FEEDBACK INCORPORATED

CNC932 IBM OPTION

VOLUME 2 — SUPPLEMENTARY MANUAL

USE WITH FEEDBACK MANUALS

CNC932 Vol. 1 & and the manual

for the MIC926 interface card.

This manual makes reference to the products of IBM Corporation, P. O. Box 1328 West, Boca Raton, Florida 33432.

 CNC932/IBM ISSUE 2 REV.0 DECEMBER 1986

FEEDBACK INC., 620 SPRINGFIELD AVE., BERKELEY HEIGHTS, N.J. 07922

CNC932 IBM OPTION

VOLUME 2 — SUPPLEMENTARY MANUAL

CONTENTS

SEE ALSO VOLUME 1 — PROGRAMMING IN BASIC

CHAPTER 1 - PREPARATION OF USER DISK

1.1 — COMMENTARY

For the convenient use of the CNC932DD software it is very desirable that the disk you use should be self-starting (boots up automatically). To achieve this it is necessary to combine it on the same disk with the IBM Disk Operating System **(PC-DOS)**. It is also convenient to have IBM Advanced BASIC **(BASICA)** on the same disk. Because of international licensing and copyright restrictions, FEEDBACK cannot supply you with an operational **User Disk**, carrying **DOS** and **BASICA**.

We provide a master program disk from which you can prepare a **User Disk** using your own **DOS** and **BASICA** disks.

You have to carry out the following procedure only once to create your original user disk. To provide additional user disks, you need only copy the first one you created.

NOTE: USE THE FEEDBACK MASTER DISK ONLY TO CREATE USERS DISKS. NEVER USE THE MASTER DISK FOR EXPERIMENTAL WORK. ENSURE THE MASTER DISK IS WRITE PROTECTED AT ALL TIMES. WHEN NOT IN USE, FILE MASTER DISK AWAY SAFELY.

1.2 — MAKING THE USER DISK

A. This can be done on an appropriate PC with or without the CNC hardware connected. Hardware connection details are given in **Fig. 2.2**.

B. Hardware/software required

- Feedback master disk
- Clean, blank disk, 5 1/4" double sided, double density, soft sectored
- IBM-PC (or appropriate compatible) with 128K memory and dual disk drives
- System disks for **DOS** and **BASICA**

C. For IBM:

- Boot PC as usual with boot disk in drive **A**. This disk must carry **DOS** and **FORMAT** program.
- At prompt, type **FORMAT B: /S /V**
- At prompt, insert clean, blank disk in Drive **B**.
- When prompted, enter volume name **CNC.**
- When asked, **FORMAT ANOTHER (Y/N)** answer: **N**
- Remove boot disk from drive **A**:
- Transfer newly formatted disk from drive **B**: to drive **A**:
- Insert FEEDBACK master disk in drive **B**:

D. Perform ONE of the following steps depending on the specific configuration of your equipment.

a) If you will be using a Version 1 MIC926* interface card with an IBM monochrome card and monitor, type — **B:CNC M1**

b) If you will be using a Version 1 MIC926 interface card with any other graphics card or monitor, type — **B:CNC C1**

c) If you will be using a Version 2 MIC926 interface card with an IBM monochrome card and monitor, type— **B:CNC M2**

d) If you will be using a Version2 MIC926 interface card with any other graphics card or monitor, type — **B:CNC C2**

* Version 1 cards are not silk screened on the component side. Version 2 cards have silk screening on the component side. All MIC926 boards supplied after August, 1986, will be Version 2.

E. Computer responds with file copying message and lists copied files. This list should agree with the directory listing on the Feedback master disk label.

F. Remove master disk from Drive **B** and replace with **BASIC** disk. Follow on-screen directions. Remove disk from Drive **B**. You are now finished with the master disk which should be returned to secure storage.

G. The computer is actually now ready to run the CNC program and you can do this by just following the on-screen directions.

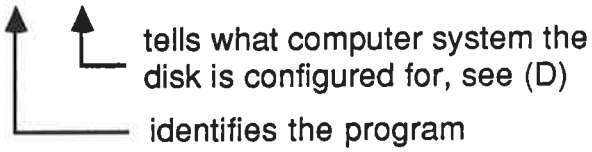
H. If you wish to make further copies of your **User Disk** at this time, you must power down the computer to break out of file in which you are currently working. Then restore the boot disk to Drive **A**. Power up the system and respond to Date/Time questions.

I. At prompt, type **DISKCOPY A: B:** Follow the on-screen directions to make more copies on clean, blank disks **(TARGET DISKS)** using your newly created user disk as **SOURCE DISK.**

J. If you do not want to run the CNC program or make further copies at this time, remove newly created user disk from Drive **A**, label it and store it safely.

label should:

* Identify the disk — **CNC. M1**



* Give the date created — so you can know you are working with a current disk

* Give the version number of the **Master Disk** used to create the **User Disk.** — V3.0 - this is so you can check with FEEDBACK on details of the program and keep track of updates.

* See **figure 4.2** for example

CHAPTER 2 - SYSTEM SET UP

2.1 — HARDWARE CONNECTIONS

2.1.01 Refer to **Fig 2.1** on next page and ensure the jumper links inside theCNC932 control unit are set for operation with IBM-PC.

2.2.02 Refer to **Fig 2.3** to configure the MIC926 card for **'MICA'** operation. Follow detailed instructions of Section 2 of MIC926 manual to install MIC926 card in the IBM-PC (Compatible computers will probably require minor changes in procedure). Set for base address of 768 (i.e. all address selector switches to **'OFF'**

2.3.03 Refer to **Fig. 2.2** that follows for connection details. If you have a multispeed computer you must set the clock frequency to 4.7 MHz

2.4.04 The **User Disk** (i.e. combined **CNC932 control + DOS + BASICA**) will boot-up automatically when inserted in Drive **A** and the computer switched on. The screen presents introductory information and a menu of available facilities for selection. The screen directions are clear and easy to follow and, when read in conjunction with the CNC932 manual, allow all of the coursework in Vol. 2 of the CNC932 manual to be carried out.

2.5.05 The following brief notes will add to the screen menu information on the principal facilities provided by CNC932DD.

2.3.0l. **TEST**

Turns the CNC932 cutter off and raises it to maximum height before executing 5 complete cycles of full-scale **X** and **Y** motion. This routine to some extent mimics a standard industrial warm-up program which is performed before any machining is attempted.

2.3.02 **ENTER**

Similar to **EDIT** (see below) but preceded by the option of erasure of a part program, called **MANUSCRIPT**, currently in memory.

2.3.03 **LIST**

Lists the contents of the currently stored **MANUSCRIPT**, either to a printer or to the screen.

2.3.04 EDIT

Allows the creation of a new **MANUSCRIPT** and the editing of existing ones. Uses 'arrow' keys to change line numbers and move cursor. Detailed screen directions are provided.

2.3.05 SAVE

Stores a current **MANUSCRIPT** on disk under a chosen file name. Also permits examination of the existing directory of files.

2.3.06 LOAD

Brings a chosen file name into current **MANUSCRIPT** storage, at the same time deleting any **MANUSCRIPT** already there. Also allows examination of the file directory.

2.3.07 MANUAL

Provides manual operation of the **X**, **Y**, and **Z** axes from the computer keyboard.

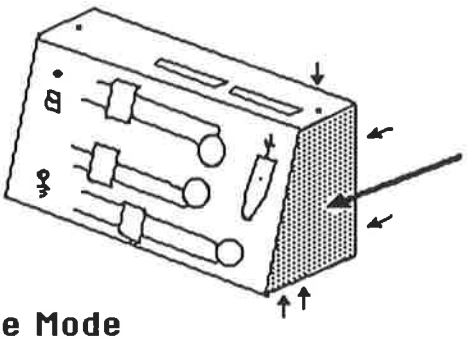
2.3.08 EXECUTE

Allows execution of the current **MANUSCRIPT**, however, it has been put there i.e. by **ENTER**, **EDIT** or **LOAD**.

2.3.09 TERMINATE

Terminates the CNC program and returns control to **BASICA**. To restart the CNC program press **F2 (RUN)** if the program is still in memoroy. Otherwise use standard reboot procedure.

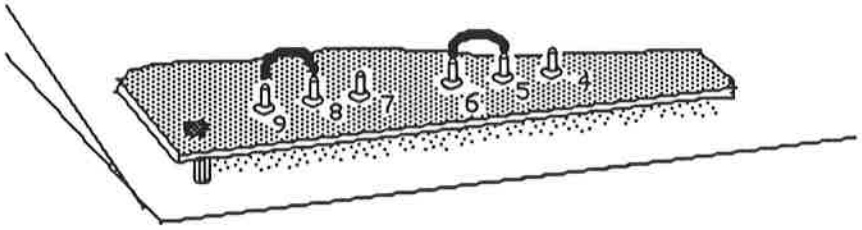
Jumpers for use with the IBM and Apple Computers



Remove the six screws holding this end cover and remove the cover to change the jumper links.

Apple Mode

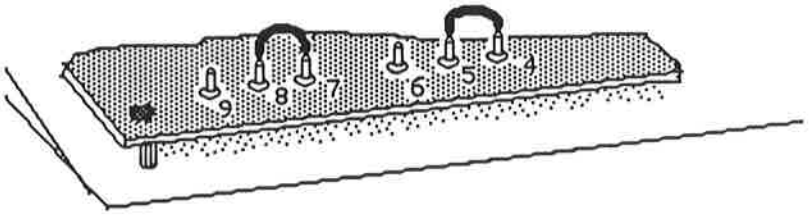
Inside you will see the numbered locations for the jumpers. For use with the APPLE connect 9 to 8 and 6 to 5 as shown



Only the jumpers shown should be in place. Replace the cover and the six screws.

IBM Mode

Inside you will see the numbered locations for the jumpers. For use with the I.B.M. , connect 8 to 7 and 5 to 4, as shown.



Only the jumpers shown should be in place. Replace the cover and the six screws.

FIGURE 2.1 - APPLE/IBM JUMPERS

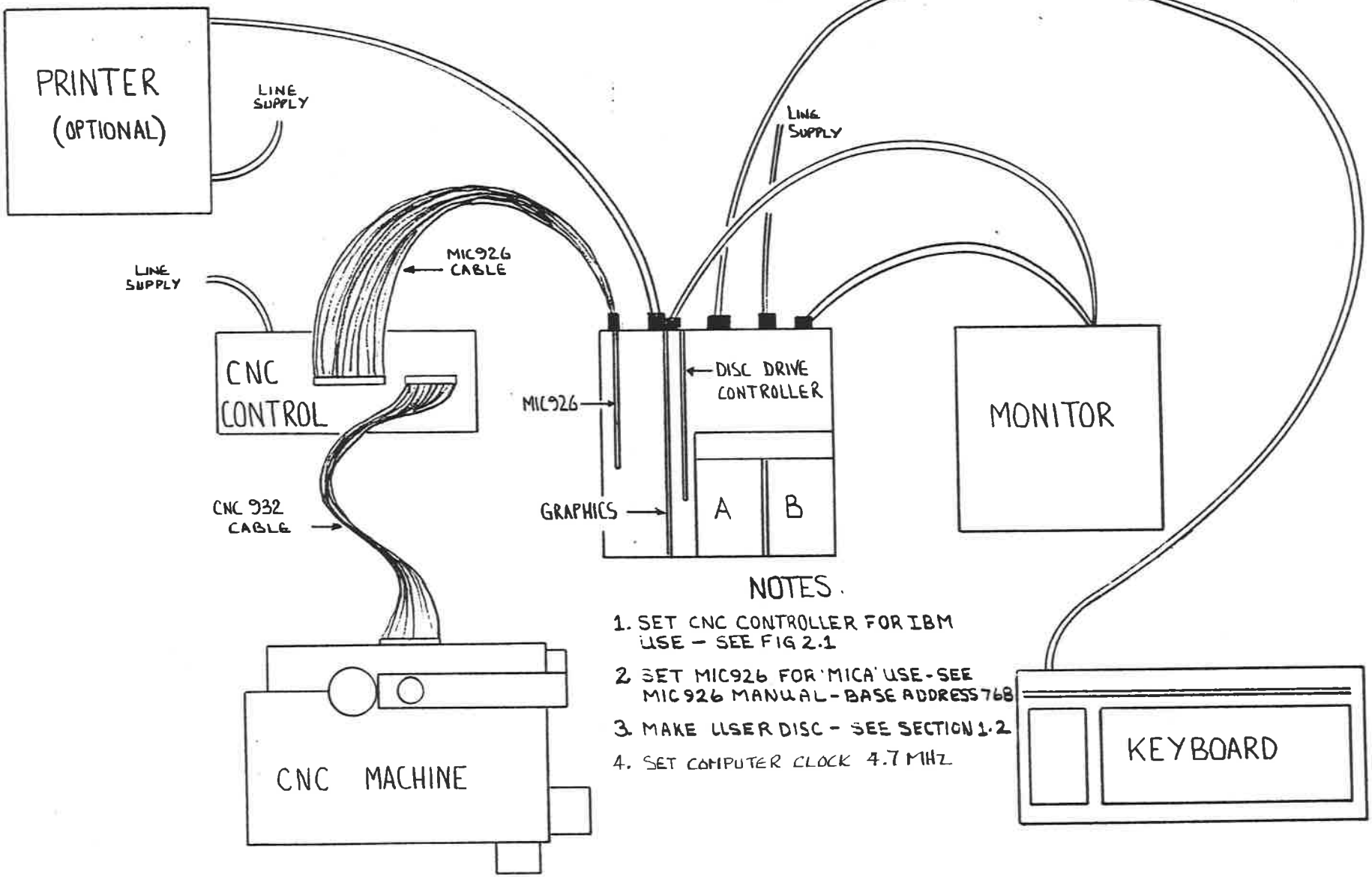
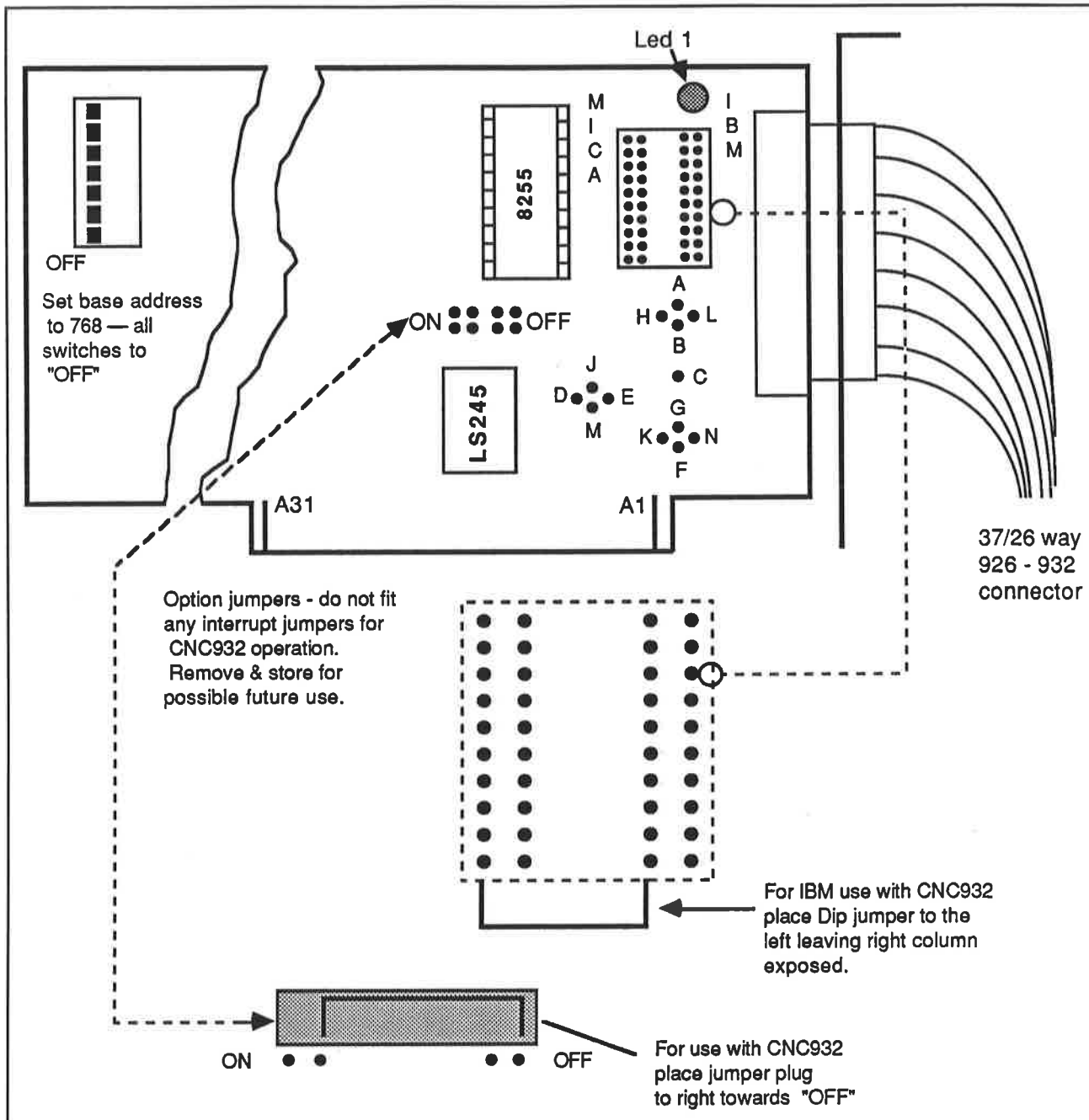


FIG 2.2 TYPICAL CNC932/IBM SYSTEM SET-UP



NOTE: The separate MIC926 manual supplied with the 926 interface provides full details of installation, operation and maintanance of this board.

FIGURE 2.3 - MIC926 JUMPER LOCATION & PLACEMENT DIAGRAM FOR USE WITH CNC932

2.3 — SHORT FORM OPERATIONAL TEST

2.3.01 Verify that the system is properly interconnected per Directions of **Figure 2.2**.

- **a.** Power to computer & 932 control unit plus printer & monitor if necessary.
- **b.** 932 CNC control to MIC926 port in computer
- **c.** 932 control to machine tool
- **d.** Proper cutting bit in chuck
- **e.** Operational printer — optional
- **f.** Power **OFF** to all equipment

2.3.02 Insert instructor provided disk in drive **A**

2.3.03 Turn **ON** power to computer, monitor & printer

2.3.04 Wait for

- **a.** Screen adjust message (answer Y or N)
- **b.** Turn on power to CNC932 control unit, switch **ON** spindle motor
- **c.** Printer connected message (Verify printer condition answer Y or N)

2.3.05 Wait for screen showing **MAIN MENU** selections

2.3.06 CNC Program and computer are now ready for your use

2.3.07 Assume you want to move the machine manually

- **a.** Press **7**
- **b.** Follow directions on screen
- **c.** When finished press **(D)**one

2.3.08 Assume you want to machine the shape shown as Problem **1** on page 5.3

- **a.** Place workpiece on bed of machine tool and clamp it in place per instructors directions.
- **b.** Press **6** key to **LOAD** manuscript
- **c.** Press **Y** followed by **RETURN** (shows files)
- **d.** After reading screen press **C**
- **e.** Key in **PROB1.CNC**
- **f.** Press **3** key to **LIST** manuscript
- **g.** Press **S**

h. After reading screen press **C**
i. After reading screen press **C**
j. Press **8** to **EXECUTE** the manuscript
k. Machine will go to home position
l. Use **+** and **-** keys to adjust tool tip to be 8mm above workpiece surface
m. When satisfactory press **D** key
n. Tool will proceed to cut part to a depth of 2mm
o. At completion of machining tool will return to home position

2.3.09 When **MAIN MENU** screen appears, press **9** key to **TERMINATE** the CNC Control Program and return to **BASIC A.**

2.3.10 Completion of this procedure confirms system is correctly connected and operating satisfactorily.

CHAPTER 3 — A GUIDED TOUR

3.1 — COMMENTARY

The objective of this chapter is to familiarize you with the features, presentation and operation of the FEEDBACK CNC program.

To do this we "walk through" the options provided by the program menu showing for reference, the screen displays that you can expect to see.

Some simple exercises are included to provide "hands-on" familiarity with most of the available functions.

This is a very "user friendly" program, the screens are written in plain English and explain clearly your options at each step.

The program contains many "error traps" to help avoid problems. In most cases wrong entries will simply be rejected, if a wrong entry does interfere with program operation the program will usually return automatically to the **MAIN MENU** and you can reenter the program from there.

If you do find yourself in a "lock-up" condition refer to the HELP page at the end of this section for rebooting instructions.

When you have worked through this section you will be able to 'drive' the CNC program.

Please note this section does not teach you how to write a part production program. That is dealt with in Section 4 and in the supporting manuals.

3.2 ANNOTATED SCREENS

In the series of actual screens, as seen on the computer's video display, several levels of intensity are used:

1. Regular (Generally used for titles and background information)
2. Regular Blinking
3. High Intensity (Generally used for current state data)
4. Blinking High Intensity (Generally used for soliciting a Response)

Note also that the actual screen display generally includes a blinking cursor somewhere. Cursors have been omitted from the printed screens that follow to improve clarity.

3.3 — INITIALIZATION

3.3.01. Set up your computer and CNC machine as **Figure 2.2**.
Note that your computer clock frequency should be 4.7 MHz

3.3.02. Insert a valid **USER DISK** in drive **A**. If you haven't made a **USER DISK** yet do so now following instructions of Chapter 1 — **DO NOT BE TEMPTED TO USE YOUR MASTER DISK FOR THESE EXERCISES.**

3.3.03 If you have a printer connected; switch that **ON** and also the computer. Do not power up the CNC machine yet. Remove any tool from the chuck and raise the spindle motor in the arm clamp so that the chuck clears the workpiece clamps.

3.3.04 Respond to the date and time prompts on the computer. When the program title screen appears switch **ON** the CNC control unit.

3.3.05. When the program starts, **Screen 1** appears on the video display. The purpose of actions in response to the questions on the display is to:

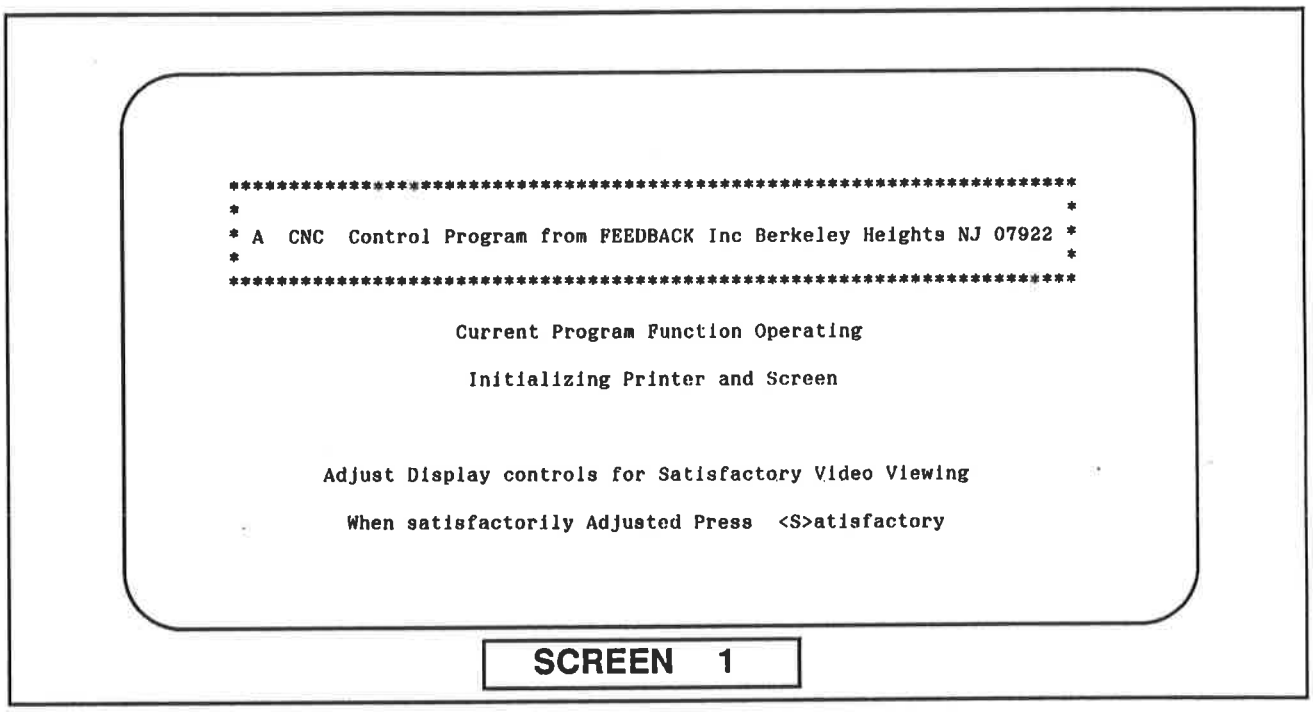
a. **Permit adjustment of the CRT** to the users ambient lighting and other conditions. The actual screen you see has two levels of brightness on the display. Adjust CRT controls so brightness levels are distinct.

b. **Indicate printer status.** If the question is answered **N**, and a printer is connected, the printer output from the Feedback program cannot be obtained. If the user does attempt to obtain printing, the program will issue a message:

PRINTER NOT AVAILABLE

3.3.06. If the question is answered **Y**, and a printer is not connected, the resulting action is dependent on the **DOS** version being used; the computer system may "lock-up" requiring a **CTRL-ALT-DEL** to continue; alternatively, a time-out message may appear requiring a program **RE-RUN**.

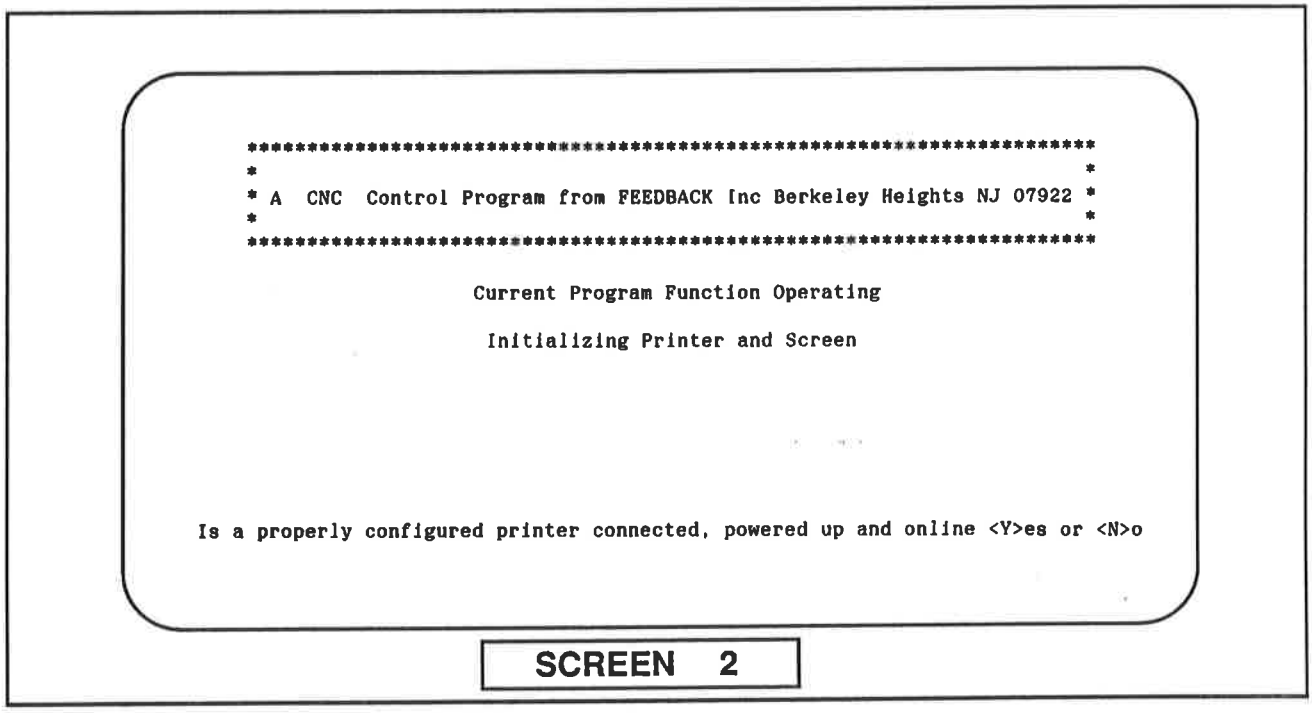
Other errors in program operation may terminate program execution, for example — file errors. However the program itself makes all possible effort to continue operation.



SCREEN 1

3.4.01. Use the video monitor controls (usually Brightness and Contrast) to adjust screen for satisfactory viewing.

3.4.02. Press **"S"**. Wait for **Screen 2.**

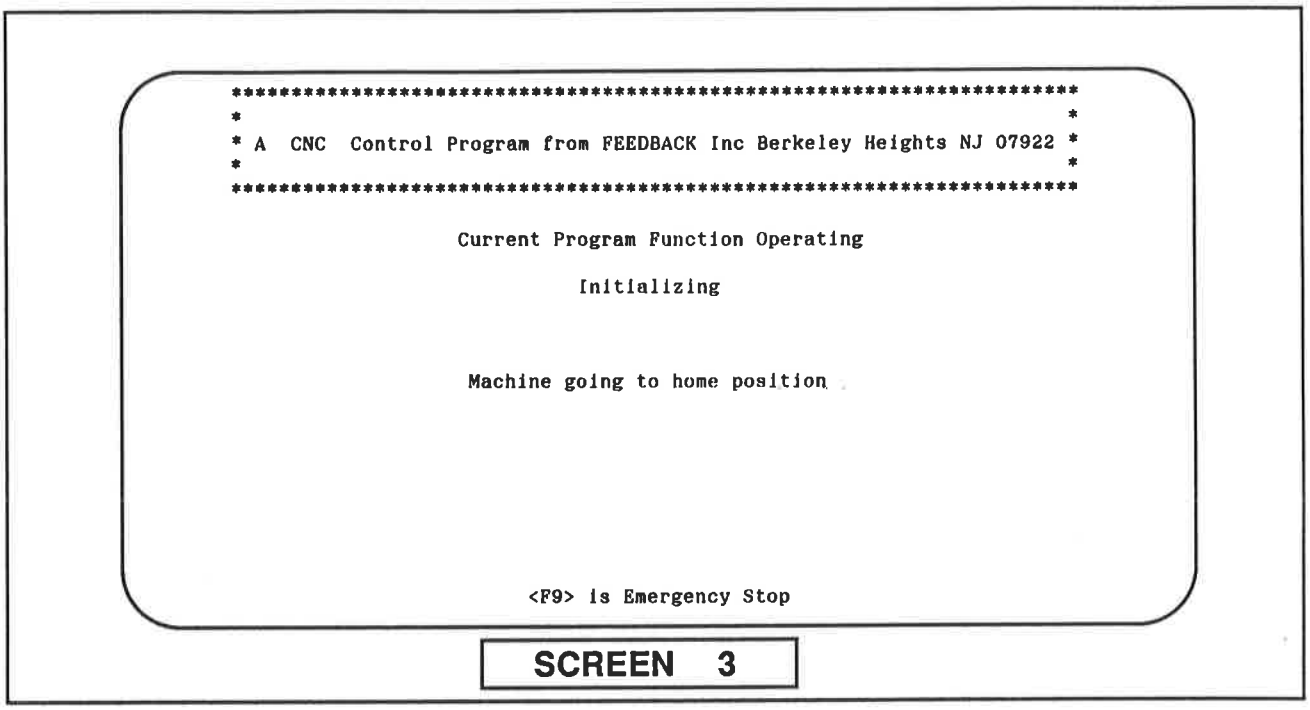


SCREEN 2

3.4.03. Verify printer power, connection, configuration and on-line status.

3.4.04. Press **"Y"** or **"N"** as appropriate.

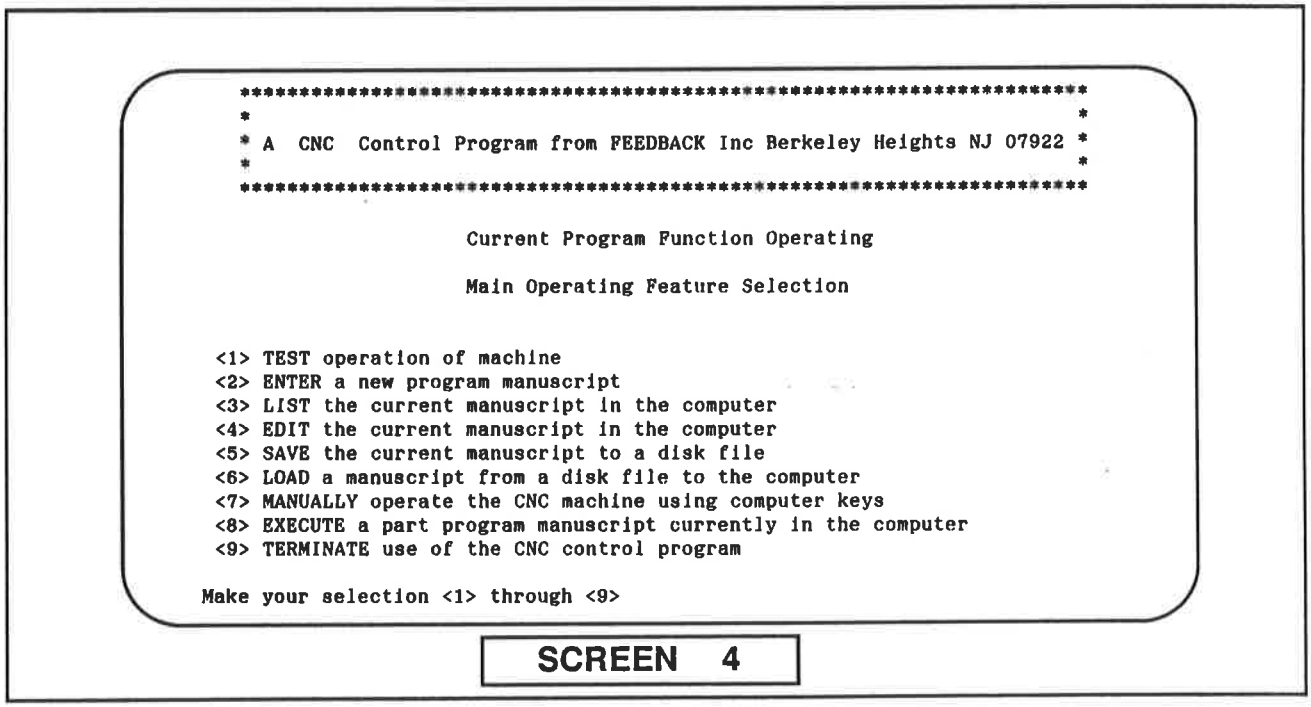
3.4.05. After a short delay **Screen 3** will appear as the machine goes to home position.



SCREEN 3

Wait for machine to return to home Position. Home is indicated by **"X limit"** and **"Y limit"** lamps on the 932 CNC Machine control Unit both lighting and going out followed by the stopping of the **X** and **Y** motions.

3.4.06. Wait for **Screen 4**, the **MAIN MENU.**



SCREEN 4

3.5 — MAIN MENU

This screen indicates the nine principal options of the program. This screen will be used repeatedly during program operation. There will be a slight delay in display of the **MAIN MENU** when returning from any of the following actions. **Wait** until the Menu appears. Program housekeeping is accomplished while the **MAIN MENU** is waiting to appear.

1. **TEST** physically exercises the CNC machine and warms it up prior to part production.

2. **ENTER** enables entry of a program manuscript into the computer. Prior to actual entry of a program manuscript, any existing program manuscript in the computer is destroyed.

3. **LIST** displays the manuscript currently in the computer on either the screen or printer.

4. **EDIT** is used to make changes in the manuscript currently in the computer.

5. **SAVE** permits the manuscript in the computer to be recorded on disk. The program manuscript remains in the computer.

6. **LOAD** is used to transfer a manuscript from disk to the computer. When a manuscript is loaded, the load process destroys any program manuscript previously entered in computer RAM.

7. **MANUAL** permits use of computer keyboard keys to move machine tool per directions on screen.

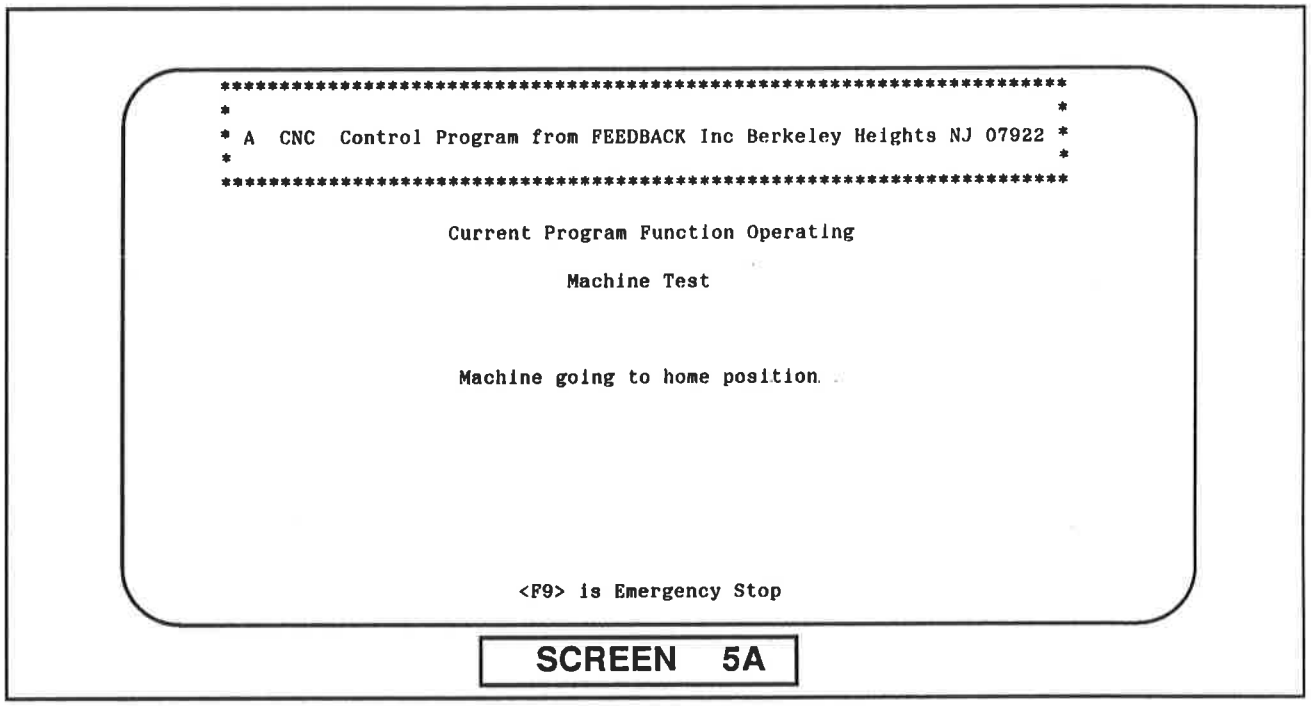
8. **EXECUTE** operates the machine tool to produce the part described by the part program manuscript currently in the computer.

9. **TERMINATE** stops program operation and exits to **BASICA.**

3.6 — TEST

3.6.01 This function is used to test and warm-up machine prior to fitting tool and starting part production.

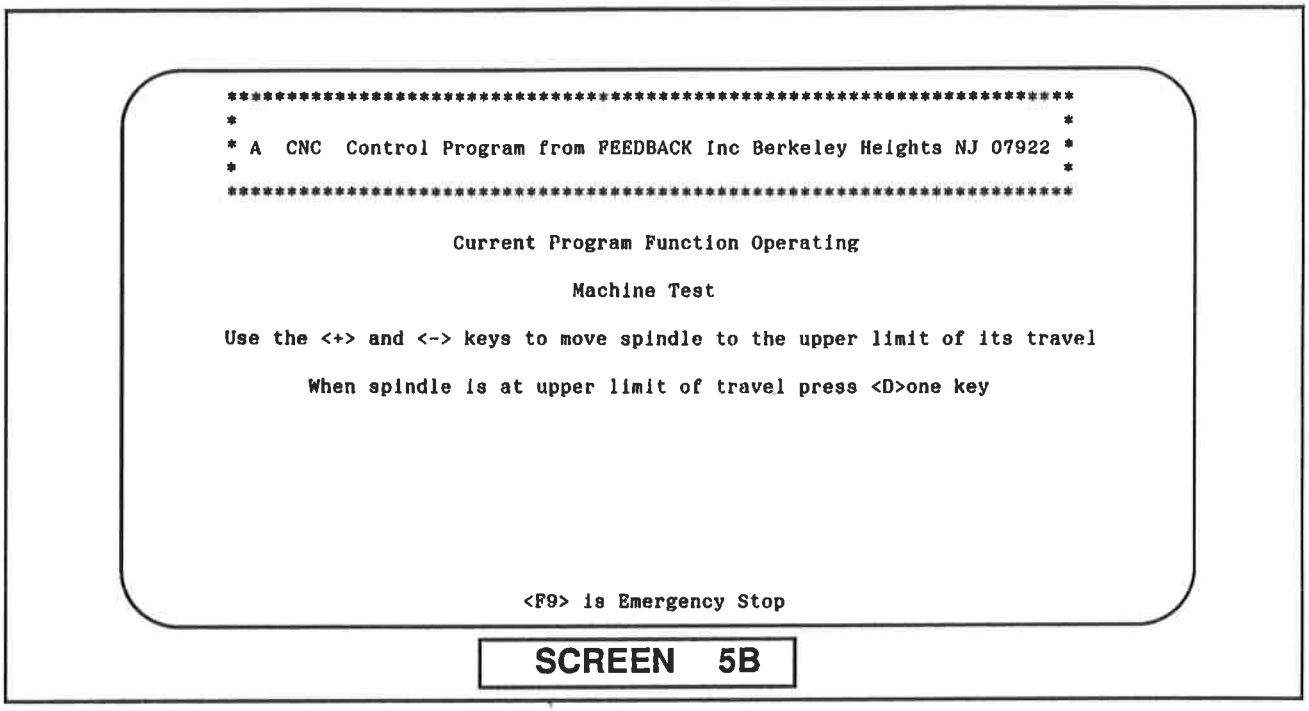
3.6.02. If you select **option 1** from **MAIN MENU**, **Screen 5A** appears while machine automatically returns to **HOME**.



SCREEN 5A

NOTE: TOOL SHOULD NOT BE INSERTED IN MACHINE CHUCK UNTIL MACHINE HAS RETURNED TO **'HOME'**

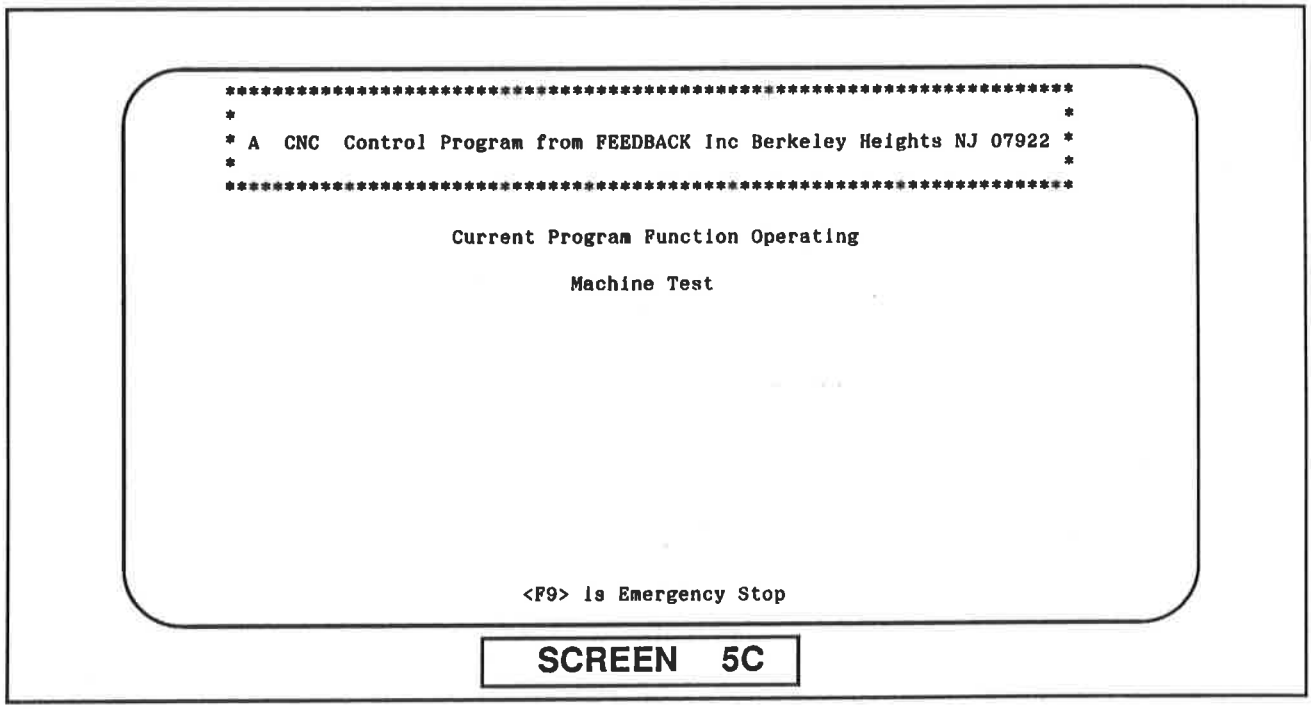
3.6.03. When machine reaches **HOME Screen 5B** appears. Use **'+'** key to move **Z axis** to upper limit. When machine reaches upper limit back off one step using **'-'** key.



SCREEN 5B

3.6.04. Now fit tool and manually set position of spindle motor in clamp arm to ensure tool tip clears the top of the workpiece.

3.6.05. When satisfied with tool position press **D, Screen 5C** will appear and the CNC machine will cycle several times the complete regular range of **X** and **Y** motions 5 times and then reutrn to **HOME** (Datum).



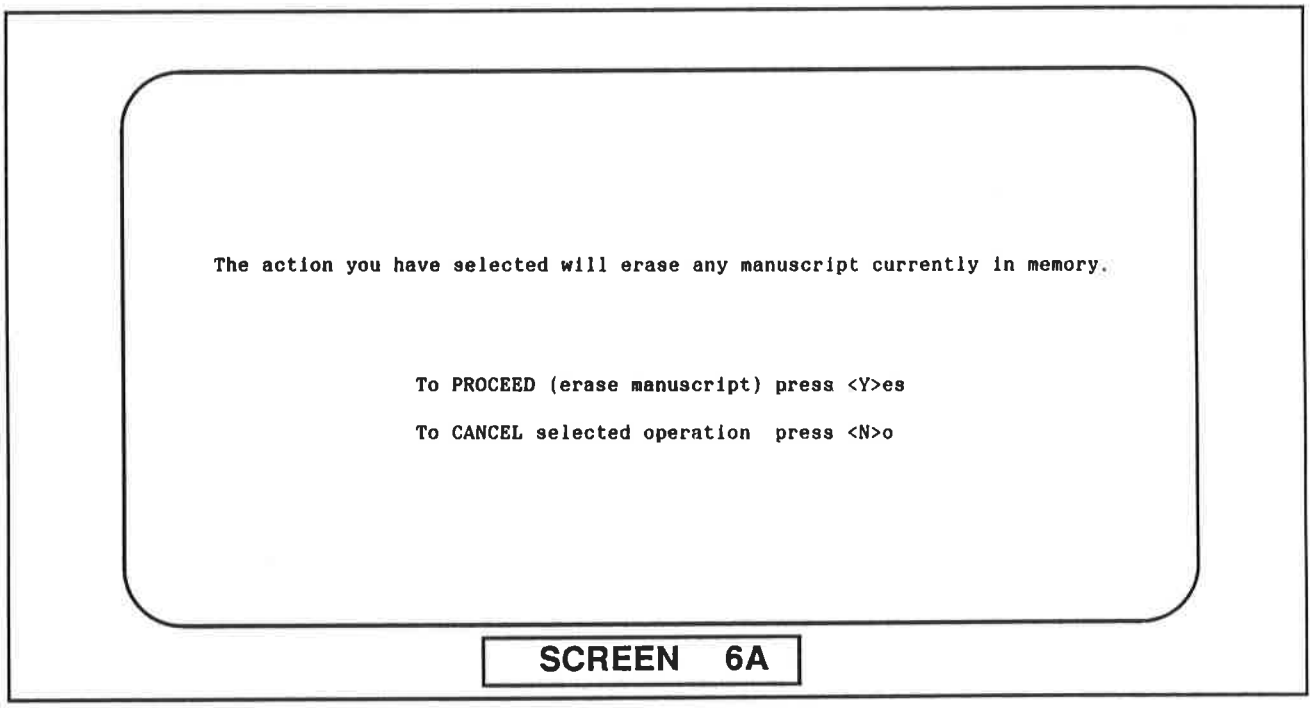
SCREEN 5C

3.6.06 At the end of the test sequence the program will return to the **MAIN MENU (Screen 4)**. You may then make a fresh selection.

3.7 — ENTER

3.7.01. **NOTE:** In most cases you will find it easier to leave the **'NumLck'** key **OFF** and use the numeric key pad for edit commands while making numeric enteries from the main keyboard.

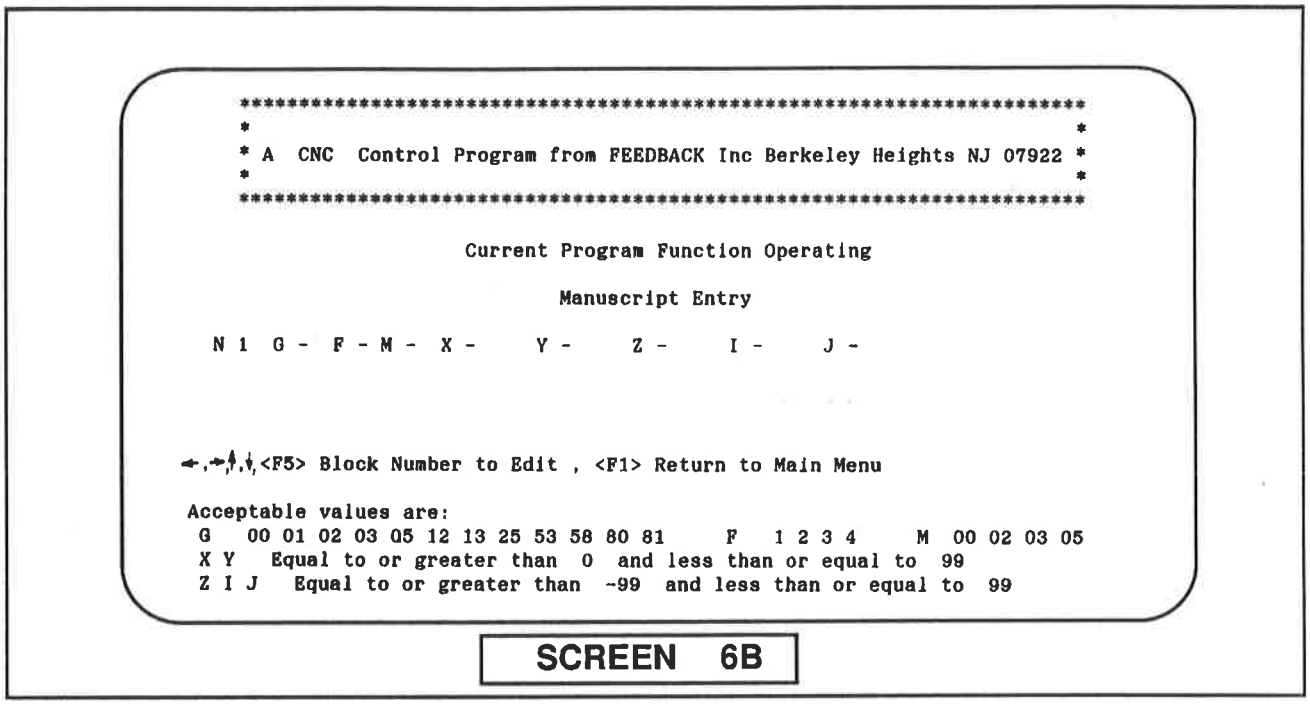
3.7.02. A warning Screen, **Screen 6A**, appears after you select **option 2** from the **MAIN MENU**. This gives you a second chance to **'SAVE'** any manuscript currently in RAM.



SCREEN 6A

3.7.03 If you press **N** the program returns to **MAIN MENU** and you can then select another option.

3.7.04 If you press **Y** there will be a short wait before **Screen 6B** appears.



SCREEN 6B

3.7.05. Use the practice program of **Fig. E1** below for the **ENTER** exercise started on the following page.

Listing of part manuscript prob1

N	G	F	M	X	Y	Z	I	J
1	00	-	-	40	40	-	-	-
2	01	1	03	-	-	-10	-	-
3	-	-	-	70	-	-	-	-
4	02	-	-	85	25	-	-	-15
5	-	-	-	70	10	-	-15	0
6	01	-	-	40	-	-	-	-
7	02	-	-	25	25	-	0	15
8	-	-	-	40	40	-	15	0
9	-	-	02	-	-	-	-	-

FIGURE E1 — PRINTER LISTING

3.7.06 ENTER OPERATION

A. The Sequence Number (line number) is set to **1** by the computer. As later lines are added, the sequence number will be incremented in steps of one.

B. The display provides space for the entry of an appropriate number of characters for each CNC word.

C. The lower portion of the display shows the valid ranges of characters and values for each CNC word.

D. A flashing cursor character appears inviting the first character entry for the **"G"** word. At this point, the program is waiting for the first character of the **"G"** word. For example, now strike the **"0"** key (top row of keyboard). Note that the **"0"** appears on the screen. The cursor has moved to the right by one position inviting entry for the second character of the **"G"** work. Looking at the lower portion shows that only one of the following characters are acceptable.

"0", "1", "2", "3", "5", "8"

Now strike the **"3"** key. The screen will show **"G 03"** and the cursor will be flashing the right of the **"3"** character.

However, **"03"** which is a correct **G** word entry is not the **G** word entry required by block **1** of **Fig E1**. Change the entry by striking the, **"Backspace"** key. This will position the cursor over the **'3'**, strike **8** and the entry **08** will be made.

Now strike the **"ENTER"** (Return) or '➡' and the cursor will position itself for entry of the **F**(eedrate).

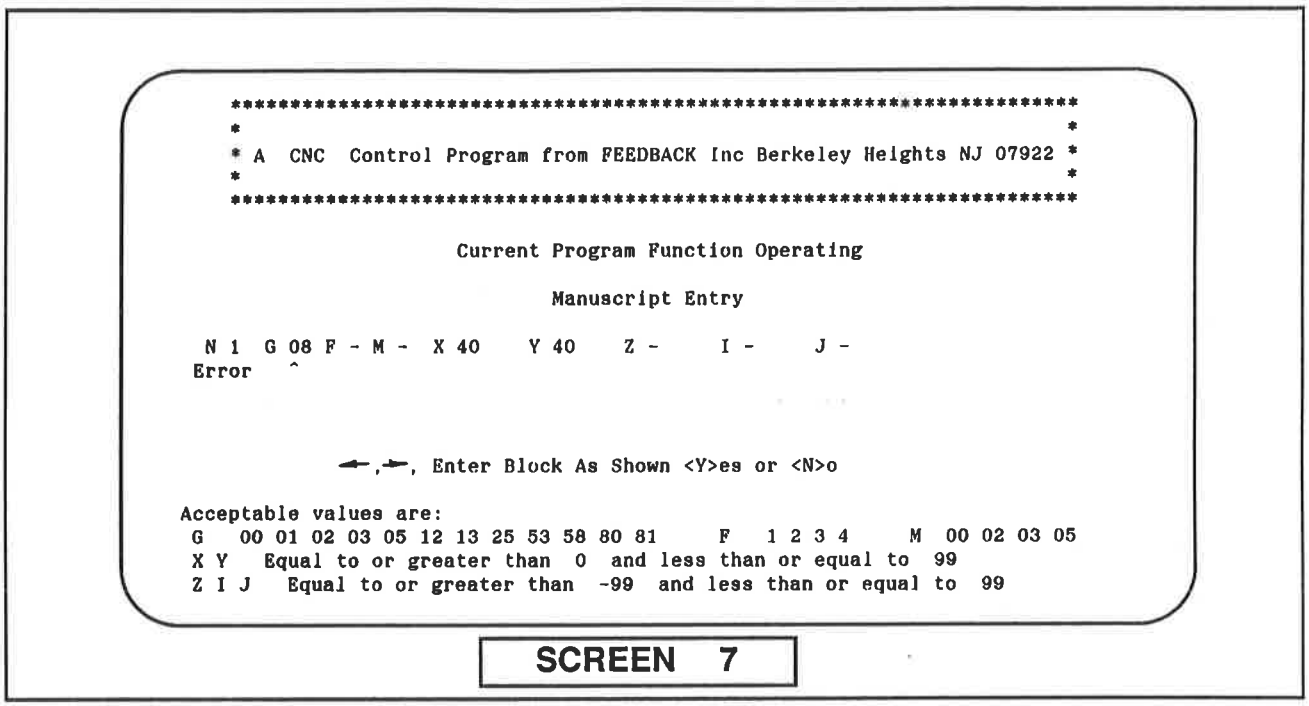
3.7.07. Now press the following keys in sequence per **Fig. E1** and observe the screen after each key press.

KEY TO PRESS	RESPONSE/COMMENT
ENTER (Return)	no explicit **F** value
ENTER (Return)	no explicit **M** value
4 **0** **ENTER** (Return)	**X** word
5 **BACKSPACE** **4** **0** **ENTER** (Return)	Wrong value Remove it **Y** word
ENTER (Return)	no explicit **Z** value
ENTER (Return)	no explicit **I** value
ENTER (Return) **YES**	no explicit **J** value

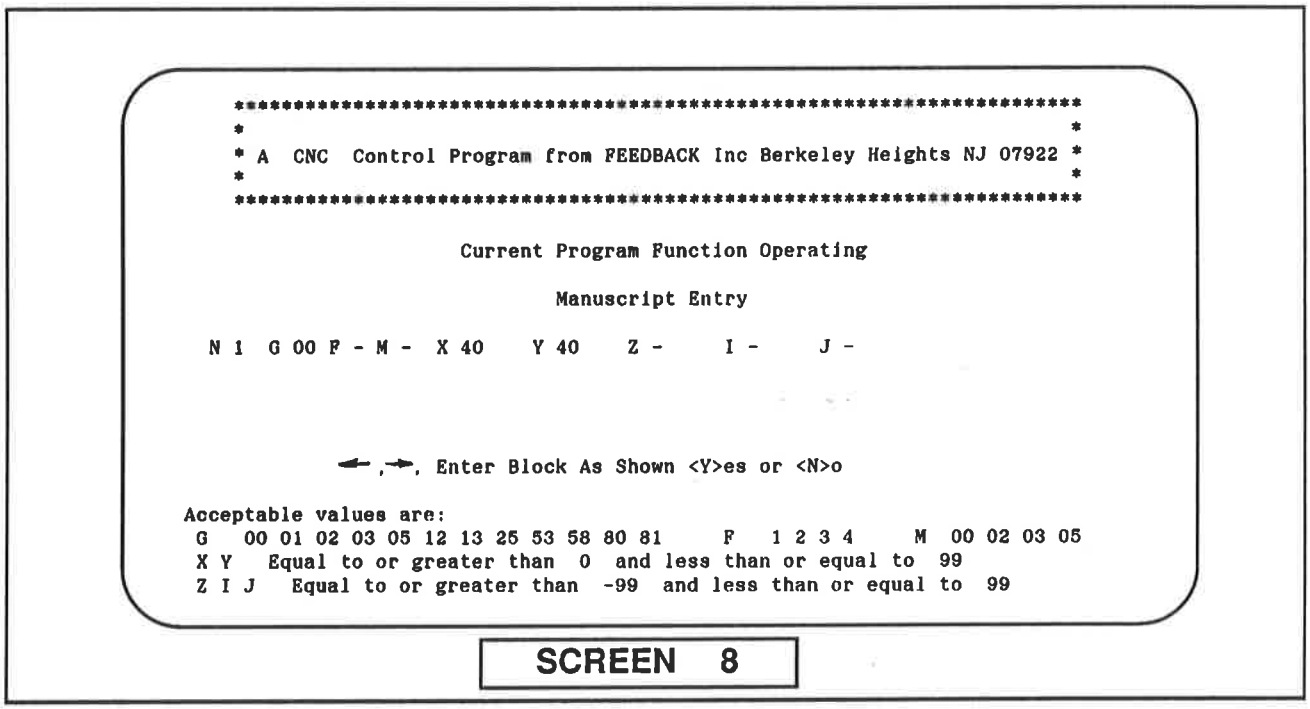
The CRT now appears as in **Screen 7.**

Now strike the following keys:

0 **0** **ENTER** (Return)	The correct value for **G** is entered



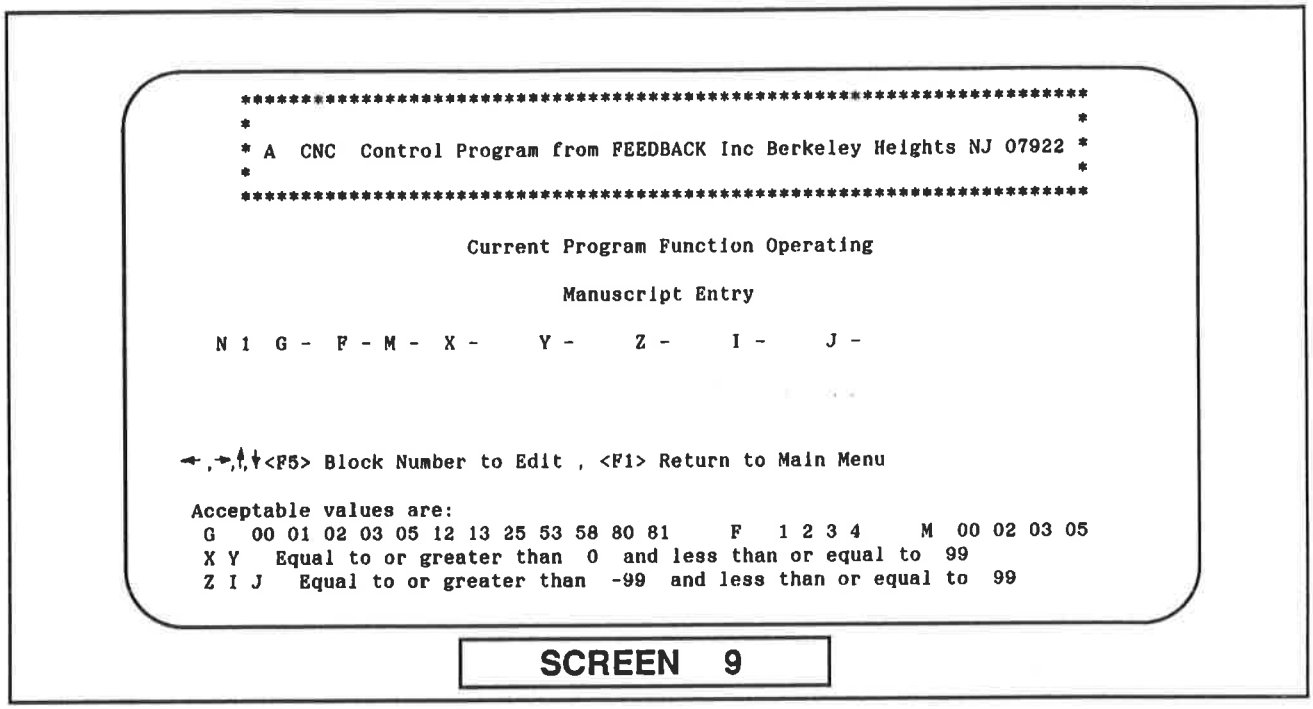
SCREEN 7



SCREEN 8

3.7.08. The display now appears as **Screen 8.** The entries for each word as a whole have been checked and found to be in range. The user has the option of rejecting the entire block with a **N.**

A **N** answer results in **Screen 9** which requires full re-entry of the Block.



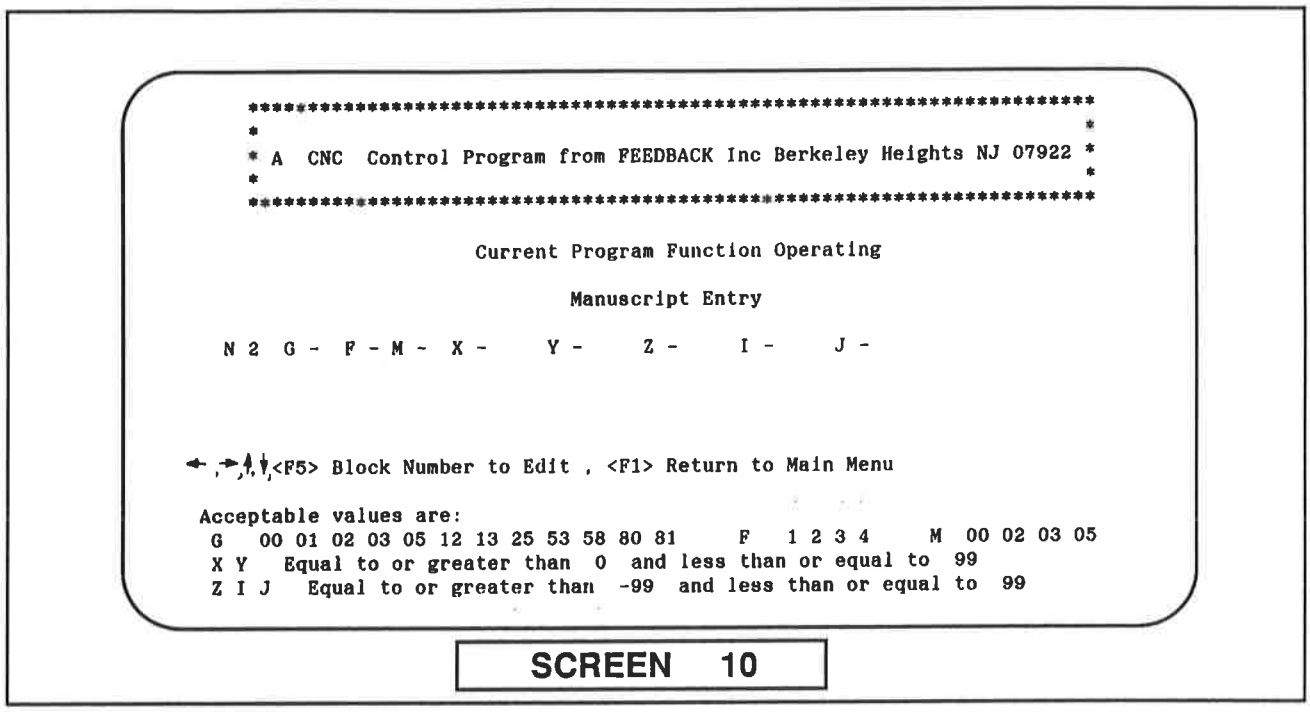
SCREEN 9

3.7.09 To re-enter block strike the following keys:

KEY TO PRESS	MEANING	
0		
0		
ENTER	**G** word	**G** word value **"00"**
ENTER	**F** word	No explicit value
ENTER	**M** word	No explicit value
4		
0		
ENTER	**X** word	**X** word value **"40"**
4		
0		
ENTER	**Y** word	**Y** word value **"40"**
ENTER	**Z** word	No explicit value
ENTER	**I** word	No explicit value
ENTER	**J** word	No explicit value

A correctly entered screen will appear as in **Screen 8.**

3.7.10. A **"Y"** answer results in **Screen 10** to permit entry of block **2.**



SCREEN 10

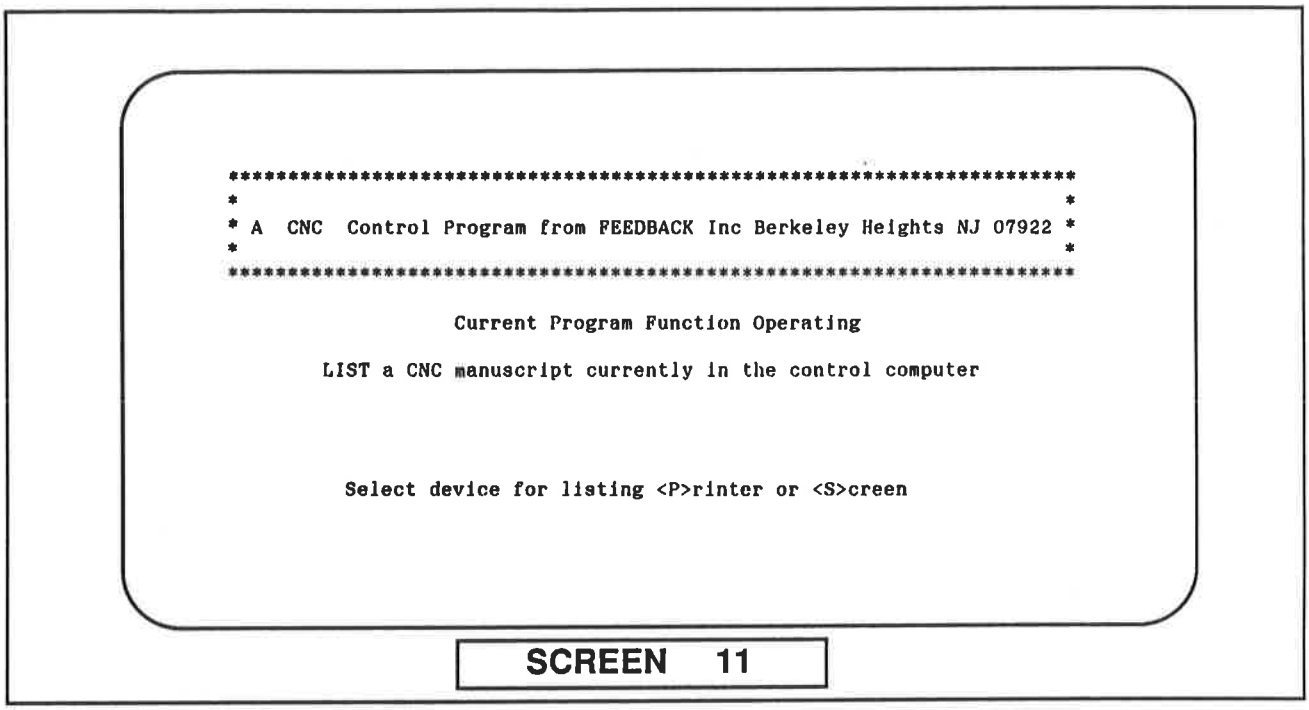
Continue making the additional entries per **Figure El.**

When you have successfully entered block **N9** containing only the explicit **M** word value **02** and have indicated that the entry is correct by answering the "satisfactory" question with a **Y**, the program will return to the **MAIN MENU (SCREEN 4)** after a delay.

3.7.11. The part program you have entered is now stored in computer RAM. You can now **LIST** it out to printer or screen through menu **option 3, EDIT** it by use of **option 4, SAVE** it to disk via **option 5** or use it to machine a part by implementing the **EXECUTE** function — **option 8.**

3.8 — LIST

3.8.01. Screen 11 appears after pressing key **3** at the **MAIN MENU**.



SCREEN 11

3.8.02 If you select the **(P)**rinter option the computer will ask you to enter a title of not more than 20 characters. Upon entry of the title the printer will print out the current manuscript with title and automatically return to **MAIN MENU** on competion. If the manuscript of **E1**, Page 10 is still in the computer you will get a printout as **E2** with whatever title you have chosen.

3.8.03. Selection of the **S**(creen) option will result in a similar display on the monitor screen but you are then required to use key **'C'** to return to **MAIN MENU** or to continue the listing if it contains more than 8 lines (note a 2 line overlap is provided when listing is continued).

Listing of part manuscript user defined heading

N	G	F	M	X	Y	Z	I	J
1	00	-	-	40	40	-	-	-
2	01	1	03	-	-	-10	-	-
3	-	-	-	70	-	-	-	-
4	02	-	-	85	25	-	-	-15
5	-	-	-	70	10	-	-15	0
6	01	-	-	40	-	-	-	-
7	02	-	-	25	25	-	0	15
8	-	-	-	40	40	-	15	0
9	-	-	02	-	-	-	-	-

FIGURE E2 — SAMPLE (P)RINTER LISTING

3.9 — EDIT

3.9.01 With no program manuscript currently loaded in the computer, striking key **"4"** at the **MAIN MENU** produces **Screen 12.**

```
**************************************************************************
*                                                                        *
* A  CNC  Control Program from FEEDBACK Inc Berkeley Heights NJ 07922    *
*                                                                        *
**************************************************************************

                    Current Program Function Operating

                            Manuscript Editor

  N 1  G -  F - M -  X -     Y -     Z -     I -     J -



←,→,↑,↓<F5> Block Number to Edit , <F1> Return to Main Menu

 Acceptable values are:
  G   00 01 02 03 05 12 13 25 53 58 80 81     F   1 2 3 4     M  00 02 03 05
  X Y   Equal to or greater than  0  and less than or equal to  99
  Z I J   Equal to or greater than  -99  and less than or equal to  99
```

SCREEN 12

3.9.02. Now lets enter a specimen program to give us something to test the editor on.

Press F1 to return to **MAIN MENU**

Press 6 to select **LOAD**

Press (Y)es at prompt

Press (N)o at next prompt

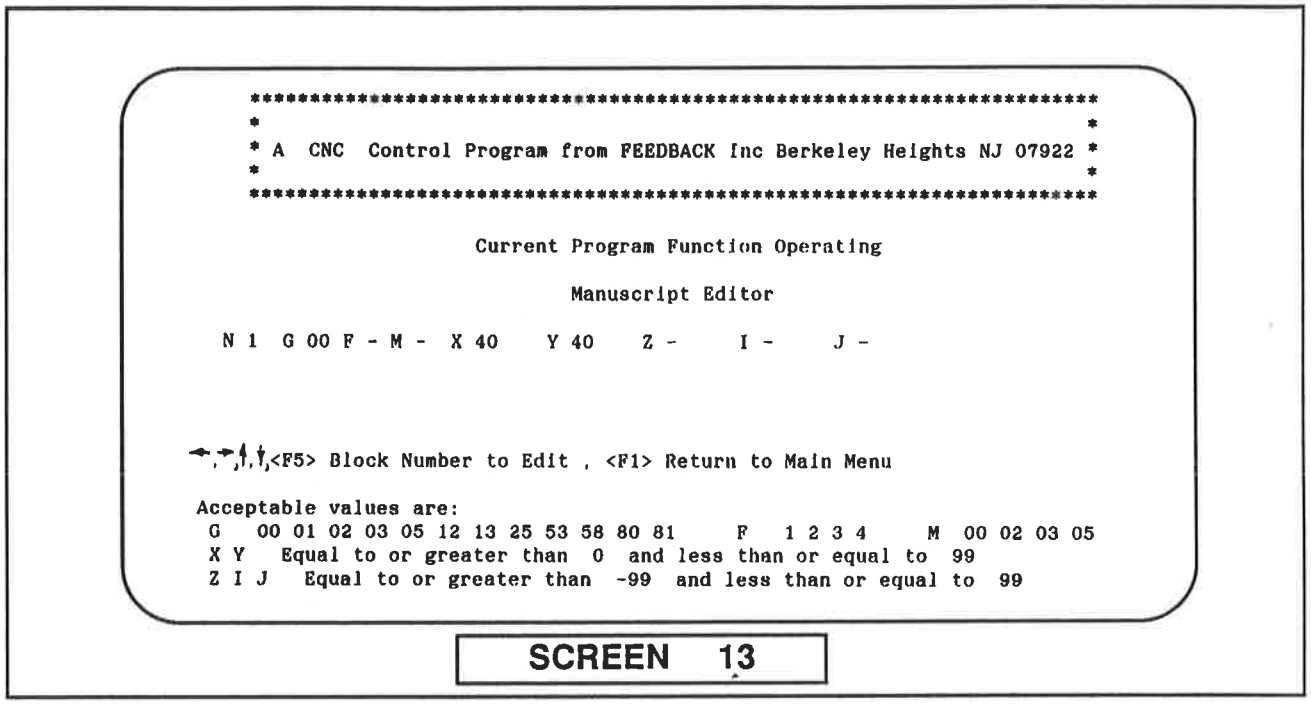
Enter filename **PROB1.CNC**

Screen will show short message confirming file is being loaded, it then displays a **'WAIT'** message, and when loading is complete you are automatically returned to **MAIN MENU.**

To check the file has been loaded correctly you can Select **(3)** and follow the prompts to **LIST** the newly loaded file on the screen. The listing will be the same as **Figure E1** on Page 10.

When you are happy with the listing **press C** to return to **MAIN MENU**.

3.9.03. Now select **'4'**, the **EDIT** option again. This time you will see **Screen 13**.



SCREEN 13

This is **Line 1** of **Figure E1** and the cursor will be blinking under the first **0** of **G00**.

Let's examine the functions of the edit keys.

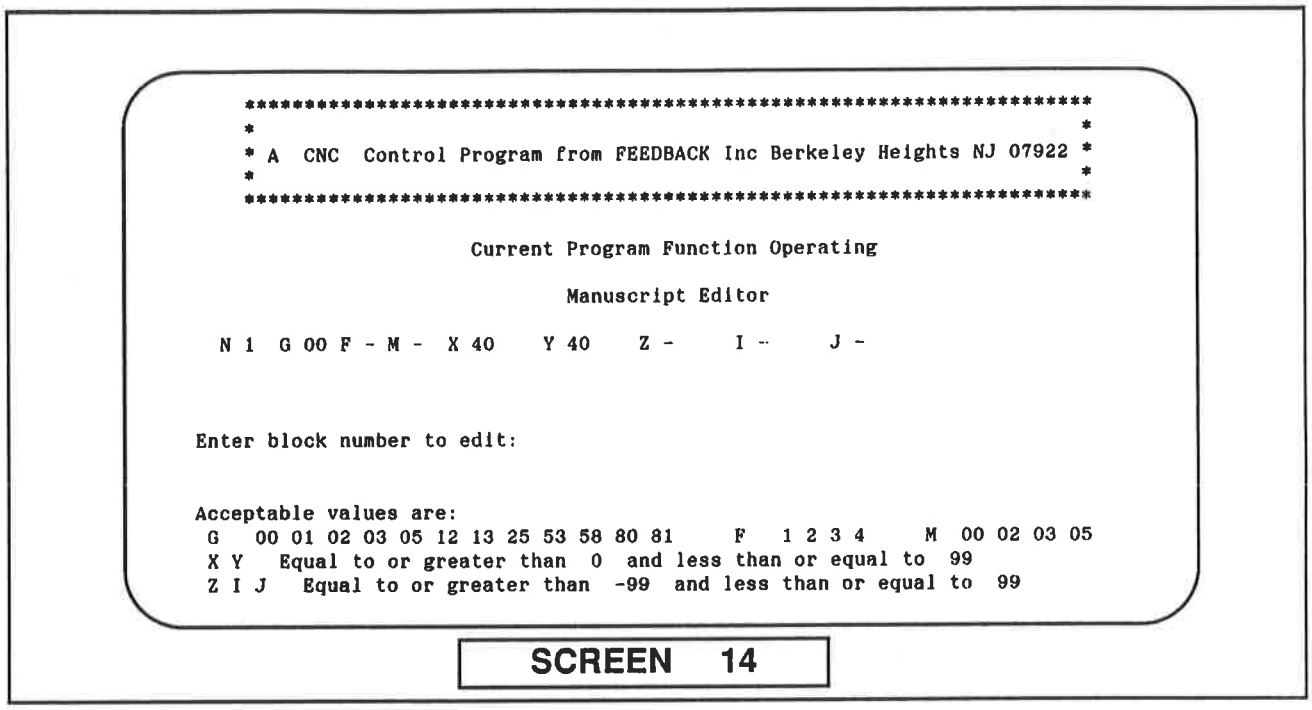
A. → : moves cursor to **right** along selected line, figures directly above the cursor can be edited. Notice that when a new digit entry is made you lose the option of ↑ and ↓ keys until the complete line is entered. When cursor is driven fully to right it 'wraps around' to the beginning of the same line. Note cursor 'jumps' to valid entry points and will not allow positioning at intermediate points — i.e, you must always edit the **complete** function block.

B. ← : Same function as 1 except cursor moves to **left**.

C. ↓ : When shown as an option on screen this key steps you through to the next line of manuscript. There is no automatic **'END OF FILE'** detection on this key and thus you can use it to step beyond the last line of current manuscript. This key is used to increment the manuscript **AFTER** you have corrected a line.

D. ↑ : When shown as an option on screen this key enables you to back through the manuscript to the preceding line. It automatically locks at **Line 1**.

E. F5 : Selecting this option results in **Screen 14**, so press **F5**.



SCREEN 14

Simply enter the line number at the prompt and press **RETURN** (enter). The program automatically goes direct to the requested line for editing — this of course is more convenient than use of ↑ or ↓ for long programs with widely separated line editing requirements.

F. F1 : This key is used to exit the edit program and return to **MAIN MENU**.

3.9.04. To enter a correct line either :

A) If you have examined a line and find it correct without the need for any changes you can enter the complete line by using the ↑ or ↓ key to move on to the preceding or subsequent line when ↑ or ↓ is shown as an option on the screen.

B) You can use **F5** to move to another, specified line.

As an example enter **5** and press **RETURN** to show **N5**, line 5 of the program.

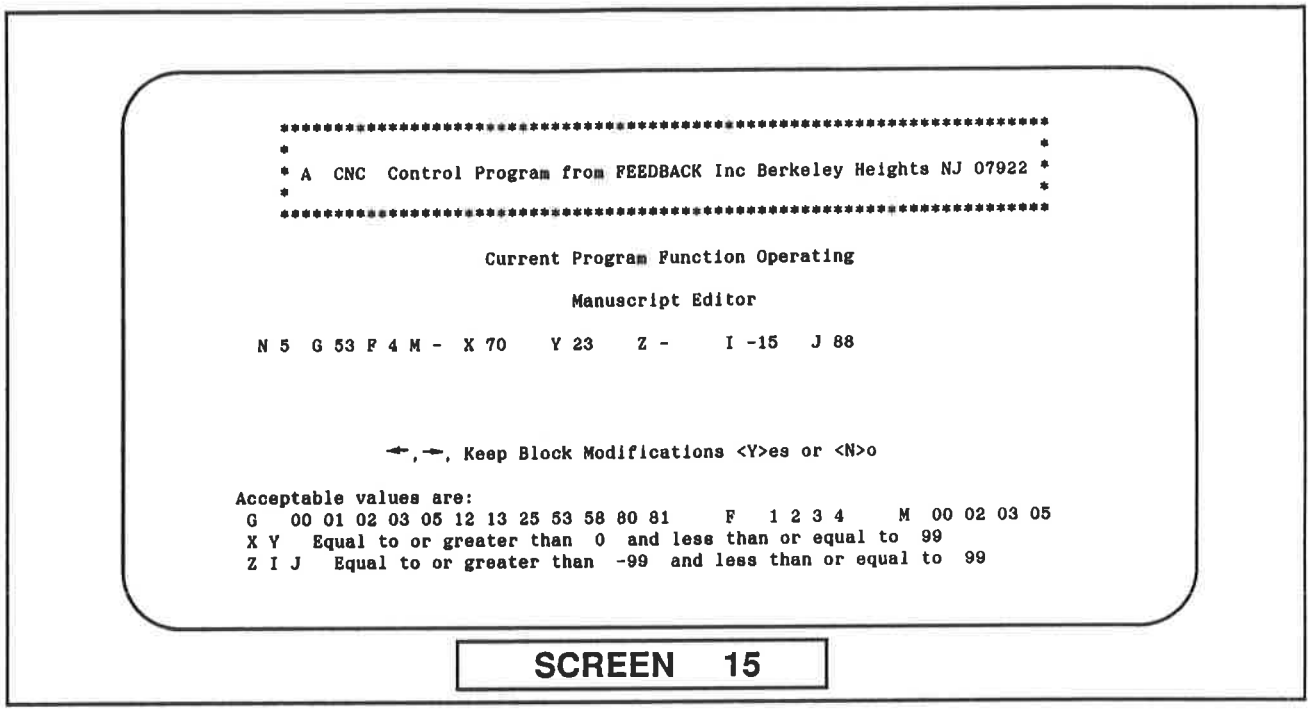
The line should read:

N5 G- F- M- X70 Y10 Z- I -15 J0

Use the editor to change this to:

N5 G53 F4 M- X70 Y23 Z- I -15 J0

3.9.05A If you have made the changes at 3.9.04B you will be presented with **Screen 15**.

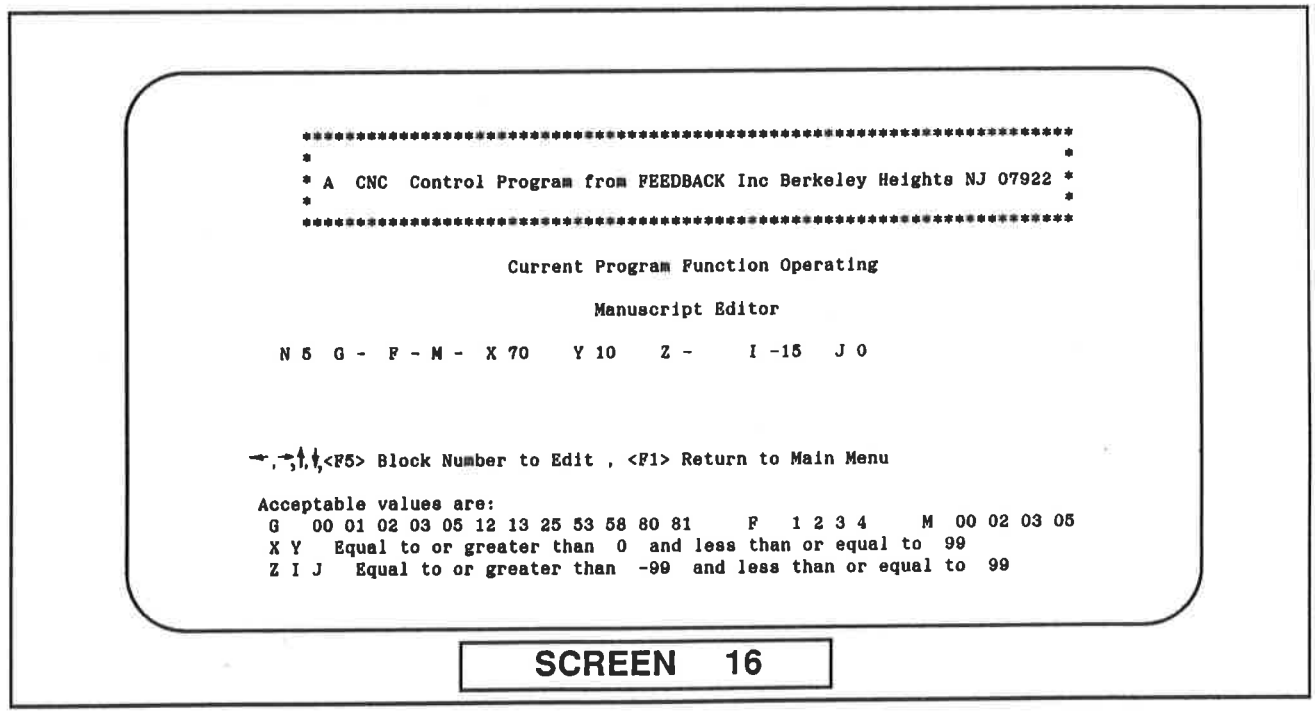


SCREEN 15

B) Selecting option **(N)**o deletes corrections made and displays **Screen 16**, giving you a chance to redo the corrections. The values now shown are correct and could be entered by use of ↓ or ↑.However to complete this example type in:

N5- GO2 F- M- Y40 Z- I15 J0

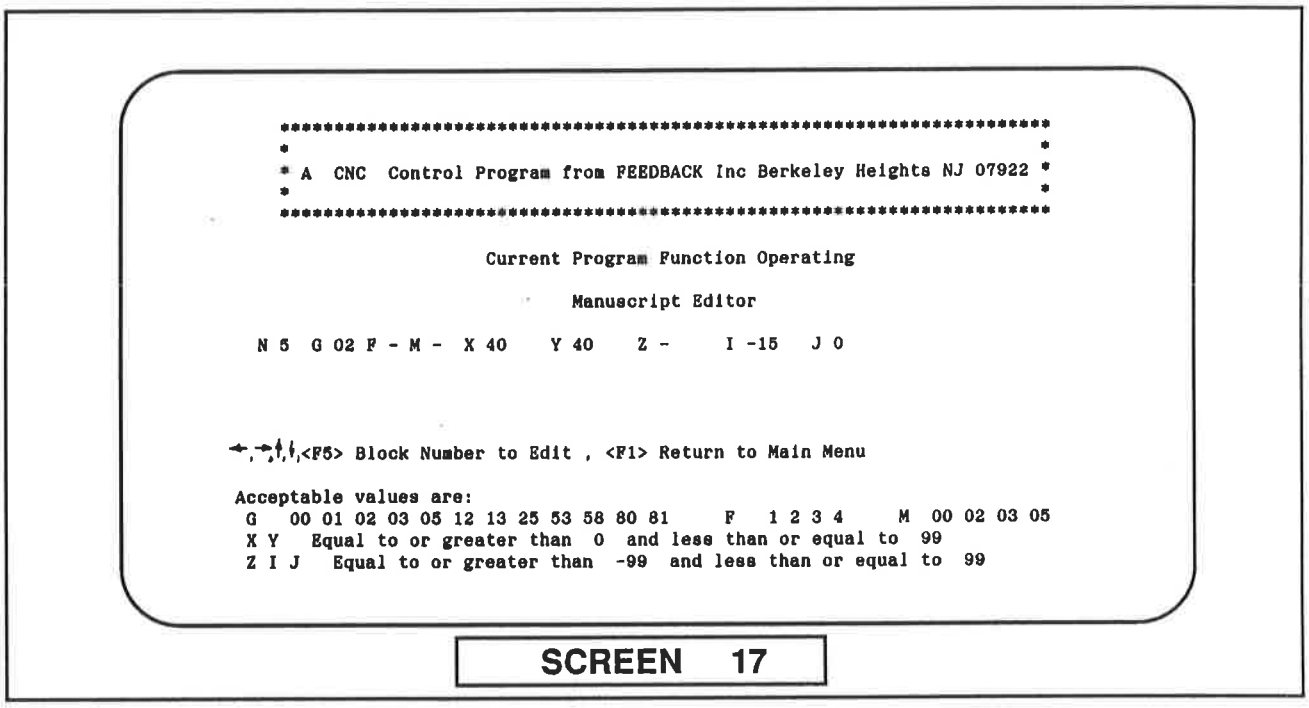
C) Now select option **(Y)**es to enter the revised line and bring up **Screen 17** with the just entered line still displayed. Use ↑, ↓ or **F5** as appropriate to move on to the next line to be edited, or **F1** to get back to **MAIN MENU**, use ↓ if you are just incrementing through the program.



SCREEN 16

3.9.06 **BACKSPACE :** can be used instead of ← .

3.9.07 **^** **:** "carrot" characters automatically appear under incorrect codes when you try to enter a line with errors. For example use **F5** to select line, of the current manuscript. If the manuscript has been changed per 3.9.05B you should see **Screen 17**.



SCREEN 17

3.9.08 Note that line **5** has indeed been changed and that it is now incorrect when compared with **Figure E1** of Page 10 which reads:

N5 G- F- M- X70 Y10 Z- I-15 J0

<u>To correct it :</u>

Press C to complete the listing

Press C again to return to **MAIN MENU**

Select option 4 — **EDIT**

Press F5 and enter **5** when prompted for line number (—**press RETURN** (enter) after **5** because the computer does not know if it is waiting for a one, two, or three digit number — you have to tell it)

Now practice with →, ←, **BACKSPACE** and **RETURN** keys to restore line **5** to read;

N5 G- F- M- X70 Y10 Z- I15 J0

When the line is correct press **(Y)**es to enter it in memory.

Return to **MAIN MENU** (press **F1**) and list **PROB1.CNC** to screen again.

3.9.09. If you have used the **EDITOR** correctly you will see that line **5**has been changed back to agree with original entry of line **5, Figure E1** Page 10.

Now return to EDIT mode and display line 8. This should read:

N8 G- F- M- X40 Y40 Z- I15 J0

From the keyboard change line 8 to read :

N8 G11 FØ M12 X40 Y40 Z- I15 J0

Press **(Y)**es to enter these changes, you will then see **Screen 18** indicating **ERROR** in **G, F** and **M** entries. Note that you cannot exit from this screen until you attempt correction of the line with errors.

```
**************************************************************************
*                                                                        *
* A  CNC  Control Program from FEEDBACK Inc Berkeley Heights NJ 07922    *
*                                                                        *
**************************************************************************

                    Current Program Function Operating

                            Manuscript Editor

 N 8  G 11 F Ø M 12 X 40    Y 40    Z -     I 15    J 0
Error    ^     ^    ^

              ←,→, Keep Block Modifications <Y>es or <N>o

Acceptable values are:
 G   00 01 02 03 05 12 13 25 53 58 80 81     F   1 2 3 4     M  00 02 03 05
 X Y   Equal to or greater than  0  and less than or equal to  99
 Z I J   Equal to or greater than  -99  and less than or equal to  99
```

SCREEN 18

3.9.10. Now **EDIT** line 8 by use of ← and → as appropriate to enter values.

N8 G00 F2 M02 X88 Y40 Z- I15 J8

Press (Y)es option to enter this line into computer memory.

You will then see **Screen 19.**

```
**************************************************************************
*                                                                        *
* A  CNC  Control Program from FEEDBACK Inc Berkeley Heights NJ 07922 *
*                                                                        *
**************************************************************************

                    Current Program Function Operating

                              Manuscript Editor

 N 8  G 00 F 2 M 02 X 88    Y 40    Z -     I 15    J 8



←,→,↑,↓,<F5> Block Number to Edit , <F1> Return to Main Menu

Acceptable values are:
 G   00 01 02 03 05 12 13 25 53 58 80 81     F   1 2 3 4     M  00 02 03 05
 X Y   Equal to or greater than  0  and less than or equal to  99
 Z I J   Equal to or greater than  -99  and less than or equal to  99
```

SCREEN 19

3.9.11. Because all the numeric values are within the correct range the computer has accepted this as a valid program line and will have changed the file **PROB1.CNC** in RAM accordingly.

To check this use **F1** to return to **MAIN MENU**, select option **3**, choose **(S)**creen listing and observe the display on **Screen 20.**

```
**************************************************************************
*                                                                        *
* A  CNC  Control Program from FEEDBACK Inc Berkeley Heights NJ 07922 *
*                                                                        *
**************************************************************************

                    Current Program Function Operating

              LIST a CNC manuscript currently in the control computer

 N   G   F   M     X      Y      Z      I      J
___ ___ ___ ___  _____  _____  _____  _____  _____
 1  00   -   -   40     40     -      -      -
 2  01   1  03   -      -     -10     -      -
 3   -   -   -   70     -      -      -      -
 4  02   -   -   85     25     -      -     -15
 5   -   -   -   40     40     -     -15     0
 6  01   -   -   40     -      -      -      -
 7  02   -   -   25     25     -      0     15
 8  00   2  02   88     40     -     15      8


Press  < C >  key to continue listing
```

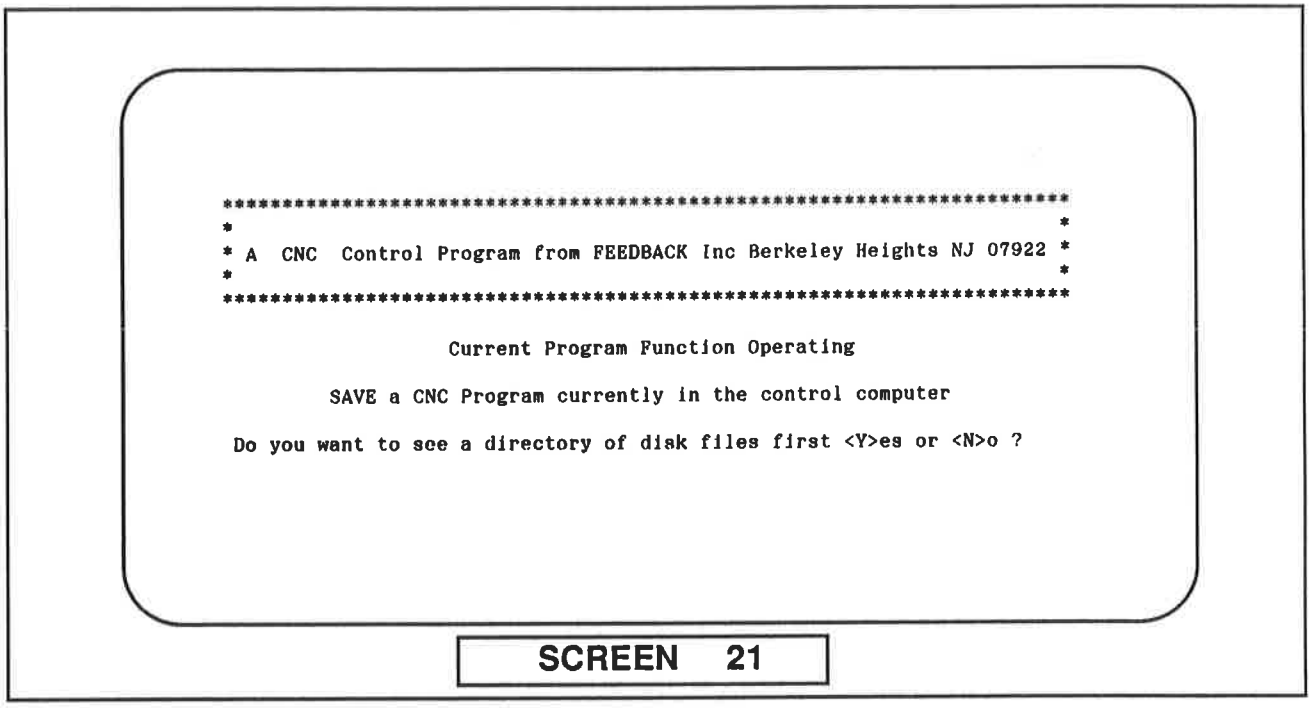
SCREEN 20

3.9.12. Note that although the line entry has been accepted and stored in RAM it is not correct when compared with the original listing for line 8, **Figure E1**, page 10. This shows that the program error traps can only check that values are within range — they can't tell whether or not you have chosen the right numbers within the acceptable range.Note that you have only changed the RAM file - the original file on disk has not been altered because you do not **SAVE** the altered manuscript back to disk.

3.10 — SAVE

3.10.01 This function **SAVES** the current manuscript in the computer RAM by writing it as a file on a disc.

Select **option 5** from **MAIN MENU** and you will see **Screen 21**.



SCREEN 21

This screen offers you the option of first reviewing the directory of files currently on the disc. This option is available so that you can:

A) Check to see if the file you are about to **SAVE** is already on the disc — in which case you don't need to save it again.

B) Check the file titles in use to make sure you don't duplicate a title.

C) Check that there is enough space left on the disk to hold the file you want to **SAVE**. The amount of space left on the disc is shown at the end of the directory as :

(???????) bytes free

3.10.02. If you choose option **(Y)**es you will see a display similar to **Screen 22.** Remember the list of file titles and amount of space free will be different from this example if you have saved any additional files or deleted any original files.

```
A:\
COMMAND .COM      CNC     .BAS      FIG2$18 .CNC      FIG2$1  .CNC
FIG2$5  .CNC      FIG2$7  .CNC      FIG2$8  .CNC      FIG2$12 .CNC
FIG2$14 .CNC      FIG2$15 .CNC      FIG2$16 .CNC      FIG2$17 .CNC
PROB1   .CNC      PROB2   .CNC      PROB3   .CNC      PROB4   .CNC
PROB5   .CNC      PROB6   .CNC      PROB7   .CNC      ROBIN   .CNC
BASICA  .EXE      MAKE    .BAT      AUTOEXEC.BAT
 192512 Bytes free



Press  < C > to continue your work
```

SCREEN 22

Press <C> to continue your work.

If you **press <C>**, or if you initially select **(N)**o option you will see **Screen 23.**

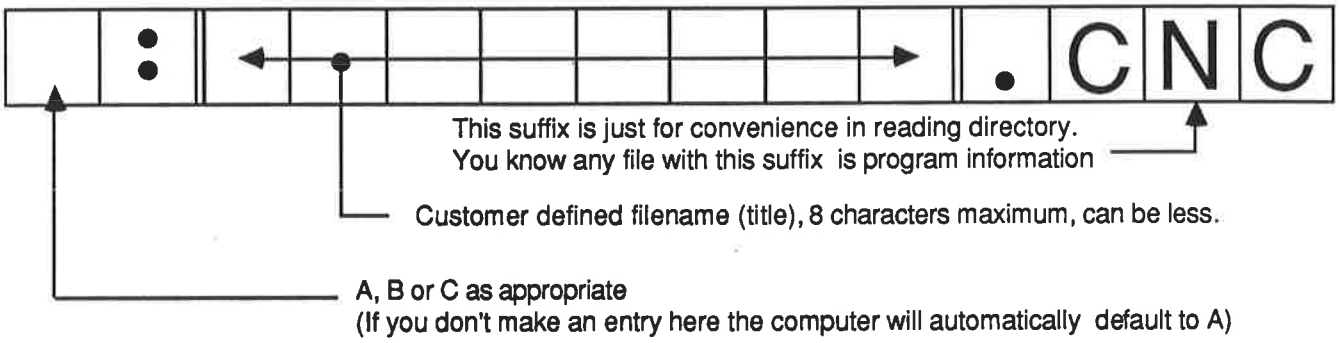
```
**************************************************************************
*                                                                        *
* A  CNC  Control Program from FEEDBACK Inc Berkeley Heights NJ 07922    *
*                                                                        *
**************************************************************************

                    Current Program Function Operating

            SAVE a CNC Program currently in the control computer


Enter filename using IBM conventions as drive-id:filename.extension

For example  B:SAMPLE.CNC  followed by operation of the < ENTER > key

     Name of file to save is:
```

SCREEN 23

3.10.03 Choose a filename which is not already used in the disk directory and enter this at the prompt.

The format of the filename is important. It must identify the drive to be used. If you have your user disk in one drive you can still save to a **FORMATTED DISC** in another drive. It must provide a unique name of not more than 8 characters — including any spaces or punctuation used.

It must terminate with **.CNC**



3.10.04 Lets try an example to see how it all works:

A. Return to **MAIN MENU** — if **Screen 23** is still showing without a filename entered you can get to **MAIN MENU** by pressing **RETURN**, waiting for the error message and then pressing any key.

B. If a filename has been entered just press **RETURN**. The computer will write a disc file under that name and then return to **MAIN MENU**.

C. From the **MAIN MENU** select **option 6 — LOAD**

D. At the prompt select **(Y)**es to proceed.

E. At the next prompt select **(Y)**es for directory and identify the active disk when asked.

F. A display similar to **Screen 22** will appear, any file you have entered per step **B** will be added to the original directory. If you have a printer connected make a print out of the current directory by pressing **SHIFT PRT SC**.

G. Press **"C"** and when the prompt screen **(Screen 23)** appears enter name of file to load as **PROB1.CNC** and press **RETURN**.

H. When **PROB1.CNC** has been **LOADED** from disc into computer RAM the program will return to **MAIN MENU**.

I. To ensure we now have a manuscript in memory select **option 3** and when **Screen 11** appears press **S** to list the manuscript on Screen.

J. You will see that we have actually entered into memory the data of **Figure E1** Page 10 again. We have done this just as a convenient way of providing a known file that we can now try to **SAVE**.

K. **Press C** to complete the listing process and then **C** again to get back to main memory.

L. Now select **option 5 — SAVE**. When **Screen 21** reappears **press N** to bypass the directory listing.

M. Now enter the filename

A : EXAMPLE.CNC

NOTE: If you have a write protected disk in **A** you must have a formatted disk in **B** and change the filename to **B: EXAMPLE.CNC** or save to a hard disk if installed by using filename:
C : EXAMPLE.CNC.

N. **Press RETURN**. Computer displays a **WAIT** message and returns to **MAIN MENU** when the **SAVE**-process is complete.

O. Now, to prove that we have indeed **SAVED** a new file select menu **option 6 — LOAD.**

P. At the prompt press **Y** to erase the current manuscript.

Q. When **Screen 26** appears press **Y** and identify the disk to which you **SAVED** the manuscript. A display similar to screen **22** will appear but with **EXAMPLE.CNC** added as the last file in the list :

R. Now **press C** and at the prompt type the filename

A : EXAMPLE.CNC

S. **Press RETURN**, the computer gives you a loading message and when the process is complete returns to **MAIN MENU**.

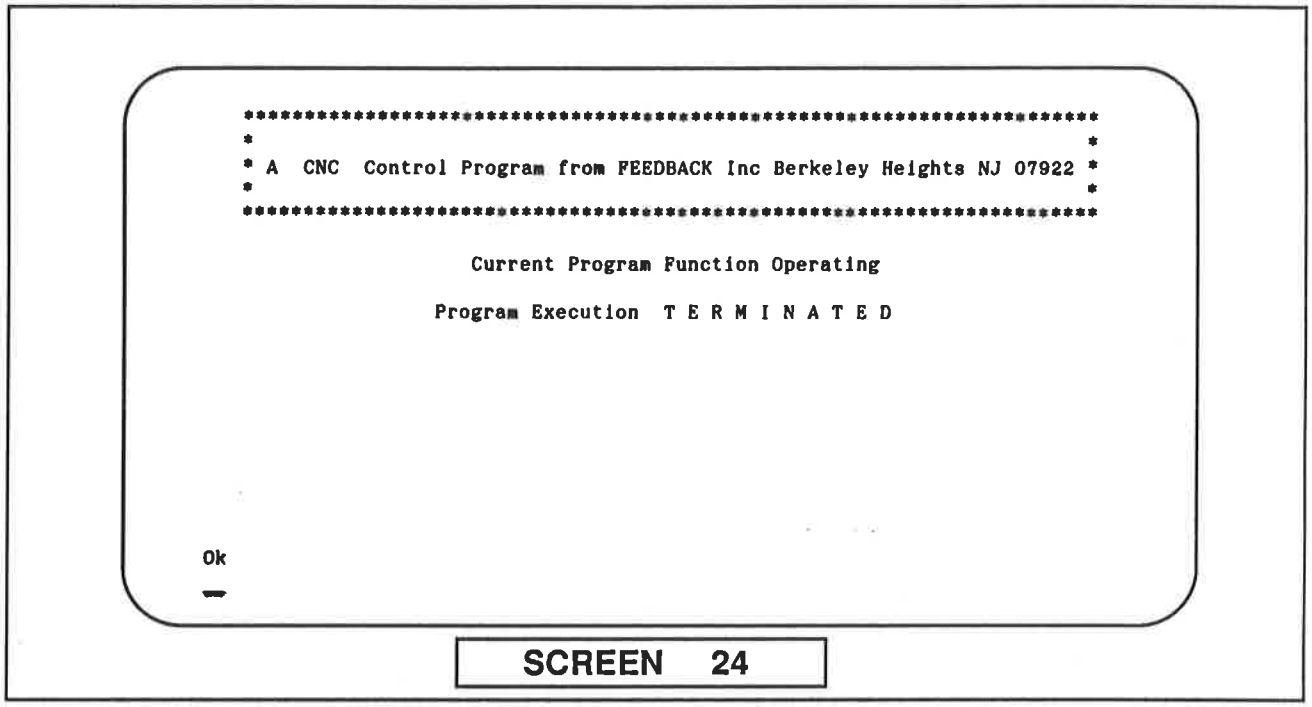
T. Select **option 3** and list the current manuscript to **(S)**creen. You will see we have **SAVED PROB1.CNC** under a new filename — **EXAMPLE.CNC** and have retrieved this new file from disk and listed it on screen.

3.10.05 **A.** We could now take a moment to clean up the disk and remove unwanted files created at steps **B** and **N**. This is not necessary but might be a useful experience. **BE VERY CAREFUL NOT TO DELETE WANTED FILES. NEVER EXERCISE FOLLOWING STEPS ON MASTER DISKS.**

B. Follow computer prompts to return to **MAIN MENU**.

C. Select **option 9** and follow prompts to erase manuscript & terminate program.

D. When you see **Screen 24** the prompt **OK** above a blinking cursor indicates that you have exited the CNC program to **BASICA.**



SCREEN 24

E. Now type **KILL "EXAMPLE.CNC"** press **RETURN**. This will delete the **EXAMPLE** file from the disk. When deletion is complete the computer will signal with **OK** and a blinking cursor.

F. Repeat this process to delete any other unwanted files.

G. If you want to get back into the CNC program now you can do this several ways. The easiest is to press **F2** (the **RUN** key), this puts you back at the start of the program and displays the **INITIALIZATION SCREEN** (**Screen 1**, Page 3). Alternatively, because the **ERROR TRAP** routines of the CNC program are still active, you can generally return from the **BASIC PROMPT (OK + Cursor)** direct to the CNC **MAIN MENU**.

To do this type in some **'garbage'** e.g. **(SHIFT) @ # $ % — ENTER**.

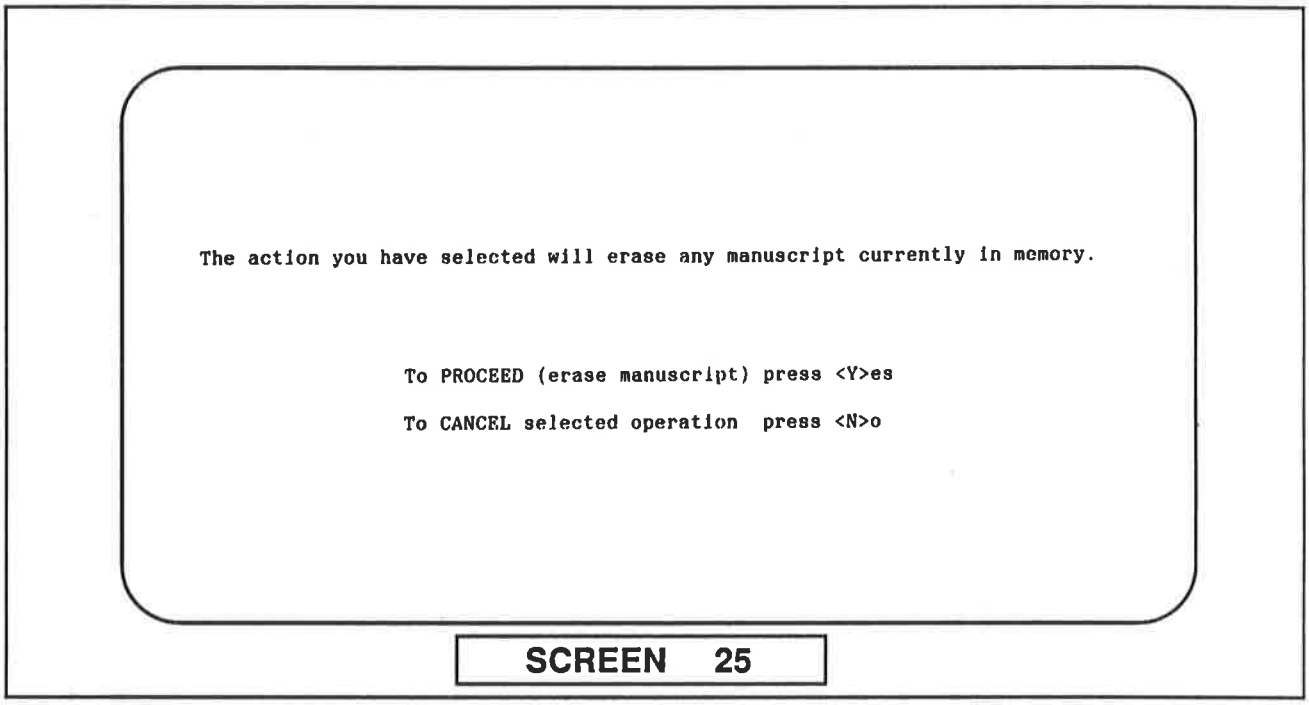
The computer will display an error message. Just press **RETURN** and the computer should display the **MAIN MENU** — there may be occasions when this doesn't happen depending on the precise condition of the error traps. If you do get **'LOCKED UP'** you can just power down and power back up. The program will automatically reboot if a valid **'USER DISK'** is in the computer.

Alternatively you can reboot by simultaneously pressing,

CONTROL ALT DEL

3.11 — LOAD

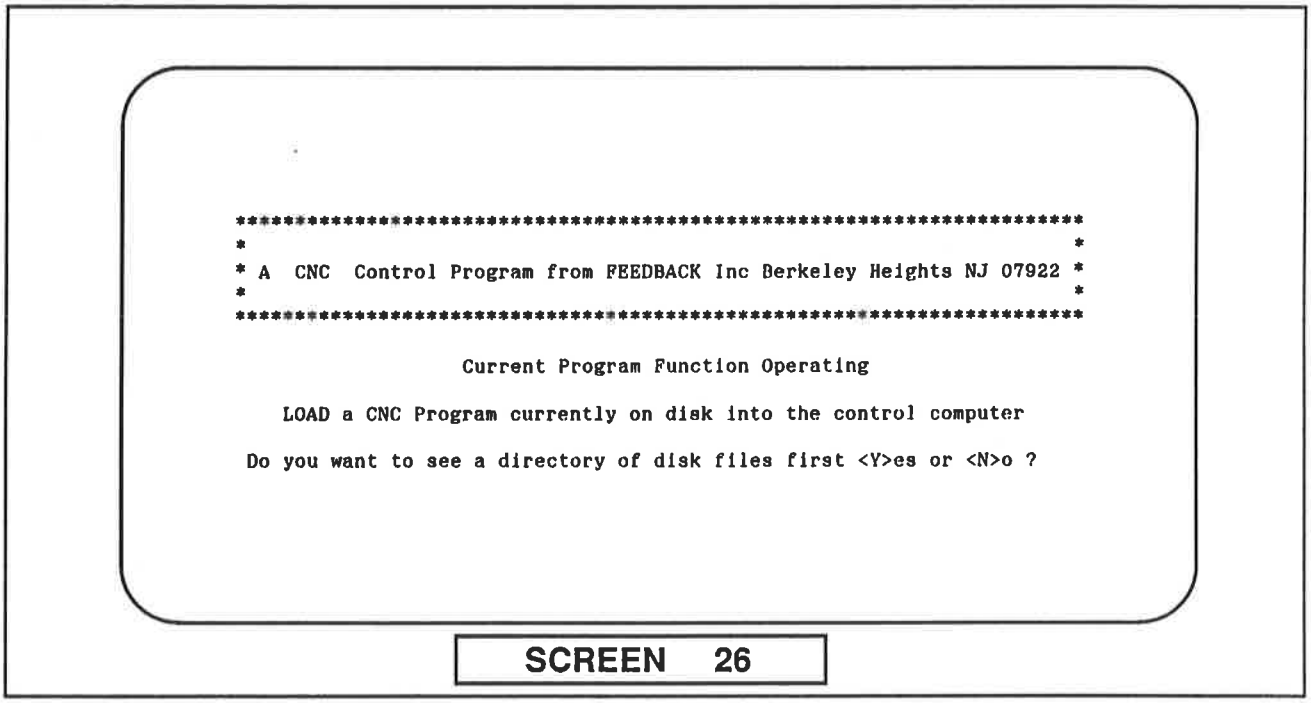
3.11.01 Pressing key **6** selects the **LOAD** option from **MAIN MENU**. The action of **LOADing** will destroy any program currently in RAM. Thus you are reminded of This and offered an opportunity to cancel the command at **Screen 25**.



SCREEN 25

3.11.02 Pressing **(N)**o will return you to **MAIN MENU.**

3.11.03 Pressing **(Y)**es will bring up **Screen 26.**



SCREEN 26

3.11.04 Pressing **(Y)**es for directory will bring up a prompt asking you to identify the drive in which is the disk for which you want the directory.

Enter **A, B** or **C** as appropriate and a catalog of all files currently on the identified disk will be displayed.

The directory of a newly made user disk in drive **A** will appear as **Screen 27.**

```
A:\
COMMAND .COM      CNC     .BAS      FIG2$18 .CNC      FIG2$1  .CNC
FIG2$5  .CNC      FIG2$7  .CNC      FIG2$8  .CNC      FIG2$12 .CNC
FIG2$14 .CNC      FIG2$15 .CNC      FIG2$16 .CNC      FIG2$17 .CNC
PROB1   .CNC      PROB2   .CNC      PROB3   .CNC      PROB4   .CNC
PROB5   .CNC      PROB6   .CNC      PROB7   .CNC      ROBIN   .CNC
BASICA  .EXE      MAKE    .BAT      AUTOEXEC.BAT
 192512 Bytes free



Press  < C > to continue your work
```

SCREEN 27

3.11.05 We have used **PROB1** as an example for other operations, so lets press **C** to get to the file selection screen **(Screen 28).**

```
**************************************************************************
*                                                                        *
* A  CNC  Control Program from FEEDBACK Inc Berkeley Heights NJ 07922    *
*                                                                        *
**************************************************************************

                  Current Program Function Operating

   LOAD a CNC Program currently on disk into the control computer



Enter filename using IBM conventions as drive-id:filename.extension

For example  B:SAMPLE.CNC   followed by operation of the < ENTER > key

Name of file to load is:
```

SCREEN 28

Type **PROB1.CNC** at the blinking cursor prompt and press **RETURN**.

The screen confirms that **PROB1.CNC** is being **LOADed** and then displays a **WAIT** message while the action is completed.

When the file has been successfully **LOADed** into computer RAM the program returns you to **MAIN MENU**.

Just to confirm that you have loaded a program select **option 3** from the menu and list the current RAM contents to **(S)**creen. You should get the table of values as **Figure E1,** Page 10.

3.12 — MANUAL CONTROL

3.12.01 Selection of **option 7** from the **MAIN MENU** puts the operator in manual control of the CNC machine.

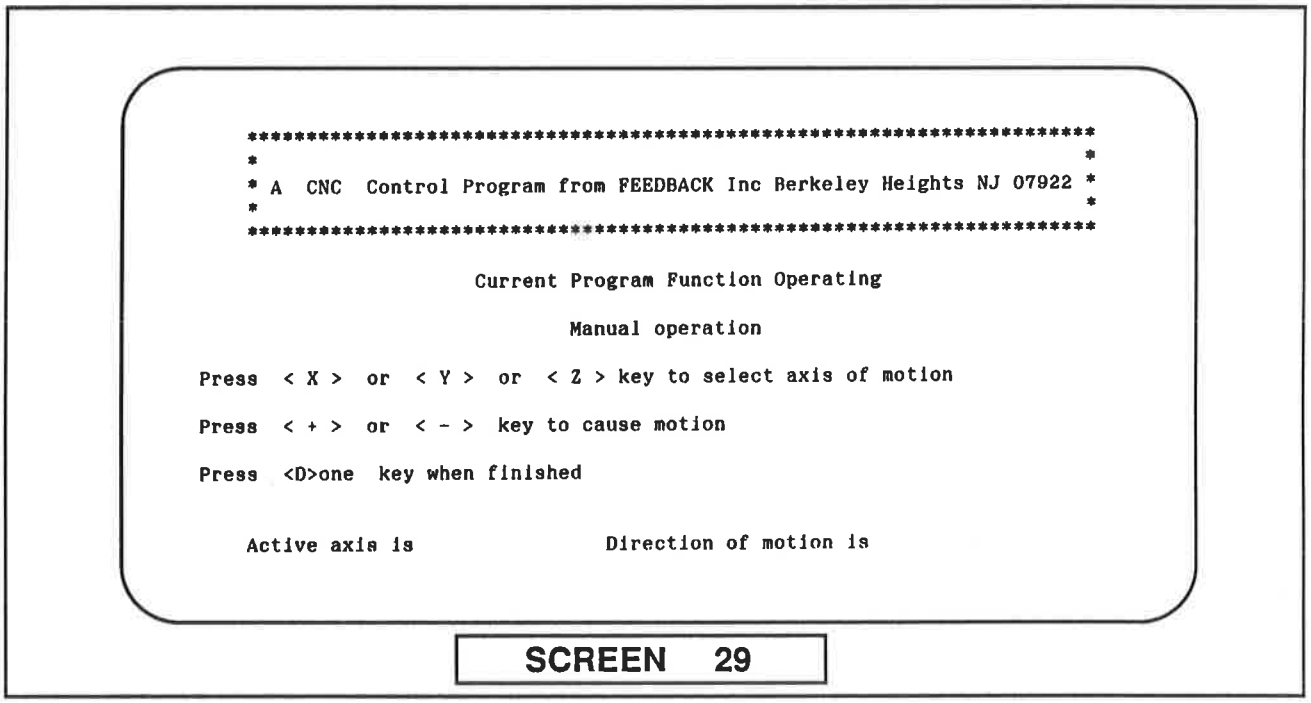
The **X, Y,** & **Z** axes can be driven one at a time in either direction by use of **+** and **-** keys. While the machine is in motion the appropriate **+** or **-** symbol is displayed on the screen.

The **+** or **-** keys on the numeric keypad or the shifted characters on the keyboard are both recognized in this operation.

A single key tap will move the drive motors one step, holding the keys down causes continuous motion until the key is released.

When the machine is satisfactorily positioned press **(D)**one to return to **MAIN MENU**.

3.12.02 **Screen 29** is the control screen for this operation. If you have not already done so press **7** to get this screen.



SCREEN 29

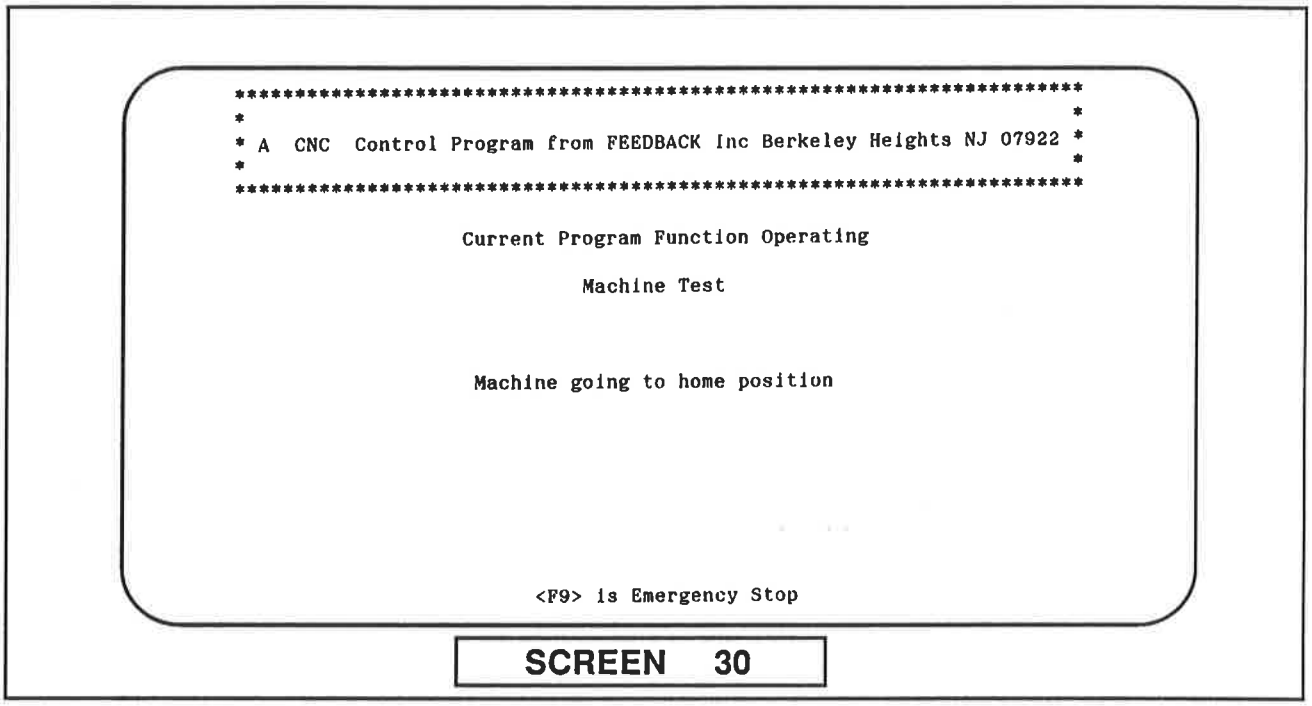
3.12.03 Now type **X** for the axis and hold down the **+** key. The LED indicators on the CNC control module will illuminate to indicate that **X** is enabled and that the axis is moving in **+** (positive) direction.

3.12.04 Release **+** key and type **Y**, the computer screen will change to indicate **Y** as the chosen axis. Holding down the **+** or **-** key will cause the **Y** indicators on the CNC control module to come on. If you try to drive any axis past its limits the appropriate **LIMIT** warning light will light up on the control module.

Observe the CNC machine bed, use the **X,Y** and **Z** control options to position it as you desire, then press **(D)**one to return to **MAIN MENU**.

3.13 — MANUAL OPERATION FROM EMERGENCY STOP

3.13.01 When the CNC machine is active in **TEST (menu option 1)** or **EXECUTE (menu option 8)**. The monitor will display a screen similar to **screen 30**.



3.13.02 When the **"<F9> is Emergency Stop"** message is displayed you can stop all machine motions (**X, Y, Z** and spindle motor) by pressing **F9**.

3.13.03 Exercising this option brings up the **MANUAL** control screen **(Screen 29)**.

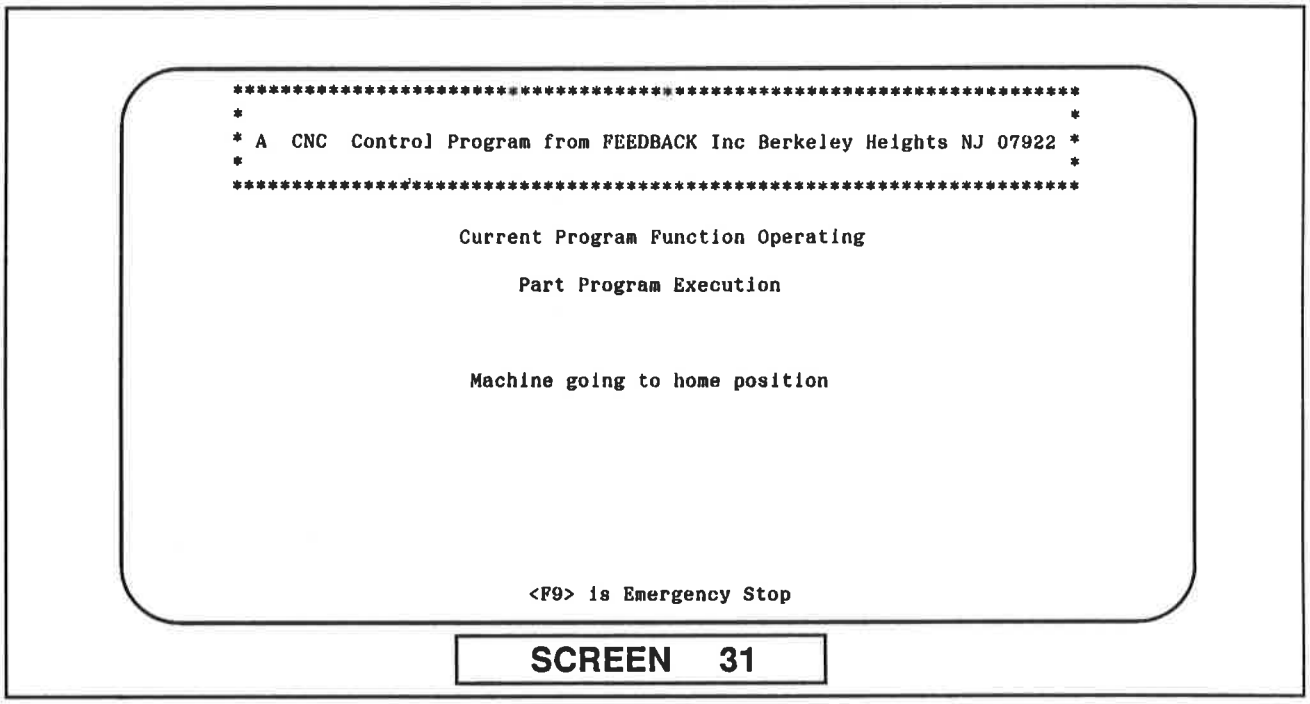
Following the on screen directions you can then move each axis individually to its desired position.

When **"remedial manual positioning"** is complete press **(D)**one and you will be returned to **MAIN MENU.**

3.14 — EXECUTE

3.14.01 This is the function that enables you to machine a part from a program (manuscript) stored in computer RAM. This manuscript can be typed in under **ENTER** or **EDIT** option, or a prerecorded manuscript can be **LOAD**ed from disk.

This function is activated by pressing key **8** in the **MAIN MENU**. The part program to be executed must already be in the computer. **Screen 31** is displayed while the machine is returning to home position.



SCREEN 31

3.14.02 Observe the machine in operation and be prepared to exercise the **EMERGENCY STOP (F9)** if there is any danger to tool, workpiece or operator.

If you do exercise **EMERGENCY STOP** you can use the **X**, **Y** and **Z** manual control facility to move the machine to safety. When **(D)**one you are returned to **MAIN MENU**.

When the CNC machine reaches its **HOME** position **Screen 32** is displayed.

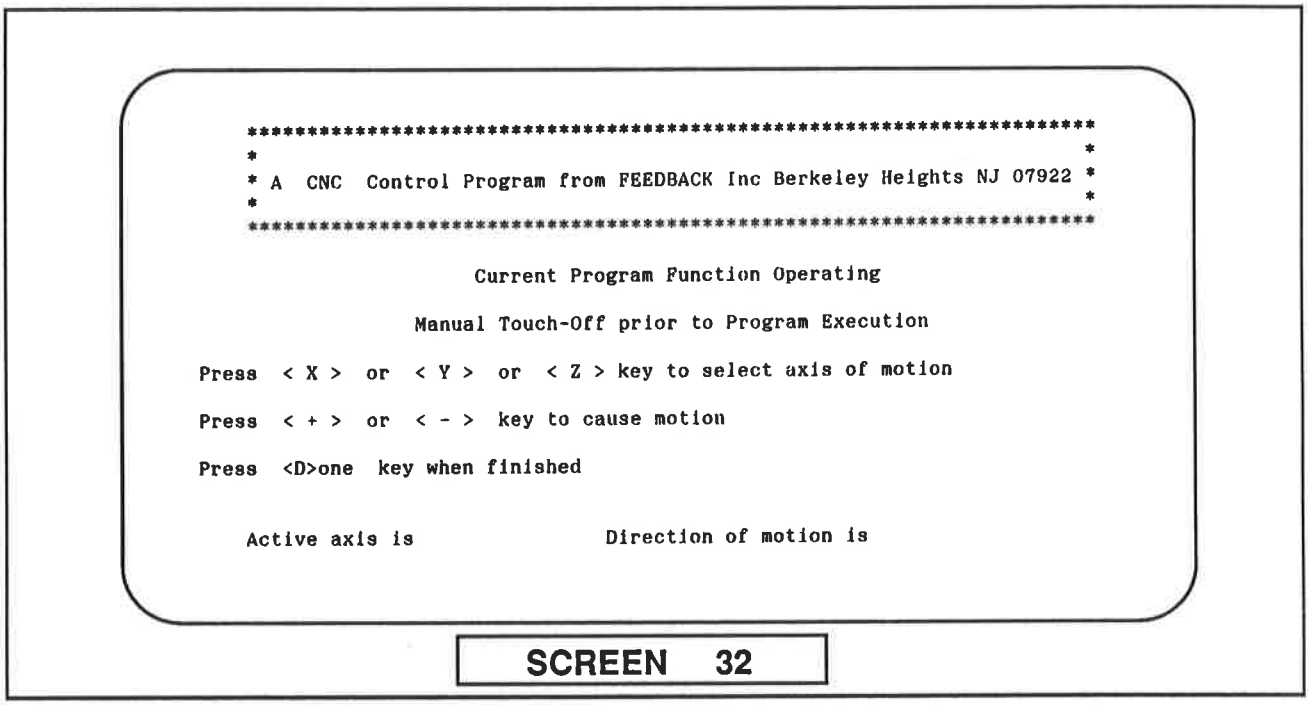
3.14.03 This screen functions to enable touch off. Touch off is the process of moving the machine manually so as to establish the correct part-tool relationship to permit correct part production. The conditions and dimensions for touch off are established by the programmer at the time the program is written. These conditions and dimensions must be passed to the machine operator, along with the program listing, in the program documentation package. For example to produce the part of the manuscript in **Figure E1**, with a depth of cut of 2mm a typical touch off procedure would be:

a. Move the tool towards the surface of the part material using the **Z** axis and the key.

b. With a piece of thin shim stock (.0005) on the surface of the part, and directly below the tool, continue to move the tool downward until the tool contacts the shim stockand friction is felt when the shim stock is moved.

c. **Figure E1** shows a **Z** axis motion of 10mm, thus to get a depth of cut of 2mm, the tool must now be raised a total at 8mm. Do this by selecting the **Z** axis and pressing the **+** key 8 times.

d. When accomplished press the **(D)**one key.



Axis selected will be displayed.

A "+" or "-" will be displayed while the machine is in motion.

3.14.04 When **MANUAL TOUCH-OFF** is completed a part can be machined. However, before we proceed check the following.

A) Correct tool and workpiece are fitted in the CNC machine

B) Correct initialization **(HOME)** and **MANUAL TOUCH-OFF** procedures have been completed.

C) A valid **MANUSCRIPT** (machining program) has been **ENTERED** or **LOADED** into computer RAM.

3.14.05 As an example lets return to **MAIN MENU** and start the sequence from scratch.

If **Screen 32** is still displayed press

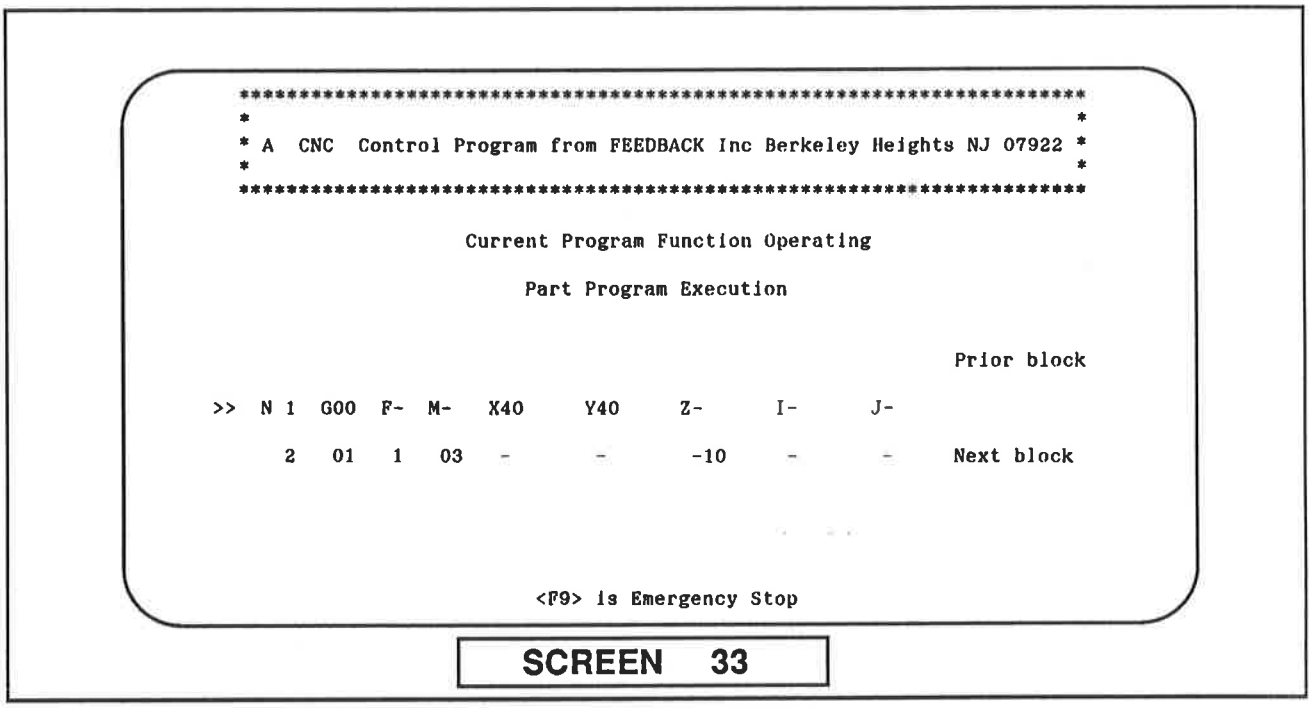
RETURN

RETURN

This will bring up the **MAIN MENU**.

Select **option 6** and **LOAD PROB1.CNC** as we did in 3.11.05. This provides us with a manuscript in memory from which to machine a part.
When **LOADing** is complete and computer returns to **MAIN MENU** select **option 8 — EXECUTE.**

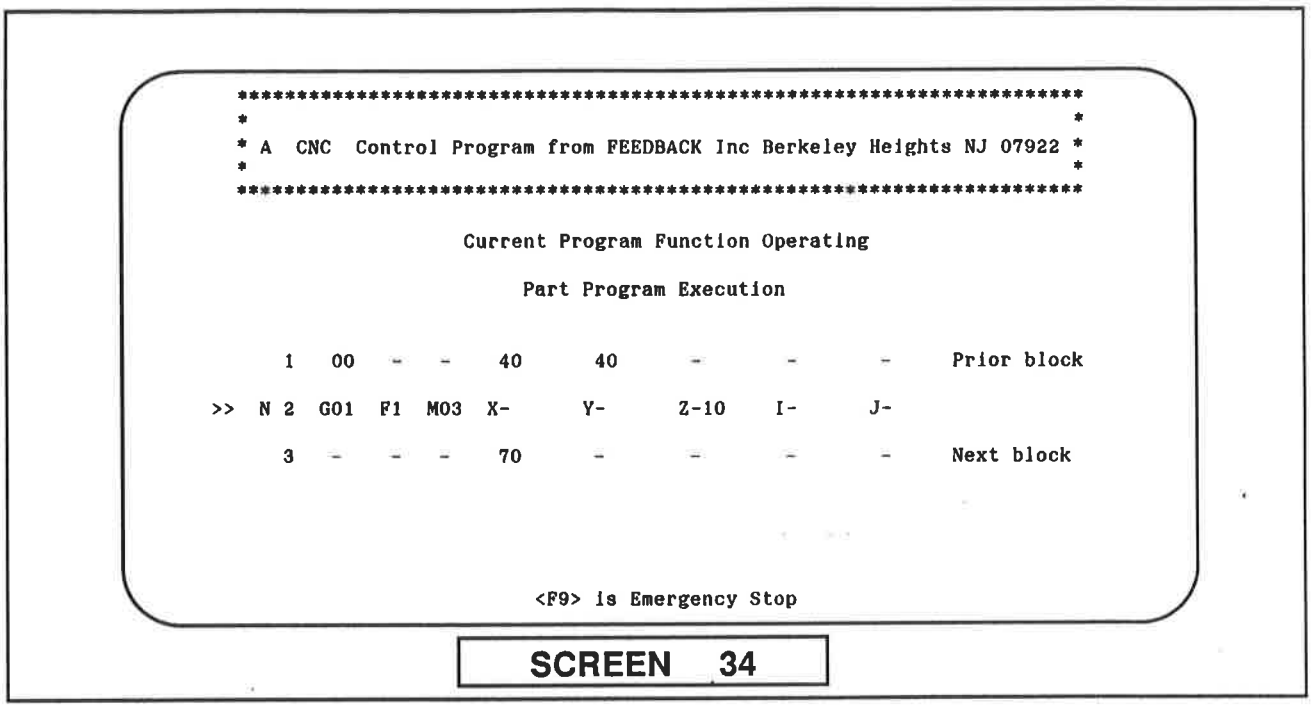
3.14.06 **Screen 31** will reappear, and CNC will check its home position. When initialization is complete **Screen 32** comes back. By pass the **MANUAL TOUCH OFF** procedure by just pressing **(D)**one for this example. **Screen 33** will appear and the CNC machine will commence the machining program of **PROB 1.**



SCREEN 33

You can recognize line **1** as the first line of **Figure E1** page 10. The **"Next Block"** is obviously line **2** of the same figure. Because we are displaying line **1** there is no **"Prior block"** to display.

You allow the machine to complete line **1** the display changes to **Screen 34.**



SCREEN 34

Review of the last two screens shows that as the program is executing the prior block, current block executing and the next block to be executed are displayed.

3.14.07 During execution of the part program, should an incorrect **"G"** or **"M"** code be encountered, program execution will terminate.

3.14.08 During part program execution, incorrect **"I"** and **"J"** values will cause part program execution to be terminated.

3.14.09 Incorrect numbered values for **"X', "Y", "Z", "I"** and/or **"J"** may cause a machine Jam-up.

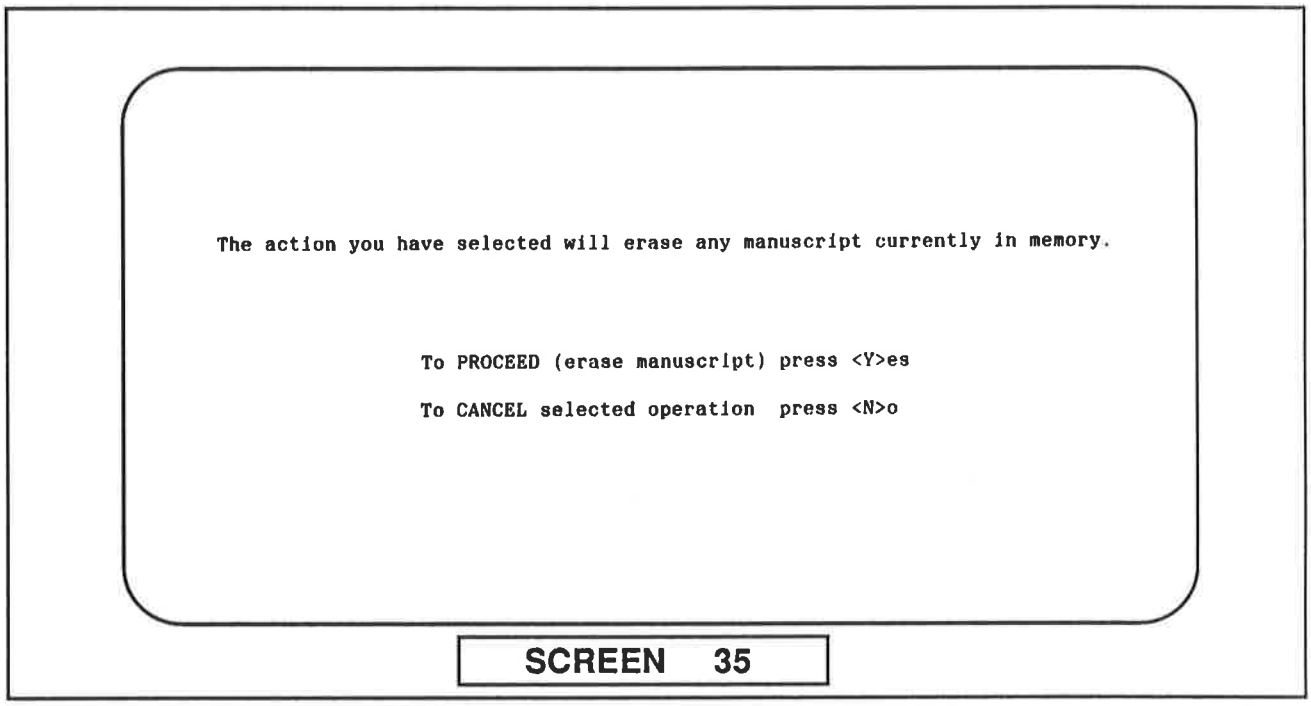
3.14.10 When production of the part is complete, the program returns to the **MAIN MENU** after a delay.

If yhou wish to exit the program before completion use the **EMERGENCY STOP** option **(F9)**.

3.15 — TERMINATION

3.15.01 Option **9** of the **MAIN MENU** allows you to terminate the CNC program.

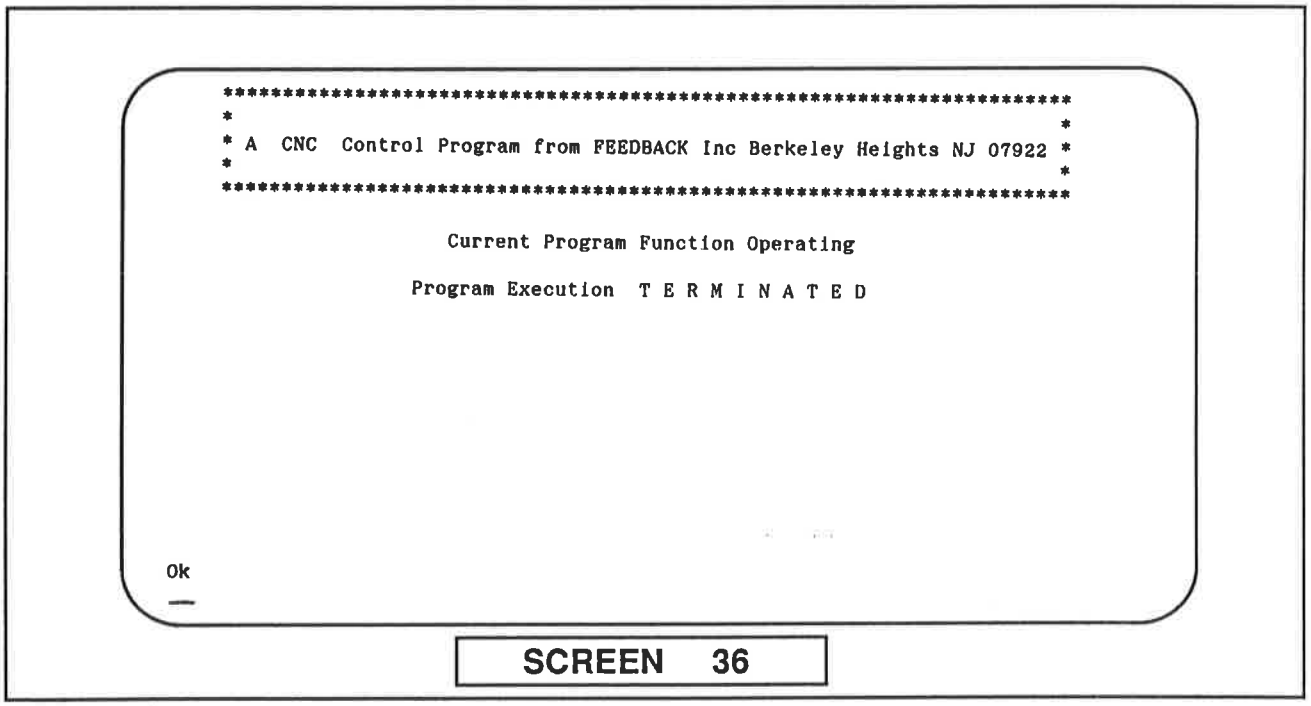
Pressing **9** will bring up the caution screen — **Screen 35**.



SCREEN 35

Selecting **(N)**o will return you to **MAIN MENU**.

3.15.02 Selecting **(Y)**es will terminate the CNC program and leave you in **BASICA**. This is confirmed by **Screen 36**.



SCREEN 36

3.16 — HELPFUL HINTS

3.16.01 The CNC program has been written to be as user friendly as we could make it and as representative of contemporary industrial practice as possible within the technical and economic confines of current educational budgets and resources.

Don't be afraid to experiment with the program. Providing you exercise reasonable care in keeping the tool from fouling and damaging the machine tool bed or part clamps there is very little that can go dramatically wrong.

Helpful screen displays and extensive use of error traps in the program minimize potential "lock-ups" and "program bombs"

However, to understand how the program operates and to be able to extend and modify the fundamental program you will need to study the **BASIC** and **DOS** manuals for the computer you are using.

3.16.02 IF THE SYSTEM DOESN'T WORK:

A. Check that the set-up is as **Figure 2.2**.

B. Make sure the CNC control module circuit jumper has been set for IBM use as **Figure 2.1.**

C. Make sure an MIC926 interface has been correctly configured and fitted to the computer as page 2.1.

D. Make sure all connectors are properly tightened.

E. Make sure the pointer and disk drive you are using are compatible with the computer.

F. Make sure computer clock frequency is set for 4.7MHz

G. Make sure all system components have been switched on and that indicator lamps show power applied.

H. Make sure you have installed a **USERS DISK** properly made per instructions of Chapter 1 and that you have correctly identified the version required as on page 1.2 section D.

I. Make sure the spindle motor is switched **ON** — the switch is on the top end of the spindle motor.

J. A few helpful 'computer hints' follow — please note these are no real substitute for a proper understanding of your computer operating system and **BASIC** language.

3.16.03 A FEW COMPUTER POINTERS

A. To reboot a running system

— press **CTRL ALT DEL**

— or simply powerdown and power back up with autobooting users disk in computer

B. To break out of a running program

— press **CTRL BREAK**

reboot as above, **CTRL ALT DEL**

C. To **LOAD** a **BASIC** file from disk:

When not in the CNC program environment

Type **BASIC** or use **F3**

When operating within the CNC program environment use **LOAD** from **MAIN MENU.**

D. To **SAVE** a **BASIC** file from disk:

When not in the CNC program environment type **SAVE** or use **F4.**

E. To get from **BASIC** to **MS-DOS**:

Type **SHELL**, this puts you in **DOS** but leaves **BASIC** in the computer.

F. To get from **MS-DOS** to **BASIC**:

Type **EXIT**

G. To delete unwanted files from disk:

Exit CNC program and enter **BASICA** — menu **option** 9

Type **KILL** "filename" **(RETURN)**

CHAPTER 4 - CNC PRINCIPLES & OPERATION

4.1 — THE CNC932 SYSTEM

4.1.01 INTRODUCTION

The CNC932 Computer Numerically Controlled Machine is a working miniature machine tool capable of being controlled by any ordinary microcomputer compatible with a Feedback MICA interface. It is intended to be used 'hands-on' as a practical trainer for students of computer numerical control (CNC).

This manual is intended for students interested in part-programming (telling a machine what shape to make the workpiece) and other aspects of CNC. It treats the CNC932 as a practical CNC machine in miniature. For computer-related aspects, consult the other manual, Volume 1 — **Programming in BASIC**.

4.1.02 This Volume 2 shows, with hardly any reference to computers or computer programs, how students can acquire the skills of CNC by:

* using internationally-recognised part-programming languages to describe the geometry of a part and the actions which a machine must take in order to produce that part.

* entering, editing and listing part-programs using a keyboard and symbols similar to those on a full-size machine tool.

* using the CNC932 as a working CNC machine tool in miniature, to run and test their part-programming exercises without the heavy risks or cost incurred when full-size machines are used.

4.1.03 The CNC comprises:

A) a model of a drilling/milling machine

B) a control unit with integral power supply

C) this manual

D) a separate manual (Volume 1) for use by people who wish to study the principles underlying the software

E) one 2mm end mill, two 1.5mm drills and two 2mm drills

F) a sample supply of working material, Toysteel.

4.2 — ANCILLARY REQUIREMENTS

These may be summarised as:

A) a computer equipped with disc drive and, if listings are required, a printer

B) an interface

C) a disc containing software (a computer program) which tells the computer how to behave as a CNC controller

D) blank discs

To use the CNC932 machine a computer is necessary, together with a Feedback MICA (or MICA-compatible) interface suited to the computer. For the exercises presented in this volume, which use the CNC932 as a working CNC machine tool, software is also required.

To load the software the computer must be equipped with a disc drive. The software should be ordered separately, and so should the MICA interface, specifying which microcomputer they are required for.

A supply of further discs will be needed for the purposes described later.

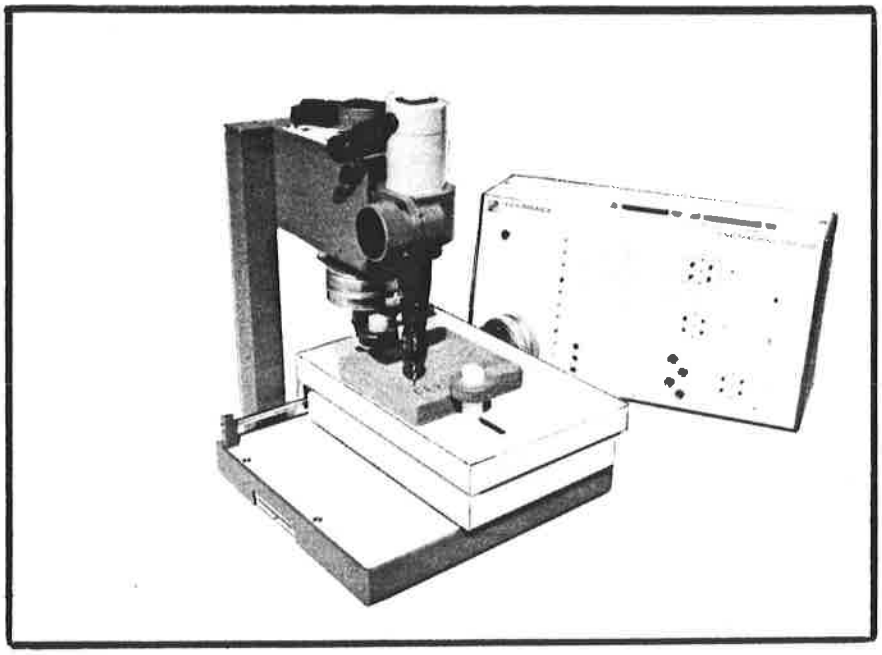


FIGURE 4.0 THE CNC MACHINE TOOL WITH ITS CONTROL UNIT

4.3 — SOFTWARE NOTES

Copyright subsists in the software, which must not be reproduced in any form beyond the strict limits stated here.

Discs or the information on them can be destroyed by various kinds of accident, by misuse and by wear. Accordingly the master software disc which is supplied should be used only to create **User Disks**, and only copies should ever be used by students. For details of how to copy discs see chapter 1.

Copying of the software is allowed only to the extent required for the use of one CNC932 machine, unless otherwise agreed in writing. Without limiting the generality of the foregoing, sale of copies of the software or material based on it without written permission is expressly forbidden.

Copied software discs used by students may be write protected. This means putting an adhesive tab over a slot in the edge of the disc's permanent case, to prevent anything being recorded on the disc. (It can have information read from it, but will not accept information put into it).

Discs need careful handling. Exercise 1 contains instructions which are as detailed as practicable, and more detail will be given in the literature of the computer maker or disc drive maker. However, instructors may have different equipment, or ideas on how to use it.

For instance, some instructors may prefer to have discs and computers set up by laboratory staff. If students are to be discouraged from handling discs at all, it may be necessary to have a copy of the software and the student's work on the same disc, which would need to be **NOT WRITE PROTECTED.**

On the other hand, students may be less distracted if all that appears on their working disc is their own output, in which case a separate write-protected disc with a copy of the software is appropriate, and students will also require a spare disc for saving their workpiece-programs. This should be formatted in accordance with the computer maker's instructions, but not write protected.

The manual envisages these possibilities, but students may need local guidance on exactly what they are expected to do.

Discs and disc drives should be protected as effectively as possible from dust and swarf generated by the machine, since they can destroy both discs and the drive. Careful siting and the provision of adequate storage for discs are essential.

4.4 — PRINTER

A printer provides the facility to record part-programs on paper, using the **LIST** command. This is not a necessity, but will be convenient for instructors who wish to grade their student's work and can provide students with a record for future reference. The printcan of course be used with any other applications of the computer also if required.

4.5 — MILLING CAPABILITY

This machine is not intended for any but light milling, using small cutters on selected materials. An especially suitable material is Toysteel, obtained in the UK from Denford Machine Tools Limited, Birds Royd, Brighouse, HD6 1NM, West Yorkshire. Machineable wax is another suitable material, obtainable from Portisle Limited, 8 Charman's Close, Horsham, West Sussex, or from Freeman Manufacturing & Supply Company, 1246 West 70th Street, Cleveland, Ohio 44102.

4.6 — HISTORICAL BACKGROUND

4.6.01 Origins

Some of the basic ideas which lead to computer numerical control (CNC) are very old. The making of records in some form of symbols (such as numbers or writing) is an example. Another is the idea of using numbers to describe the size of things. Such numbers are always associated with a unit' in workshop practice this is very often standardized as a millimetre (mm). The idea of an aid to calculation is as old as the abacus, found in very early civilisations. Since the time of ancient Egypt geometry has been an essential system of ideas for defining shapes.

An important further idea, only a few hundred years old, is that of using co-ordinates, which are simply groups of numbers by which the position of any point may be specified. By specifying enough points in this way, any shape can be specified as accurately as desired.

A machine tool creates a desired shape by performing motions of various kinds which cause the shape to be cut or formed in some way. Machine tools can be provided with numbered scales on each of their axes of motion, so that the operator can read off a dimension easily, or perhaps even set a co-ordinate.

4.6.02 Automatic Machines

As the demand for engineering products increased, engineers devised ways of making machines perform an operation over and over again. This enabled repetition work to be done much faster and more cheaply than if a man had been needed to control the machine. It made new demands of skill on the people who 'set' the machines, but each setter could look after several machines.

The disadvantage of simple automatic machines is that any except the most minor alteration to the kind of task a machine can do is impossible, or costs so much in setting-up time that it is not economic unless large batches are to be manufactured. Numerical control was devised as a response to this, aimed at making more complex settings possible and/or taking less time to alter them.

4.6.03 Numerical Control

Numerical control puts together two uses of numbers. One is the idea of co-ordinates and the other is that of using numbers as coded instructions which will alter the behaviour of a machine. Putting these together, it becomes possible to tell a machine to do simple things like start and stop, and to relate these functions to having reached a particular co-ordinate or dimension of the workpiece.

A system of numerical control needs:

A) someone to specify the shape of the required part

B) someone or something to translate this into a specification of the required motions of the machine

C) some way of getting this specification into the machine in terms that it can 'understand'

D) devices within or attached to the machine which make the machine move in the desired way

The specifications of the shape and/or the required motion must be recorded so that they can be used more than once, and if necessary amended from time to time. Ways of conveying information from the stored record to the machine are usually based on electrical signals, and the final drive to the machine is generally electrical or hydraulic.

Early systems of numerical control generally used holes punched in paper tape as the record and data transfer medium. The tape might be punched on a machine with a typewriter mechanism and a mechanism which punched a particular coded pattern of perforations for each symbol typed. This produced simultaneously a human-readable typed document and a tape which could be read by a reading device mounted on or near the machine tool.

There were disadvantages of such a scheme, including:

E) the rate at which people type errors, and the clumsy process necessary to correct them.

F) the necessity to re-do in every detail all the coded instructions often for reasons very simple in concept, such as a change in the scale of some part.

G) lack of flexibility in the available coded instructions, with no provisions for adding or inserting further ones as the need arose.

4.6.04 Computer Numerical Control (CNC)

A computer is:

A) a modern and effective way of handling numbers.

B) capable of providing communication, with all the necessary changes in language, coding etc, between widely differing devices.

C) flexible in operatin because a new program can be inserted, causing the behaviour to change.

D) potentially cheap, because one mass-produced design of computer may be able to serve in several different designs of machine tool, which purpose designed electronics could not.

It is not surprising therefore that computers can be made to control machines, and that this practice is becoming increasingly usual. Adding the computer's initial C to the NC of numerical control gives CNC.

To use CNC, people must be able to tell the computers what they want the machines to do. It is obviously helpful, if as far as possible machines 'understand' the same language, so that people do not have to learn new ways of instructing each particular computer or machine.

International standards have been agreed for such languages, and these will be used as far as possible in this manual.

4.7 — FAMILIARIZATION EXERCISE IN COMPUTER CONTROL

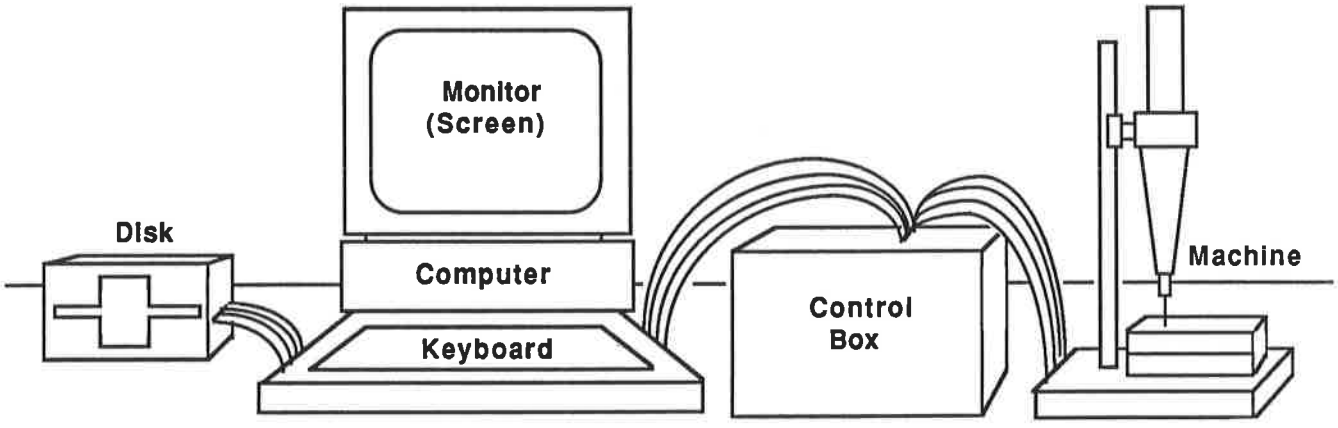


FIGURE 4.1

4.7.0I Introduction

Figure 4.1 shows the communication path between you and the machine. You can communicate with the computer by means of its keyboard **(INPUT)** and display screen **(OUTPUT)**. It communicates with the control box via a cable. Finally the control box converts the computer's low power signals into high power signals to drive the machine's motors.

(The machine also sends data back the other way, but that will come later).

No computer can do anything useful unless it has been told what to do. It needs lots of instructions, and the complete list of them is called a **'program'.** The computer programs you need have been recorded on a special sort of disc that the computer can read quickly. (Workshop CNC systems usually have the program 'built in', so that you do not have to worry about it).

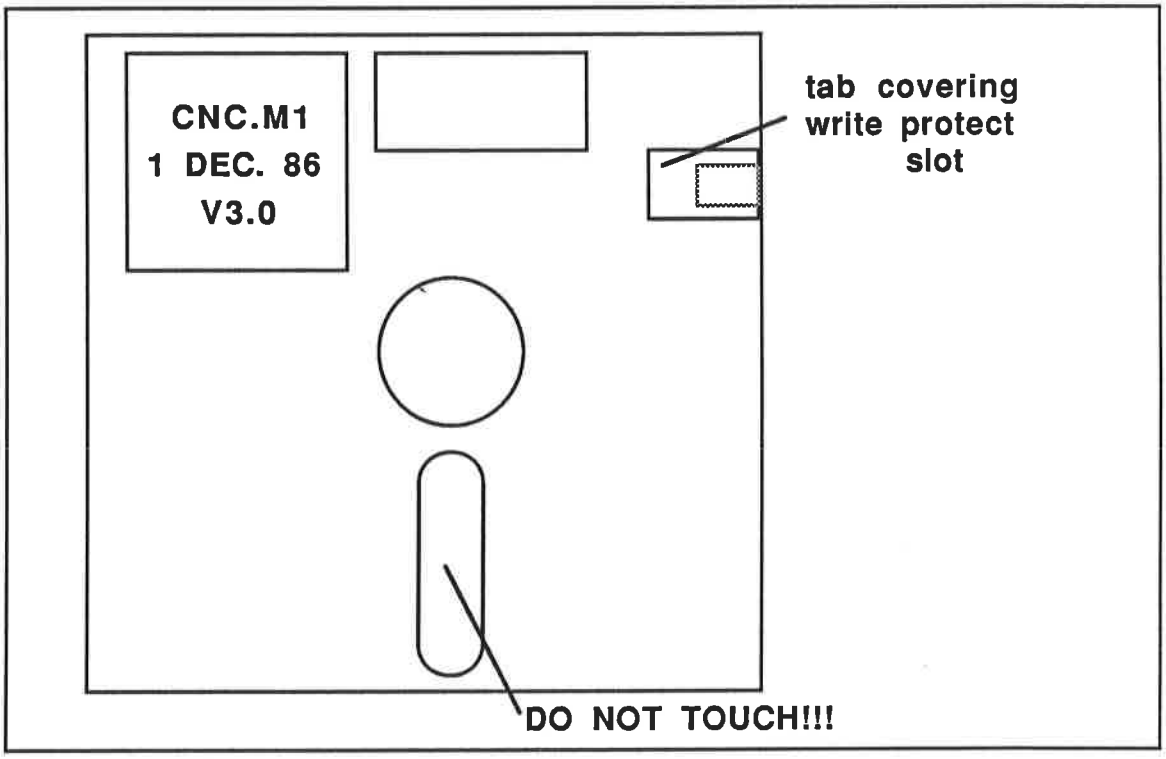


FIGURE 4.2

4.7.02 Preparation

Your instructor may have provided special instructions to get you started. If so follow them, but if not carry on as follows. Find the **USER DISK** - made as instructed in Chapter 1. The **DISK** looks like **Fig. 4.2.** It should bear a label identifying it as the **CNC USER DISK** and give the date it was created and the version number of the master disk that was used, — see Chapter 1, section 1.2J.

Some instructors will arrange for it to be in the disc drive ready for you. The disc drive is the device which reads the disc. It has a door (usually a small flap in the center of the slot which accepts the disc); the door should be open. Open it if necesary, usually by raising the flap **GENTLY.**

Otherwise you should find the disc (and sleeve) in an outer protective cover.

Important - The disc itself should never be touched. The smallest amount of dirt or of grease from the fingers could damage it and the disc drive (reding device). Abrasive dust created by the machine's operation is especially destructive, so keep the disc away from it.

The actual disc is coated with magnetic material on which computer programs are recorded. The process is similar in some ways to the technique of recording on tape (e.g. in cassettes). It is somewhat flexible, but should always be handled with care (rather like an expensive gramophone record).

As with a cassette tape, careless recording can write over existing information and destroy it. To minimise the chance of this happening, the disc may be provided with a 'write protect' facility which operates when the self-adhesive strip covers the write protect slot, **figure 4.2. — Do not remove it. — Discs are delicate.**

4.7.03 Rules For Handling Discs

DON'T bend a disc

DON'T touch the disc itself

HOLD a disc by the label, between finger and thumb of **ONE HAND** only

KEEP discs **AWAY** from:

stong sunlight
extremes of heat, cold or damp
strong magnetic fields
dirt and swarf

TAKE CARE to follow the instructions

4.7.04 Loading The Computer Program

Switch **OFF** the CNC control unit. Do not switch it on until told.

After checking that the disc drive door is open, switch on the power to the monitor (television-like display device) and printer if connected. If the program disc is not in the drive, grasp it (always gently) by the top edge (edge nearest the top of the printed label). Continue holding it like that as you remove it from its protective cover and insert in into the slot of the disc drive. Insert it **GENTLY** until it is all inside, close the door and switch **ON** the computer.

The disc drive lamp will come on and the drive will click and whirr for a few moments while the computer **READS** the disc and **LOADS** the CNC control program into RAM. When the **LOAD** process is complete, the drive lamp extinguishes and a prompt appears on the computer screen asking for a new data. Just hit **RETURN** and the computer will advance to another prompt asking for a new time. Just hit **RETURN** again.

After a few moments of disc drive activity, the CNC title screen will appear:

```
*************************************************************************
*                                                                       *
* A  CNC  Control Program from FEEDBACK Inc Berkeley Heights NJ 07922   *
*                                                                       *
*************************************************************************

                    Current Program Function Operating

                     Initializing Printer and Screen


               Adjust Display controls for Satisfactory Video Viewing

                 When satisfactorily Adjusted Press  <S>atisfactory
```

FIGURE 4.3 CNC TITLE SCREEN

Now switch **ON** the CNC control moduleand the machine spindle moto. Respond to the prompts on the monitor to bring up the **MAIN MENU** of the control program. If you experience difficulty refer to the guides in Chapter 3.

From now on you may think of the keyboard and screen as being those of a simple machine controller.

You can now go on to Exercise 7.7.05. But if the computer is switched off, even for a moment, Exercise 4.7.04 must be done again before anything else. (Otherwise the computer 'forgets' what to do).

What has happened is this. The computer understood the sequence of keys which you pressed as a coded instruction to read a program from the disc and then 'execute' it, i.e. do what it says. What the program says to the computer is (in effect) 'Behave like a CNC Controller'. Without the program it won't know how to.

4.7.05 Exercising The Axes of Motion

The machine shown diagrammatically in **Figure 4.4A** is intended to have a workpiece mounted on the table. This might typically be a block of material to be drilled. When giving the machine instructions, it is convenient to think of how the cutter moves relative to the block. For this purpose, we can establish axes fixed in relation to the block, as in **Fig 4.4.B.**

Examination of the machine will show that it is straightforward to move the drill head relative to the block (or other workpiece). Thus an upward movement of magnitude **Z** will raise the drill head (relative to the workpiece as well as absolutely); if **Z** is negative the cutter will approach the workpiece, moving downward.

Considering horizontal motions is not quite so straightforward. The drill head will not move in the **X** or **Y** directions. Instead, a movement of the table moves the workpiece. If a movement of the cutter relative to the workpiece is to be x units in the **+X** direction, then the table is moved x units in the opposite direction, which is called **X', Figure 4.4.** Similarly, a motion of magnitude and sign **+y** of the table in the direction **Y'** has the same effect as would be obtained if the drill head were to move **+y** in the **Y** direction.

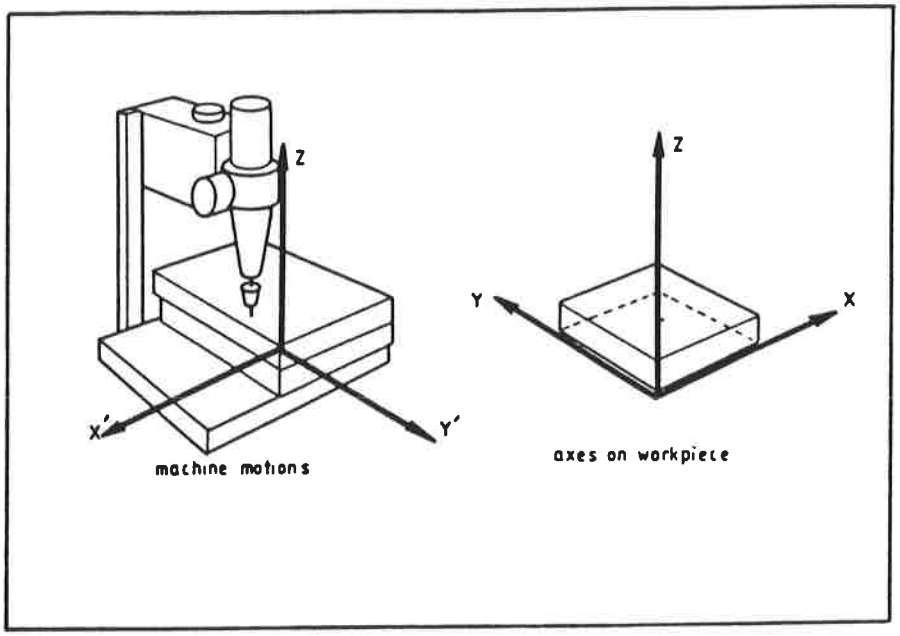


FIGURE 4.4 AXES OF MOTION

However, virtually all the work covered by this volume deals with motion ***of the cutter relative to the workpiece.*** All references to motion should be understood in this way unless stated otherwise. We shall therefore ignore the distinction between **X** and **X'**, **Y** and **Y'** from here on.

Now, from the **MAIN MENU** displayed on the computer screen, select option 7 **MANUAL** control. This will allow you to control the CNC932 manually and thus gain familiarily with its motion.

Take a few minutes to become comfortable with the **MANUAL CONTROL SCREEN** and the actions it calls for. Practice moving the **X, Y** and **Z** axes from the keyboard. Note that one sharp tap of the + or - key moves the relevant axis a lmm step.

Notice that if you hold the keys down or tap them rapidly, the machine moves continuously. Some keyboards store these drive signals in the keyboard buffer and this causes the machine to continue moving after you release the key. Be careful not to loose control of the machine in this way. Refer to Chapter 3 for help if necessary.

4.7.06 Coordinates

When you are confident that you can control the machine manually place a sheet of paper on the machine's table. Mark on it a point underneath the drill. (All through this exercise, do not worry too much about accuracy - it's just the basic ideas we want to learn).

Drive the machine about 30mm in the positive **X** direction and mark an **X** on the paper under the drill.

Reset to datum and then drive the machine 30mm in the positive **Y** direction. Make a mark **Y** under the drill.

Now take the paper off the machine and join **0** to **X** with one straight line, and **0** to **Y** with another straight line. The result should be somewhat similar to), **X** and **Y** in **Figure 4.5**.

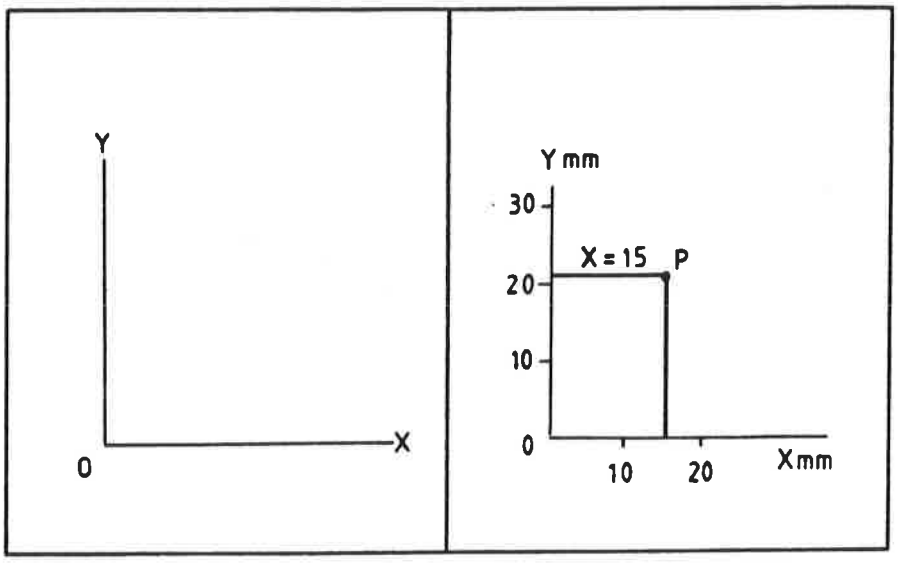


FIGURE 4.5

FIGURE 4.6

Figure 4.6 shows a point **P** which is at a distance **X** from one line and at a distance **Y** from another. The distances **X** and **Y** are called the **'co-ordinates'** of **P**. The two lines are called **'axes'**. By choosing different values for **X** and **Y** we can specify the positions of other points, such as **Q** and **R.** For the point **0** where the lines meet, both **X** and **Y** are zero. This point **0** is called the **'origin'** by mathematicians, and we have seen that it is related to the datum on the machine.

Draw the point **P** on your piece of paper at **X + 15**mm and **Y + 21**mm. Imagine that it is a place at which you wish to drill a hole in the paper.

Reset to datum. Then place the paper on the machine table with the origin **0** under the drill and **X** and **Y** pointing in the proper directions.

Position the drill at **P** by use of the **X** and **Y** axis control commands. Carefully count the individual steps applied to each axis.

point	co-ordinates X	Y	distance travelled since last point X	Y
datum	0	0		
P	15	21	+15	+21

FIGURE 4.7A

Complete a table as shown in **Figure 4.7**. The measured coordinates (in mms) should correspond with the number of keystrokes required to effect the move.

Remember keystrokes are sign conscious. If you used any **-** strokes they must be subtracted from the number of **+** strokes to give you the effective number of keystrokes used (there may be some small numerical difference because we have not been striving for accuracy in this exercise).

A typical drilling operation will need to drill several holes.

Move the machine a short distance away, using separate short strokes on the **+** and **-** keys, again counting the steps taken. Mark the position reached as **Q** on the paper and enter the co-ordinates of **Q** in the next line of your table. Enter also the number of steps, and the direction (**+ or -**), to get there. Your table might now look like this:

point	co-ordinates X	Y	distance travelled since last point X	Y
datum	0	0		
P	15	21	+15	+21
Q	23	18	+ 8	- 3

FIGURE 4.7B

Notice that the new co-ordinates do not correspond to the number of steps taken, but to the steps which would need to be taken if you reset the machine to datum first.

Verify this by resetting to datum and counting out the co-ordinate values on each axis while returning to **Q.**

Get similar results for a third point **R.**

4.7.07 Effect Of Changing The Work Requirement

Consider what would happen if, on the next workpiece, either point P or point Q were not required. Draw up a new table showing the different numbers of steps needed to get to the remaining two holes, keeping the co-ordinates the same.

It is partly because the number of steps changes like this that absolute co-ordinates are preferred to 'dead reckoning' (from one point to the next). We shall see in the next Exercise how co-ordinates are typically used.

4.7.08 Summary

In this exercise a computer program was read from a disc. This told the computer to behave like a simple machine-controller. You were then able to control the machine by pressing keys on the computer.

The idea of co-ordinates was introduced, and the relation between the axes of machine motion and axes of a set of co-ordinates was demonstrated. A datum setting of the machine is a known starting position from which co-ordinates are measured. It was shown that by describing points in terms of coordinates measured from the fixed axes (corresponding to the datum setting), we avoid having to respecify them if the path to a point is changed.

4.8 — COMPUTER NUMERICAL CONTROL

4.8.01 Introduction

Execise 5.7 showed that a computer can control a machine. It did so in direct response to operator commands related to the axes **X, Y and Z.** In this exercise we shall see how we can, with the help of the computer:

A) use co-ordinates of points directly, letting the computer do the calculations of how many steps from one point to the next

B) store a sequence of commands

C) perform miscellaneous functions

D) execute a program as many times as required by using the stored program again

In Exercise 4.7 the computer was programmed to behave as a very simple controller. This time it must have a different program, to introduce the N (for numerical) of CNC.

4.8.02 Preparation

The vertical feed should be raised to the top of its travel using the hand-wheel. The clamps will be required to hold workpieces, so these should be fitted and moved to the extreme ends of their slot in the table.

Now if your computer is running with the CNC disk installed, use the keyboard to get back to the **MAIN MENU** - see chapter 3 if you have difficulty.

If your system is powered down, power up and **BOOT** the control program as Section 4.7.04.

We want to tell the computer how to make a sample part, in jargon we need to **ENTER** (type in) a **MANUSCRIPT** (this is a description of a machine activity, called a 'part program' or 'workpiece program', not the computer program which tells the computer how to behave like a CNC machine, that is called the **'control program'**.

A program is a series of **'blocks'** each showing a set of functions and/or co-ordinates. The series of blocks making up one part machining program is called a **MANUSCRIPT**. We shall write the contents of one block in a single line. A typical example is:

N	G	F	M	X	Y	Z	I	J
1	00	-	-	60	30	-	-	-
2	-	-	02	-	-	-	-	-

N is simply a serial number for the line.

FIGURE 4.8

Figure 4.9 shows the codes related to several of the letters appearing in **Figure 4.8.**

ENTERING A BLOCK

G00	Point to point positioning
G01	Linear Interpolations
G02	Clockwise Circular Interpolation
G03	Counterclockwise Circular Interpolation
G05	Hold
G12	Clockwise Hole Milling
G13	Counterclockwise Hole Milling
G25	Jump to Block Number
G53	Reset change of Reference Point
G58	Change Reference Point
G80	Cancel Fixed Drilling Cycle
G81	Fixed Drilling Cycle
M00	Program Stop
M02	End of Program - Return to Datum
M03	Spindle motor ON
M05	Spindle Motor OFF

F Feedrate may have the following values only :

1, 2, 3, 4 millimeters per second

The **X** and **Y** co-ordinates are specified with respect to the machine datum. **X** and **Y** coordinates may take values in the range 0 thru 99.

I and **J** Coordinates are specified with respect to the start of a Circular Arc. They may have values in the range -99 thru +99. **I** and **J** coordinates must always be explicitly stated.

A Block containing an explicit **Z** value must contain **"-"** as **X** and **Y** entries. No other **X** and **Y** entry is acceptable.

(These are the codes recognized by CNC932, see page 4.30 for a more complete listing of standard codes)

FIGURE 4.9 — CODES VALID FOR CNC932

We will use only some of them for the moment. **G00**, i.e. the value **00** set into the **G** column means "positioning". **M02** means the "end of program".

Notice that the computer fills in **"-"** in any blank spaces.

4.8.03 G00 Point - to - Point positioning

G00 is never used to machine material. It moves the machine to the values of **X** and **Y** as quickly as possible to take up reference positions such as the beginning of a part program or returning to **HOME**. It therefore does not need data about feedrate and other things.

The Block with sequence number **(N)1** says:
G00 position (as fast as possible) to the values given namely
X60 the X - coordinate is to be 60
Y30 the Y - coordinate is to be 30

When entering a block, the computer will give you a chance to correct any mistakes. Answer the questions carefully.

4.8.04 M02 End of Program Return to Datum

Setting **M02** signifies the end of the program. All programs must end with a block containing the single value **M02**. Entering the block containing only **"M02"** is the only program way of leaving **EXECUTE** mode. It also returns the machine table to the datum **(X = 0, Y = 0)**, it also raises the tool to Z=0 and turns off the spindle motor.

4.8.05 Edit

You can check or alter your program by entering **EDIT** mode from the **MAIN MENU "4"**. You can check the **EXECUTION** of your program by typing **"8" EXECUTE** from the **MAIN MENU**.

4.8.06 Example

Boot up the CNC program. The machine will set itself to datum. Select **'ENTER'** and type in Fig. 4.8. Return to **MAIN MENU** and select **'EXECUTION'**. The machine will execute **block I**, that is, position itself to the position **X60, Y30.** It will then, immediately perform the next block of the program; since this is **'end of program'** it will come straight back to the datum again. Note the path in returning to datum; for as long as possible the machine travels in both axes. When no further travel is needed in one axis, the travel necessary for the other axis is completed.

4.8.07 G01 Linear Interpolation; M03 Spindle ON/M05 Spindle OFF

You may have noticed that the **G** function **00** does not generally produce motion in a single straight line.

When the machine 'interpolates' it means that we have control of the path. Linear interpolation means that the path will be a straight line directly from one point to the next.

Suppose we wanted to produce the item in **Fig. 4.10**. The machine uses absolute programming, i.e. all measurements are referred to the datum. Let us redraw the item to show the measurements used for programming as in **Fig 4.11.** This will greatly simplify writing the program.

Figure 4.12 is a listing of a suitable program.

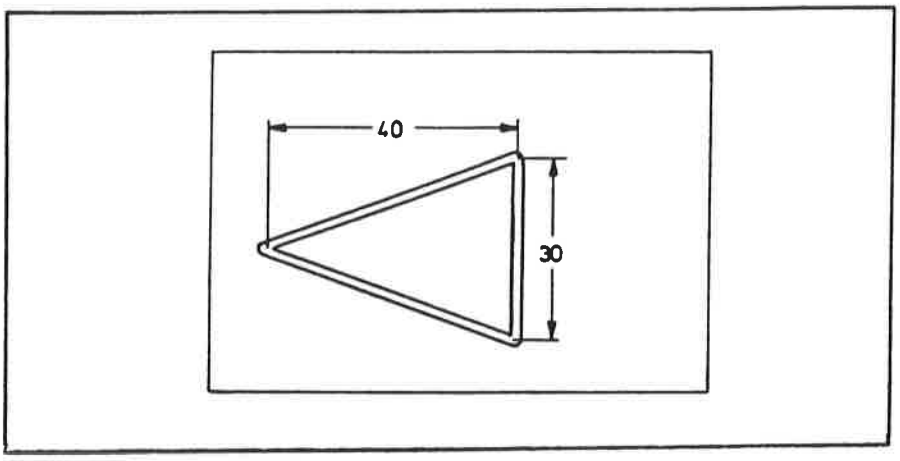


FIGURE 4.10

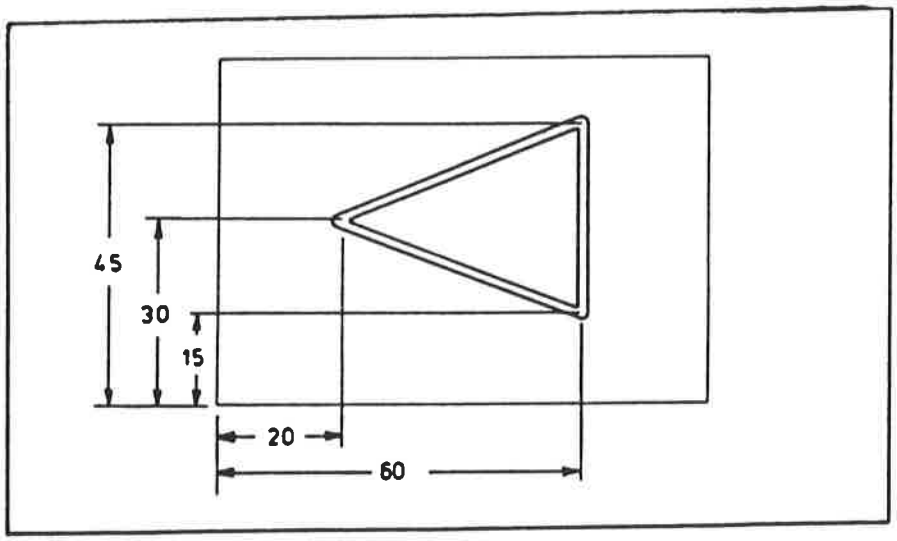


FIGURE 4.11

N	G	F	M	X	Y	Z	I	J
1	00	-	-	20	30	-	-	-
2	01	2	03	-	-	-6	-	-
3	-	4	-	60	45	-	-	-
4	-	-	-	-	15	-	-	-
5	-	-	-	20	30	-	-	-
6	-	-	02	-	-	-	-	-

FIGURE 4.12

Let us consider what each line does.

Line 1 is simple positioning as in the previous example, the machine goes to **X20** and **Y30**.

Line 2 uses the **G01** function, linear interpolation. Since **Z** is the only co-ordinate which has changed, from **0** to **-6**, the cutter moves downwards, the negative direction. (If **6** had been entered it would have tried to move upwards). The downward feedrate is set by **F=2**, so the cutter descends at 2mm/second into the material to start the cut. At the same time setting **M=03** switches on the spindle motor (in the cutter head); a complementary function **M=05** will stop the spindle, but is often not needed because the required program ending function **M02** includes the **M05** function.

Line 3 is again linear interpolation., The cutter height **Z** is unchanged, so the cutter moves at constant height relative to the material to position **X60**, **Y45**, cutting a groove. (The material actually moves past the cutter, but it is easier to think of motion relative to the workpiece).

Lines 4 and 5 are similar, **line 4** doing linear interpolation to **X60, Y15** and **line 5** from there to **X20, Y30**.

Line 6 signifies end of program, on which the cutter is always raised to **Z=0** and stopped, and the table is then sent back to the datum.

4.8.08 Reference Points

It is worth mentioning that the **X-Y** reference is set by two detectors which give a reference signal to the computer. The reference point for the **Z** axis is where the cutter was set at the start of the run, so that in this example if the cutter had been 5mm above a flat workpiece before starting it would have made a 1mm cut.

Enter the program and check it carefully. Use **ENTER** and **EDIT** from the Main Menu.

The **'touch-off function'** at the start of execution allows the operator to position the machine on all axes, as required, prior to cutting material. **'Executing'** a program means getting the machine to execute all the commands in it. Mistakes are easy to make, so it is always a good idea to see what the machine does before running with a workpiece present. So see that the table is clear and the cutter head at its highest point (which should leave much more than 5mm clearance from the table). When the machine runs, if it looks like it is doing something dreadful, use the emergency stop Key **"F9"**. The emergency stop procedure will permit you to manually move out of the "dreadful" case.

This procedure should always be followed with each of your programs to save possible damage to the machine, cutter workpiece or even you!

After any errors have been corrected, (Use the **Edit** functions to accomplish the corrections) mount the workpiece on the table. Set the switch of the spindle motor to **'on'**. Then **Execute** the program to cut the workpiece.

4.8.10 Save

If all went well you may like to save a copy of the program on disc. You should have been told which disc to use. Make sure you use the correct one, or you may be in trouble. If you have to change discs, remember the rules for disc care and store the removed disc in its protective cover.

When your workpiece-program disc is in the drive and the door closed, select from the main menu. When a program is **SAVEd** it must have a title, and you must be careful not to use an existing title. (If you do you will destroy the existing program file).

You may enter a title of up to eight characters, see Chapter 3 page 3.26 - for details of title format. You could use your intitals, with a serial number perhaps. Make a careful note of the title so that you can get the program back later. You will need to spell it exactly the same way then.

4.8.11 List

If your system is provided with a powered up and properly configured printer you should be able to print the current program (i.e. the most recently **ENTERed**, **EDITed**, or **LOADed** one) by selecting **"LIST"** from the main menu. The program can alternatively be displayed on the CRT screen. **8** lines at a time are displayed on the screen. For larger programs that will not fit on a single screen an overlap of two program lines is presented on each screen.

4.8.12 Load

This function reads from a disc a workpiece program which has previously been recorded on it. Before loading therefore, the correct disc must be in the drive with the door closed. Select **LOAD** to copy a program from disks to the computer. Prior to the loading you may actually display the disk's directory.

To check that your program is now loaded, you can use **LIST**.

4.8.13 G 02 Clockwise Circular Interpolation

One of the powerful features of CNC machines is their ability to automatically describe circles and circular arcs. Suppose we modify the design of the previous workpiece to that of **Figure 4.13**.

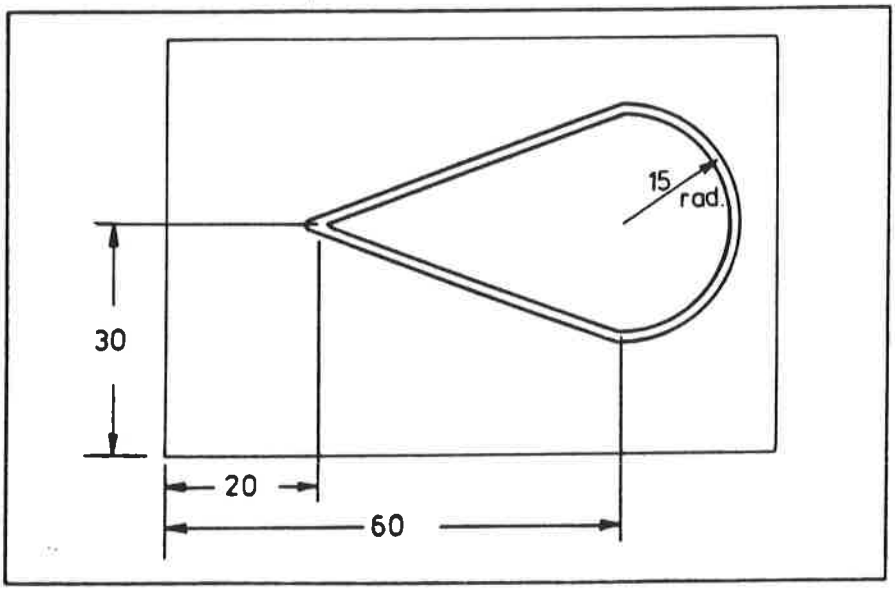


FIGURE 4.13

Remember that in the program all **X,Y** & **Z** dimensions are specified in relation to the reference point. A program to produce this item might look like **Figure 4.14**.

N	G	F	M	X	Y	Z	I	J
1	00	-	-	20	30	-	-	-
2	01	2	03	-	-	-6	-	-
3	-	4	-	60	45	-	-	-
4	02	-	-	75	30	-	0	-15
5	-	-	-	60	15	-	-15	0
6	01	-	-	20	30	-	-	-
7	-	-	02	-	-	-	-	-

FIGURE 4.14

Lines 1 to **3** are the same as in the previous example.

Line 4 calls **G02**, clockwise circular interpolation. The **G02** and **G03** functions both require values **I** and **J** to be specified. These specify the position of the center of the circular arc relative to the cutter position at the start of the cut (and are not absolute co-ordinates like **X, Y** and **Z** but are rather relative coordinates). Thus **line 3** brought the cutter to **X60, Y45**. From here the center of the arc is at the same **X** value, so in **line 4** the difference **I** is **O**. The center of the area is at **Y = 45-15,** so **J** is **-15**.

Line 5 is like **line 4,** except that the cutter is now at the point **P**, from which the distance to the circular center is **I = -15** for the **X**-component and **J = O** for the **Y**-component.

Lines 6 and **7** are the same as the last two lines of the previous example.

Enter the program of **Figure 4.14**. Test run it without a workpiece as before. Then **EXECUTE** it after setting the cutter height to 5mm above the workpiece and checking that the spindle motor switch is on (switch side with **"I"** depressed).

4.8.14 G03 Anti-Clockwise Circular Interpolation

This is similar to **G02**, but the path bends the other way. Suppose we wish to reverse the direction of the cutter path in the previous example. **Figure 4.15** shows a program.

N	G	F	M	X	Y	Z	I	J
1	00	-	-	20	30	-	-	-
2	01	2	03	-	-	-6	-	-
3	-	4	-	60	15	-	-	-
4	03	-	-	75	30	-	0	15
5	-	-	-	60	45	-	-15	0
6	01	-	-	20	30	-	-	-
7	-	-	02	-	-	-	-	-

FIG 4.15

FIGURE 4.15

The first three lines are straightforward by now.

Line 4 uses the **G03** function. Notice how the value of **J** is now positive, because the center of the arc is now in the positive **Y** direction from the current position (set by **line 3**).

Line 5 is the second 90 degree arc. In this case **"I"** is negative, because, just as in the previous example, the center of the arc is to the left (**-X** direction) of the cutter position.

4.8.15 Non-quadrant arcs

In the last two examples each circular arc began and ended with either **I** or **J** zero. When this does not occur, i.e. when the arc radius at the beginning or ending is not parallel to the **X** or **Y** directions, a little simple calculation is needed to work out the values.

Suppose for instance that the cutter were to move clockwise around the path shown in Figure 4.16, having an arc **AB** with center **C** and radius **R**. For this arc the values of I and **J** are **0** and **R** respectively. The problem is to specify the co-ordinates of **B**, let us call them **XB, YB. Figure 4.16** shows this region (not to scale) in more detail.

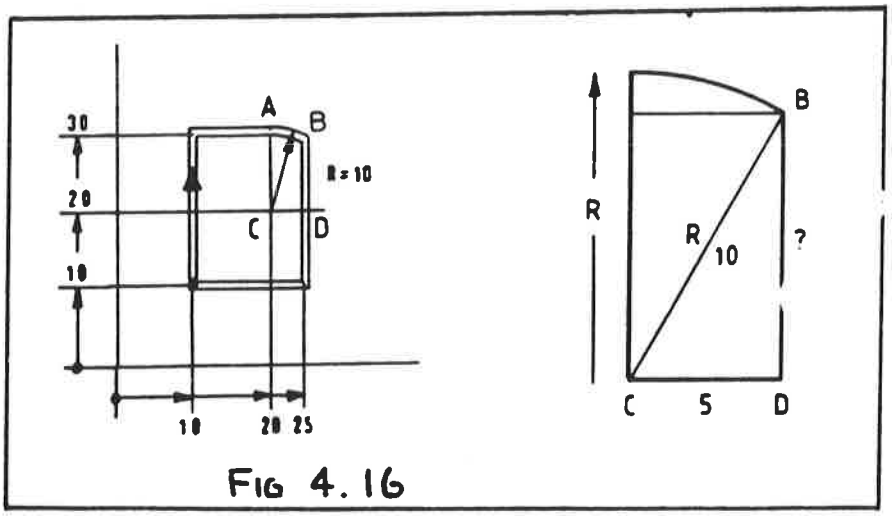


FIGURE 4.16

We can use the right-angled triangle BDC. Its hypotenuse is the radius R. CD is 5mm. Calculation of the remaining side BD is straightforward in this case, using Pythagoras' Theorem.

From Pythagoras, $CB^2 = CD^2 + DB^2$

$\therefore DB^2 = CB^2 - CD^2$

$\therefore DB = \sqrt{(CB^2 - CD^2)}$

Inserting millimeter dimensions,

$DB = \sqrt{(10^2 - 5^2)}$

$= \sqrt{(100 - 25)}$

$= \sqrt{(75)}$

$= 8.66$ approximately.

The value of **Y** for point **B** is this amount greater than that for point **C**, i.e it is:

$Y = 20 + 8.66$

$= 28.7$

After rounding to the nearest figure recognized by the machine (nearest 0.1mm). In other cases trigonometry may be required for calculations. Even in this simple case those practiced in the use of some modern calculators could use the simple relationships shown in **Figure 4.17** to do the calculation more quickly.

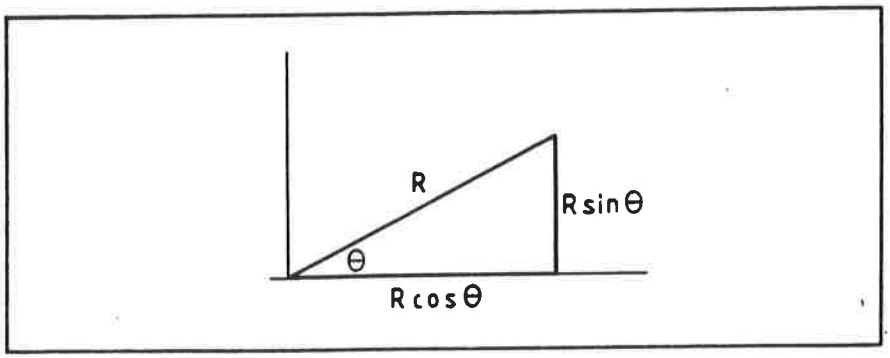


FIGURE 4.17

N	G	F	M	X	Y	Z	I	J
1	00	-	-	10	10	-	-	-
2	01	2	03	-	-	-6	-	-
3	-	4	-	-	30	-	-	-
4	-	-	-	20	-	-	-	-
5	02	-	-	25	28.7	-	0	-10
6	01	-	-	-	10	-	-	-
7	-	-	-	10	-	-	-	-
8	-	-	02	-	-	-	-	-

FIGURE 4.18

However, in this simple case, not even the simplest trigonometry is needed, since DB can be calculated by Pythagoras' rule.

By now you know how to test run a program and then use it. It will be assumed that your instructor has given guidelines on how many workpieces you are actually to cut.

4.8.16 G12 Hole Milling Clockwise

Hole milling is a function which allows a cutter to produce a hole larger than its own diameter. We shall use the example of using a 2mm cutter to produce a 10mm diameter hole. This is done by cutting along a circular path. The center of the circle is the point at which the previous line left the cutter, and it returns to the same point, **Figure 4.19**.

Figure 4.20 shows a program which will mill two 5mm holes at **X30**, **Y30** and **X50**, **Y50**.

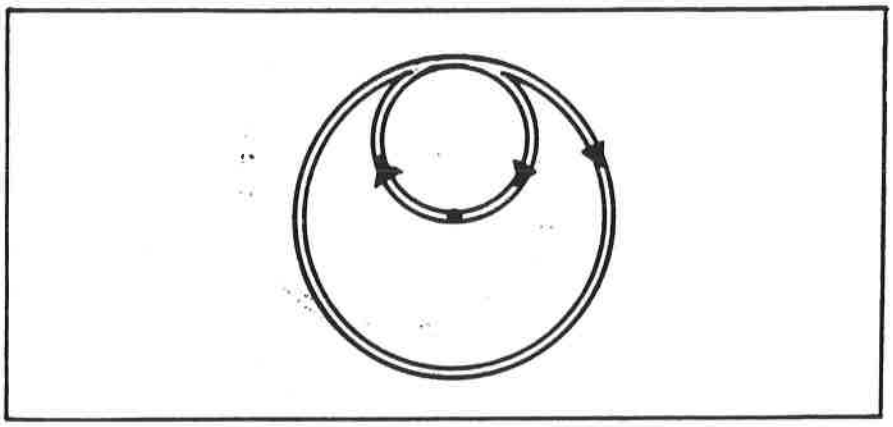


FIGURE 4.19

N	G	F	M	X	Y	Z	I	J
1	00	-	-	30	30	-	-	-
2	01	2	03	-	-	-6	-	-
3	12	-	-	-	-	-	5	2
4	01	-	-	-	-	0	-	-
5	00	-	-	50	50	-	0	-
6	01	-	-	-	-	-6	-	-
7	12	-	-	-	-	-	5	2
8	-	-	02	-	-	-	-	-

FIGURE 4.20

The only lines of a new type are **lines 3** and **7.** They use only the **I** and **J** co-ordinates. **I** is the **radius of the hole** and **J** is the **diameter of the cutter**; don't mix them up or the program will not run.

4.8.17 G13 Anti-Clockwise Hole Milling

This is just the same as **G12** except the cutter is moved in the opposite direction around the path. The availability of **G12** and **G13** gives the choice of up-milling or down-milling.

4.8.18 Canned Cycles

G12 and **G13** are examples of **'canned cycles'**. A canned cycle is a sequence of operations which could be programmed in terms of the elementary functions on several lines, but which has its own name **(G12 or G13)** by which it can be programmed in a single program line.

Using them has two advantages: it shortens programs and, because the detail of the function has been worked out and tested previously there is less chance of making mistakes. The path shown in **Figure 4.19** would require eight lines, not one, if the circular arc function **G02** or **G03** were used.

4.8.19 G25 Jump to Block Number (LOOP)

If there is some task in your program that must be executed several times, (usually at different locations) it is possible to go back to an earlier block in the program and repeat blocks using the **G25** function. **G25** obviously needs to be told where to go back to. The value of **J** supplies this. It also needs some further information to prevent it looping back every time the **G25** is reached for ever. **I** specifies the number of times the program is to jump back.

The most common use of **G25** is in conjunction with **G58**, so we will look at that before giving an example.

4.8.20 G58 Change of Reference Point

If the reference point were changed before executing a program, the shape produced by it would appear in a different place. Suppose that the shape in **Figure 4.11** (the second program) were to be milled several times in different parts of a workpiece. It would be possible to write out several copies of the program with the **X** and **Y** values altered accordingly, but a simpler way is provided by using just one copy several times and changing the reference point instead.

G58 shifts the reference point in **X,Y,** and **Z** by specified amounts. (A specification is required for each of them; **"-"** will not do).

To demonstrate its use let us cut two triangles like **Figure 4.11** with the program of **Figure 4.21**.

N	G	F	M	X	Y	Z	I	J
1	00	-	-	20	30	-	-	-
2	01	2	03	-	-	-6	-	-
3	-	4	-	60	45	-	-	-
4	-	-	-	-	15	-	-	-
5	-	-	-	20	30	-	-	-
6	-	-	-	-	-	0	-	-
7	58	-	-	0	40	0	-	-
8	25	-	-	-	-	-	1	1
9	-	-	02	-	-	-	-	-

FIGURE 4.21

The first five lines are like the original triangle program. (If you **SAVEd** that program earlier you could save typing by **LOADing** it now).

Line 6 has had the **M02** instruction removed and **Z0** added to take the cutter out of the workpiece.

Line 7 changes the reference point by 40mm in the **Y** direction. Notice that all three of **X**, **Y** and **Z** are specified by numbers, which can be zero or negative but not **"-"**.

Line 8 causes the program to loop back to **line 1** (the **J** value). It does this only once (specified by the **I** value).

Line 9 which ends the program resets the machine to its fixed reference point or datum.

A common use of **G58** and **G25** is to take several passes at a cut with increasing depth, by changing the reference in **Z** and looping back.

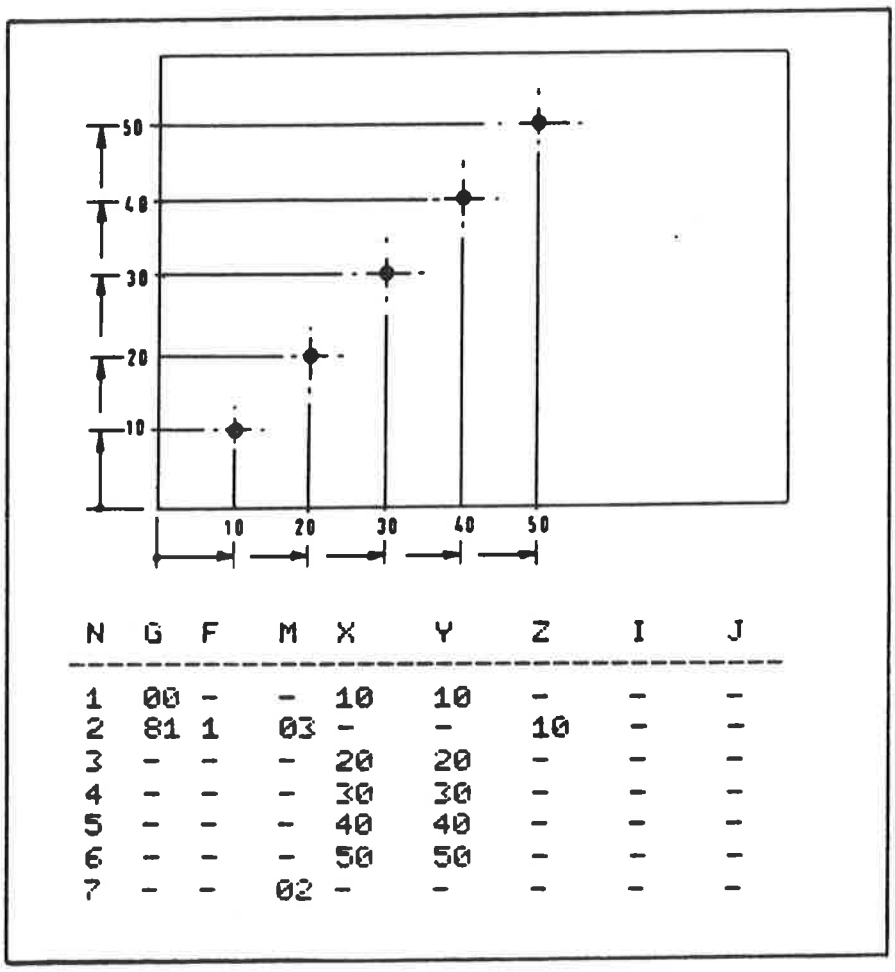
4.8.21 G53 Cancel a Change of Reference Point

This function may only be used after **G58** has been used once, and cancels the effect of the **G58**. If **G58** has been used several times, the reference should be reset by another **G58** with **X, Y, Z** values the negative of the total previous change.

4.8.22 G81 Fixed Drilling Cycle

This is another canned function. It makes jobs requiring a lot of holes very easy. Like the previous canned function it returns the cutter to the point where it was before entering the function.

Figure 4.22 is a simple example of its use.



N	G	F	M	X	Y	Z	I	J
1	00	-	-	10	10	-	-	-
2	81	1	03	-	-	10	-	-
3	-	-	-	20	20	-	-	-
4	-	-	-	30	30	-	-	-
5	-	-	-	40	40	-	-	-
6	-	-	-	50	50	-	-	-
7	-	-	02	-	-	-	-	-

FIGURE 4.22

Line 1 moves the cutter to **X10, Y10**

Line 2 uses the **G81** function which drills a hole whose depth is set by a 10mm travel of the cutter, first in the negative direction and then in the positive. Notice that it is not necessary to specify the **Z** value as negative, since the drilling action is always downward, then back up.

4.8.23 G80 Cancel Fixed Drill Cycle

Suppose that after the fixed drill cycles in the previous example, instead of returning directly to the reference point, we wish to perform other operations. We must than first cancel the fixed drill cycle by using function **G80**. After that ordinary programming can continue.

If we wanted for example a line joining the first and last holes, the last few lines of the program would become as **Figure 4.23**.

N	G	F	M	X	Y	Z	I	J
7	80	-	-	-	-	-	-	-
8	01	-	-	-	-	-8	-	-
9	-	-	-	10	10	-	-	-
10	-	-	02	-	-	-	-	-

FIGURE 4.23

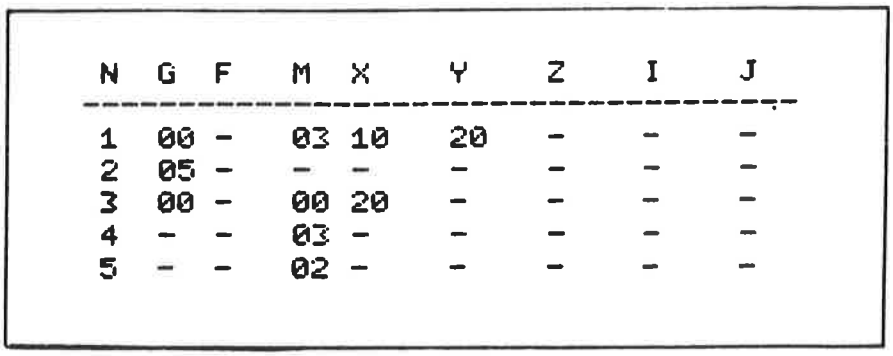
4.8.24 G05 Hold

This function causes the machine to wait (in the state it reached at the previous block) until the operator presses the **G** key. It then executes the block following the **G05** block. During the wait period no feed motion takes place, but the spindle will go on rotating if it was rotating before (and in a real machine coolant and other functions would be unaffected).

4.8.25 M00 Program Stop

This function causes the program and the machine itself to stop before execution of the subsequent block. This may be required to enable a tool change or for any other reason. Execution of the next block of the program continues after the operator presses the **G** key. The **M00** must be entered in a separate block by itself.

The difference between **G05** and **M00** is demonstrated in the very simple program of **Figure 4.24**. The machine will wait after executing **block 1** with the spindle running, and at the end of **block 3** will wait again, but this time with the spindle stopped.



N	G	F	M	X	Y	Z	I	J
1	00	-	03	10	20	-	-	-
2	05	-	-	-	-	-	-	-
3	00	-	00	20	-	-	-	-
4	-	-	03	-	-	-	-	-
5	-	-	02	-	-	-	-	-

FIGURE 4.24

It is good practice (especially in systems using stepping motors or other devices which rely on counting rather than absolute measurement) to end the program with a programmed return to **X=0, Y=0, Z=0**, followed by **G05**, before the final **M02**. Then if there is any doubt about the accuracy with which the machine has performed, a check (perhaps with dial gauges) can be performed before and after the reset to datum caused by the end of program **M02**. Also stepper motor machines can keep a running total of all program steps (the sum of positive and negative steps). Thus when it returns to DATUM all registers should show a sum of zero steps - obviously the positive steps must equal the negative steps for a full cycle. If any register shows a value other than zero this indicates a problem of some kind.

This could be:

A) Excessive wear in the machine allowing backlash.

B) Losening of drive shafts or gearing allowing backlash

C) Overloading of axis drives causing them to fall out of step

D) Interference with datum detectors by swarf build up

4.9 — OTHER STANDARD FUNCTIONS
(EXAMPLES ONLY)

The full list of standard functions should be looked up in a standards document, (e.g BS 3635 Part 1 in UK, or equivalent EIA, DIN and ISOR/-series standards). However a few examples are given below to give an idea of some of the functions covered. Note that in some cases the same function code, say **G99**, is shown with two definitions. You must refer to the programming manual for a specific machine to determine which definition applies and it specifics.

MISCELLANEOUS FUNCTIONS

These signals are used for On/Off control of a machine function. One **M** code can be specified in a block. If two or more **M** codes are specified, only the last one is effective.

The below information is furnished as a brief illustration of the implementations of **M** codes on industrial CNC machines. While EIA, ISO and AIAA standards exist (see appendix), always consult the programming and Operator Manual of a particular machine tool /control combination for exact specific details.

CODE	TITLE	DESCRIPTION
M00	Program Stop	Cycle operation is stopped. When stopped, all existing modal information remains unchanged. (Details differ among tool builders). Operator must take an action to continue program execution.
M01	Optional Stop	Operation is stopped after a block containing M01 is read. The code is effective only when the optional stop switch has been set.
M02	End of Program	Must be in the last block of a program so as to indicate end of program. May include rewind of tape.
M03	Spindle on Clockwise	Start spindle rotation in a clockwise direction as seen looking out form the spindle face.
M04	Spindle on Counter-clockwise	Start spindle rotation in a counterclockwise direction as seen looking out from the spindle face.
M05	Stop Spindle	Stop all spindle rotation
M06	Tool Change	if a sequential tool changer is used. Must also be used on machines where tools are designated by numeric position location.
M07	Coolant No2 **On** (usually flood)	usually flood

(CONTINUED)

M08	Coolant No1 **On**	Usually mist (On single coolant form machines used for all coolant control).
M09	Coolant **Off**	Turn off all coolant
M10	Turn **On** Automatic clamps	
M11	Turn **Off** Automatic clamps	
M12	Half Index Turret	Used to provide clearance on two turret machines so indexing tools may move between each other.
M13	Spindle on Clockwise (see M03) and coolant **On**	
M14	Spindle on Counter-clockwise (see M04) and coolant **On**	
M19	Oriented Spindle Stop	Stop spindle at a predetermined angular postion.
M21	Mirror Image **X**	
M22	Mirror Image **Y**	
M23	Mirror Image **Off**	
M25	**Z** Axis Home	Move **Z** axis spindle to home at rapid rate.
M30	Program Stop & Rewind	Stop all spindle rotation, tool motion and coolant flow. Unlock clamps. Reset all machine functions to default state. Rewind program. Otherwise similar to **M02.**
M40	Select Low Gear Range	
M41	Select High Gear Range	

(CONTINUED)

M47	Continue Program Execution	from start unless inhibited by interlock
M48	Cancel **M49** Override	
M49	Deactivate Manual Spindle and/or feed	override and return to programmed rate
M58	Cancel **M59**	
M59	Hold Constant RPM	
M98	Go to Subroutine	from main program
M99	End of Subprogram	Return to calling program unit.

CODE	TITLE	DESCRIPTION
G00	Rapid Transverse Positioning	move to reference positions without machining.
G01	Linear Tool Motion	at a specified feed rate
G02	Circular Tool Motion	in a clockwise direction
G03	Circular tool motion	in a counterclockwise direction
G04	A Temporary Dwell or Delay in the Tool Motion.	It is cancelled by the operator.
G05	A Permanent Hold or Delay in the Tool Motion	It is cancelled by the operator.
G06	Parabolic Interpolation	
G09	Causes an Exact Stop	of one tool motion before going to the next tool.
G10	Offset	set by program
G13	Select X Axis	
G14	Select Y Axis	
G15	Select Z Axis	
G16	Cancel Axis Selection	
G17	Selection of the X & Y Axes	plane for cutting
G18	Selection of the X & Z Axes	plane for cutting
G19	Selection of the Y & Z Axes	plane for cutting
G20	Inch Data Input	
G21	Metric Data Input	
G22	Activate Stored Axes Travel Limits.	
G23	Deactivate Stored Axis Travel Limits.	
G27	Return to Machine Home	via programmed travel on each axis
G28	Return to Machine Home	via programmed intermediate point
G29	Return to Workpiece	from machine home via programmed intermediate point.
G30	Return to Reference **G29**	
G33	Constant Lead Threading	with possible multiple pass
G34	Increasing Lead	thread cutting

G35	Decreasing Lead	thread cutting
G40	Cancel Radius compensation	radius compensation
G41	Cutter Radius	compensation **LEFT** of workpiece
G42	Cutter Radius	compensation **RIGHT** of workpiece
G43	Activate Tool Length	compensation in **SAME** direction as stored offset.
G44	Activate Tool Length	compensation in **OPPOSITE** direction as stored offset.
G46	Tool Offset	increase
G47	Tool Offset	double increase
G49	Tool Offset	double decrease
G49	Tool Length	compensation cancel
G50	Establish Floating Zero	
G53	All Positioning Data are with Reference to Machine Home.	
G54	Position Data Reference	floating zero **1st** fixture
G55	Position Data Reference	floating zero **2nd** fixture
G56	Position Data Reference	floating zero **3rd** fixture
G57	Position Data Reference	floating zero **4th** fixture
G59	Reposition Floating Zero	to new location
G60	Exact Stop of one Tool Motion	before going to next tool.
G63	Cancel Feedrate Override	
G64	Cancel **G60**	
G65	User Macro	
G66	Cancel User Macro	
G70	Inch Input Data	
G70	Finish Turning Cycle	
G71	Metric Data Input	
G72	Multiple Pass Facing canned Cycle.	
G72	Activate Third Axis	
G73	Multiple-pass Pattern Repeat Canned Cycle.	
G73	Peck Drill Cycle	

G73	Third Axis Circular Interpolation.	clockwise
G74	Peck in **Z** Axis Canned Cycle.	
G74	Tapping	
G75	Peck Cycle In **X** Axis	
G76	Multiple Pass Single Point Threading Canned Cycle	
G76	Fine Bore	
G80	Cancel Canned Cycle	
G81	Drilling Canned Cycle	feed in and rapid out
G82	Drill with Dwell	
G83	Peck Drill	multiple feed pecks & rapids ou:
G84	Basic Tapping Cycle	feed in, reverse feed out
G85	Canned Boring Cycle	feed in, feed out
G86	Alternate Boring Cycle	feed in, stop spindle & rapid out
G86	Alternate Boring Cycle	feed in, stop spindle, manual retract
G88	Alternate Boring Cycle	feed in, dwell, stop spindle, manual retract
G89	Alternate boring Cycle	feed in, dwell & feed out
G90	Single Pass Turning	
G91	Incremental Positioning	
G92	Establish Floating Zero	
G94	Inch Per Minute Programming	
G94	Single Pass Facing	
G95	Inch Per Revolution Programming	
G96	Constant Surface Speed Programming	
G97	Revolutions Per Minute Programming	
G98	Inch Per Minute Programming	
G99	Inch Per Revolution Programming	
G99	Position Data with 99	refers to machine home

CHAPTER 5 - STUDENT EXERCISES

5.1 — EXPLANATION OF USER DISK DIRECTORY

This directory can be obtained as a by product of the **SAVE** or **LOAD** options from **MAIN MENU** or by use of the directory call from **DOS**.

5.1.01 **COMMAND.COM** - resident **MS-DOS** commands - transferred to **USER DISK** from computer system disk when you make the **USER DISK**.

5.1.02 **CNC.BAS** - transferred to **USER DISK** (after this disk has been impressed with **BASICA)** from the **FEEDBACK MASTER DISK** supplied with the CNC932. This is the actual CNC control program.

5.1.03 Files **FIG2$18.CNC** through **FIG2$17** (10 files, not in numerical order) are illustrations used in the text of this manual.

5.1.04 Files **PROB1.CNC** through **PROB7.CNC** are answers to the Problems 1 through 7 on the following pages of this section. These files can be **LOADed** from disk and **EXECUTED** to produce the parts described in problems **1** through **7**.

5.1.05 File **ROBIN.CNC** is a canned engraving program. It engraves **ROBIN** into a suitable block of wax. It can be LOADed and EXECUTED in the normal way. It requires a wax or toysteel block 4 x 4.

5.1.06 **BASICA.EXE** is the **BASICA** language transferred to **USER DISK** from a **BASIC** disk when the **USER DISK** is created. **BASIC** must be on the user disk before it can interpret the CNC control program.

5.1.07 **MAKE.BAT** this is a batch file program used in creating the **USER DISK**. It is not part of the CNC control program.

5.1.08 **AUTOEXEC.BAT** this is the program that automatically boots **BASICA** and the CNC control program for you. It is not part of the operational CNC control program.

5.2 — "CANNED" MACHINING PROBLEMS

Seven typical machining problems are given on following pages:

Students should use a programming sheet such as **Figure 5.1** to derive a manuscript to machine the parts described in the problems.

They should **ENTER** the manuscripts they have developed into the computer.

They should use the **EDIT** facility to make any corrections.

They should **LIST** the manuscripts to a printer to provide a hardcopy record.

They should **SAVE** the manuscripts to disk under recorded filenames.

They should **LOAD** the manuscripts back to the computer.

They should **EXECUTE** the programs to machine the parts as described.

If they run into trouble typical manuscripts are provided on the **USER DISK** under filenames **PROB1.CNC** through **PROB7.CNC.**

Problem 7

Write a program to produce the spiral below, use 90-degree arcs to simplify the construction. Start with a 4mm radius and increase the radius in 2mm steps every 90 degrees. Start at X46, Y50.