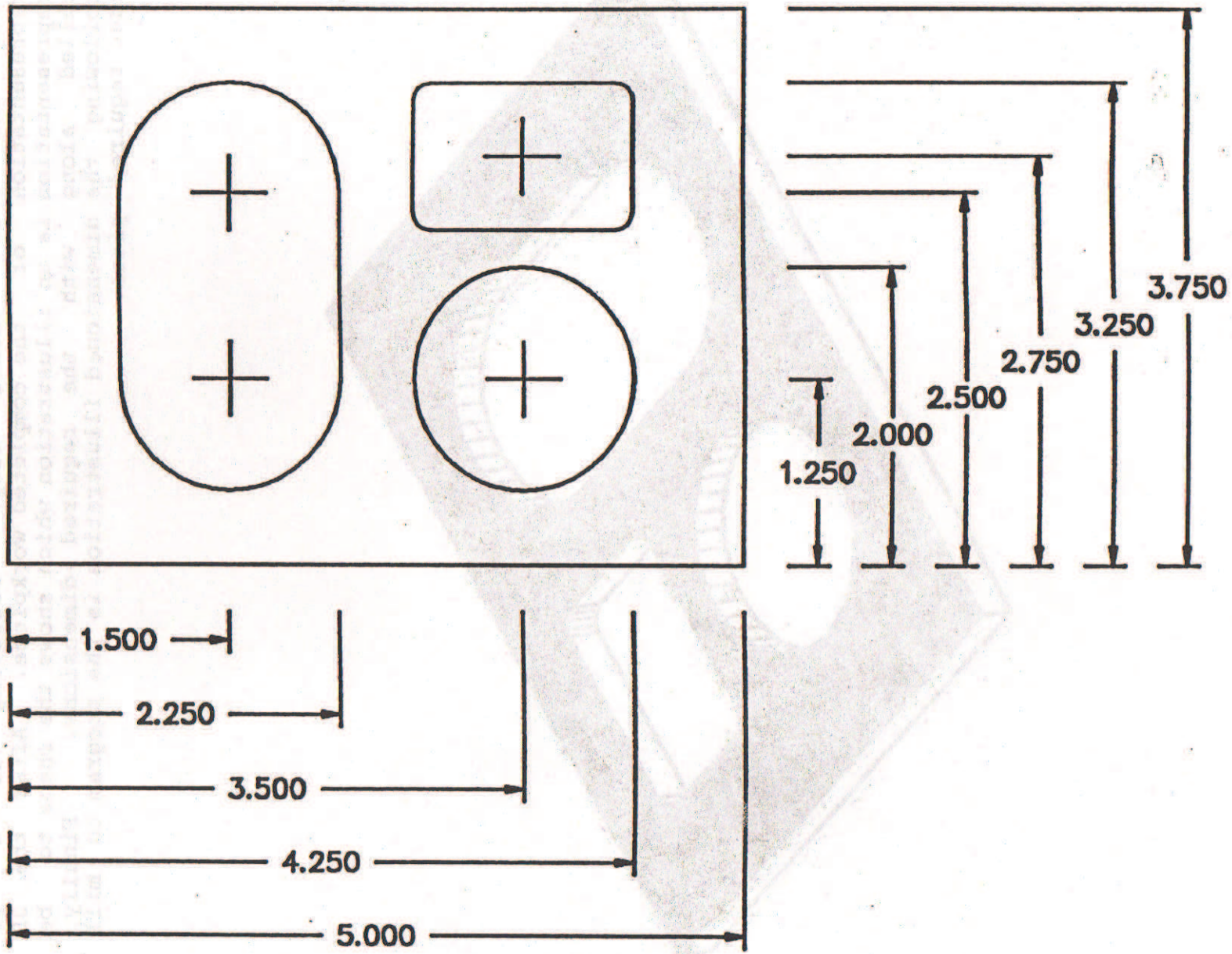

0.250
1.750
2.000
2.250
2.750
3.000
0.250
1.250
2.500
3.000
4.000
4.750
5.000

```
:       Initial Setup
G90 F6                          ;Absolute Positioning and Feedrate
G0 X-.3 Y-.005 Z.125            ;Initial Position Off of the Workpiece
T5/.75/.375 M6                  ;Tool Selection
M3                              ;Spindle On, Clockwise
:       Remove Metal Around the Oblong and the Circle
G1 Z-.15                        ;Lower the Cutter
G77 X5.6 Y1.05 Y.35             ;Face the Lower Segment
G1 Y1.395                       ;Area to the Left of the Oblong
X.6
X.5 Y1.745
X-.3
Y2.095
X.6
G0 X-.3                         ;Rapid Out and Reposition
Y2.45
G77 X5.6 Y.35 Y.35              ;Face the Upper Segment
G1 Y3.145
X5.3
G0 Y1.395                       ;Rapid Reposition
G1 X3.15                        ;Area to the Right of the Oblong
X3.25 Y1.745
X3.15 Y2.095
X3.5
X3.6 Y1.745
X3.55 Y1.55
X5.3                            ;Area to the Right of the Circle
Y1.745
X4.45
X4.5 Y2.095
X5.3
G0 X4.75 Y3.3 Z-.3              ;Outside of Rectangle, Off of the Edge
G41                             ;Cutter Compensation Left
G1 Y.25                         ;Mill the Rectangle (Clockwise)
X.25
Y2.75
X5.05                           ;Overshoot the Last Corner
G40                             ;Cutter Compensation Off
:       Mill the Circle
G0 X4.25 Y2.55 Z.15             ;Outside of Circle, Off of the Edge
G41                             ;Cutter Compensation Left
G1 Y2                           ;Mill the Circle (Clockwise)
G2 I4 J2
G1 Y1.45                        ;Overshoot the Last Point
G40                             ;Cutter Compensation Off
:       Mill the Oblong
G0 X3.3 Y1.25                   ;Outside of Oblong, Off of the Edge
G41                             ;Cutter Compensation Left
G1 X1.25                        ;Mill the Oblong (Clockwise)
G2 Y2.25 I1.25 J1.75
G1 X2.5
G2 Y1.25 I2.5 J1.75
G1 X.45                         ;Overshoot the Last Point
G40                             ;Cutter Compensation Off
:       Return to the Home Position
M5                              ;Spindle Off
G0 X-.3 Y-.005 Z.125 M2         ;Return to Home for Restart
```

CUTTER COMPENSATION RIGHT

The following sample program cuts three different shaped holes out of a rectangular plate. The program utilizes the cutter compensation right, G42, code. The original piece of metal is cut to a size 5 inches long, 3 3/4 inches wide and 1/16 inches deep. Following is a 3D representation of the completed workpiece. After the 3D representation is an illustration which shows the shape to be milled along with the required dimensions. Finally, following the dimensioned illustration is the program to mill the required piece.




3.750
3.250
2.750
2.500
2.000
1.250
1.500
2.250
3.500
4.250
5.000

```
:       Initial Setup
G90 F4                          ;Absolute Positioning and Feedrate
G0 X-1 Y-1 Z.125                ;Initial Position Off of the Workpiece
T12/.5/.25 M6                   ;Tool Selection
M3                              ;Spindle On, Clockwise
:       Mill the Oblong
G0 X1.75 Y2 Z.125               ;Inside of Oblong, Off of the Edge
G1 Z-.125                       ;Lower the Cutter
G42                             ;Cutter Compensation Right
X2.25                           ;Move to the Edge
Y1.25                           ;Mill the Oblong (Clockwise)
G2 X.25 I1.5 J1.25
G1 Y2.5
G2 X.75 I1.5 J1.25
G1 Y1.7                         ;Overshoot the Initial Cut
X1.75                           ;Move Off of the Edge
G40                             ;Cutter Compensation Off
:       Mill the Circle
G0 X3.75 Y1.25 Z.125            ;Inside of Circle, Off of the Edge
G1 Z-.125                       ;Lower the Cutter
G42                             ;Cutter Compensation Right
X4.25                           ;Move to the Edge
G2 I3.5 J1.25                   ;Mill the Circle (Clockwise)
X4.2 Y.8 I3.5 J1.25             ;Overshoot the Initial Cut
G1 X3.75 Y1.25                  ;Move Off of the Edge
G40                             ;Cutter Compensation Off
:       Mill the Rectangle
G0 X3.75 Y3 Z.125               ;Inside of Rectangle, Off of the Edge
G1 Z-.125                       ;Lower the Cutter
G42                             ;Cutter Compensation Right
X4.25                           ;Move to the Edge
Y2.25                           ;Mill the Rectangle (Clockwise)
X2.75
Y3.25
X4.25
Y2.7                            ;Overshoot the Initial Cut
X3.75                           ;Move Off of the Edge
G40                             ;Cutter Compensation Off
:       Return to the Home Position
M5                              ;Spindle Off
G0 X-1 Y-1 Z.125 M2             ;Return to Home for Restart
```

CHARACTER SET MACRO DEFINITIONS

The following macros define codes used to generate a possible character set for the CNC controller. Each macro contains a comment field describing the character it creates. Each number and letter in the character set is Ø.25 inches wide and Ø.3 inches high. The engraving depth used is Ø.Ø3 inches.

```
#1 ;'1'
G1G91Z-.Ø3
X.2
X-.1
Y.3
X-.Ø5Y-.Ø75
GØX.2Y-.225Z.Ø3
$
```

```
#2 ;'2'
GØG91X.2
G1Z-.Ø3
X-.2
Y.Ø75
G2X.Ø7Y.Ø75I.Ø75
G1X.Ø5
G3Y.15J.Ø75
G1X-.Ø5
G3X-.Ø75Y-.Ø75J-.Ø75
GØX.25Y-.225Z.Ø3
$
```

```
#3 ;'3'
GØG91Y.1
G1Z-.Ø3
G3X.1Y.1I.1
G1X.1Y.1
X-.2
GØX.25Y-.3Z.Ø3
$
```

```
#4 ;'4'
GØG91Y.3
G1Z-.Ø3
Y-.15
X.2
GØX-.Ø25Y.15Z.Ø3
G1Z-.Ø3
Y-.3
GØX.Ø75Z.Ø3
$
```

```
#5 ;'5'
GØG91Y.1
G1Z-.Ø3
G3X.1Y.1I.1
G1X-.1
Y.1
X.15
GØX.1Y-.3Z.Ø3
$
```

```
#6 ;'6'
GØG91X.1Y.3
G1Z-.Ø3
G3X-.1Y-.1J-.1
G1Y-.1
G3I.1
GØX.25Y-.1Z.Ø3
$
```

```
#7 ;'7'
G1G91Z-.Ø3
X.2Y.3
X-.2
GØX.25Y-.3Z.Ø3
$
```

```
#8 ;'8'
GØG91X.Ø75Y.15
G1Z-.Ø3
G3Y-.15J-.Ø75
G1X.Ø5
G3Y.15J.Ø75
G1X-.Ø5
G2Y.15J.Ø75
G1X.Ø5
G2Y-.15J-.Ø75
GØX.125Y-.15Z.Ø3
$
```

```
#9 ;'9'
GØG91X.1
G1Z-.Ø3
G3 X.1 Y.2 I-.15 J.2
G3 X0 Y0 I-.1 J0
GØX.Ø5Y-.2Z.Ø3
$
```

```
#10 ;'Ø'
GØG91Y.1
G1Z-.Ø3
Y.1
G2X.2I.1
G1Y-.1
G2X-.2I-.1
GØY-.1Z.Ø3
G1Z-.Ø3
X.2Y.3
GØX.Ø5Y-.3Z.Ø3
$
```

```
#11 ;'A'
G1G91Z-.Ø3
X.1Y.3
X.1Y-.3
GØX-.15Y.15Z.Ø3
G1Z-.Ø3
X.1
GØX.1Y-.15Z.Ø3
$
```

```
#12 ;'B'
G1G91Z-.Ø3
Y.3
X.125
G2Y-.15J-.Ø75
G1X-.125
X.125
G2Y-.15J-.Ø75
G1X-.125
GØX.25Z.Ø3
$
```

```
#13 ;'C'
GØG91X.2Y.1
G1Z-.Ø3
G2X-.2I-.1
G1Y.1
G2X.2I.1
GØX.Ø5Y-.2Z.Ø3
$
```

```
#14 ;'D'
G1G91Z-.Ø3
Y.3
X.1
G2X.1Y-.1J-.1
G1Y-.1
G2X-.1Y-.1I-.1
G1X-.1
GØX.25Z.Ø3
$
```

```
#15 ;'E'
GØG91X.2
G1Z-.Ø3
X-.2
Y.3
X.2
GØX-.Ø75Y-.15Z.Ø3
G1Z-.Ø3
X-.125
GØX.25Y-.15Z.Ø3
$
```

```
#16 ;'F'
G1G91Z-.Ø3
Y.3
X.2
GØX-.Ø75Y-.15Z.Ø3
G1Z-.Ø3
X-.125
GØX.25Y-.15Z.Ø3
$
```

```
#17 ;'G'
GØG91X.2Y.2
G1Z-.Ø3
G3X-.2I-.1
G1Y-.1
G3X.2I.1
GØX-.Ø5Z.Ø3
G1Z-.Ø3
X.Ø5
Y-.1
GØX.Ø5Z.Ø3
$
```

```
#18 ;'H'
G1G91Z-.Ø3
Y.3
Y-.15
X.2
Y.15
Y-.3
GØX.Ø5Z.Ø3
$
```

```
#19 ;'I'
G1G91Z-.Ø3
X.2
X-.1
Y.3
X-.1
X.2
GØX.Ø5Y-.3Z.Ø3
$
```

```
#20 ;'J'
G0G91Y.1
G1Z-.03
G3X.2I.1
G1Y.2
G0X.05Y-.3Z.03
$
```

```
#21 ;'K'
G1G91Z-.03
Y.3
G0X.2Z.03
G1Z-.03
X-.2Y-.2
X.075Y.075
X.125Y-.175
G0X.05Z.03
$
```

```
#22 ;'L'
G0G91Y.3
G1Z-.03
Y-.3
X.2
G0X.05Z.03
$
```

```
#23 ;'M'
G1G91Z-.03
Y.3
X.1Y-.15
X.1Y.15
Y-.3
G0X.05Z.03
$
```

```
#24 ;'N'
G1G91Z-.03
Y.3
X.2Y-.3
Y.3
G0X.05Y-.3Z.03
$
```

```
#25 ;'O'
G0G91Y.1
G1Z-.03
Y.1
G2X.2I.1
G1Y-.1
G2X-.2I-.1
G0X.25Y-.1Z.03
$
```

```
#26 ;'P'
G1G91Z-.03
Y.3
X.125
G2Y-.15J-.075
G1X-.125
G0X.25Y-.15Z.03
$
```

```
#27 ;'Q'
G0G91Y.1
G1Z-.03
Y.1
G2X.2I.1
G1Y-.1
G2X-.2I-.1
G0X.1Z.03
G1Z-.03
X.075Y-.1
G0X.075Z.03
$
```

```
#28 ;'R'
G1G91Z-.03
Y.3
X.125
G2Y-.15J-.075
G1X-.125
X.125
X.075Y-.15
G0X.05Z.03
$
```

```
#29 ;'S'
G0G91Y.075
G1Z-.03
G3X.075Y-.075I.075
G1X.05
G3Y.15J.075
G1X-.05
G2Y.15J.075
G1X.05
G2X.075Y-.075J-.075
G0X.05Y-.225Z.03
$
```

```
#30 ;'T'
G0G91X.1
G1Z-.03
Y.3
X-.1
X.2
G0X.05Y-.3Z.03
$
```

```
#31 ;'U'
G0G91Y.3
G1Z-.03
Y-.2
G3X.2I.1
G1Y.2
G0X.05Y-.3Z.03
$
```

```
#32 ;'V'
G0G91Y.3
G1Z-.03
X.1Y-.3
X.1Y.3
G0X.05Y-.3Z.03
$
```

```
#33 ;'W'
G0G91Y.3
G1Z-.03
X.05Y-.3
X.05Y.15
X.05Y-.15
X.05Y.3
G0X.05Y-.3Z.03
$
```

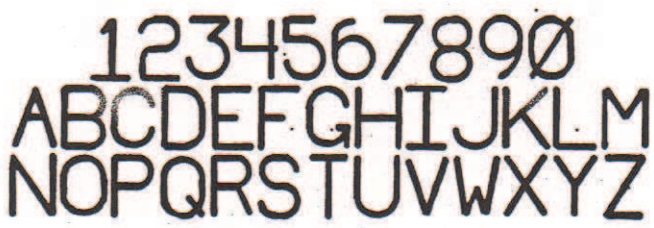
```
#34 ;'X'
G1G91Z-.03
X.2Y.3
G0X-.2Z.03
G1Z-.03
X.2Y-.3
G0X.05Z.03
$
```

```
#35 ;'Y'
GØG91Y.3
G1Z-.Ø3
X.1Y-.15
X.1Y.15
X-.1Y-.15
Y-.15
GØX.15Z.Ø3
$
```

```
#36 ;'Z'
GØG91Y.3
G1Z-.Ø3
X.2
X-.2Y-.3
X.2
GØX.Ø5Z.Ø3
$
```

The following sample program demonstrates the usage of the macros to produce the defined letters and characters. Following the sample program is the actual output generated when this program was executed.

```
GØG75F2Ø  ;INITIAL SETUP
G91X-1.225Y-.35 ;POSITION FOR 1-Ø
=#1
=#2
=#3
=#4
=#5
=#6
=#7
=#8
=#9
=#1Ø
G91X-2.875Y-.35 ;POSITION FOR A-M
=#11
=#12
=#13
=#14
=#15
=#16
=#17
=#18
=#19
=#2Ø
=#21
=#22
=#23
G91X-3.25Y-.35 ;POSITION FOR N-Z
=#24
=#25
=#26
=#27
=#28
=#29
=#3Ø
=#31
=#32
=#33
=#34
=#35
=#36
G91X-1.675M2 ;STOP FOR RESTART
```



INTERFACING THE CNC4 TO AN EXTERNAL DEVICE

The Centroid CNC4 has the capability to trigger and to receive a trigger pulse from an external device. This ability makes it possible to interface the CNC4 to a wide variety of external devices such as a pneumatic drill head (Spindle Wizard for example), punch press, or any device whose activation can be started with a trigger pulse or switch closure. If the external device signals the completion of its cycle, the CNC4 can respond to that information. If the external device does not signal the end of its cycle, then a dwell time can easily be programmed so that the CNC4 does not try to position with a tool still in the workpiece.

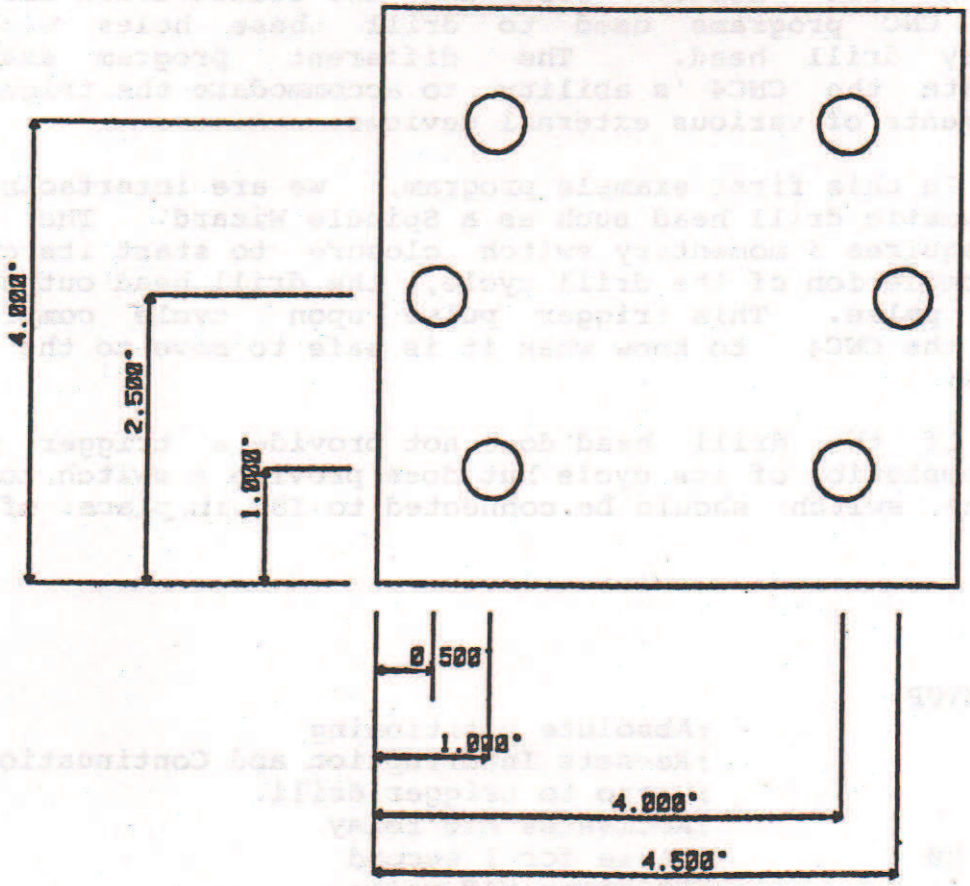
The following illustration shows a square workpiece with six .500" holes. Following the illustration are some example CNC programs used to drill these holes with an auxiliary drill head. The different program examples illustrate the CNC4 's ability to accommodate the triggering requirements of various external devices.

In this first example program, we are interfacing to a pneumatic drill head such as a Spindle Wizard. The drill head requires a momentary switch closure to start its cycle. Upon completion of the drill cycle, the drill head outputs a trigger pulse. This trigger pulse upon cycle completion allows the CNC4 to know when it is safe to move to the next position.

If the drill head does not provide a trigger pulse upon completion of its cycle but does provide a switch toggle, then the switch should be connected to I3 in place of the relay.

```
:INITIAL SETUP
G90                     ;Absolute positioning
M96                     ;Re-sets Interruption and Continuation mode
#1                      ;Macro to trigger drill
M10                     ;Activates M10 relay
M98 G4/100 M0           ;Pause for 1 second
M11                     ;Releases M10 relay
M96                     ;Re-sets Interruption and Continuation
M97/3 M0                ;Halts program execution until switch 3 is toggled
M96                     ;Re-sets Interruption and Continuation mode
$                       ;End of macro
G0 X1 Y1                ;Rapid traverse to first hole position
=#1                     ;Initiate drill sequence macro for first hole
G0 X4 Y1                ;Rapid traverse to second hole position
=#1                     ;Initiate drill sequence macro for second hole
G0 X4 Y4                ;Rapid traverse to third hole position
=#1                     ;Initiate drill sequence macro for third hole
G0 X1 Y4                ;Rapid traverse to fourth hole position
```

```
=#1                    ;Initiate drill sequence macro for fourth hole
GØ X.5 Y2.5            ;Rapid traverse to fifth hole position
=#1                    ;Initiates drill sequence macro for fifth hole
GØ X4.5 Y 2.5          ;Rapid traverse to sixth hole position
=#1                    ;Initiates drill sequence macro for sixth hole
GØ XØ YØ               ;Rapid traverse to X and Y axis zero position
M99   M2               ;Restarts complete program when front panel start
                       ;button is depressed
```



In this program example, the CNC4 is connected to a drill head which simply requires a switch closure to activate its cycle. The drill head does not provide a switch closure or trigger pulse to signal the completion of its cycle; therefore we must provide sufficient dwell time so that the CNC4 does not try to position for the next hole while the tool is still in the workpiece.

```
:                INITIAL SETUP
G9Ø                        ;Absolute positioning
M96                        ;Re-sets Interruption and Continuation mode
#1                         ;Macro to trigger drill head to start cycle
M1Ø                        ;Activates M1Ø relay and pauses for 1 second
M98 G4/1ØØ MØ              ;Pauses for 1 second with M1Ø relay on
M11                        ;Releases M1Ø relay
M96                        ;Re-sets Interruption and Continuation mode
M98   G4/15ØØ MØ           ;Sets 15 second dwell time for drill cycle
M96                        ;Re-sets Interruption and Continuation mode
$                          ;End of macro
GØ X1 Y1                   ;Rapid traverse to first hole position
=#1                        ;Initiate drill sequence macro for first hole
GØ X4 Y1                   ;Rapid traverse to second hole position
=#1                        ;Initiate drill sequence macro for second hole
GØ X4 Y4                   ;Rapid traverse to third hole position
=#1                        ;Initiate drill sequence macro for third hole
GØ X1 Y4                   ;Rapid traverse to fourth hole position
=#1                        ;Initiate drill sequence macro for fourth hole
GØ X.5 Y2.5                ;Rapid traverse to fifth hole position
=#1                        ;Initiates drill sequence macro for fifth hole
GØ X4.5 Y2.5               ;Rapid traverse to sixth hole position
=#1                        ;Initiates drill sequence macro for sixth hole
GØ XØ YØ                   ;Rapid traverse to X and Y axis zero position
M99 M2                     ;Restarts complete program when front panel start
                           ;button is depressed
```

The following program illustrates the use of the G88 (punch\press cycle). In this program example, the auxiliary device does not provide an end of cycle switch closure or trigger pulse; therefore it is necessary to set a sufficient dwell time to allow for the completion of the drill cycle before the CNC4 re-positions for the next hole. In this example, we will assume that the total required dwell time is 15 seconds.

```
M96              ;Re-sets Interruption and Continuation mode
G4/75Ø G9Ø       ;Establishes dwell time
;                 (The required dwell time is 15 seconds total.
;                 (Since the g88 command dwells twice during the cycle,
;                 (set the dwell time to 1/2 the required time)
G88 X1 Y1        ;Rapid traverse to first hole position and drill
X4 Y1            ;Rapid traverse to second hole position and drill
X4 Y4            ;Rapid traverse to third hole position and drill
X1 Y4            ;Rapid traverse to fourth hole position and drill
X.5 Y2.5         ;Rapid traverse to fifth hole position and drill
X4.5 Y2.5        ;Rapid traverse to sixth hole position and drill
G8Ø              ;Cancel G88 Punch/Press cycle
GØ XØ YØ         ;Rapid traverse to X and Y axis zero position
M99   M2         ;Re-starts program when front panel start switch is
                 ;pressed
```

HOOKUP INSTRUCTIONS

RFI (RADIO FREQUENCY INTERFERENCE) AND ELECTRICAL NOISE

The CNC4 is a sophisticated electronic device utilizing high speed digital electronic circuitry for both logic and motor drive functions. This section of the hookup instructions will deal with the proper wiring and shielding procedure to eliminate radiated RFI from the CNC4 and also to prevent RFI from entering the CNC4 enclosure. All sensitive electronic equipment is susceptible to this kind of interference without proper shielding. Devices such as welders, large electric motors (such as spindle motors), contactors and relays, and digital devices (like computers) can all generate extremely powerful bursts of RFI noise. The AC wires connected to these devices act as very efficient antennas, radiating this noise to any other wire in the vicinity. Remember that this "noise" is in the radio frequency band. The only way to prevent this noise from interfering with the sensitive electronic circuits inside the CNC4 is to provide a shield around all wiring going into and out of the CNC4. The shield acts to "grab" the noise bursts and carry them to ground potential.

CNC4 CONTROLLER PLACEMENT

The CNC4 should be located as close to the mill as possible. A standard electronics equipment rack with 19" rail spacing is ideal. This rack should be grounded to the body of the mill. The programming terminal or computer can either be placed in the rack or a little further away so as to prevent damage by flying metal chips and cutting lubricant. Clear plastic molded covers are available for most keyboards. These covers prevent contamination of the keyboard yet still allow easy typing and are highly recommended.

MOTOR CABLE SHIELDING

CENTROID recommends the use of a 4 conductor cable with foil shield and drain wire. The conductor size should be no smaller than 18 GA wire. The drain wire should be connected to the motor cases on the mill end, and to the rear panel binding post on the rear panel of the CNC4. In addition,it is recommended that you run the shielded cable through liquid tight flexible conduit. 3/8" conduit should be sufficient. Be sure to terminate the conduit to the motor using the proper end fittings. Most stepping motors large enough for milling machine use will have standard pipe threads in the motor case. Centroid 1350 oz/in and 2130 oz/in stepping motors are so equipped. The other end of the conduit should be routed to a bracket with holes so that the proper conduit end fittings can be attached. If possible attach this metal bracket directly to the CNC4 rack or to the back of the mill. Keep the motor cable length as short as possible yet still allow room for the axes to move freely.

LIMIT SWITCH CABLE SHIELDING

If your mill is equipped with limit switches, it is best to wire all of the switches in series so that if one of them is tripped the circuit opens. The CNC4 requires that limit switches be normally closed. Use 2 conductor shielded cable with a foil shield and drain wire to connect the switches together. Route the switch wire through liquid tight flexible metal conduit. Connect the drain wire to the switch body (if it is metal) or to the mill itself as close to the switch as possible. Connect the drain wire on the CNC4 end of the cable to one of the rear panel binding posts. Run the conduit to the same bracket that the motor cable conduits are attached to.

M - FUNCTION RELAY CABLE SHIELDING

The CNC4 M - function outputs are designed to drive relays only. The switch contacts on these relays provide a simple way to turn on or off external devices such as spindle motors and coolant pumps under CNC program control. The relays can also be pulsed, providing a convenient method of supplying timed trigger pulses to external devices. With the optional BCD I/O board, the CNC4 can also control tool-changers, or spindle speed controls that respond to BCD switching.

!!! CAUTION !!!

M function relays should be placed in a suitable metal enclosure. DO NOT CONNECT THE RELAY OUTPUTS OF THE CNC4 TO ANYTHING EXCEPT THE COIL OF A SUITABLE RELAY. DOING SO WILL CAUSE SEVERE DAMAGE TO THE CNC4 AND MAY PRESENT A SERIOUS SHOCK HAZARD TO THE OPERATOR.

Use a 2 conductor shielded cable with a foil shield and drain wire. Connect the drain wire to the relay enclosure or to the body of the mill, as close to the relay as possible. Connect the controller end of the drain wire to one of the binding posts on the CNC4 rear panel. Use liquid tight conduit if possible and attach the conduit to the relay enclosure and the bracket containing the motor and switch input conduits.

RS-232 SERIAL CABLE SHIELDING

All RS-232 cables provided by Centroid are fully shielded and ready to go. If you need to make additional cables or are interfacing into an existing cable connector and need to re-wire, it is essential that you use shielded cable, especially if the system is operating in a high RFI noise area.

AC WIRING

AC wiring, especially the wiring in the building, is very important. Clean, safe, and properly grounded AC lines are essential for safe and reliable operation. If the AC line level drops low during operation of large machinery or during periods of peak use, steps must be taken to ensure stable line voltage at all times.

The CNC4 has a built in AC line filter which will filter out RFI noise carried in on the AC line. Since all of the shielding utilized so far is connected to the chassis of the CNC4, the AC line ground must be properly connected to earth ground.

GENERAL MOTOR HOOKUP

All motor, switch, relay, and power connections are made on the rear panel. The "CNC4 Rear Panel Drawing" shows where each connection is located.

The "General Motor Wiring Drawing" shows a typical motor connection for a 4, 6 and 8 lead motor. The motor is connected such that there are two phases, each phase consisting of two wires. Note that 8 lead motors can be wired in either a series or parallel configuration. The parallel configuration will produce slightly less dynamic torque but will allow the motor to run much faster. The series configuration will produce slightly more dynamic torque but will limit the motor's top speed. When choosing a motor, it is best to choose a motor large enough to provide sufficient dynamic torque when wired in the parallel configuration. Six lead motors are internally wired in the series configuration and cannot be changed. Although the CNC4 will drive these motors, they may not run as fast as an equivalent 4 lead bipolar or 8 lead motor wired in the parallel configuration. Contact Centroid for assistance in choosing a suitable motor for your application.

SPECIFIC MOTOR HOOKUPS

Before wiring your motors to the CNC4 terminals, conduct the "Motor Continuity and Ground Test". The test is outlined on the following page.

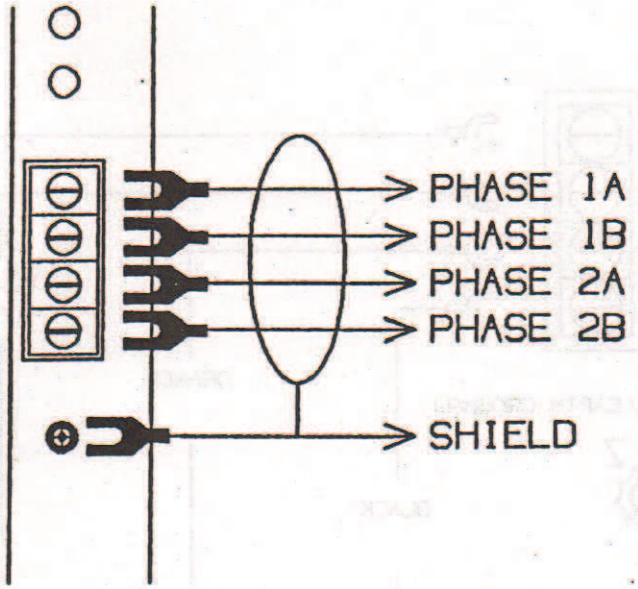
Installation diagrams are contained in this manual for many of the common motors. Find the diagram which corresponds to your motor and connect a suitable length of cable to the motor leads or terminals. The CNC4 end of the cable should be terminated with spade terminals. In Most cases, motors purchased from Centroid will be wired and ready to connect to the CNC4.

*** * IMPORTANT * ***

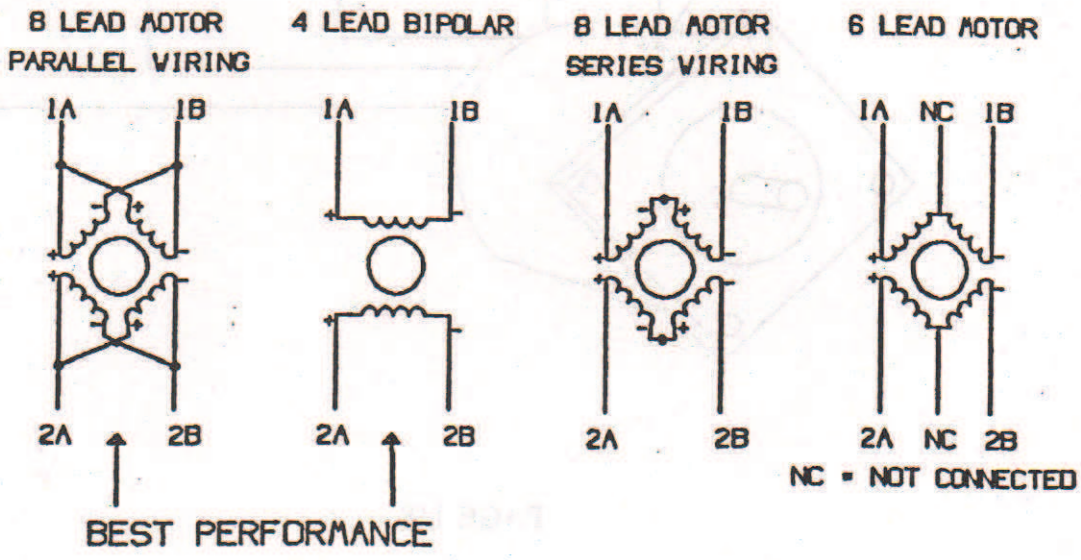
Do not randomly connect the CNC4 to your motors --
DAMAGE TO A CNC4 AND MOTOR CAN OCCUR !!

After the motors are connected to the CNC4, you may need to adjust the CNC4's drive card output current to meet your motor's requirements. Again, if you ordered the motors from Centroid along with the controller, the drive current is pre-set to the motors based on the information supplied to us at the time of purchase.

GENERAL MOTOR WIRING

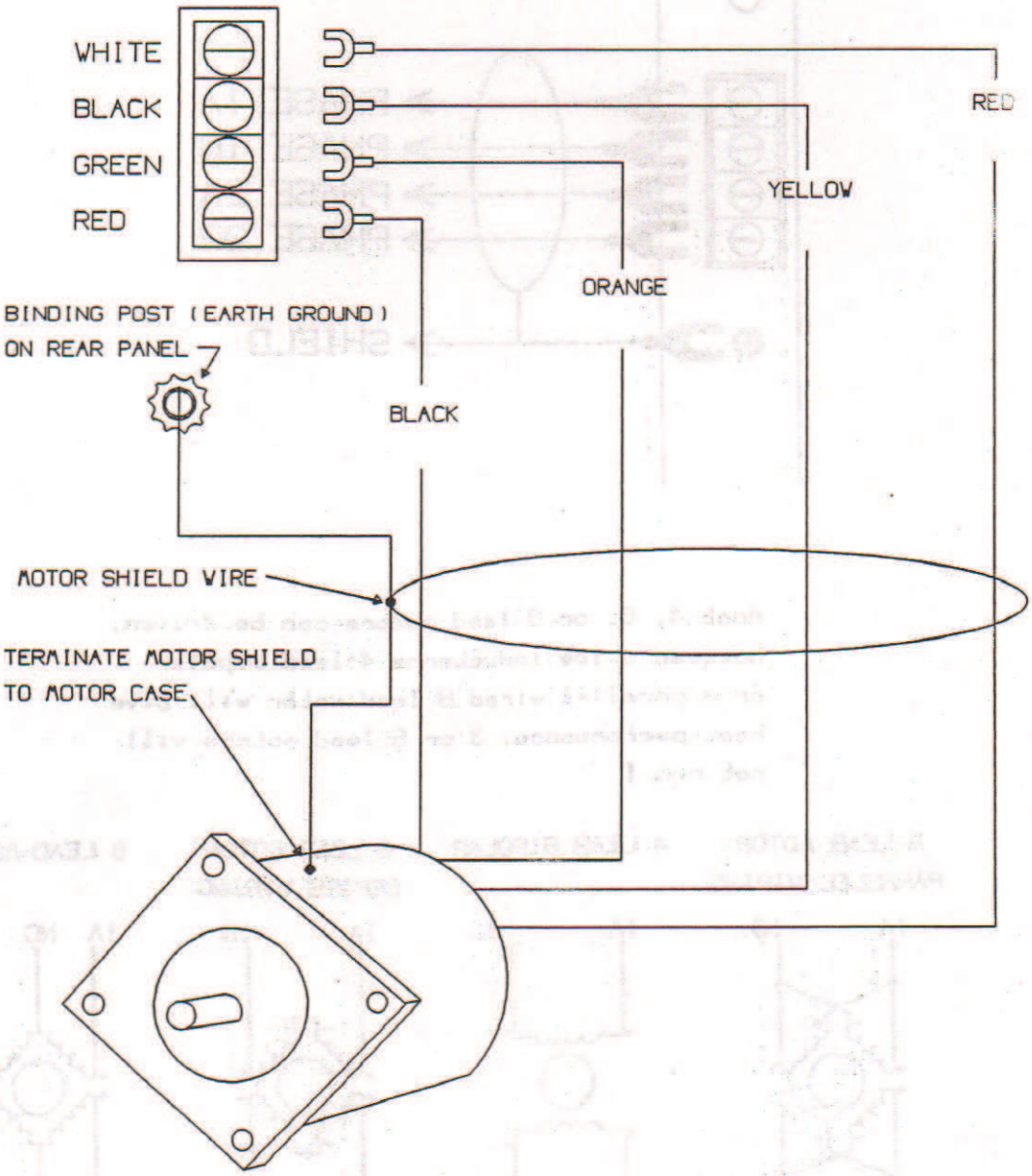


Most 4, 6, or 8 lead motors can be driven, however a low inductance 4 lead bipolar or a parallel wired 8 lead motor will give best performance. 3 or 5 lead motors will not run !

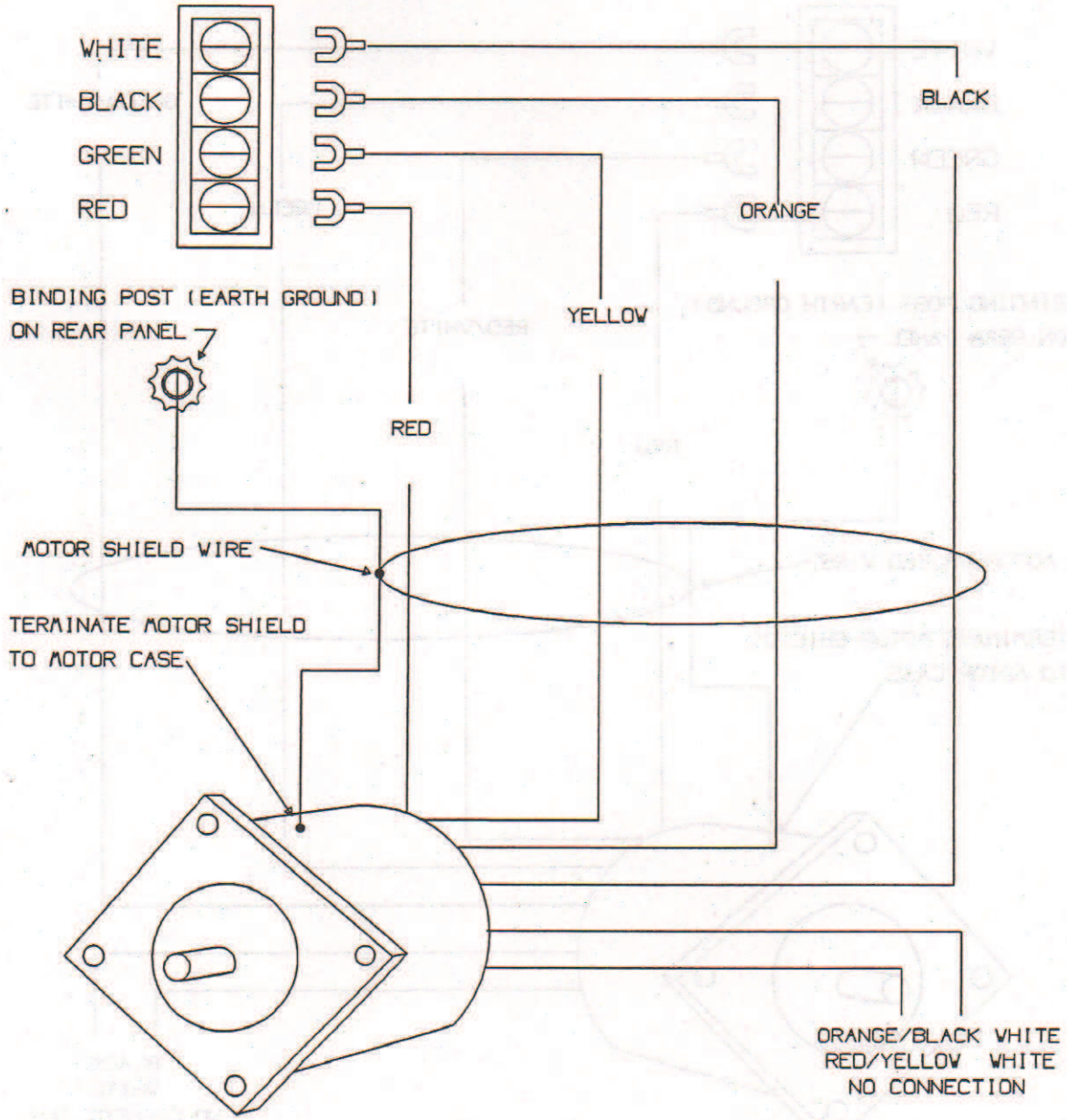


MAE-CENTROID-SIGMA 4 LEAD

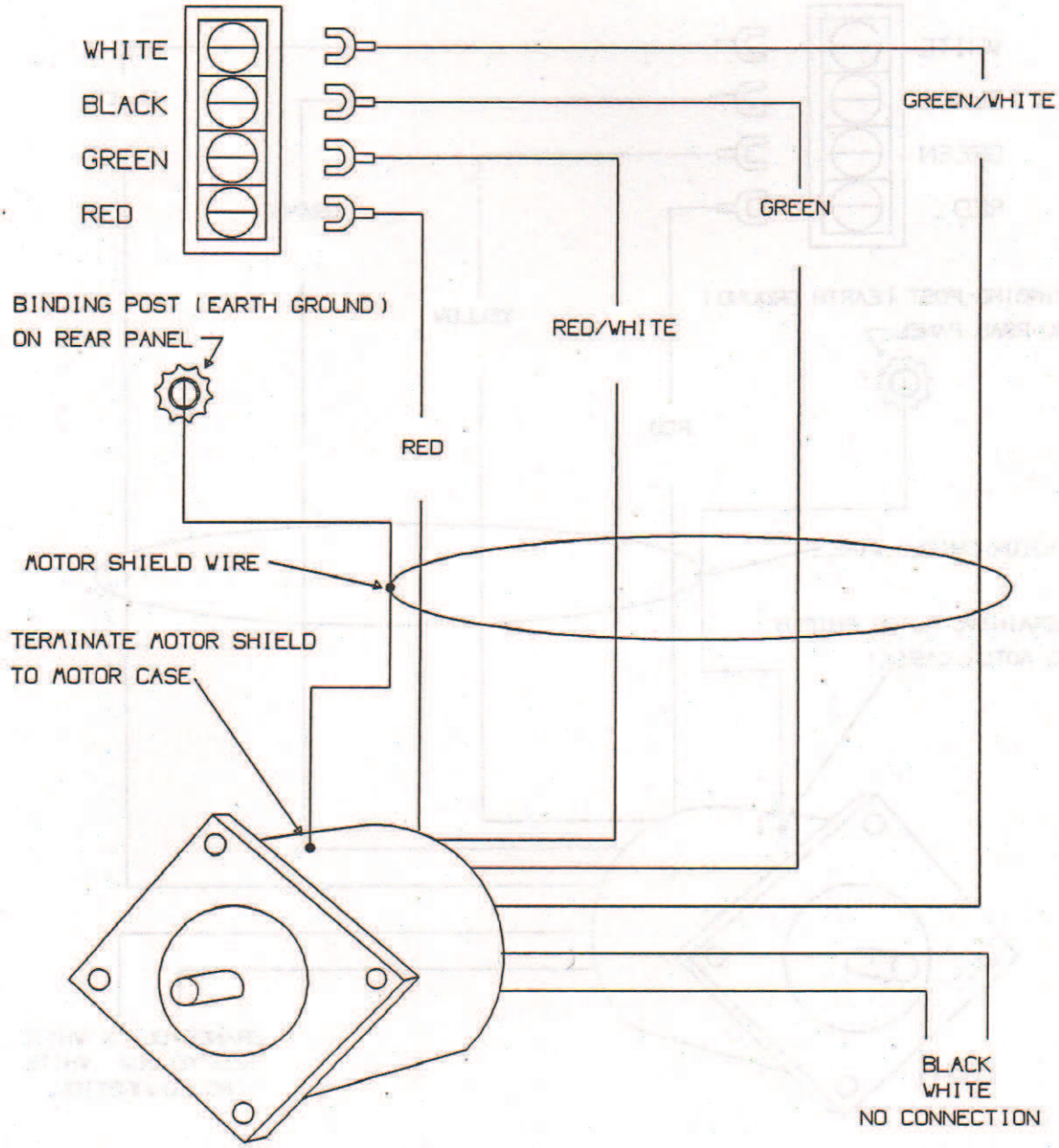
PARALLEL HOOKUP - HIGH CURRENT CARD



SIGMA 6 LEAD
HIGH CURRENT CARD



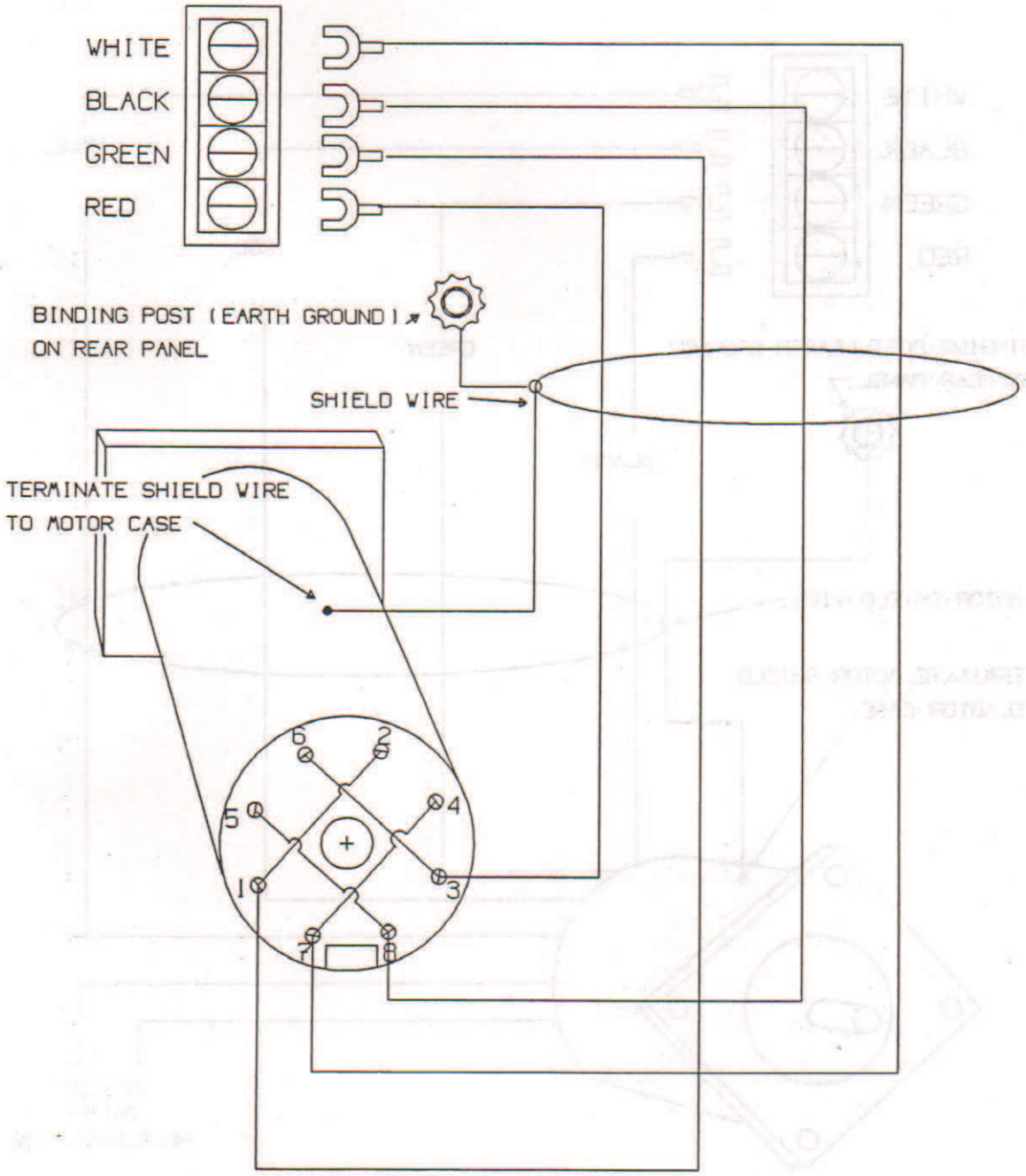
SUPERIOR ELECTRIC (SLO-SYN)
MEASURE MATIC (NU-SYN)
EASTERN AIR DEVICES
RAPIDSYN



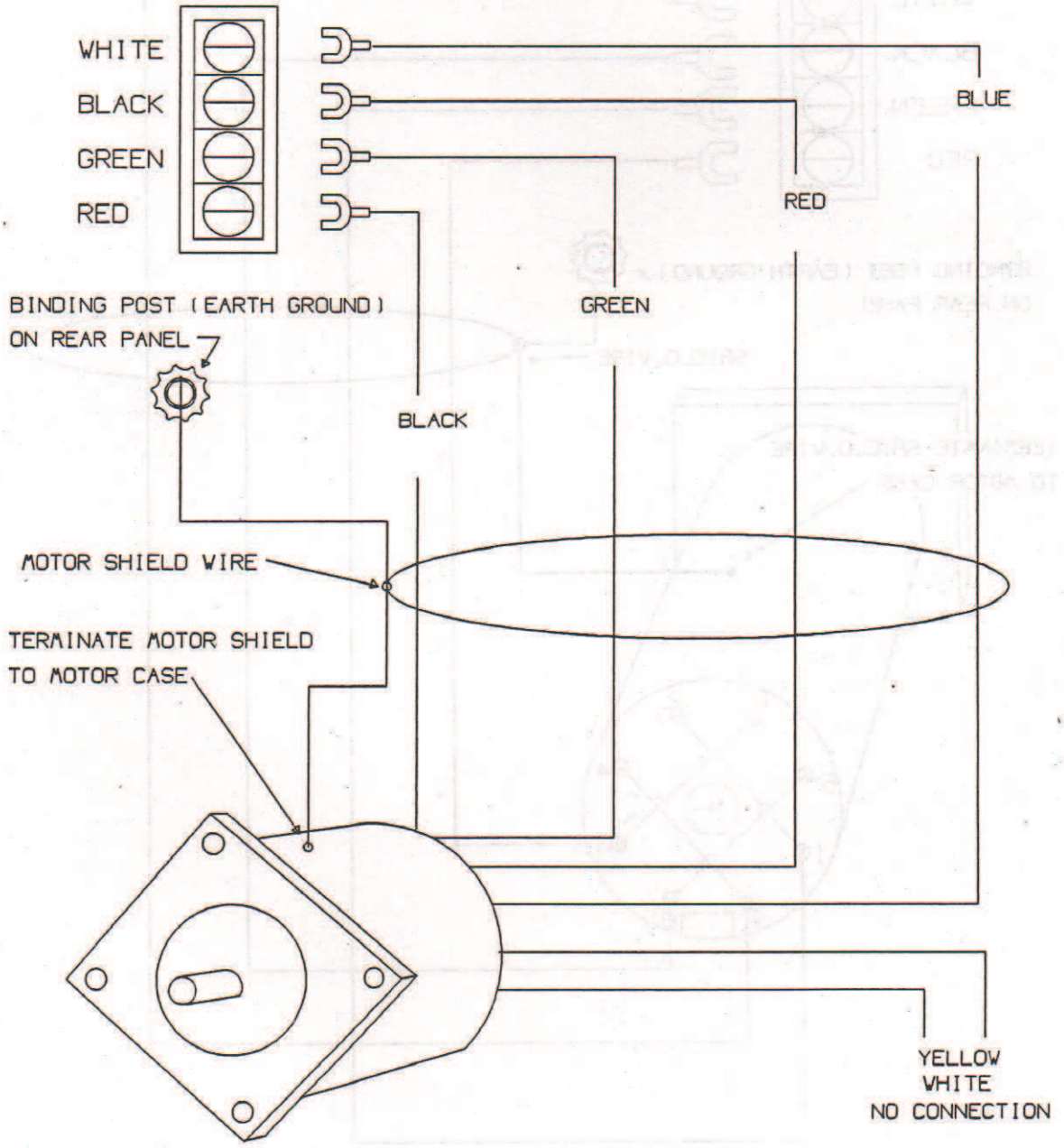
SUPERIOR (SLO-SYN)

8 LEAD WITH SCREW TERMINALS

PARALLEL WIRING



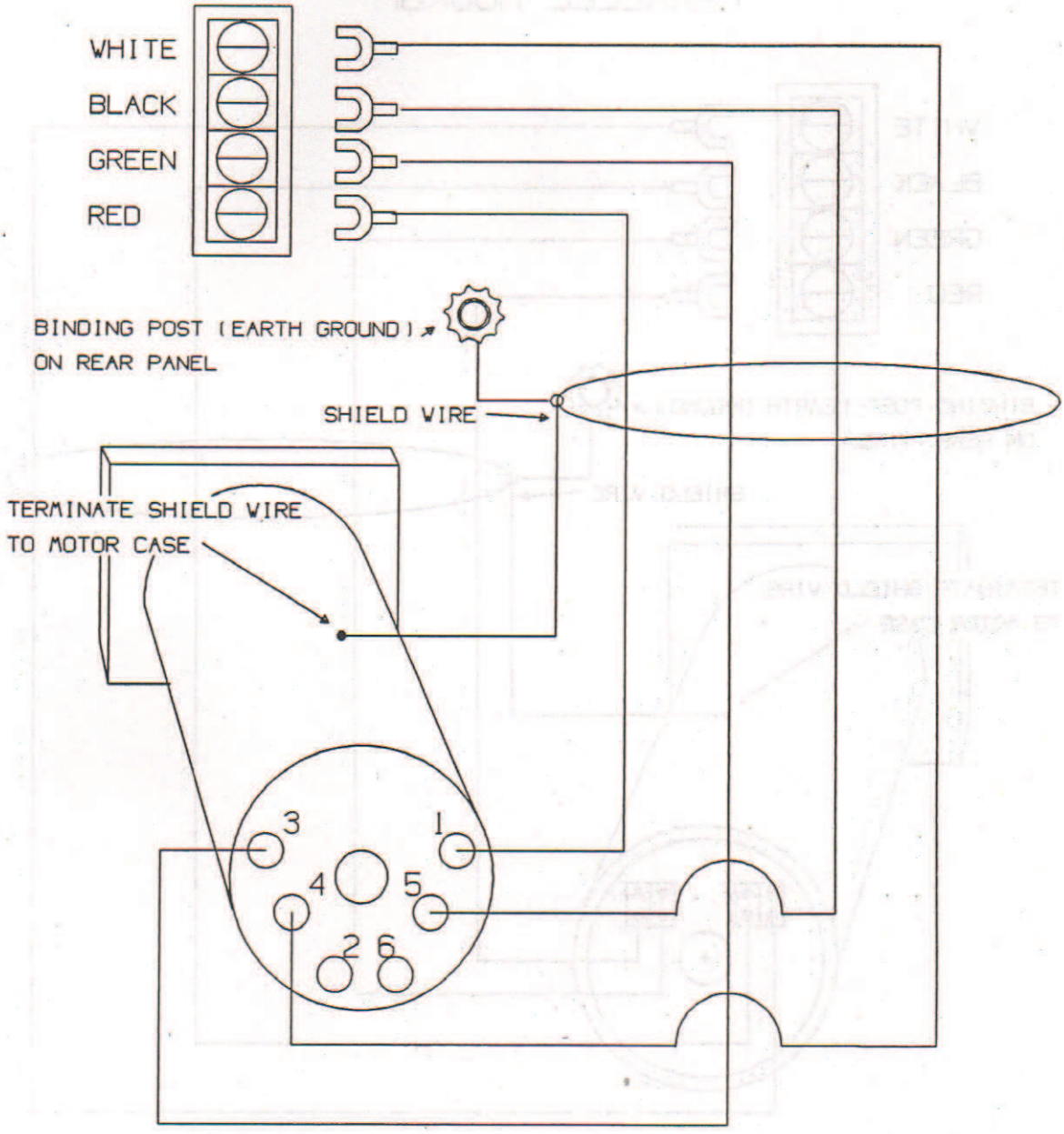
ORIENTAL (VEXTA) 6 LEAD HIGH CURRENT CARD



SUPERIOR (SLO-SYN)

6 LEAD WITH SCREW TERMINALS

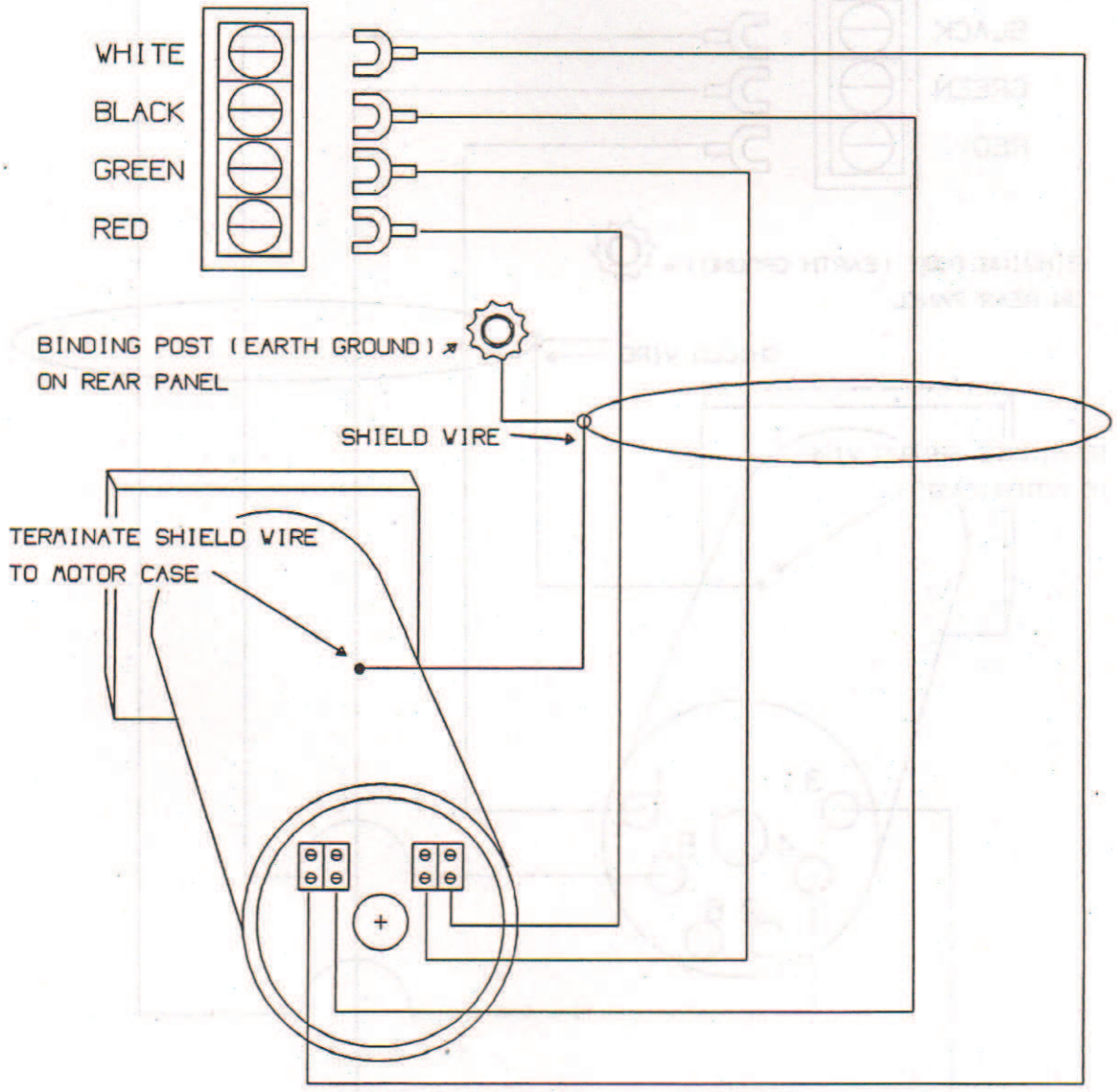
HIGH CURRENT CARD



CENTROID- MAE 2130. MAE 1350

ENCODER - READY WITH 4 TERMINAL BLOCKS

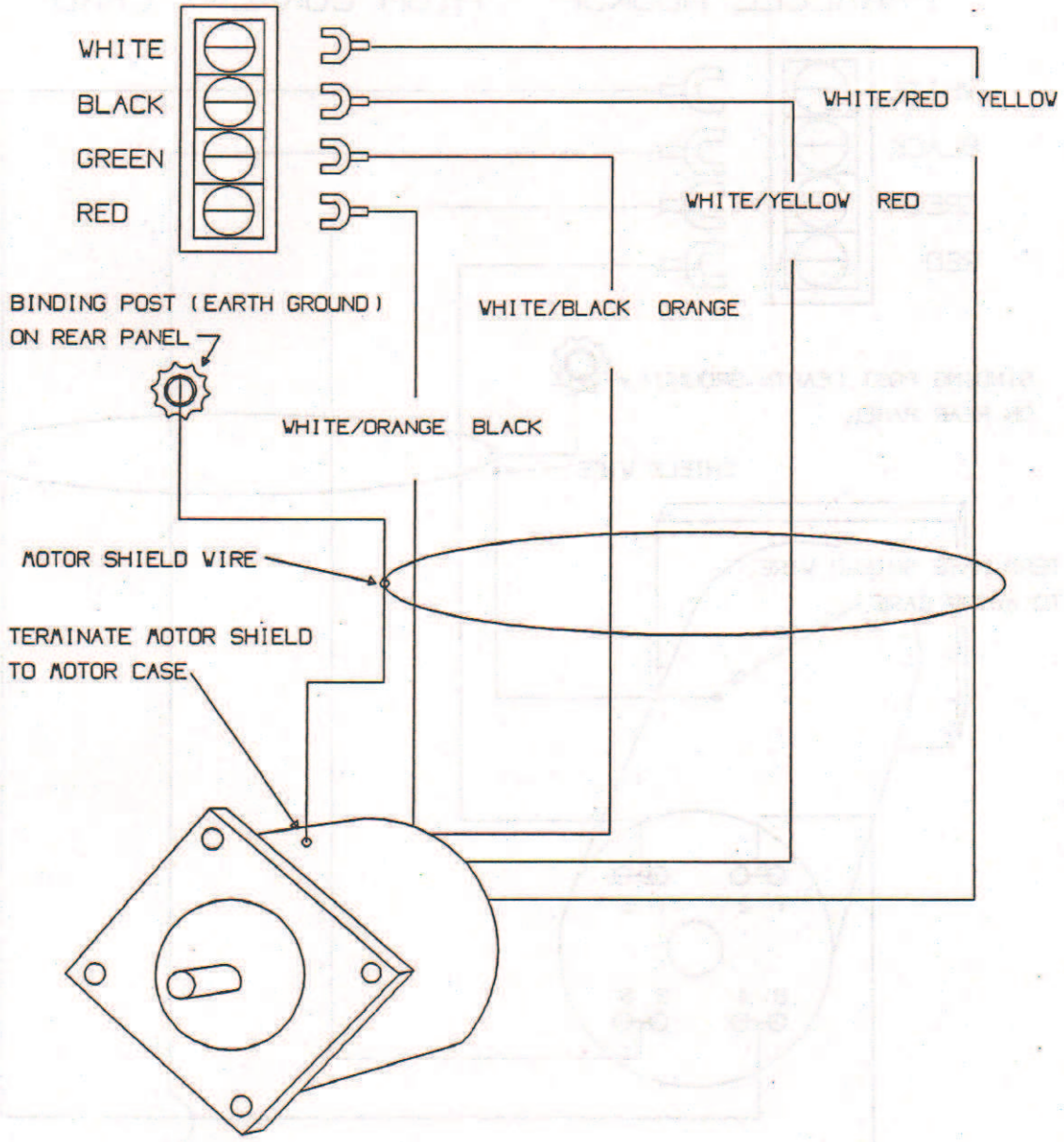
PARALLEL HOOKUP



CENTROID MODEL MAE 450. MAE 150
SIGMA

8 WIRE - PARALLEL HOOKUP

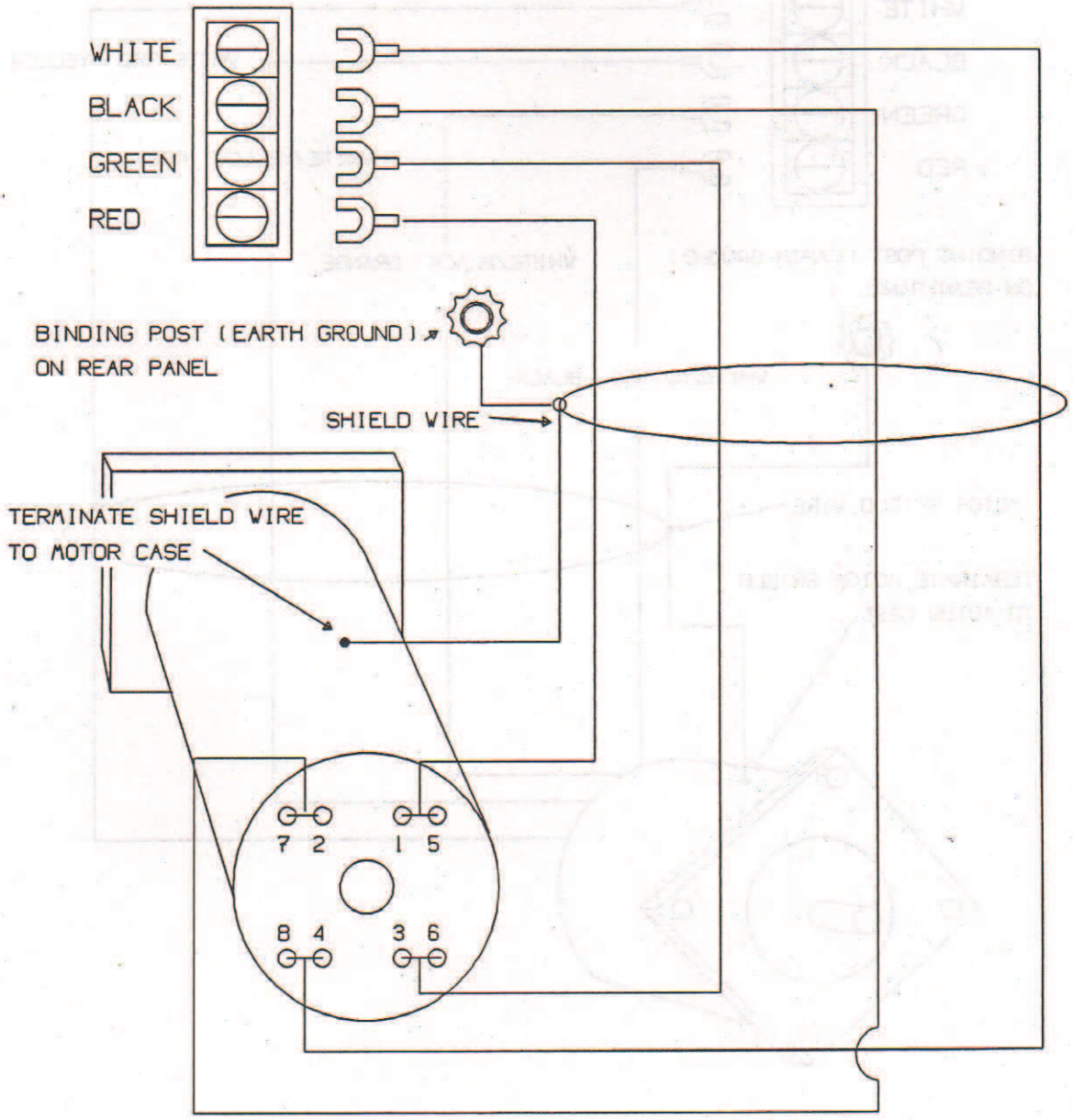
HIGH CURRENT CARD



SIGMA

8 LEAD WITH SCREW TERMINALS

PARALLEL HOOKUP - HIGH CURRENT CARD



I/O BOARDS

The I/O boards each provide 8 programmable switch inputs, 8 relay driver outputs, +12 and +5VDC VDC out for use with Hall effect sensors or opto-interrupters, and ground terminals. Each I/O board contains a 20 position terminal strip. Each terminal position its labeled according to its function.

The optional second I/O board can be used for straight I/O or can be used in conjunction with the Automatic Tool Changer setting available in the Setup Menu to control any device which uses a BCD number for control. To use the Automatic Tool Change feature of the CNC4, the optional second I/O board MUST be installed.

I/O BOARD SWITCH INPUTS

The Centroid I/O board switch inputs are normally high and must be pulled down to ground for the CNC4 to sense a toggle.

LIMIT SWITCHES

As shown in the figure, "CNC4 I/O Board Connections", micro switches can be connected in series for use as axis travel limit switches. Switches are connected between one of the input terminals (I1 through I8 only) and one of the ground terminals (G) or one of the rear panel binding posts. For those users who are going to use Hall effect sensors or opto-interrupters, Centroid has provided +12 and +5 VDC. If your electronic sensor requires something other than +12 or +5 VDC, choose a series limiting resistor to limit current to the device.

Note that electronic switches can not be wired in series. If you are using electronic switches as limit switches, then each switch must be connected to a separate input and each input assigned to a specific axis and direction (Setup Menu option 7). Also, the switches must be normally in a low state. The CNC4 will see an electronic switch in the high state as being tripped and will not allow the axes to move.

Terminal positions on I/O boards:

First I/O Board		Second I/O Board		
Terminal Number	Function	Terminal Number	Function	BCD
R1	Relay 1 (M3)	1	Relay 9	1
R2	Relay 2 (M4)	2	Relay 10	2
R3	Relay 3 (M8)	4	Relay 11	4
R4	Relay 4 (M10)	8	Relay 12	8
R5	Relay 10	10	Relay 13	10
R6	Relay 20	20	Relay 14	20
R7	Relay 7	SPD	Relay 15 Spindle Strobe	***
R8	Relay 8	TCH	Relay 16 (Tool Change Strobe)	***
+12	+12 VDC for relays	29	+12 VDC for relays	***
G	GND	30	GND	***
G	GND	31	GND	***
+5	+5 VDC	32	+5 VDC	***
i8	Input 8	i16	Input 16	***
i7	Input 7	i15	Input 15	***
i6	Input 6	i14	Input 14	***
i5	Input 5	i13	Input 13	***
i4	Input 4	i12	Input 12	***
i3	Input 3	i11	Input 11	***
i2	Input 2	i10	Input 10	***
i1	Input 1	i9	Input 9	***

*** Does not apply

I/O BOARD RELAY DRIVER OUTPUTS

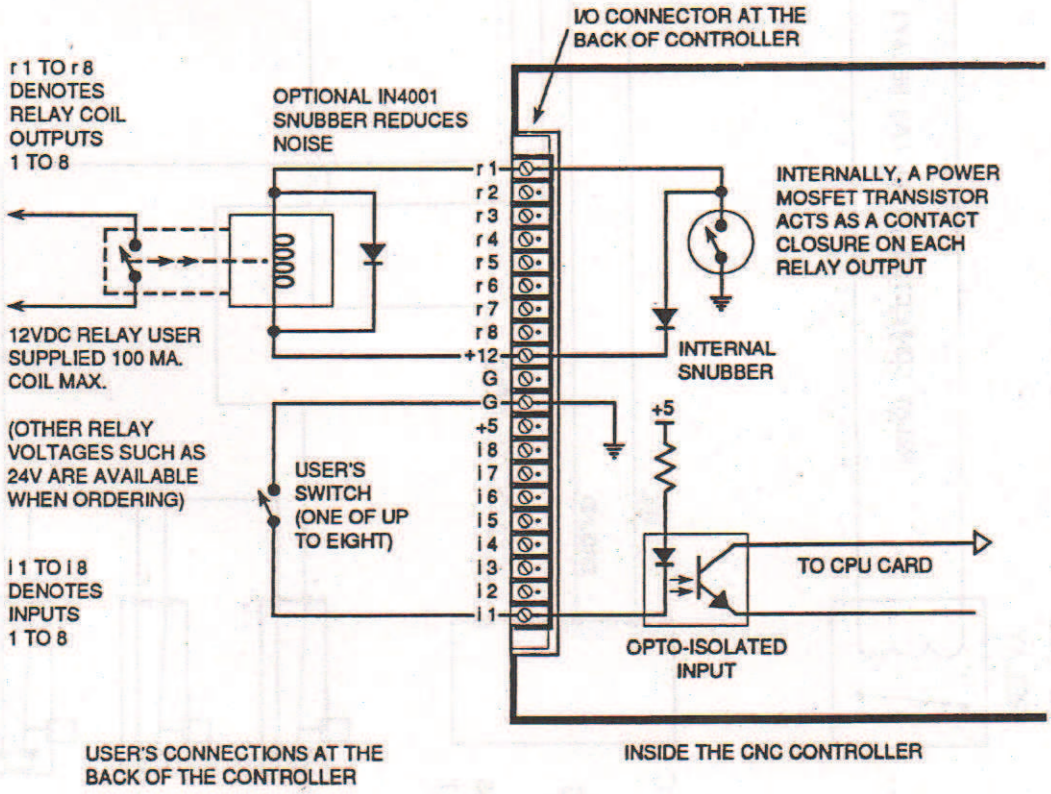
!!! CAUTION !!!

DO NOT CONNECT THE RELAY OUTPUTS OF THE CNC4 TO ANYTHING EXCEPT THE COIL OF A SUITABLE RELAY. DOING SO WILL CAUSE SEVERE DAMAGE TO THE CNC4 AND MAY PRESENT A SERIOUS SHOCK HAZARD TO THE OPERATOR.

The figure "CNC4 I/O Board Connections" illustrates how to connect a relay output to a relay output (R1 through R8). A 12 VDC relay coil that draws no more than 100 milliamps should be used. Relays are connected between one of the relay terminals (R1 through R8) and +12 VDC.

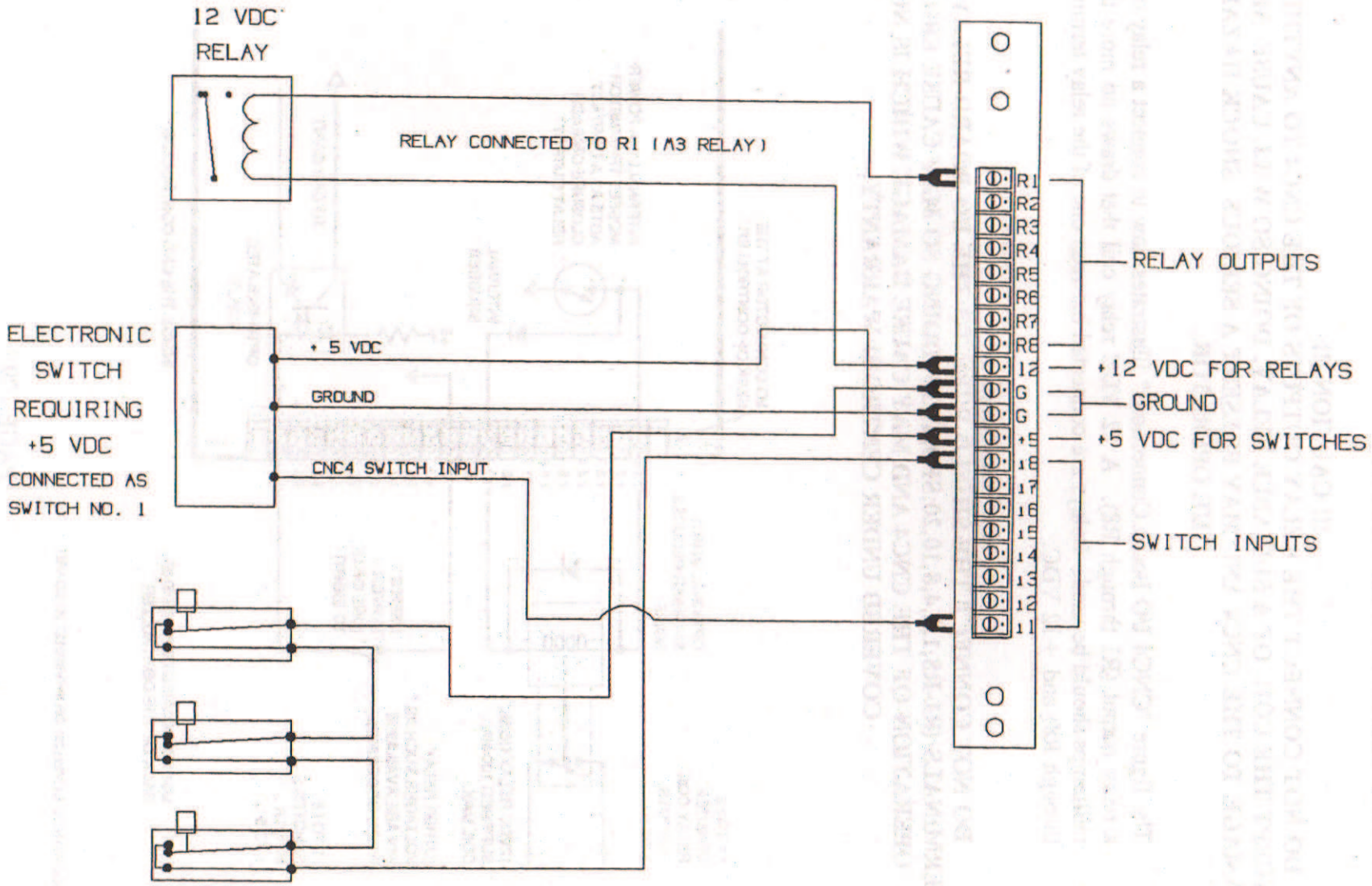
!!! WARNING !!!

DO NOT CONNECT THE SHIELD WIRE TO THE I/O BOARD RELAY TERMINALS (R1-R8,1,2,4,8,10,20,SPD,TCH). DOING SO MAY CAUSE ERRATIC OPERATION OF THE CNC4 AND MAY CAUSE DAMAGE WHICH IS NOT COVERED UNDER CENTROID WARRANTY.

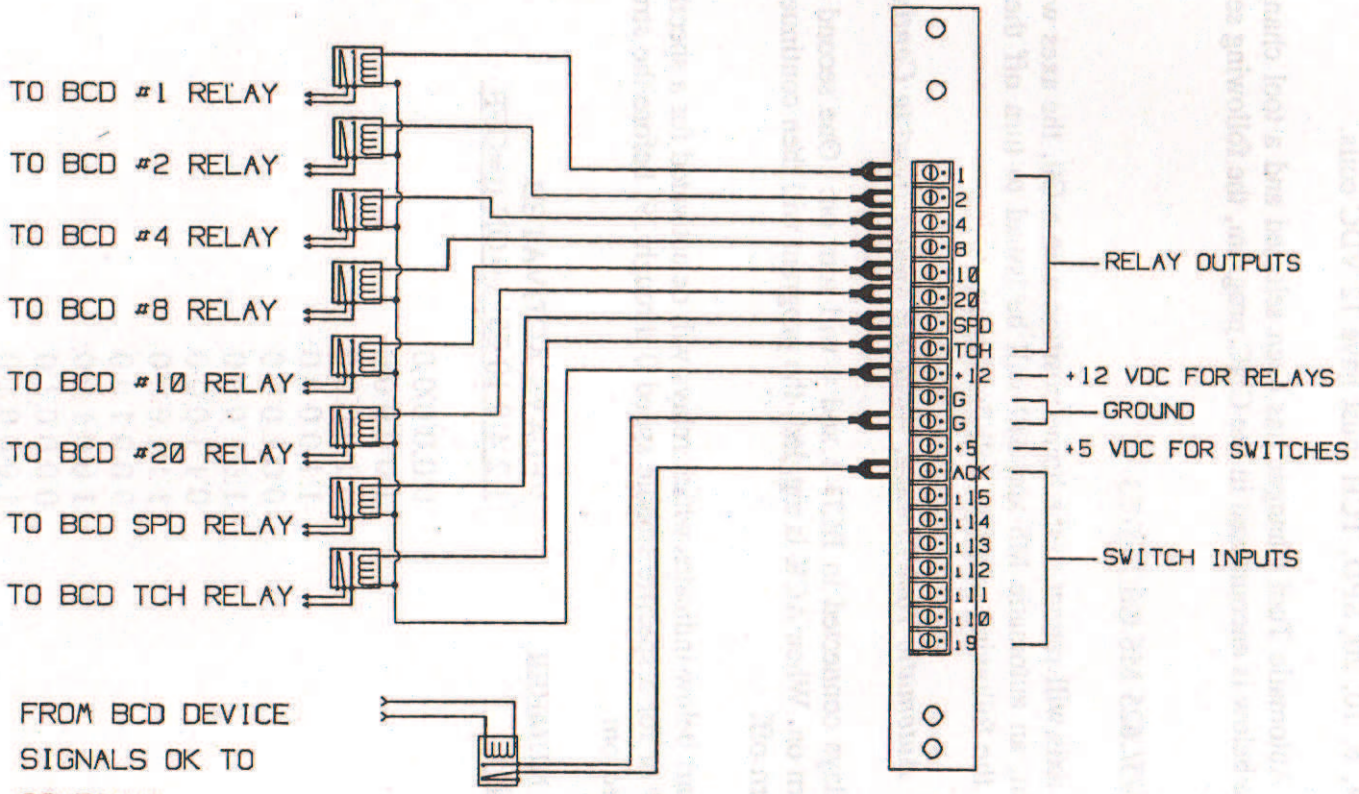


CNC4 WITH I/O REV891231 OR REV890127 I/O BOARD

CNC4 I/O BOARD CONNECTIONS



CNC4 BCD BOARD (2ND I/O BOARD)



SELECT AUTOMATIC TOOL CHANGER
FROM SETUP MENU TO USE BCD BOARD

OPTIONAL SECOND I/O USED TO CONTROL A BCD INTERFACE

The figure "CNC4 BCD Board (2nd I/O Board)" illustrates the second I/O board connections to a BCD interface. Note that the relays connected to pins R1 through R8, 1, 2, 4, 8, 10, 20, SPD, TCH must have 12 VDC coils.

When Automatic Tool Changer has been selected and a tool change command such as the one below is encountered in the CNC program, the following sequence of events will occur:

T5/-.975/.625 M6 G0 X-5 Y3

The Z axis will retract to the home position with a G0, the axes will go to the X -5 Y3 position, an automatic M5 and M9 will be issued to turn off the spindle and coolant pump, the following message will be output:

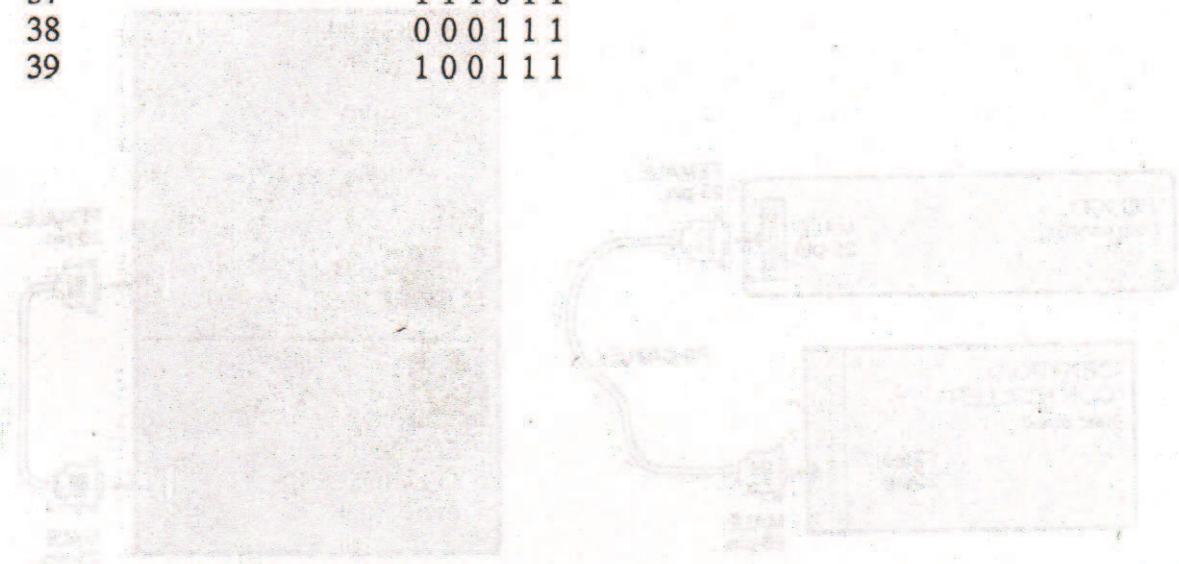
Automatic Tool Change, Awaiting Switch Input to Continue.

The relays connected to BCD 1 and 4 will turn on. One second later the TCH relay will turn on. When ACK is toggled, the program will then continue, and all BCD relays will turn off.

The chart below indicates which relays will be activated for a specific tool number from 0 to 24 or for a specific spindle speed 0 through 39, before the strobe (TCH) or (SPD) is turned on:

TOOL NUMBER BCD Nos.	RELAYS ACTIVATED 1 2 4 8 10 20 1=ON 0=OFF
0	0 0 0 0 0 0
1	1 0 0 0 0 0
2	0 1 0 0 0 0
3	1 1 0 0 0 0
4	0 0 1 0 0 0
5	1 0 1 0 0 0
6	0 1 1 0 0 0
7	1 1 1 0 0 0
8	0 0 0 1 0 0
9	1 0 0 1 0 0
10	0 0 0 0 1 0
11	1 0 0 0 1 0
12	0 1 0 0 1 0
13	1 1 0 0 1 0
14	0 0 1 0 1 0

15	1 0 1 0 1 0
16	0 1 1 0 1 0
17	1 1 1 0 1 0
18	0 0 0 1 1 0
19	1 0 0 1 1 0
20	0 0 0 0 0 1
21	1 0 0 0 0 1
22	0 1 0 0 0 1
23	1 1 0 0 0 1
24	0 0 1 0 0 1
25	1 0 1 0 0 1
26	0 1 1 0 0 1
27	1 1 1 0 0 1
28	0 0 0 1 0 1
29	1 0 0 1 0 1
30	0 0 0 0 1 1
31	1 0 0 0 1 1
32	0 1 0 0 1 1
33	1 1 0 0 1 1
34	0 0 1 0 1 1
35	1 0 1 0 1 1
36	0 1 1 0 1 1
37	1 1 1 0 1 1
38	0 0 0 1 1 1
39	1 0 0 1 1 1



CONTROLLER CABLE CONNECTIONS

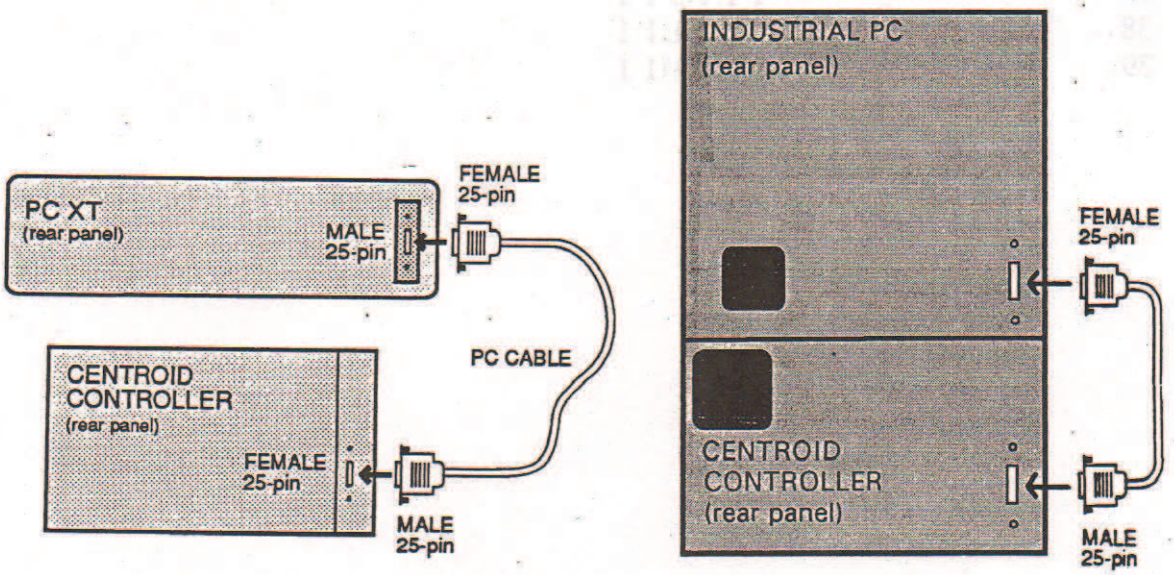
Your PC (or compatible) must have a serial card (RS- 232) and an 80 column monitor to operate with the CNC4. In addition, you will need communications software such as CENCOMM by Centroid, CROSSTALK by Microstuff, Inc., MIRROR, or PROCOMM PLUS.

The IBM-PC can be connected to the CNC4 using the standard IBM serial port, COM1 or COM2. Most terminal communications programs provide ways to setup the communications port parameters under program control.

The RS-232 cable between the IBM-PC and the CNC4 can be a standard commercially available straight through cable. If your PC has a DB-9 pin connector for its serial port, a standard commercially available 25 to 9 pin adaptor must be used in addition to the 25 wire straight trhough cable.

CONNECTING THE INDUSTRIAL PC OR COMPATIBLE TO THE CONTROLLER

Connect the controller as shown below:



RS-232 SPECIFICATIONS FOR THE CNC4

- The Centroid CNC4 displays in 80 columns
- Controller has a female DB-25 S connector
- Baud rate is set to 9600 when shipped:
  - baud rate can be adjusted by switches inside controller
  - baud rates available: 300,600,1200,2400,4800,9600,19200
  - changing the baud rate is described in the manual section "Changing the CNC4 Baud Rate"
- The communications protocol is as follows:
  - No parity
  - 8 Data bits
  - 1 Stop bit
  - No handshaking

TROUBLESHOOTING COMMUNICATIONS PROBLEMS

If the CNC4 initially will not communicate with your terminal or computer, or if communications between the CNC4 and your terminal or computer cease, refer to the list below for help in finding the problem:

1. Check your computer or terminal for proper configuration (baud rate, port selection, etc.). Check all cables between the computer or terminal and the CNC4.
2. Review your RS-232 cable connections with the applicable diagrams shown in this manual. If you are using one of the computers listed, your cable connections must match the diagram exactly. If you are using a dumb terminal, you may have to experiment with different hand-shaking tie back combinations. Most initial communications problems are caused by improper RS-232 cable connections.
3. If you are using a computer, check your communications software settings (baud rate, port selection, data bits, etc.) for proper setup.
4. Check the Computer or terminal for proper operation. If possible, substitute a different computer or terminal and see if the problem goes away. Try running a backup copy of your communications software, if you are using a computer.

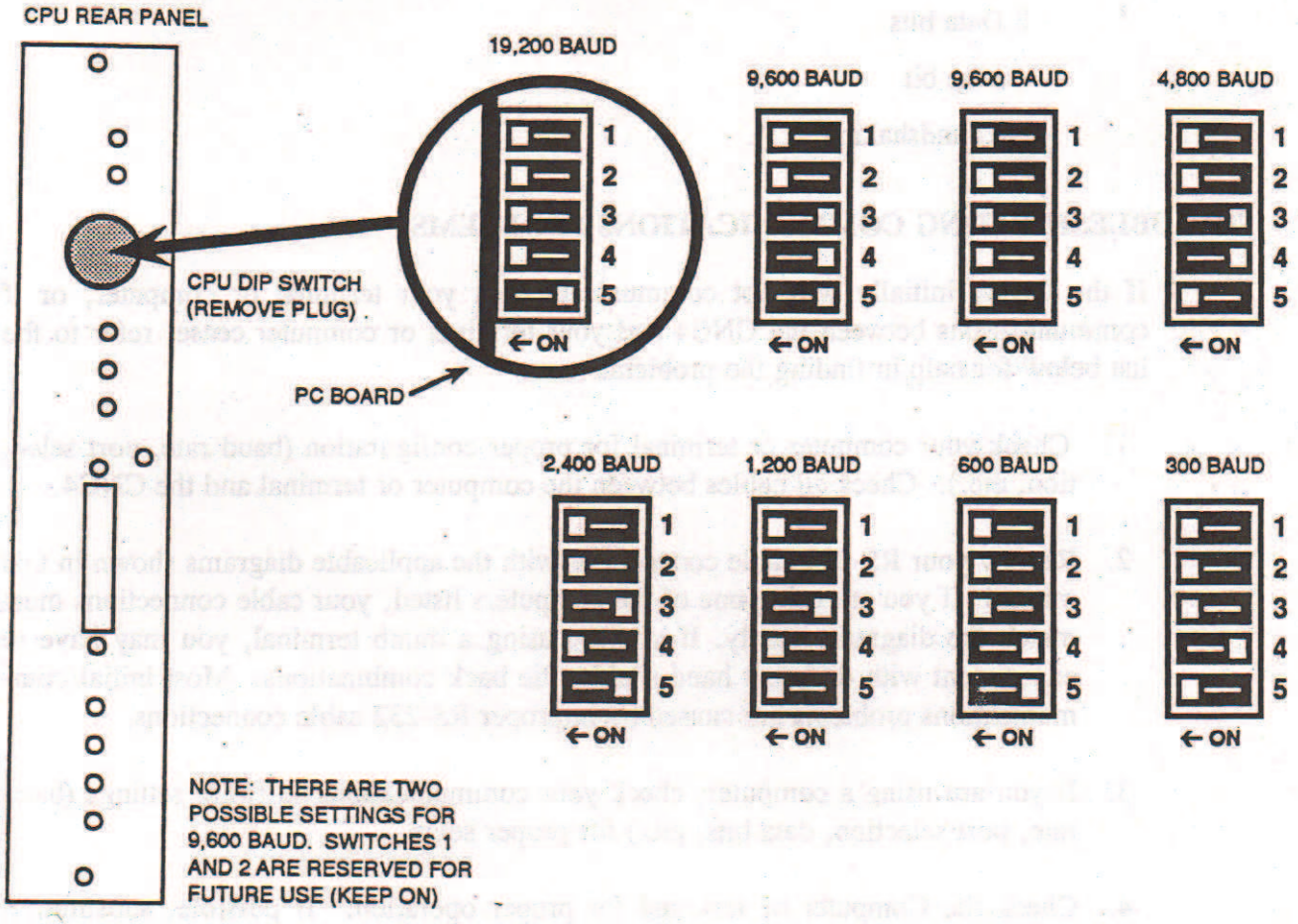
CHANGING THE CNC4 BAUD RATE

The CNC4 is shipped with its RS-232 transmission baud rate set at 9600. This setting can be changed if necessary to 19,200, 4800, 2400, 1200, 600, or 300 baud as required by your terminal, computer, or other device.

Baud rate changes can be accomplished by setting the baud rate dip switches (just above the RS-232 connector on the rear panel) to the appropriate settings as shown in the figure "RS-232 Baud Rate Adjustment".

Since these DIP switches are read only on initial power up, you must turn the CNC4 off before re-setting the BAUD rate. The CNC4 will read the new DIP switch setting on the next power up.

RS-232 BAUD RATE ADJUSTMENT



TROUBLESHOOTING POSITIONING PROBLEMS

Assuming that there are no mechanical problems such as loose pulleys or couplings, binding caused by improper machine lubrication, or excessive backlash, most positioning errors are caused by three things:

1. Improper chopper drive output current settings.

If chopper drive current setting is too high or too low, you may notice erratic operation during very slow step rates. Excessive heat in the motors (too hot to touch !) is an indication of excessive current. If chopper drive current is set too low, your motors may stall under a heavy load, or they may stop during acceleration to high speed.

2. Improper selection of motor parameters

If the operator selects the wrong motor parameters for his installation, performance will be poor. It is also possible for the operating limits of the motors to be exceeded. The Maximum Rate is the parameters which must chosen carefully. Some experimentation may be necessary to find the optimum settings for positioning accuracy and repeatability. See the manual section "Setup Menu Option 7, Motor Parameters and Limit Switches".

IMPORTANT

Remember that although the CENTROID CNC4 is capable of producing rates of 100,000 steps per second, most motors are not be capable of running that fast. Do not try to run the motors faster than they will reliably go!

3. Motor is too small for its intended use.

Stepper motors develop their maximum torque when the motor is not moving and power is applied to the windings. This is known as holding torque. Torque falls off as motor speed increases. If your dynamic (moving) torque requirement is 150 oz/in and you choose a motor rated at 150 oz/in of holding torque, your speed will be severely limited. If however, you choose a much larger motor, it will still have the required 150 oz/in of torque at a much higher speed.

MOTOR DRIVE CURRENT ADJUST (BACKGROUND INFORMATION)

For good performance, it is imperative that the motor drive current is set properly for the motors you intend to drive. If you purchased motors from Centroid when you purchased your CNC4, or if you specified the motor type to be used, then the motor drive currents will be pre-set and should not require further adjustment.

In general, motor output current should be set to a value which is as close to (but not more than) the maximum rated current for your motor. Sometimes motors have two different current ratings, one for parallel windings and one for series windings. In addition there may be 2 sets of parallel or series current ratings; 1 for unipolar and 1 for bi-polar drives. The CNC4 uses bi-polar drives.

For example, if you have an 8 lead stepper motor which you have wired in the parallel configuration. The motors bipolar parallel current rating is 6.3 amps per phase. You would set the motor drive output current for that motor to a level which is as close to, but not over 6.3 amps. The closest setting would be 6 amps.

If you have a six lead (series) motor that you wish to connect to the CNC4. The motor's bi-polar current rating is 15 amps. In this case, you must set the drive output current to 8 amps.

HIGH CURRENT MOTOR DRIVE - OUTPUT CURRENT ADJUSTMENT

You must remove the CNC4 front panel to adjust the motor drive output current.

**** IMPORTANT ****
MAKE SURE THE CNC4 IS UN-PLUGGED
FROM THE AC POWER OUTLET
BEFORE THE FRONT COVER IS REMOVED

The following steps should be followed exactly for front panel removal:

1. Turn the CNC4 front panel power switch to the OFF position.
2. Unplug the CNC4 from the AC outlet and let the unit sit for a few minutes so that the motor power supply filter capacitor can discharge.
3. Remove the 8 #6-32 phillips head screws holding the front panel in place.

4. Gently pull apart the connector attached to the wires going to the power switch-circuit breaker. The plug is polarized - it will only fit together one way. Gently pull apart the small connector leading to the PAUSE button, and then gently remove the ribbon cables from the feedrate override card.

5. The figure "CNC4 with Front Panel Removed" illustrates the position of the drive cards inside the CNC4. The drive cards have an 8 position dip switch which is used to select the output current for each motor.

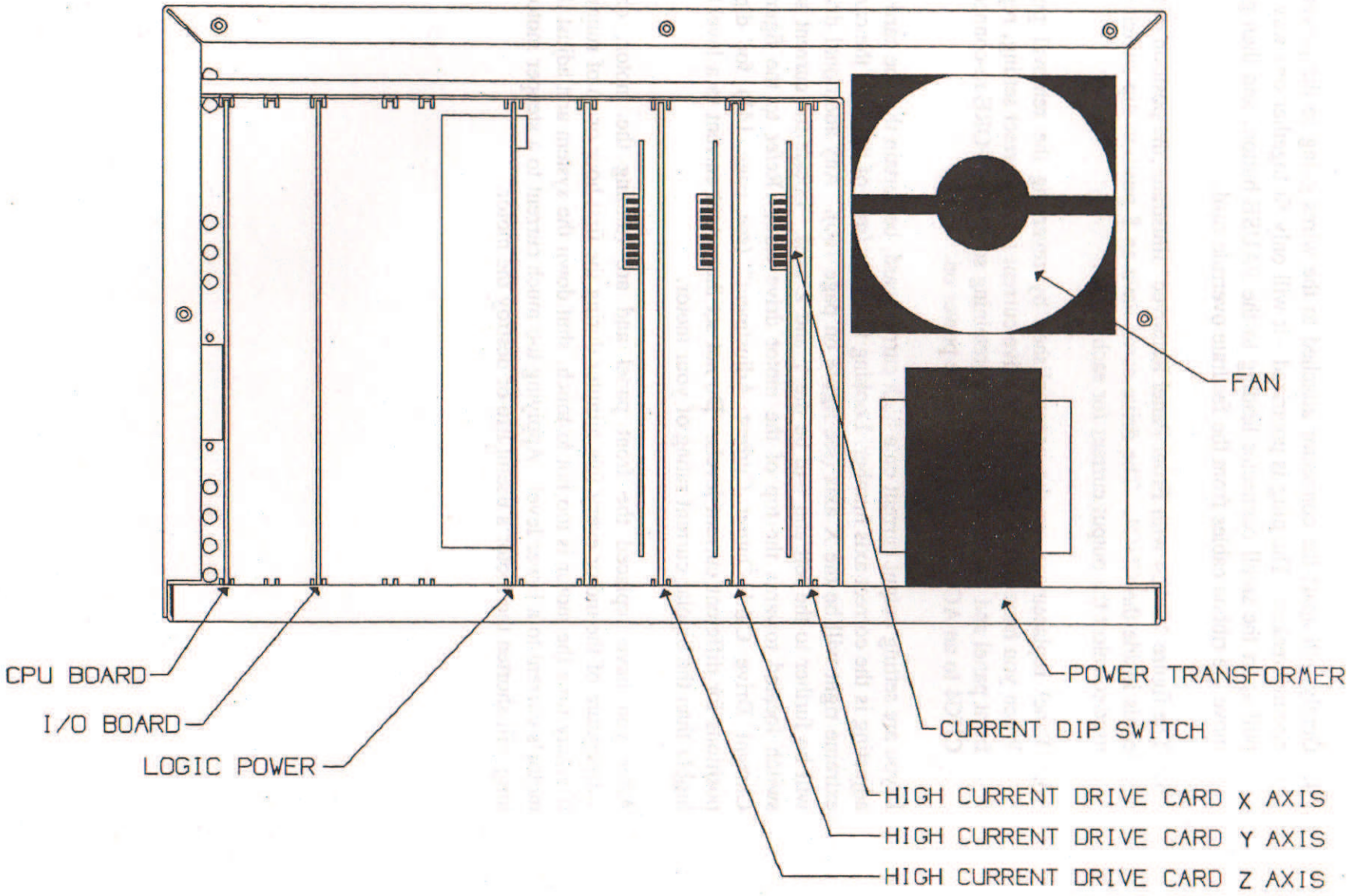
6. Panel replacement can be accomplished by reversing the removal procedure. When you have adjusted the motor drive current to the correct setting, replace the front panel and install the front panel retaining screws BEFORE re-connecting the CNC4 to an AC outlet and turning the power on.

If you are setting output current on a high current card, be certain that the card you are adjusting is the correct axis number. Looking from the front of the unit, the card on the extreme right will be the X axis (see figure on page 140). Any additional drive cards will be further to the left and will be the Y and Z axes. Locate the current select dip switch located towards the top of the motor drive card. Refer to the figure "High Current Drive Card Output Current Adjustment" (see page 141) for dip switch positions for different current levels. Do not set the output current to a level which is higher than the bipolar current rating of your motor.

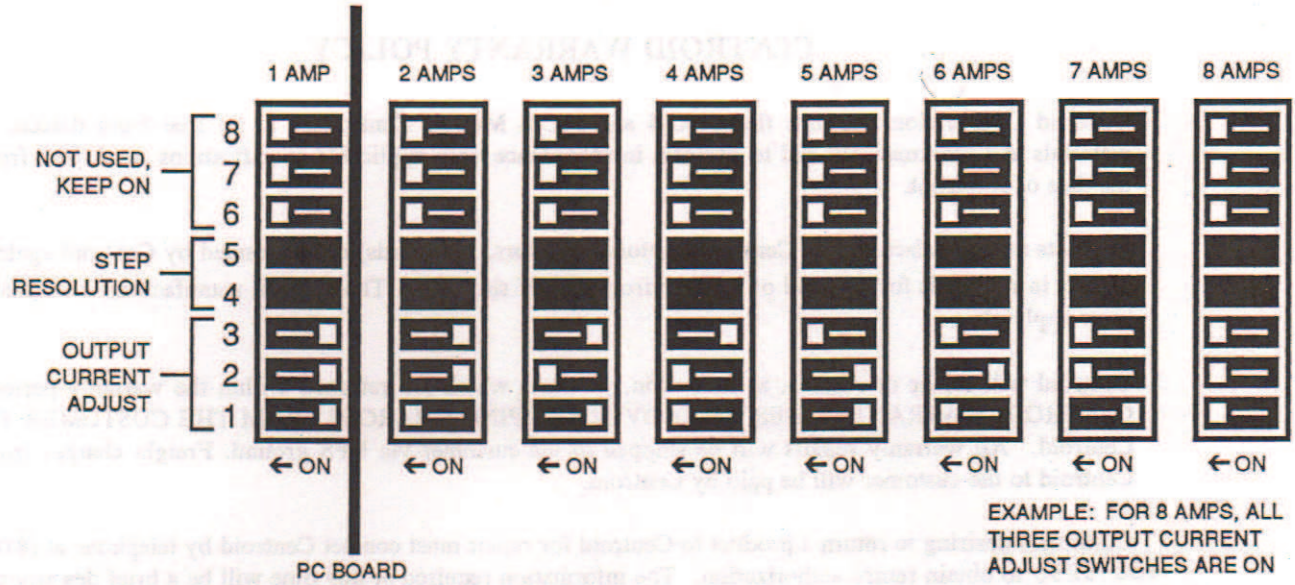
After you have replaced the front panel and are running the motor, check the temperature of the motor every few minutes during the first hour or so of running time. If at any time the motor is too hot to touch, shut down the system and adjust the warm motor's current to a lower level. Applying too much current to a stepper motor for too long will shorten the motor's useful life or destroy the motor.

CNC4 WITH FRONT PANEL REMOVED

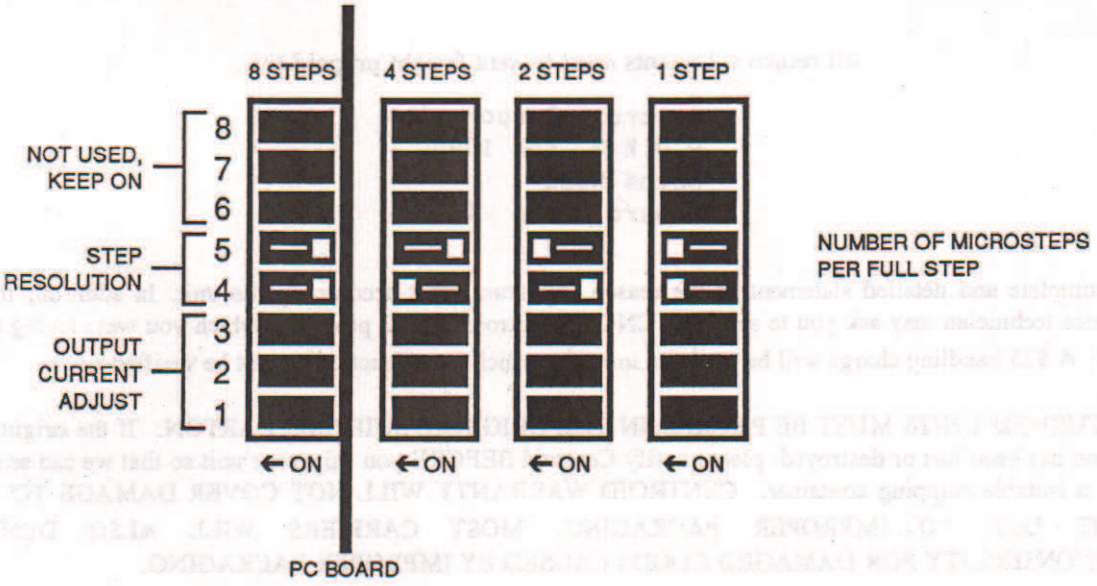
VIEWED FROM THE FRONT



HIGH-CURRENT DRIVE CARD OUTPUT CURRENT ADJUSTMENT



HIGH-CURRENT DRIVE CARD STEP RESOLUTION ADJUSTMENT



MICRO1 REV. 890913 02-13-90

MICRO1 REV. 891102

CENTROID WARRANTY POLICY

Centroid Corporation warrants the CNC-4 and MC-8 Motion Controllers to be free from defects in materials and workmanship and to perform in accordance with applicable specifications for 1 year from the date of shipment.

Products not manufactured by Centroid (motors, monitors, keyboards) are warranted by Centroid against defects in materials for a period of 30 days from date of shipment. The original manufacturer's warranty may apply also.

Centroid will repair or replace, at its option, products which are returned within the warranty period. CENTROID WARRANTY DOES NOT COVER SHIPPING CHARGES FROM THE CUSTOMER TO Centroid. All warranty repairs will be shipped to the customer via UPS ground. Freight charges from Centroid to the customer will be paid by Centroid.

Customers desiring to return a product to Centroid for repair must contact Centroid by telephone at (814) 353-9290 to obtain return authorization. The information required at this time will be a brief description of the problem, the model and serial number, and the date of purchase. All returns to Centroid must have a return authorization number written on the shipping carton or shipping label. UNAUTHORIZED RETURNS TO CENTROID WILL BE REFUSED.

All return shipments must be sent freight prepaid to:

Centroid Corporation
R.D.# 3 Box 104D
Gates Road
Howard, PA 16841

A complete and detailed statement of the reason for return must accompany the unit. In addition, the service technician may ask you to send any CNC or Centroid BASIC programs which you were trying to run. A $25 handling charge will be made on units for which a malfunction cannot be verified.

RETURNED UNITS MUST BE PACKED IN THE ORIGINAL SHIPPING CARTON. If the original carton has been lost or destroyed, please notify Centroid BEFORE you ship your unit so that we can send you a suitable shipping container. CENTROID WARRANTY WILL NOT COVER DAMAGE TO A UNIT DUE TO IMPROPER PACKAGING. MOST CARRIERS WILL ALSO DENY RESPONSIBILITY FOR DAMAGED GOODS CAUSED BY IMPROPER PACKAGING.

Centroid warranty will not cover damage to a CNC or MC controller caused by abuse, negligence, or ignoring the operating precautions contained in the manual. In addition, Centroid warranty will not cover damage to products not manufactured by Centroid which have been abused, shipped, or handles improperly.

Centroid warranty will not cover damage to any product which shows evidence of attempted repairs by anyone other than Centroid. Centroid warranty will not cover damage to a stepping motor which has been dropped, shipped in improper containers, or taken apart.

CENTROID

CENCOMM

Terminal Communications Software

User's Manual

REV 910730

INTRODUCTION

General Information

CENCOMM is an easy to use terminal communications program written for the IBM PC, XT, AT or compatible computer. This software allows the user of Centroid MC or CNC controllers to create an ASCII text file, transmit the file to the controller, capture a file from the motion controller, and save a file to disk.

System Requirements

CENCOMM operates on a variety of IBM PC's and compatibles. Minimum system requirements are:

- An IBM PC, XT, or AT (or compatible)
- 256K RAM minimum
- Any display card/monitor combination
- One 5.25" or 3.5" floppy drive
- RS232 Serial Port (COM 1 or COM 2)

Note: A Hercules Monochrome, EGA, VGA, or compatible graphics adaptor is required for operation of the Digital Read Out (DRO) feature.

The following hardware is optional BUT is recommended for fast operation of CENCOMM:

- Second floppy drive or hard disk

Conventions Used in This Manual

The following conventions are used throughout this manual to indicate which keys are to be pressed, or what is to be typed on the keyboard.

Text shown in **TYPEWRITER TYPE** indicates that the letters should be typed by the user from the keyboard.

Words contained within the "less than" and "greater than" signs such as **<ENTER>** or **<ESC>** indicate that those keys should be pressed. Multiple words contained within "<" and ">" such as **<ALT F1>** or **<CTRL F>** indicate that the first key should be pressed and held while the second is pressed. Words or letters not shown in **TYPEWRITER TYPE** or contained within "<" and ">" should not be pressed or typed on the keyboard.

CENCOMM <ENTER>

indicates that the letters CENCOMM should be typed into the keyboard, then the key marked ENTER should be pressed. The instruction line:

<CTRL C>

indicates that the CONTROL key should be held down and the C key should then be pressed.

Words or letters shown in ***italics*** are command prompts issued by CENCOMM. For example,

FILE TO SAVE?_
<ESC> to exit

Words or phrases contained within parenthesis, ( ), are comments or instructions to the operator. For example,

Search String: **ABC**
(where 'ABC' is the string to search for)

INSTALLATION PROCEDURE

Connecting Your Computer to the Centroid Controller

In order to communicate with a Centroid controller, you must first connect the controller to your computer's serial port. Use the cable supplied with the Centroid controller. When you have connected the controller to the computer, turn the controller on and run CENCOMM as per the following instructions.

Running CENCOMM Without a Hard Drive

First make a back-up copy of the CENCOMM distribution diskette and store the original in a safe place. NEVER use the CENCOMM distribution diskette as your working diskette. Place your back-up copy of the CENCOMM distribution diskette into the A: drive and enter the following at the DOS prompt:

CENCOMM <ENTER>

Installing CENCOMM on a Hard Drive

CENCOMM consists of only one file, CENCOMM.EXE, on your distribution diskette. To install CENCOMM on your hard drive, simply copy CENCOMM.EXE to your desired directory.

RUNNING CENCOMM

To run CENCOMM, go to the directory in which you have placed CENCOMM.EXE and type the following:

CENCOMM <ENTER>

After a few seconds, the computer screen will display a version number on top of the screen and CENCOMM will be sitting in Terminal mode. At the bottom of the screen you will notice one line which includes the following functions:

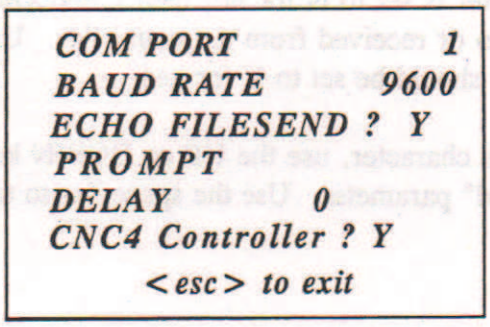
F1-Load: F2-Save: F3-Recv: F4-Send: F5-Edit: F6-Port: F7-Capt: F8-DRO: F10-Exit: Term

This line shows you what operation each function key performs. The line also displays the current mode CENCOMM is in, either Terminal or Edit. In this case, CENCOMM is in the Terminal mode as displayed at the right end of the line.

TERMINAL MODE FUNCTIONS

Setting the COM Port Options <F6>

Press <F6> on your computer keyboard. You will see a window appear in the upper left hand side of the screen which looks like this:



This box displays the current COM port settings and allows you to change the COM port settings as necessary.

COM Port

This parameter determines which of your computer's serial ports CENCOMM communicates through. CENCOMM supports either COM 1 or COM 2. You should set this parameter to match the port that the Centroid controller is connected to.

To edit this parameter, press the UP or DOWN arrow keys to move the cursor under the "COM Port" parameter. Press the space bar to toggle between 1 or 2.

Baud Rate

This is the speed at which data is sent and received between the controller and the Centroid controller. This setting MUST match the baud rate of the Centroid controller. Centroid controllers are shipped from the factory set at 9600 baud unless otherwise requested by the customer.

If you have changed the baud rate of your controller, or have ordered your controller set to something other than 9600, you can set CENCOMM to match your controller by using the UP or DOWN arrow keys to move the cursor under the "BAUD RATE" parameter. Press the space bar to toggle to the correct value.

Echo Filesend?

This parameter determines whether or not you will see a file as it is sent to or received from the controller. If this parameter is set at Y for yes, CENCOMM displays the file on the screen as it is being sent to or received from the controller. If this parameter is set to N for no, then CENCOMM will not display the file as it is being sent to or received from the controller. Unless you have a specific reason, this parameter should be set to Y for yes.

To change this character, use the UP or DOWN keys to move the cursor under the "Echo Filesend" parameter. Use the space bar to toggle between Y or N.

Prompt Character

This parameter defines a character from the controller that CENCOMM waits for before it sends the next line of the file. If you have a CNC controller, this character is normally left blank. When the prompt character is blank, CENCOMM

does not wait before it sends the next line. However, if you want to use the RUN IMMEDIATE mode to execute an extremely long CNC program, this character should be set to a question mark, "?." See the section "Line-by-Line Down-Loading" toward the end of this manual for more details about running long programs.

To edit this character, use the UP or DOWN arrow keys to move the cursor under the "Prompt Character" parameter. Type in the desired character and press the <ENTER> key. If you want to set the character as blank, press the space bar and then press the <ENTER> key.

Delay

With the great number of different computers on the market running at different speeds, it may sometimes be difficult for both the computer and the controller to send data and receive data simultaneously. For this reason, CENCOMM allows you to artificially slow down the process so that both the computer and the controller have time to receive a character, digest it, and get ready to send or receive the next character. This delay value does not change the BAUD rate, but merely induces an artificial delay between each character sent and received. Remember that at 9600 BAUD, characters are being sent across the RS-232 cable at a rate of 960 characters per second. If you are sending a file to the controller and the screen displays "garbage" or errant characters, you should increase the DELAY value.

To change the "DELAY" value, use the UP or DOWN arrow keys to move the cursor to the "Delay" parameter. Type in the new value and then press the <ENTER> key.

CNC4 Controller ?

If you are using CENCOMM with a CNC4 controller, this parameter should be set at Y for Yes. When this character is set at Y, files are automatically up-loaded and down-loaded when the F3 and F4 keys are pressed respectively.

To edit this parameter, press the UP or DOWN arrow keys to move the cursor under the "CNC4 Controller ?" parameter. Press the space bar to toggle between Y or N.

Saving the COM Port Settings

When you are done changing the COM port settings, press the <ESC> key. This causes the COM Port box to disappear and saves any changes to disk so that you will not have to re-configure the COM ports each time you run CENCOMM.

Load <F1>

CENCOMM uses an internal buffer to hold any file which is to be edited, sent, or received. Pressing the <F1> key enables you to load an existing file from disk into the CENCOMM buffer where it can then be edited or sent to the controller. Note that if <F1> is pressed and the CENCOMM buffer is not empty, the following warning message appears in the middle of the screen before the Load prompt appears:

Save current file?
Press <Y> to save the current file
Press <N> to discard the current file
Press any other key to continue

If you wish to save the CENCOMM buffer's contents to disk before loading a new file press **Y**. If you wish to clear the CENCOMM buffer press **N**. Press any other key to cancel the LOAD command and leave CENCOMM's buffer untouched.

When CENCOMM's buffer is empty, CENCOMM will display the following prompt in the upper left hand portion of the screen:

FILE TO LOAD?_
*<ESC> to exit, <Return> for directory of *.cnc*

Type in the name of the file that you want to load and press the <ENTER> key or press the <ENTER> to display all files with the "CNC" extension on the screen. Asterisks separated by a period, *.*, can be entered to see all files with any extension in a specific directory. Drive and path information may be included if the file to be loaded or the files to be listed with an asterisk do not reside in the current directory or drive. If a program is loaded in the Terminal mode the following message will appear to indicate that the file loaded correctly:

File successfully loaded.

If a program is loaded while in the EDIT mode the file will scroll on the screen while it is being loaded.

Save <F2>

This function allows you to save the contents of the buffer to a disk file. After you press the <F2> key, the following prompt appears in the upper left hand corner of the screen:

FILE TO SAVE?_
<ESC> to exit

Type in the name that you want the current file to be saved under and press the <ENTER> key. If there is already a file on disk with the same name as the one you typed, CENCOMM asks you if you want to overwrite it. If you want to write over the existing file, type **Y** and then press the <ENTER> key. If you do not want to write over the existing file, type **N** and press the <ENTER> key. Retype a different name and then press the <ENTER> key to save the file under a different name. Drive and path information can be included in the filename if you want to save the file to another directory.

Pressing the <ESC> key in response to the prompt causes CENCOMM to return immediately to Terminal or Edit mode without loading a file.

Receive <F3>

This function allows you to capture a file from a CNC4 controller. This function can only be used if the COM Port setting "CNC4 Controller ?" is set to **Y**. If CENCOMM is being used to up-load a file from another device the COM Port setting "CNC4 Controller ?" should be set to **N** and the "Capture" function <F7> should be used. As the file is being received, it is loaded into the CENCOMM buffer. Once the file is in the buffer, it can then be edited or saved to disk. Note that, if <F3> is pressed and the CENCOMM buffer is not empty, the following warning message appears in the middle of the screen:

Save current file?
Press <Y> to save the current file
Press <N> to discard the current file
Press any other key to continue

If you wish to save the CENCOMM buffer's contents to disk press **Y**. If you wish to clear the CENCOMM buffer press **N**. Press any other key to cancel the Receive command and leave CENCOMM's buffer untouched.

Send <F4>

You must be in CENCOMM's Terminal mode to send a file and the COM Port setting "CNC4 Controller ?" must be set at **Y**. Press <F1> to load the file from disk into CENCOMM's buffer. Eliminate any blank lines or junk characters before sending the file or the file may not load properly. If you are in the Edit mode press <F5> to enter the Terminal mode. Press <F4> to send the file to the controller.

Capture <F7>

Although it is more convenient to use the Receive command, <F3>, to up-load files from a CNC4 controller, the Capture command can be used. If CENCOMM is being used to up-load files from controllers other than the CNC4, the COM Port setting "CNC4 Controller ?" should be set to N and the Capture command must be used.

The method used to up-load files from controllers differs depending on the manufacturer but will be similar to the following method. To capture a file from a Centroid CNC controller, go to the CNC Edit Menu. Type **L** for list in response to the CNC Edit Menu prompt. Press <F7> to turn on the CENCOMM Capture function and then press <ENTER>. As the CNC controller lists its program on the screen, CENCOMM captures the listing and stores it in the CENCOMM buffer.
When the CNC controller has finished listing the program on the screen, press the <F7> key again. The captured file is now residing in the CENCOMM buffer. Edit out any blank lines at the top or bottom of the program before saving it to disk or the program will not load back into the controller next time. Remember to save the program to disk <F2> before you perform another Capture <F7> or Load <F1> or your captured file will be lost.

Digital Read Out (DRO) <F8>

The Digital Read Out (DRO) mode is selected by pressing the <F8> key from the CENCOMM terminal screen. Your computer must have a Hercules Monochrome, EGA, VGA, or compatible graphics adapter in order to use DRO. If proper graphics hardware is not in your computer, a message will be displayed explaining that the DRO cannot be used.

The DRO mode displays the CNC controller's XYZ coordinates in a large format on the computer screen. It also provides the CNC status messages in a smaller area below the coordinates and allows user input when prompted by the CNC controller.

Once selected, the DRO display will update and display the latest coordinate data provided by the CNC controller. Certain CNC status messages will also be displayed (sometimes very briefly) during CNC operation. However, if the CNC requires user input, the prompt will always be displayed until the user responds. Type the necessary response as you would in the normal CENCOMM mode and press <ENTER>. All other CENCOMM F-key functions are unavailable while in DRO mode.

To exit DRO mode and return to the CENCOMM terminal screen, press <F8>. CENCOMM will then display a full screen of CNC data transmitted before exiting DRO, so that the most recent actions or problems may be retraced. The number of lines printed may vary depending on the number and type of operations performed while in DRO mode. It is possible to switch back and fourth between the DRO and CENCOMM terminal screens while the CNC is running.

EDIT MODE FUNCTIONS

Edit <F5>

Pressing <F5> changes the current CENCOMM mode from the Terminal mode to the Edit mode. This function acts like a word processor and allows you to edit the contents of the CENCOMM buffer. You can use this function to make changes to a file prior to Saving <F2> or Sending <F4>. You can also create a file from scratch and then Save it, <F2>, or Send it, <F4>. The CENCOMM editor does not hold any file larger than 60,000 bytes. If a file is larger than 60,000 bytes, only the first 60,000 are loaded.

CENCOMM is a full screen editor and you can use the arrow keys to move around. The editors reminder line appears as follows:

F1-load: F2-save: F3-find: F4-replace: F5-Terminal: F6-Help: Edit

The word *Edit* at the right side of the line is an indicator letting you know you are in the CENCOMM editor. The major functions of the editor are as follows:

Load <F1> and Save <F2>

These commands are identical to the Terminal mode Load <F1> and Save <F2> functions.

Find <F3>

Search for a particular string in a file. After pressing <F7>, the prompt:

Search String: **ABC**
(where 'ABC' is the string to search for)

will appear in the lower left hand corner of the screen asking for the string that is to be searched for. Type the string and press <ENTER>. If you wish to cancel without searching press <ESC>. The Find function is case sensitive (i.e. searching for "ABC" will not find "abc"). After pressing <ENTER>, the editor positions the cursor to the first occurrence of the string in the buffer. If the string is not found then the cursor position does not change.

Replace <F4>

Find and replace all occurrences of a particular string. After pressing <F4>, two prompts:

Enter string to replace: **ABC**

Enter replacement: **123**
(all occurrences of the string 'ABC' will be replaced with '123')

will appear in the lower left hand corner of the screen. The first prompt asks for the string to replace. Type the string and press <ENTER>. The second prompt asks for the replacement string. Type the string and press <ENTER>. After <ENTER> has been pressed following the second prompt, ALL occurrences of the string to replace in the buffer file are replaced with the replacement string. To cancel without replacement, press <ESC> in response to either prompt.

Terminal <F5> (or <ESC>)

Go to Terminal mode. Pressing <F5> or <ESC> changes the current CENCOMM mode from Edit to Terminal.

Help <F6>

Pressing <F6> while in Edit mode causes the following help screen to appear:

<Enter>	Next line
<Home>	Beginning of line
<End>	End of line
<Backspace>	Delete character left
<Del>	Delete character
<Ins>	On-Insert/Off-Overwrite
<Ctrl Y>	Delete line
<Ctrl L>	Delete to end of line
<Ctrl K>	Clear buffer
<Ctrl B>	Beginning of buffer
<Ctrl E>	End of buffer
<Ctrl G>	Restore from <CTRL L> delete
<Pg Up>	Page up one page
<Pg Dn>	Page down one page
ARROWS	Move direction indicated

<ANY KEY> to continue

The help screen gives a simple explanation of the edit mode special keys. Pressing any key erases the help menu and restores the screen contents.

Edit Mode Special Keys

The following special keys are used when editing the CENCOMM buffer. A description of the special key operations follow each key name.

<Enter> Causes the cursor to go to the next new line.

<Home> Causes the cursor to jump to the beginning of the current line.

<End> Causes the cursor to jump to the end of the current line.

Backspace Deletes the character to the left of the cursor. If backspace is hit while the cursor is in column 1, the current line is appended onto the previously line. If the current line does not fit on the previous line the editor just beeps.

<Del>	Deletes character at current position.
<Ins>	When insert mode is on, the cursor looks like a big white block and characters are inserted into the buffer. When insert mode is off, the cursor is a small block and characters are replaced in the buffer. If insert mode is on and the current line is full, the editor beeps and does not allow any more characters to be typed on that line. To toggle between modes use the <INS> key.
<Ctrl Y>	Deletes the current line.
<Ctrl L>	Deletes from current cursor position to the end of the line.
<Ctrl K>	Clear the contents of the edit buffer. (Note that if you wish to save the buffer contents, you must press <F2> to save before pressing <CTRL K>).
<Ctrl B>	Moves current position to the beginning of the buffer.
<Ctrl E>	Moves current position to the end of the buffer.
<Ctrl G>	Brings back the last line deleted by <CTRL L>.
Up/Down	Moves current position up or down one page at a time.
Arrow Keys	Moves the cursor in the direction indicated by the arrow pressed

CENCOMM USER TIPS

Receive a File From the CNC Controller

To capture a file from your CNC4 controller, you must be in the Terminal mode and the COM Port setting "CNC4 Controller ?" must be set at **Y**. If in the Edit mode press <F5> to enter the Terminal mode. Use the Receive command by pressing <F3>. If a program already resides in CENCOMM's buffer you will be asked if you want to save that file first or if you want to kill it from CENCOMM's buffer. When CENCOMM's buffer is empty the Receive command up-loads the file into CENCOMM's buffer.

Use the Edit function, <F5>, to edit the file if necessary. Press <F2> to save the file to disk or the file will be lost.

Sending a Disk File to the CNC4 Controller

You must be in CENCOMM's Terminal mode to send the file and the COM Port setting "CNC4 Controller ?" must be set at **Y**. Press <F1> to load the file from disk into CENCOMM's buffer. Eliminate any blank lines or junk characters before sending the file or the file may not load properly. If you are in the Edit mode press <F5> to enter the Terminal mode. Press <F4> to send the file to the controller.

Creating a File From Scratch

Press the <F5> key to enter CENCOMM's Edit mode. If there is something already in the buffer, use <CTRL K> to clear the buffer. If you want to keep the contents of the buffer, use the Save function <F2> to save the contents of the buffer, then use <CTRL K> to clear. Now you can just start typing. When you are done with the file you can save it to disk <F2> or send it to the controller <F4>. Be sure that there are no blank lines in the beginning or anywhere in the middle of your file or it will not load properly.

Editing an Existing File

To edit an existing file, press <F1> to load the file from the disk to the buffer. If in Terminal mode, press <F5> to enter Edit mode. Make your changes and press <F2> to re-save the file to disk.

LINE - by - LINE DOWN - LOADING

Some CAD/CAM systems now generate code for two axis circular controllers, like the CNC4, to simulate a three axis circular controller. The files generated by such systems are often larger than 24,000 bytes and therefore cannot fit into the controller. Files can, however, be down-loaded line-by-line to the controller using CENCOMM if the files are smaller than 60,000 bytes. For files larger than 60,000 bytes, other communication software must be used such as PROCOMM or MIRROR. The method used to allow CENCOMM to down-load a file line-by-line is as follows:

1. Use <F6> in the Terminal mode to change the setup parameter "PROMPT" to a question mark, **?**, so that when the controller is in the Run/Immediate mode the "Block ?" prompt tells CENCOMM to send the next line of code. If PROCOMM or MIRROR is being used the "LWAIT PROMPT" character should be a question mark.
2. Also change the parameter "CNC4 Controller ?" to **N** while in CENCOMM's setup mode and press <ESC> to save the changes and exit.
3. Load the file into CENCOMM's buffer by pressing <F1>.
4. If in CENCOMM's Edit mode press <F5> to enter the Terminal mode.
5. Choose the CNC4's Run/Immediate mode from the Control Menu (option 4).
6. Press <F4> to send the file.

A few things to consider when down-loading files line-by-line are:

- Cutter paths must be precompensated if left or right cutter compensation is used.
- Macros cannot be executed when using this method.
- Loops cannot be executed when using this method.
- It takes about twice as long to execute a program that is downloaded line-by-line than if the entire file were loaded directly into the controller's buffer. It may quicker to break the program into 20,000 byte sections and download the code in sections. If this method is used, care must be taken to insure the proper modes are maintained between sections such as G90's, G17's, G75's, G40, etc.

IPC Display Instructions

The purpose of this manual is to familiarize the user with the basic operating procedures necessary to install and operate the IPC-CNC4 combination. These instructions do not cover specific CNC programming. Refer to the CNC4 Users Manual for information on G and M codes.

The IPC sits on top of the CNC4 controller. The two units are bolted together via 2 side plates. Refer to the illustration IPCINS2.DWG. The IPC is connected to the CNC4 via an RS-232 cable and provides CNC program editing, disk storage, and transfer of CNC programs to and from the CNC4. Centroid provides the necessary software to operate the IPC and the CNC4 as an integrated system.

The standard software package is CENCOMM. CENCOMM provides a text editor, disk storage, and file transfer to and from the CNC4. CNC programs can be written , saved to disk, and then transmitted to the CNC4 for execution. In addition, existing CNC programs written on another PC or output from a CAD-CAM system can be transmitted to the CNC4 for execution.

The optional software package is called INTERCON. INTERCON turns the IPC-CNC4 into a conversational CNC controller. INTERCON allows the user to program a part directly from a blueprint without having to know G and M codes.

Specific instructions for the 2 different software packages can be found in either the CENCOMM or INTERCON users manuals.

INSTALLING THE IPC :

The IPC is normally used with a CENTROID control support swing arm. This arm attaches to the body of the mill and holds the IPC and CNC4 in a comfortable position. There are separate instructions for attaching the support arm to the mill. Once the arm is installed, the IPC can be bolted to the arm, and the CNC4 can be bolted to the IPC. The IPC - CNC4 is illustrated in the drawing IPCINS2.DWG which can be found at the end of this manual. Once the support arm, IPC, and CNC4 are bolted together, you are ready to connect the IPC to a 117 VAC power source and install the RS-232 communications cable between the IPC and CNC4. Connections to the CNC4 are covered in the CNC4 USERS MANUAL or the READ ME FIRST instructions included with CENTROID retrofit kits.

AC POWER:

The IPC and CNC4 both require 117 VAC to operate. This can either come from the optional HEAVY POWER ENCLOSURE or from a standard wall outlet. If you are using a wall outlet, be certain that there are no large motors or other high current devices running off of the same circuit. A separate circuit with its own circuit breaker is best.

INSTALLING THE RS-232 CABLE:

The IPC is connected to the CNC4 via a special "Y" cable. This cable has 1 male and 2 female DB-25 connectors. Insert the male end into the CNC4 RS-232 socket (See IPCINS1.DWG).. Secure this connector with the 2 screws located on either side of the connector. Do not over tighten these screws . Now insert the longer of the two female ends into the IPC COM 1 socket. Secure the connector with the 2 screws located on either side of the connector. Do not over tighten these screws.

The remaining female end allows you to connect a remote PC. With a remote PC running CENCOMM, you can edit and send files to the CNC4 from you office. If you are not connecting a remote PC at this time, use nylon cable ties to secure the remaining end out of the way.

TURNING THE IPC ON

Before turning the IPC on, you should make all the necessary connections to the CNC4 controller. Refer to the "READ ME FIRST" document included with retrofit kits or the CNC4 operating manual.

Now insert the appropriate disk into the IPC. If you purchased INTERCON, insert the INTERCON BOOT DISK. If you did not order INTERCON, then insert the IPC BOOT DISK. Remember that when inserting a disk into the IPC disk drive the label should face to the right and the metal sliding door on the disk goes in first. Note that if you purchased the hard disk option with your IPC, do not insert a floppy disk at this time.

After the correct disk has been inserted into the IPC, turn the power switch, located on the IPC rear panel, to the on position. It may take up to 1 minute for the IPC to "boot" (load the operating system and run CENCOMM or INTERCON). When the IPC is done "booting", the screen will display either the CENCOMM version and copyright notice (if you are using CENCOMM) or the CENTROID logo and then the INTERCON main screen (if you are using INTERCON). At this point, the IPC is ready for operation. If you are using INTERCON, refer to the INTERCON users manual. If you are using CENCOMM, you can communicate directly with the CNC4.

USING CENCOMM

When the IPC displays the CENCOMM copyright screen, you can verify the RS-232 connection between the IPC and CNC4 by pressing the ENTER key on the IPC keyboard. You should see the CNC4 CONTROL MENU, 9 menu choices , and the OPTION?_ prompt at the bottom of the screen. If you do not see the CONTROL MENU, check to be sure that the CNC4 is turned on, and that the RS-232 cable ends are plugged into the correct sockets on the CNC4 and IPC.

If you do see the CONTROL MENU, refer to the CENCOMM users manual for instructions on how to use the CENCOMM text editor, and how to send a CNC file to the CNC4. The CNC4 users manual covers G codes, M codes, and proper CNC program syntax.

USING INTERCON

After the IPC "boots" and the CENTROID LOGO is displayed on the screen, you can either press any key on the IPC keyboard or wait for a few seconds to see the main INTERCON SCREEN. To verify RS-232 communications between the IPC and the CNC4, press the F5 (Jogging and zeroing) key on the IPC keyboard. If there is an RS-232 communications problem between the IPC and the CNC4, the following message will be displayed on the screen for a few seconds:

ERROR SETTING UNITS OF MEASURE

If this error message appears, check to see that the CNC4 is turned on, and review the RS-232 "Y" cable connections. If INTERCON displays the JOGGING AND ZEROING screen, then RS-232 communications between the CNC4 and the IPC have been established. You are now ready to refer to the INTERCON USERS MANUAL.

OPTIONS

HARD DISK DRIVE

The IPC is available with an optional 40 MB hard drive. If your IPC has this option, do not insert a floppy disk into the IPC when initially powering up the IPC. Hard drive IPC's will "boot" automatically when the power is turned on.

PRINTER PORT

The IPC is available with an optional rear panel printer port. The connector is located on the rear panel (See IPCINS.DWG) and is a female DB-25 connector. This port is for connection to a printer with a standard Centronics parallel interface.

EXTERNAL KEYBOARD INPUT AND SELECTOR

The IPC is available with an external keyboard input connector and selector switch. This feature allows you to plug a standard PC-AT compatible keyboard into the rear panel of the IPC. The selector switch selects either the internal (membrane) keyboard or the external keyboard. The IPC will not work with an XT keyboard.

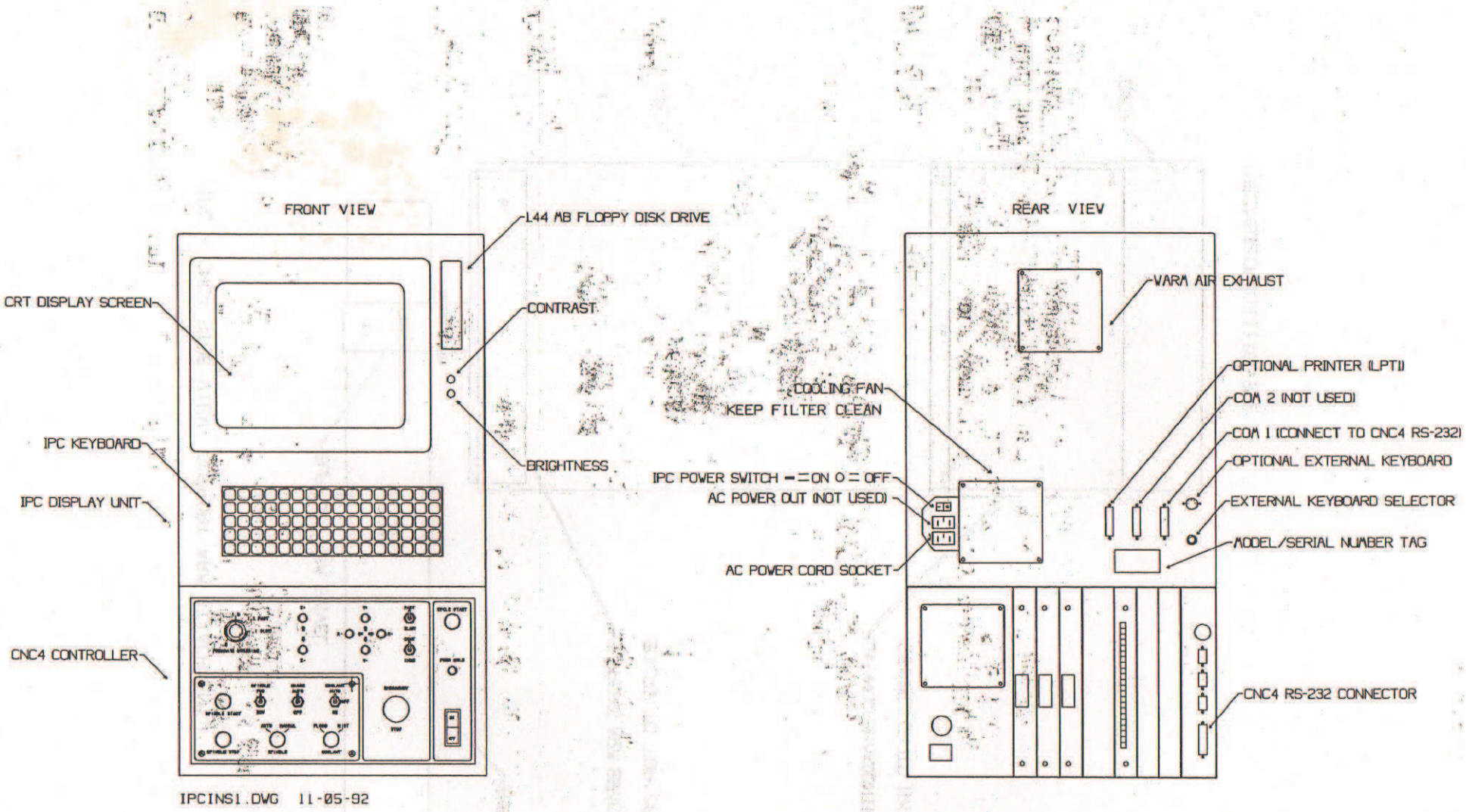
IPC TECHNICAL SPECIFICATIONS

For those users who are familiar with PC's and their operation, the following is provided for your general information:

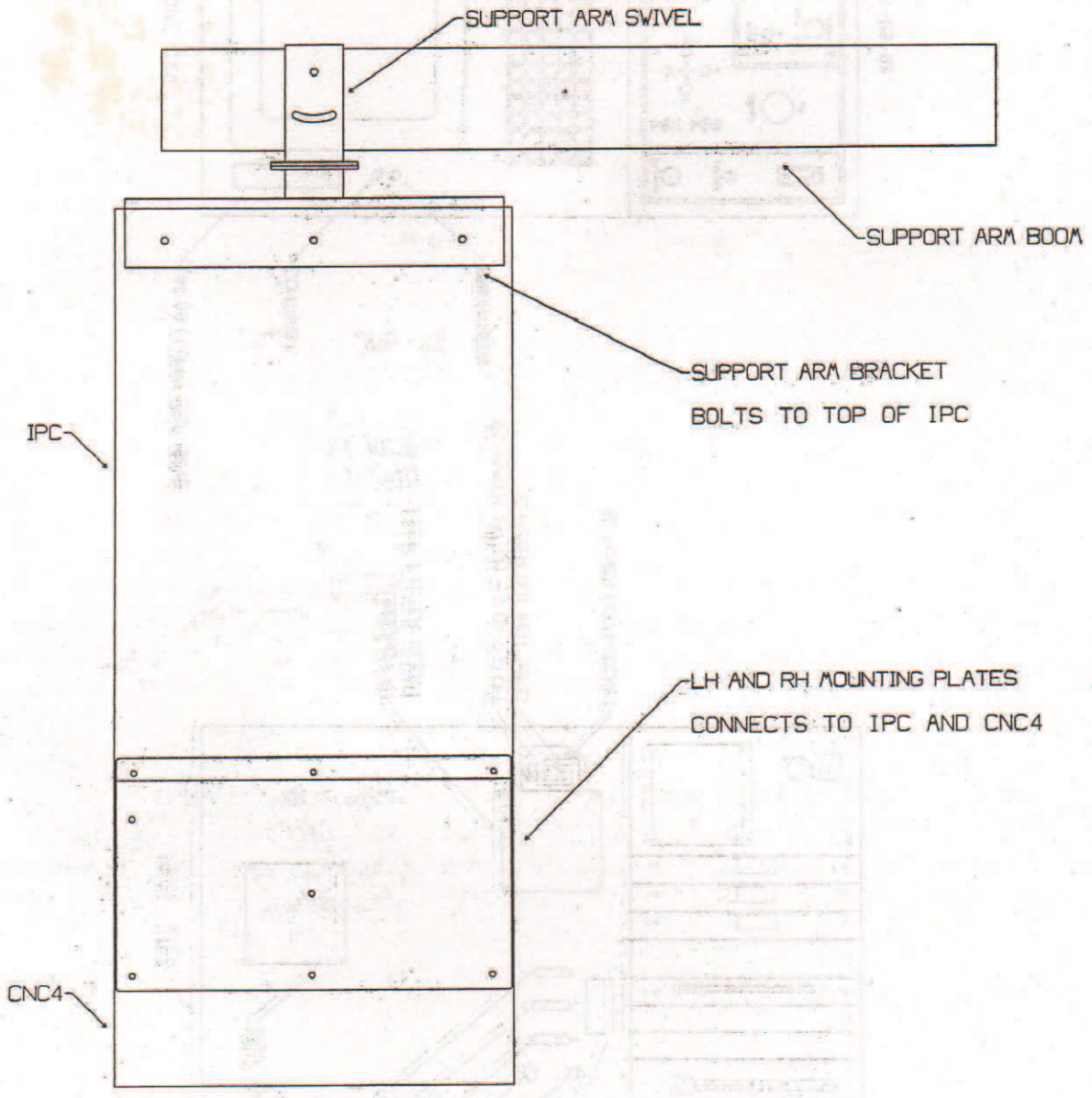
PROCESSOR TYPE : 80286, 12 MHZ
RAM: 1 MB 8 ea. 44256 4 ea. 41256 100 ns
DRIVE INTERFACE CARD: 2 Floppy, 2 IDE hard disk
DISPLAY ADAPTOR: 16 bit SVGA 256 K video RAM
DISK DRIVE: 1.44 MB 3.5" floppy (A:)
CRT DISPLAY : 14" monochrome VGA
OPERATING SYSTEM: MS DOS 5.0
KEYBOARD: QWERTY, membrane type
OPTIONAL HARD DISK: 3.5" frame size, 42mb and up (C:)
POWER SUPPLY: 200 watt 120/240 VAC input 50/60 HZ

IPC TECHNICAL SUPPORT

If you have questions or are experiencing difficulty with your IPC display unit, contact CENTROID at 814-353-9290 Monday thru Friday, 8-12 and 1-5, EST. Please have your IPC serial number and a brief description of your problem ready. If at all possible, a phone located close to the IPC will make problem solving go much faster.


FRONT VIEW
1.44 MB FLOPPY DISK DRIVE
CRT DISPLAY SCREEN
CONTRAST
BRIGHTNESS
IPC KEYBOARD
IPC DISPLAY UNIT
CNC4 CONTROLLER
REAR VIEW
VARA AIR EXHAUST
COOLING FAN
KEEP FILTER CLEAN
IPC POWER SWITCH — ON O OFF
AC POWER OUT (NOT USED)
AC POWER CORD SOCKET
OPTIONAL PRINTER (LPT1)
COM 2 (NOT USED)
COM 1 (CONNECT TO CNC4 RS-232)
OPTIONAL EXTERNAL KEYBOARD
EXTERNAL KEYBOARD SELECTOR
MODEL/SERIAL NUMBER TAG
CNC4 RS-232 CONNECTOR
IPCINS1.DWG 11-05-92

IPC - CNC4 SIDE VIEW SUPPORT ARM ATTACHMENT



IPCINS2.DWG 11-05-92