

TABLE OF CONTENTS

Chapter 1	STANDARD EQUIPMENT	1-1
Chapter 2	MICROMILL SPECIFICATIONS	2-1
	Cabinet	
	Mechanical	
	Electrical	
	Extra Equipment	
	Control Software Features	
Chapter 3	SAFETY FEATURES	3-1
	Safety Precautions	
Chapter 4	MACHINE CONNECTIONS	4-1
Chapter 5	INSTALLATION & EXECUTION OF THE CONTROL SOFTWARE	5-1
	Installing the Software	
	Running the Software	
Chapter 6	HOMING THE MACHINE	6-1
Chapter 7	LOADING STOCK	7-1
Chapter 8	REMOVING AND MOUNTING TOOLING	8-1
	Removing a Collet and/or Cutter	
	Mounting a Collet and/or Cutter	
	Removing Quick Change Tooling	
	Mounting Quick Change Tooling	
Chapter 9	TOOL OFFSETS	9-1
Chapter 10	MICROMILL OPERATING PANEL	10-1
Chapter 11	USING THE QWERTY KEYBOARD	11-1
	Keys	
	Operation	
	Selection	
	Editing	
	Operation Select	
	Execution	
	Axis Direction	
	Editing a CNC Program	

Chapter 12	CONTROL SOFTWARE - MAIN MENU	12-1
	Edit Only	
	Edit and Simulate	
	Simulate Only	
	Link to Controller	
	CNC Files	
	Print	
	Remote Link	
	Settings	
	Utilities	
	Quit	
Chapter 13	CONTROL SOFTWARE - SIMULATION MENU	13-1
	Check Syntax	
	Run Program	
	Dry Run	
	Set Datum	
	Set View	
	3D View	
	Postprocess	
Chapter 14	CONTROL SOFTWARE - MANUAL DATA INPUT	14-1
	Entering Data into the Editor	
	Directives	
	Inserting Directives into the Editor	
	G codes, M codes, and Coordinates	
Chapter 15	MANUFACTURING PARTS ON THE MICROMILL	15-1
	Cutting a Part	
	From Start to Finish (chart)	
Chapter 16	BASIC MOVEMENTS	16-1
	Basic Moves	
	Absolute Programming	
	Incremental Programming	
	Cutting Motions	
	G00, G01, G02, G03	
	Axis Definition	
Chapter 17	CUTTER RADIUS COMPENSATION	17-1
Chapter 18	TUTORIALS AND COMMENTS	18-1

Chapter 19	MACHINE MAINTENANCE	19-1
	Planning	
	Machine Lubrication	
	Adjusting the Belt Tension	
APPENDIX 1	CUSTOMIZING SETTINGS FILE	A1-1
	Fanucsmc.opt	
	Fanucsmc.go	
APPENDIX 2	GLOSSARY OF TERMS	A2-1
APPENDIX 3	LIST OF ABBREVIATIONS	A3-1
APPENDIX 4	ELECTRICAL CONTROL PANEL DIAGRAM	A4-1
APPENDIX 5	MICROMILL NOMENCLATURE	A5-1
APPENDIX 6	SUMMARY OF TYPICAL ADDRESS CHARACTERS	A6-1
APPENDIX 7	G AND M CODE LISTING	A7-1
APPENDIX 8	FAST KEYS / HELP SCREENS	A8-1
APPENDIX 9	SAMPLE PROGRAMS	A9-1
APPENDIX 10	TECH TIPS	A10-1
	INDEX	I1

1

STANDARD EQUIPMENT

Standard Equipment Supplied with the Micromill

- CNC Software for MS-DOS compatible computer.
- RS232 Machine Cable
- Instruction Manual
- Interlocking Machine Guard
- Automation Access Port
- Worklight
- 2 Machine Keys
- Spare Fuse
- Drawbar
- 1/4" collet
- 3/16" end mill
- 2 hex wrenches
- Spindle bar

This page is intentionally left blank.

2 MICROMILL SPECIFICATIONS

Cabinet

Length	27" (686mm)
Width	26.5" (648mm)
Height	27" (686mm)

Mechanical Specifications

Table to Spindle	8" (200mm)
Throat	2.25" (57mm)
Travel X axis	9" (229mm)
Travel Y axis	3" (76mm)
Travel Z axis	6.5" (165mm)
Hole through Spindle	0.405" (10mm)
Spindle Nose Thread	3/4 16 TPI
Spindle Taper	#1 Morse
Table Size	2.75" x 13.5" (70mm x 330mm)
Spindle Speed	0-3000 RPM

Electrical Specifications

Mains Supply	1 Phase 110/115 VAC
Spindle Motor	1/2 HP 90 VDC
Axes Motors	Stepper Motors 200 steps/rev.

Extra Equipment

A full range of optional equipment and accessories are available including tooling, work holding, courseware, curriculum, and Mill CAM Designer CAM software.

Tooling and workholding equipment can be supplied to suit a customers requirements.

Denford reserves the right to change specifications without prior notice.

For further details, call Denford Machine Tools at 1-800-886-9750.

Control Software Features

1. Fanuc Industrial Format Programming through a standard QWERTY keyboard or the optional Desk-Top Tutor.
2. Simultaneous 3 axis contour control.
3. Full circular and linear interpolation.
4. Programming format choice available from 1 of 2 industrial keypads (option)
5. Imperial or metric programming.
6. Combined use of absolute and increment in the same block.
7. Internal / External Screwcutting (Simulation only)
8. Manual and programmable machine stops.
9. Floating datum facility.
10. Programmable dwell.
11. Canned turning cycles.
12. Roughing and finishing cycles.
13. Output to printer or punch tape.
14. Tool offsets for 16 tools.
15. Save tool offsets to file for recall.
16. Cutter radius compensation.
17. Dry run facility.
18. Single block and auto execution.
19. Datum shift.
20. Block skip function.
21. Auxiliary inputs and outputs.
22. Full G and M code listing with help instructions - context sensitive
23. Single step or continuous program execution.
24. Axes jog on all axes with variable feedrates.
25. Programmable feedrate 30.5"/min.
20"/min. on Z axis.
26. Comprehensive toolpath graphics including 3D simulation, toolpath plot, and machining process simulation with tool motion.
27. Zoomed and sectional view with rotation.
28. Tool animation.
29. Directory listing.
30. Program merge facility.

-
31. Full edit mode allowing alter, delete, and insert.
 32. Programs stored on floppy, hard, or network drive.
 33. Cycle start / Feed hold / Cycle stop.
 34. In excess of 6000 G codes in editor.
 35. Comprehensive error messages.
 36. Continuous and incremental jog modes.
 37. Feedrate override.
 38. Screen axis display gives absolute or distance to go values.
 39. Control can be integrated into Local Area Networks allowing access to shared / group data.
 40. Control text can be loaded into word processor for translation into different languages.
 41. D.N.C. facility for both drip feed or full program data transfer, no program size limitation.
 42. On board controller diagnostics.
 43. Over 50 industrial G and M codes available.

3

SAFETY FEATURES

- Ergonomically placed emergency stop button.
- Full program stop from keyboard.
- Toolpath graphics to verify programs prior to machining.
- Software axes limits.
- Illuminated key operated power switch.
- Illuminated spindle switch.
- Program termination if guard is opened during machining.

Operating Temperature: 32-115 °F

Relative Humidity: Less than 75%

Safety Precautions

The Micromill has safety devices fitted to protect the machine and operator from unexpected accidents.

1. Tidyness

Do not place objects on or around the machine so that it interferes with the operation of the machine.

2. Power Source

When finished with the machine for the day, be sure the power is off.

Be sure that the machine is switched off before any maintenance work is carried out.

3. Lubrication

Check the state of slideway and lead screws lubrication daily.

4. Recommendations on Safety of Operation

Always wear clothing that is suitable for operating the machine.

Hold the workpiece securely in fixture or vise. The machine will not operate if safety guard is open.

It is very dangerous to touch any rotating part on the machine. Be careful when adding or removing tooling.

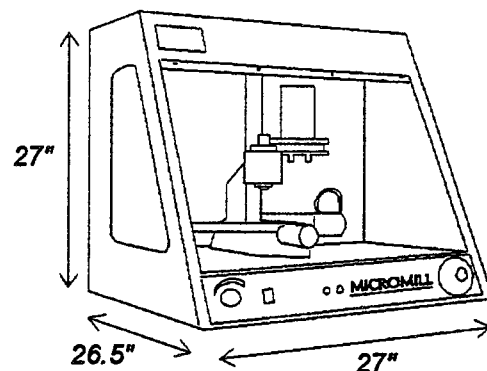
When an emergency stop is required, press the emergency stop button on the lower front of the machine. This will cut power to the spindle and axes. Reset the machine by first pressing the emergency stop button in while turning it to the right. This will make the button go in the release position (pop-out). Next, use the key to restore power to the machine.

NOTE: The Esc key will also stop the machine without locking up the machine.

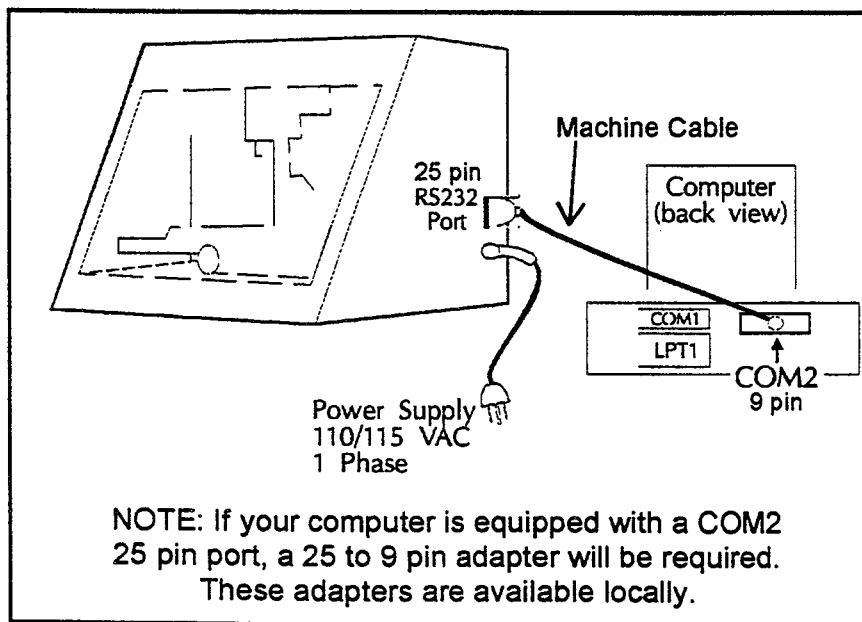
4 MACHINE CONNECTIONS

1. Unpack the Micromill and its accessories from the shipping container.

2. Place the Micromill on a suitable work surface. Before moving the Micromill, ensure that all cables have been disconnected or cleared. The machine should never be moved while connected to the electrical supply. It is important that the personal computer is located next to the machine so that the operator has access to the computer. The illustration to the right gives the dimensions of the machine.



3. Connect the RS232 machine cable to the 9 pin COM2 port on the computer and connect the other end of the cable to the RS232 25 pin port on the side of the Micromill as shown in the diagram below.



Note:
See page A1-2 to change COM port settings in the software if this is required for your specific configuration.

4. Connect the power cord and turn on the main power by using the main power key. Be sure the emergency stop button is in the release position. To release the emergency stop button, push in while turning the button to the right.

This page is intentionally left blank.

5

INSTALLATION & EXECUTION OF THE CONTROL SOFTWARE

The Micromill Control (Operating) Software is supplied on one 3.5" high density floppy disk. If you require a 5.25" disk, contact Denford Machine Tools at 1-800-886-9750.

Installing the Software

1. Go to the DOS prompt.
2. Place the Micromill Operating Disk into drive A
(If your 3.5" floppy drive is labeled as B, substitute B for A)
3. Change to the A drive by typing **A:** then press [Enter].
4. Type **INSTALL** and press [Enter].
5. Answer the questions on the screen. Make sure to load the program into the Denford directory. You will see the files on the screen as they are being loaded.
6. Press the N key to answer no to the "copy another disk" question.

IMPORTANT NOTE:

After installing the control software, the statement

SET DENFORD_DATA=C:\DENFORD\DATA must be in your AUTOEXEC.BAT file on the computer in order for the software to work properly. You must add the line shown above to the autoexec.bat file. When you do add this line, insert it somewhere in the middle of the file. DO NOT add it as the first or last line. Inserting this line enables the control software to save all CNC files to the Denford\Data directory.

It is recommended that you make a back-up copy of your autoexec.bat file on disk before making any alterations.

Consult your computer's operating system manual if you do not know how to edit the autoexec.bat file.

Running the Software

Before executing the control software, keep in mind the following:

1. It is necessary to have the machine turned on and connected to the computer whenever the software is being used. The machine is the security device for the software.
2. You must exit Windows before executing the software. If you use the DOS prompt icon in Windows to load DOS, communication errors may occur.
3. Make sure the Num Lock on the keyboard is OFF before using the software.
4. On some computers, users may experience a flashing screen. This can be resolved by making changes in Windows under Control Panel. Select the Keyboard icon and set the Delay Before First Repeat to *Short* and the Keyboard Repeat Rate to *Slow*.

Running the software from the hard drive

1. At the DOS prompt, type **CD\IDENFORD** and press [Enter]
2. Now type **FANUCSMD** and press [Enter]

Running the software from floppy disk

1. At the DOS prompt, type **A:** and press [Enter]
2. Now type **FANUCSMD** and press [Enter]

6 HOMING THE MACHINE

When you send a machine to its home or datum position, you are setting the X, Y, and Z zero positions for all three axes. Each zero position is the extreme limit each axis can move. *It is necessary to home (datum) the machine whenever it is turned on.*

1. Execute the Micromill Control Software. (see page 5-2)
2. Press the **Alt** + **H** keys to access the home screen. Make sure the emergency stop button is in the release position.
3. Press the **Z** key and the Z axis will go to its home position.
4. Press the **Y** key and the Y axis will go to its home position.
5. Press the **X** key and the X axis will go to its home position.

HINT:

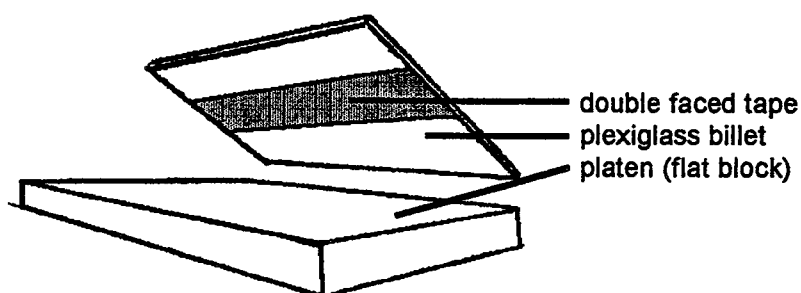
If you press the **5** key on the numeric keypad of the keyboard after you press the Alt + H keys, all three axes will automatically go to their home positions.

This page is intentionally left blank.

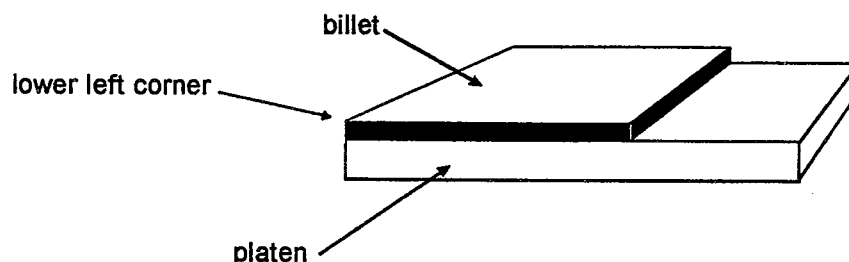
7 LOADING STOCK

Your Micromill is supplied with a fixture assembly and platen (unless a specialized work holding device, such as a vise, is fitted). The instructions below apply to mounting a piece of plexiglass on a machine equipped with the standard fixture assembly.

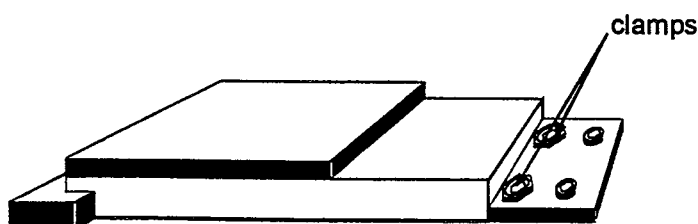
1. Locate a piece of plexiglass. Remove the protective paper from one side of the plexiglass billet. Do not remove the protective paper from the other side of the billet.



2. Put a strip of double sided tape on the side of the billet that has the protected paper.
3. Align the lower left corner of the billet to the lower left corner of the platen.
4. Press the plexiglass billet to assure that it is secure.



5. Place the platen with the billet securely attached in the fixture found on the machine table.
6. Turn the clamps 1/4 turn clockwise to secure the platen.



This page is intentionally left blank.

8

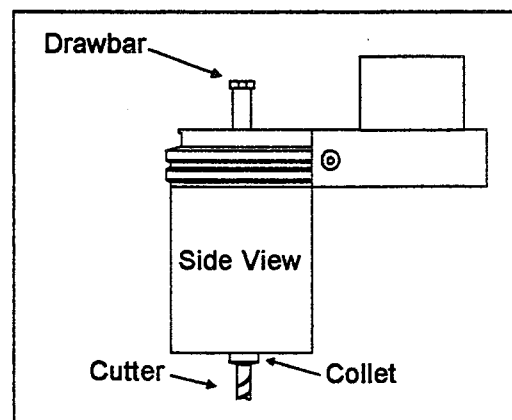
REMOVING AND MOUNTING TOOLING

This chapter consists of four sections:

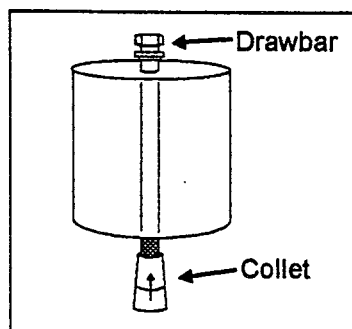
1. Removing a Collet and/or Cutter.
2. Mounting a Collet and/or Cutter.
3. Removing Quick Change Tooling.
4. Mounting Quick Change Tooling.

Removing a Collet and/or Cutter

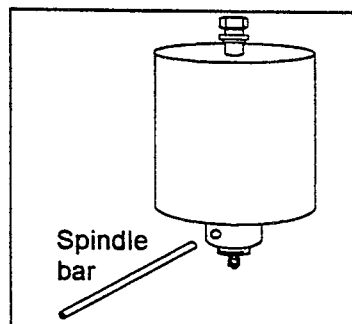
1. Loosen and turn the drawbar counterclockwise three turns. Do not over-loosen the drawbar.
2. Use a hammer to tap the top of the drawbar.
3. The cutter will loosen or completely break free from the collet.
CAUTION: Be careful, the cutter is very sharp.
4. If you also want to remove the collet, unscrew the drawbar and the collet will break free.



Mounting a Collet and/or Cutter



1. Insert the collet into the spindle and start to thread the collet onto the drawbar. Do not completely tighten the collet.
Note: If the collet is already in the spindle proceed to the next step.
2. Place the desired cutter into the collet and tighten the spindle, using the spindle bar, onto the collet.
Note: Various cutters require different size collets.



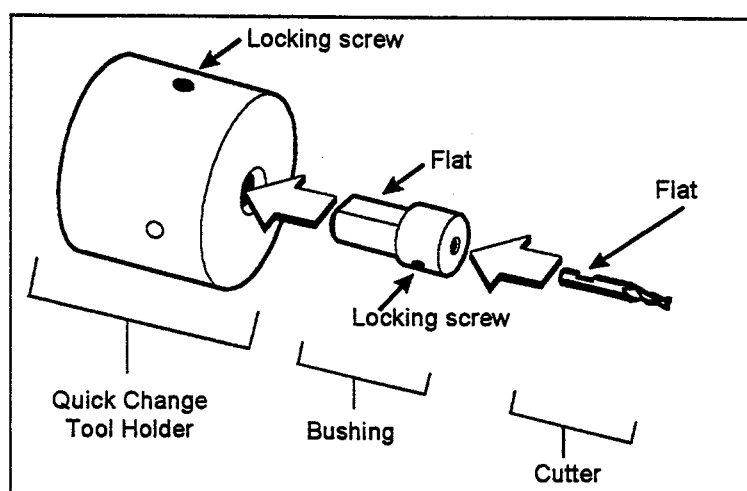
Removing Quick Change Tooling

1. First loosen the drawbar.
2. Next, insert the spindle bar into the hole in the spindle and loosen the spindle.
3. Now unscrew the quick change tooling from the spindle.

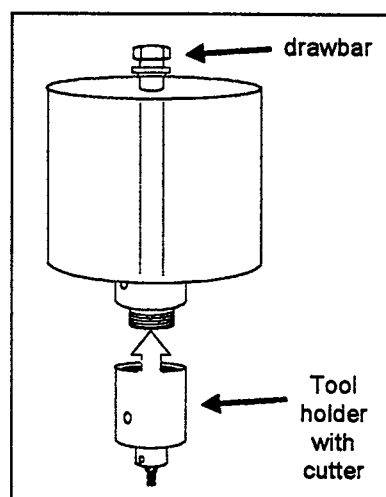
Mounting Quick Change Tooling

By using quick (fast) change tooling, several tools can be set up in the holders and when the offsets are set, parts may be manufactured using multiple tools.

1. The first step is to mount a cutter into the quick change tool holder. Refer to the diagram below. Tightening the locking screws will secure all components in place.
Note: The bushing may already be inserted into the tool holder.



2. Once the quick change tooling assembly is complete, thread it onto the spindle.
3. Tighten the drawbar and the tool holder.



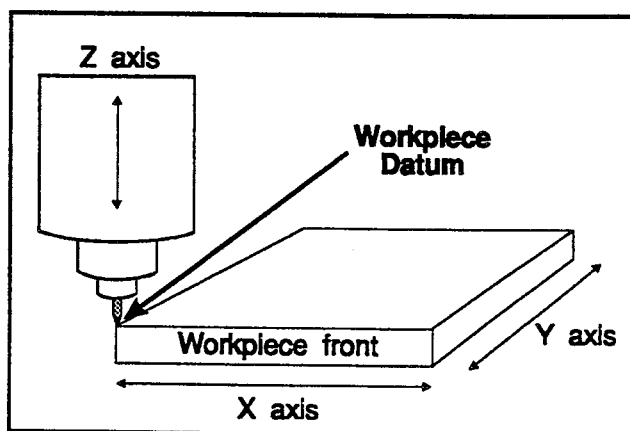
9 TOOL OFFSETS

The machine datum (home) must be set before setting tool offsets. Instructions for homing the machine are found on page 6-1.

1. Press the **Alt** + **J** keys. You want to make sure you are in the CONTINUOUS mode. Pressing Alt + J again will toggle between continuous and step modes. The bottom section of the screen will show you what mode you are in. If it does not say CONTINUOUS, press Alt + J until it shows the correct mode.
2. To datum the X axis, press and hold down the **←** or **→** keys until this axis reaches the left hand side of the billet.

NOTE: To make each axis move faster while moving the tool to the part datum, press and hold down the **5** key on the numeric keypad while moving each axis.

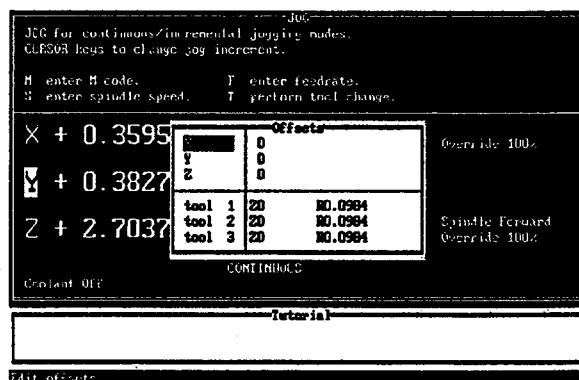
3. To datum the Y axis, press and hold down the **↑** or **↓** keys until this axis also reaches the lower left corner.
4. To datum the Z axis, first turn on the spindle. This is a precautionary measure to prevent the tooling from breaking if you happen to go to far into the billet. Next, press and hold down the **Page Down** or **Page Up** keys and move the tool to lightly touch the top of the part to set this axis.



5. Press the **F9** key

6. Cursor to **Edit Offsets** and press [Enter].

7. The screen will now show the Offsets menu. Using the arrow keys, highlight **X**, press the **X** key, now type in the value for X which is shown to the left on your screen. Press [Enter].



8. Highlight **Y**, press **Y** the key, type in the Y value shown at the left. Press [Enter].

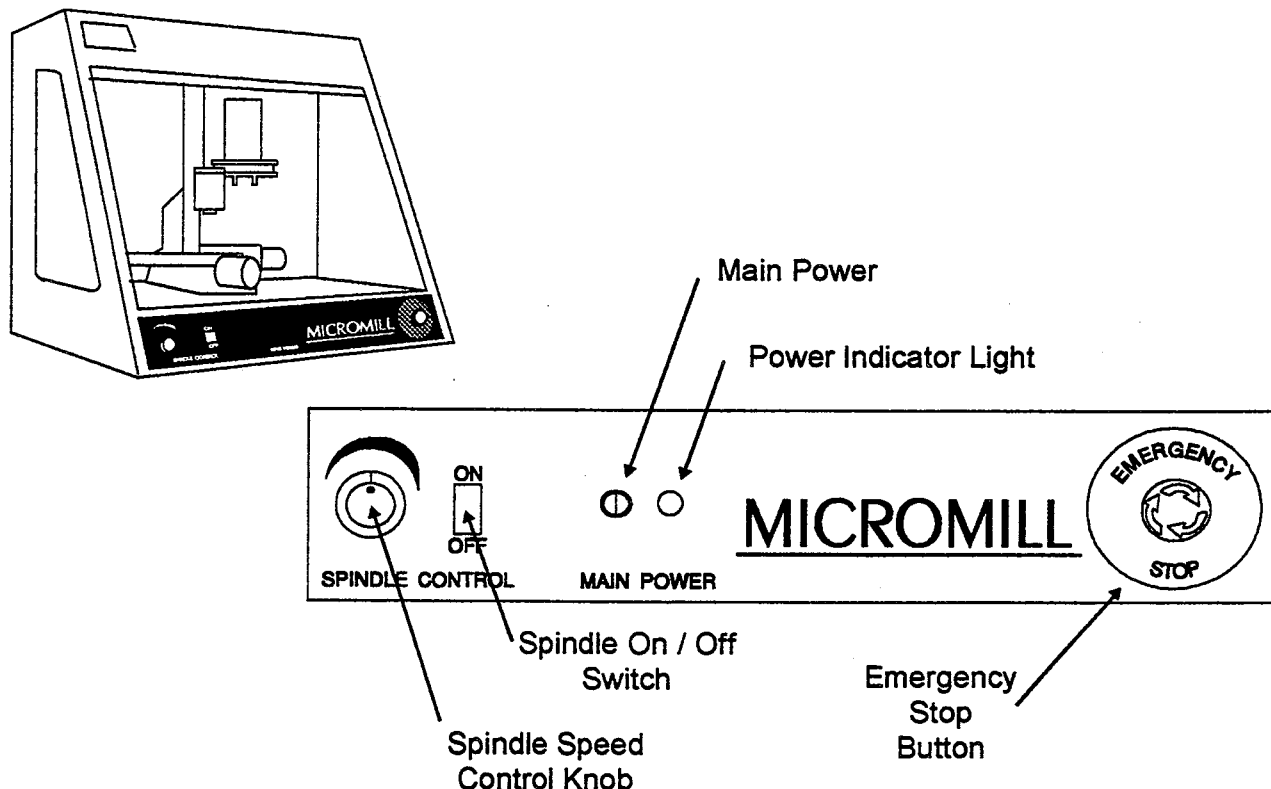
9. Highlight **Tool 1** (shown in the lower half of the offsets window) then press the **Z** key. Type in the Z value shown at the left. Press [Enter] to accept.

10. Now press the **R** key and enter in the tool radius for tool 1. Press [Enter]. (This step is not necessary if you are using an engraving cutter). The radius of the tool is used when G41 and G42 (offset left and right) are used in a CNC program (see page 17-1)

11. To save the offsets, press the **F9** key, cursor to **Save offsets**, then press [Enter]. Type in a filename (you can only use numbers for the filename), and press [Enter]. i.e.: 1111 and press Enter.

Be sure you do not type the .FAO extension after the numerical filename.

10 MICROMILL OPERATING PANEL



Spindle Speed Control Knob

The spindle speed control knob controls the speed independently of the program. Turn the knob clockwise to increase the speed or counterclockwise to decrease the speed.

Spindle On / Off Switch

This switch turns the spindle on or off.

Main Power

This is where the power to the machine is activated. A key is required to turn the machine on.

Power Indicator Light

The red light will illuminate when the machine is on.

Emergency Stop Button

Pressing this button will cut all power to the three axis drives and also the spindle.

NOTE: If the emergency stop button is pressed in when you turn on the machine, the machine will not power up. Turn the Emergency stop button to the right to disengage the button.

This page is intentionally left blank.

This section describes the keys used when utilizing the Micromill Control Software.

Operating the QWERTY Keyboard

Make sure the Num Lock light is off before using the Micromill Control Software.

The following keys and key combinations will allow you to create and edit programs in the control software.

Operation Keys



Resets alarm messages and resets program to start in edit mode.
Also used as an emergency stop when a program is running.



Moves cursor through program and menus.

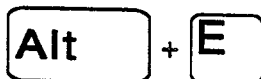


Moves cursor through program page by page. Also moves through multiple help screens.

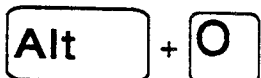
Selection Keys



UTILS - Toggles between directives. Toggles between TOOL DEF, BILLET, etc. (see pg. 14-2)





EDIT - Selects edit mode




MENU OFFSET - Toggles between M.D.I. (Manual Data Input) and Tool settings

Editing Keys

 +  ALTER - Alters the data at the cursor position. Replaces selected data with the current command on the edit line.

 INSERT - Inserts data

 DELETE - Deletes data

 ENTER - Registers End of Block (EOB)
/ ; # (characters shown are toggled by the Enter key)

Operation Select Keys

Alt + **A** AUTO MODE - Select this mode to execute the program.

F9 SIMULATION MENU - Opens the simulation menu.

← **→** Feed Override when program is running

Alt + **E** EDIT - Select to edit a program.

Alt + **S** SINGLE BLOCK - Allows single step execution of program.

Alt + **? /** BLOCK SKIP - Used to ignore block when running program. (It is activated when "/" is keyed in front of a block of information. "/" is keyed in, when in edit mode, by pressing Enter twice then Insert)

Alt + **H** HOME - Begins the process to home (datum) the machine.

Alt + **J** JOG - Moves axes around in Continuous mode, at a pre-established feedrate. Change from Continuous to Jog Step mode by pressing Alt+J again. When in Jog Step mode the amount of movement per step can be adjusted by pressing Alt + cursor up / down. The axis moves at 0.0001", 0.0002, 0.0196", and 0.1969" increments.








Alt + **M** Displays the "distance to travel" on each axis when running a program. This toggles between "Actual Position" and "Distance to Travel"

Execution Keys

Alt + **A** (from the datum or jog screen) then **F9** then **Execute CNC**
CYCLE START - Starts program.

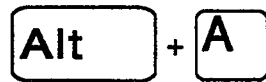
Space Bar **CYCLE STOP** - Activates feed hold mode. Press Enter to continue.

Axis Direction Keys

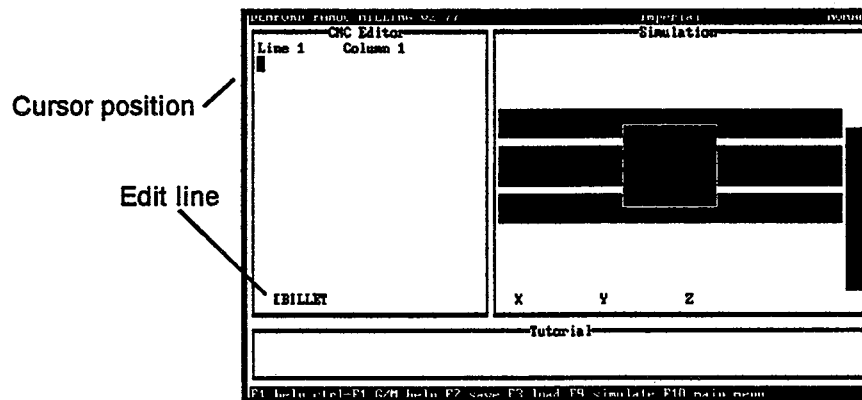
	-X	Movement in -X direction
	+X	Movement in +X direction
	-Z	Movement in -Z direction
	+Z	Movement in +Z direction
	-Y	Movement in -Y direction
	+Y	Movement in +Y direction
	TRVRS	Rapid Traverse when pushed with any axis direction key

Editing a CNC Program

When editing a CNC program, the following keys may be used.



ALTER - Alters information at the cursor position with the information on the edit line.



Moves cursor in appropriate direction.



Deletes at the cursor



Inserts at the cursor



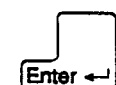
Moves up a page



Moves down a page



Clears the edit line

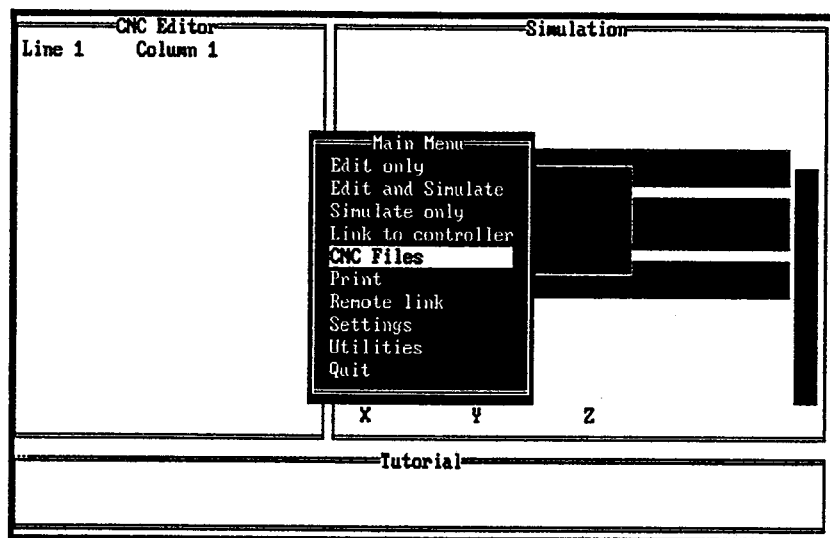


Selects End of Block

This page is intentionally left blank.

12 CONTROL SOFTWARE - MAIN MENU

Pressing the **F10** key at any time will activate the Main Menu.

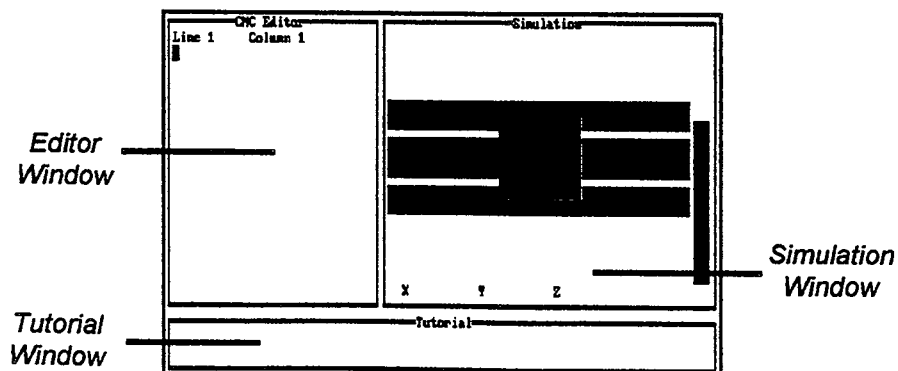


EDIT ONLY

Displays full screen editor with 241 character sideways scrolling facility.

EDIT AND SIMULATE

Displays Editor, Simulation, and Tutorial windows. If the CNC line is longer than the edit window, it scrolls sideways. During program edit, simulation can be activated at any time. After program execution, the cursor returns to its last position in the Editor.



SIMULATE ONLY

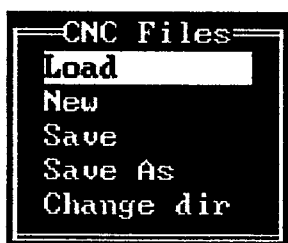
Resets the simulation window to a full screen view. The tutorial window is still displayed at the bottom of the screen. If an error occurs during program execution, the Edit and Simulate mode will be automatically selected and the problem code highlighted.

LINK TO CONTROLLER

This function is not used with the Micromill.

CNC FILES

A very powerful filing system is integrated within the software. The following points should be noted when using the dialog box for loading, saving, or listing files.



Load

If there is a program currently in the editor, you will be asked whether to merge the new program. Press the Y key to merge or the N key to clear the current program from the editor and to load the desired file.

After selecting Load, a dialog box will display:



To display a list of all the files available, press the Enter key. All the files are listed in alphabetical order. Use the cursor up and cursor down keys to select a file, then press Enter to load the selected file.

Note: the F3 key is the quick load fast key.

New

Clears the current program from the editor. If the program has been changed since the last "save" you will be prompted "current program not saved - save it first?" Answer "Y" to save the program first or "N" to clear the Editor.

Save

Saves the current program in its default directory and with its default filename. If the program has no name it will prompt for a name to save it as. The extension assigned to the filenames will be .FNC.

If you want to save your program with a new filename, do not select this option.

Note: the **F2** key is the quick save fast key.

Save as

Saves a new program with a filename. This option should be used for the first initial time you save a program. Once you have saved the program with a new filename and the program has been altered in some way, you should select the save option.

Change dir

Lets the user select a different path for the files to be saved to and loaded from. It is recommended that the default path be used.

For example:

After selecting Change dir, a dialogue box will appear. Press Enter to view the available directories to choose from. Cursor to the desired directory and press Enter.

If you want to load a file from the new directory, first press the F3 key. If the dialogue box is empty, press Enter to view the files. If the dialogue box shows a different directory then what you specified, press the Delete key once, press the Backspace key once and then press Enter. This will let you view the files in the new directory.

PRINT

Allows you to print a paper copy of your program in a paginated form. The layout for the copy can be set in the *Setting Menu* under *Print Page Layout*.

a) *Line format with errors*

Prints the program as displayed in the Editor with errors highlighted.

b) *Line format no errors*

Prints the program as displayed in the Editor without highlighting errors.

Note: errors can be checked with the Dry Run option, but a print out with errors can be useful for most any program.

REMOTE LINK

Used to send or receive information from a remote device, for example, a computer, a punch tape reader, or a data carrier.

Load from device

If there is a program currently in the Editor, you will be asked whether to merge the program from the Remote Device.

Press the Y key to merge or the N key to clear the current program from the Editor and to Load the selected program from the Remote Device.



Send to device

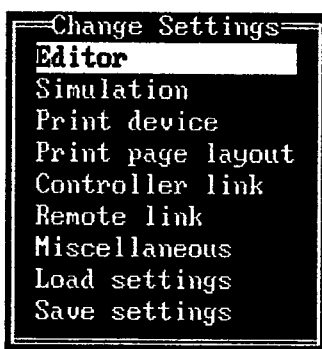
You will be prompted with "Ready to Send?". Press the Y key to send or the N key to abort. During transmission a "Transmitting to device" message with the number of bytes and lines sent will be displayed in a window. A "Transmission completed" message confirms that the whole program has been sent. Press the Esc key to clear the window.

SETTINGS

There are many settings within the software that allows customizing to suit the user. Once you have set all the options, remember to save the settings.

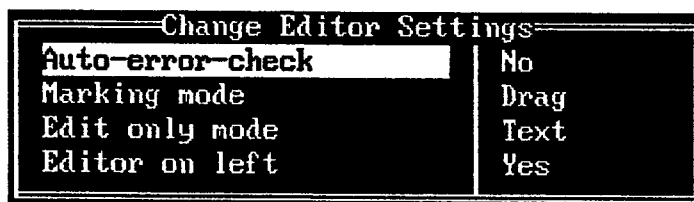
Editor

By selecting *Editor* from the Change Settings Menu, the *Change Editor Settings* menu will appear.



Auto-error-check

Toggles Yes or No for automatic error checking of each program line. With error checking on, pressing the Enter key will display a description of any error. Pressing the Esc key will clear the message and the cursor will then highlight the location of the error.



Marking mode

Toggles the marking mode between Anchor and Drag.

Note: this option has no effect on the Micromill and is only displayed for compatibility with other Denford software products.

Edit only mode

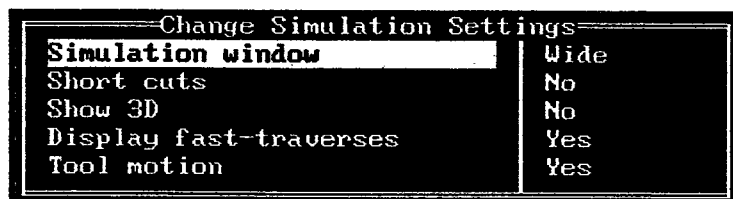
Toggles between Text or Graphics mode when the edit only option is selected from the main menu. The only advantage of selecting Text mode over the Graphics mode is a slight speed difference when scrolling. If Edit and Simulate is selected from the main menu, then Graphics mode is set automatically. This function has an advantage when working with a 286 computer.

Editor on left

Allows you to re-position the editor on the screen. You can have the editor on the left or right side.

Simulation

By selecting *Simulation* from the Change Settings Menu, the *Change Simulation Settings* menu will appear.



Simulation window

Switches between a normal size simulation window to a wider window. Displays a larger view of the simulation if the Wide option is selected.

Short cuts

This option is active when the tool motion is set to ON. This then speeds up the simulation by not displaying all the tool movements. (i.e.: drilling cycle)

Show 3D

Enables or disables the 3D view option when simulating a program.

Display fast traverse

Toggles fast traverse display on or off. If fast traverse is set to Yes, the software displays both fast traverse and programmed feedrates. Fast traverse is displayed as a yellow line in graphic simulation.

Tool motion

Toggles tool display on or off. If Yes is selected, the tool shape is animated and displayed. If set to No a toolpath plot only is shown.

Print device

By selecting *Print device* from the Change Settings Menu, the *Change Print Device* menu will appear.

Change Print Device	
Device: Type	Dos device
Name	PRN
RS232: Baudrate	
Parity	
Data bits	
Stop bits	
Protocol	

Device Type:

This can be set to:

DOS device

A DOS device to normally the parallel port which can be set to LPT1, LPT2, or PRN. Select this option if your printer has a parallel port.

Serial

There are several settings required with the serial port:

- Baudrate
- Parity
- Data bits
- Stop bits
- Protocol

To change any of the parameters, make sure that the device is set to Serial. Highlight the option and press Enter to scroll between the various options.

File

This option is used to save the program as a file on disk for printing at a later date. The current filename will be used with an extension of .LST

Print page layout

By selecting *Print page layout* from the Change Settings Menu, the *Change Print page Layout* menu will appear.

Change Print Page Layout	
Page width	80
Page depth	60
Left margin	0
Top margin	0
Bottom margin	0
Column width	80
Carriage return nulls	0
Line feed nulls	0
Form feed nulls	0
Print line feeds	Yes

By changing the following parameters, a customized printout can be set. If you have several printers, save each individual setting as a different filename.

- Page width
- Page length
- Left margin
- Top margin
- Bottom margin
- Column width
- Carriage return nulls
- Line feed nulls
- Form feed nulls
- Print line feeds

Line feeds and form feeds depend on the printer's settings. If you are not sure, try different options until a layout form is to your requirement.

Controller link

This function is not used with the Micromill.

Remote link

By selecting *Remote link* from the Change Settings Menu, the *Change Remote Link* menu will appear.

Change Remote Link	
Device name	COM1
Baudrate	4800
Parity	Odd
Data bits	7
Stop bits	1
Send line-feeds	Yes
End-of-file	ctrl-Z
Send XOFF at % full	100

The settings for the remote link are for the serial port. This sets the protocol when communicating to an external device such as a remote computer or paper tape punch.

Miscellaneous

By selecting *Miscellaneous* from the Change Settings Menu, the *Change Misc Settings* menu will appear.

Change Misc Settings	
Global units	Metric
User's name	A N Other
Screen text size	Large

Global units

The global units toggle between Imperial (inches) or Metric programming.

User's name

If the user's name is used, it will be printed out on all CNC program printouts. This can only be numerical characters when using the Micromill. It is suggested, for use in an educational setting, that a student ID number be assigned as the user's name.

Screen text size

Toggles between 25 or 43 lines on screen. NOTE: There will be no change of text size with newer super VGA drivers.

Load settings

Any number of settings files can be stored to disk with the extension .OPT

Select the load settings option, and the default settings file appears. The default filename is FANUCSMD. If you want a listing of all available settings files, clear the edit window with the Esc key and press Enter.

Save settings

This allows you to save to current settings. Type in a filename or accept the default and press Enter. The filename can only be composed of numerical characters.

UTILITIES

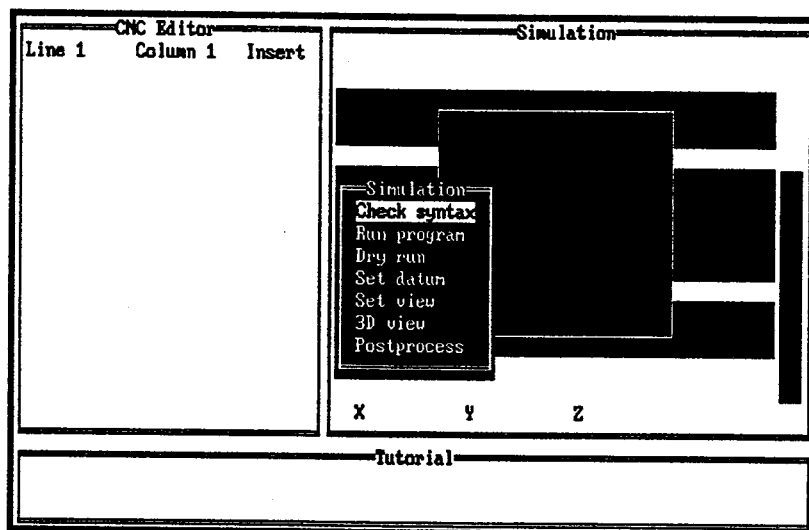
The utilities function is not utilized with the Micromill.

QUIT

This function when selected allows the user to exit the Micromill software and return to DOS.

13 CONTROL SOFTWARE - SIMULATION MENU

Pressing the **F9** key at any time will activate the Simulation Menu from the Editor or Simulation screens.



CHECK SYNTAX

This facility checks through the whole program for format errors.

RUN PROGRAM

This starts the on screen simulation of the program.

DRY RUN

This facility runs the program without an on screen display. This provides fast overtravel checking, so that it is possible to avoid a "run in" or exceeding the machine limits when the program is run/executed.

SET DATUM

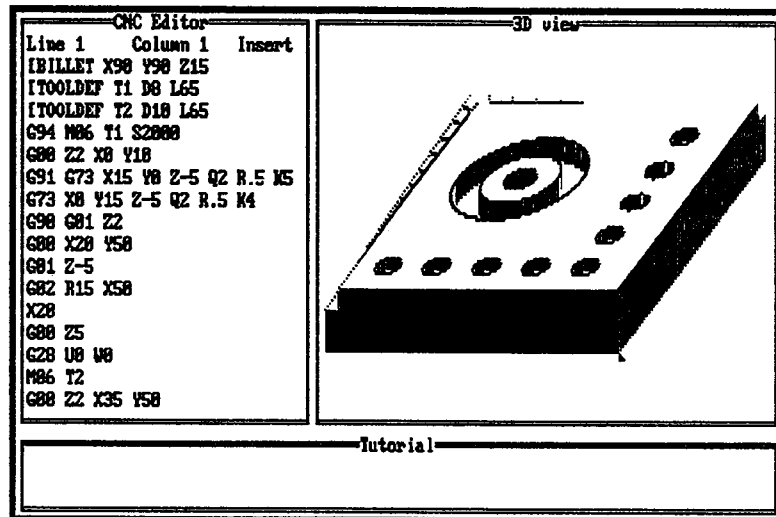
This allows you to change the datum position on the billet in the Simulation window. Select the option to display the crosshairs and then use the arrow keys to move to the desired position. Press Enter to set the new datum position.

SET VIEW

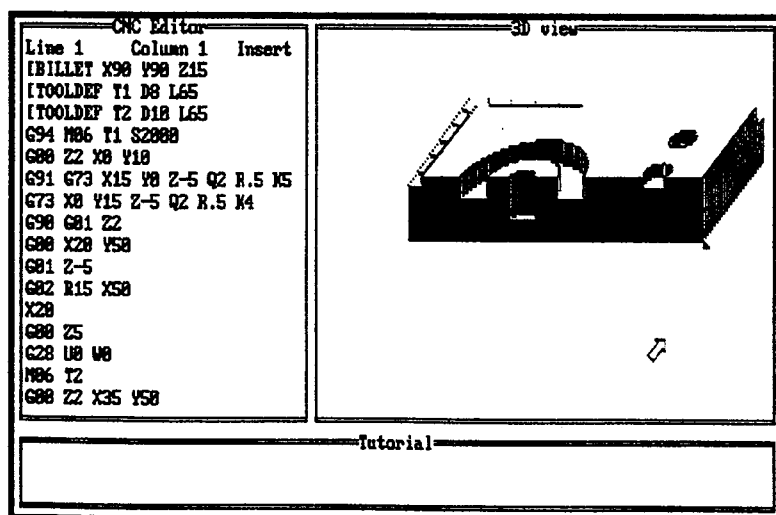
Use this facility to indicate the view you require for on screen simulation. Use the spacebar or the arrow keys to change the view. Press Enter to confirm.

3D VIEW

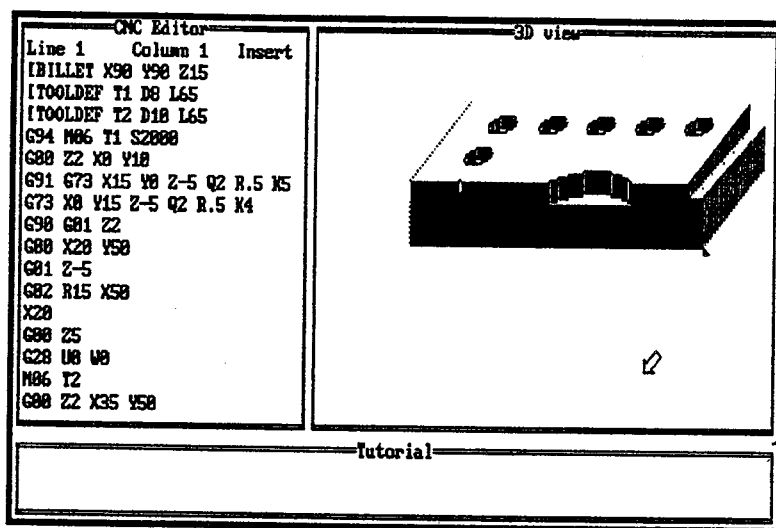
Shows the simulation in a three dimensional view.



You can also view sections of the 3-dimensional piece. To do this, first view your part in 3D. Next use the arrow keys to move the pointer to select a section of the part. Now press the Page Down key to view



Pressing the Page Up key will let you rotate the part 180° on the screen.



To restore the 3D image to its full view, use the arrow keys to move the pointer down and press the Page Down key.

POST PROCESS

This function creates an intermediate text file with an extension of .MNC for use with Denford's optional Universal Post Processor to create CNC files compatible with non-Denford CNC machines.

This page is intentionally left blank.

14 CONTROL SOFTWARE - MANUAL DATA INPUT (MDI)





This chapter briefly explains the basic procedures used to manually create a CNC program in the Micromill Control Software.

Note: Make sure the NUM LOCK on your keyboard is off before using the software.

ENTERING DATA INTO THE EDITOR

1. Type data. The information you type will be displayed on the edit line. (see pg. 11-5)

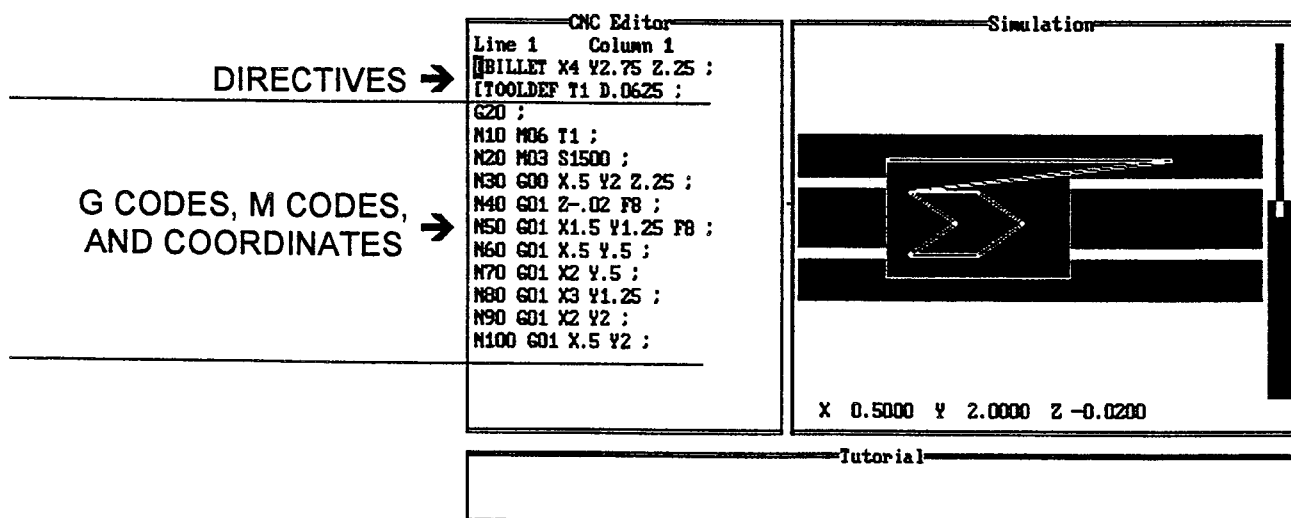
2. Press the  key after each code is typed.

For example: G00  X1.5  Y1.5  Z.01 

3. Press the  key to register End of Block (EOB) followed by the  key.

A semicolon (;) will be shown to signify the end of a block.

The screen shown below is an example of a simple CNC program which was manually written in the control software. The topics covered in this chapter are labeled below.



DIRECTIVES

Pressing the  key will toggle through the directive options.

When manually writing a CNC program in the control software, you must first select the appropriate directives before proceeding with G codes, M codes, and coordinates. The following is a list of directives.

[BILLET

Billet Definition - This directive allows the billet to be given a size.

The billet definition should be placed at the start of a program.

For example:

[BILLET X4 Y2.75 Z.25; defines the billet as 4" long, 2.75" wide, and .25" deep. If the unit of measurement is set to metric, specify the billet size in metric units.

[TOOLDEF

Tool Definition - This directive is used to set the tool number, tool diameter, and tool length.

T is the prefix for the tool number

D is the prefix for the tool diameter

L is the prefix for the tool length. The tool length is the distance from the bottom of the spindle nose to the end of the cutter. The tool length is used to check overtravel on the Z axis.

For example:

[TOOLDEF T1 D.0625; defines that Tool 1 has a tool diameter of .0625".

[CLEAR

This clears the tutorial messages window.

[EDGEMOVE

Since the software checks for overtravel, the [EDGEMOVE directive allows the billet to be moved to the same position as on the machine. It is used by the on-screen simulation.

For example:

[EDGEMOVE X.5 Y.5; this will place the billet's bottom left corner at .5" and at .5" from the X0 and Y0 position of the machine before offsets are loaded.

[STEP

This directive switches the program to single step execution.

[NOSTEP

This directive turns off single step execution on-screen and when linked to the machine.

[SHOW

This allows operations to be simulated on screen.

[NOSHOW

Deactivates the on screen simulation.

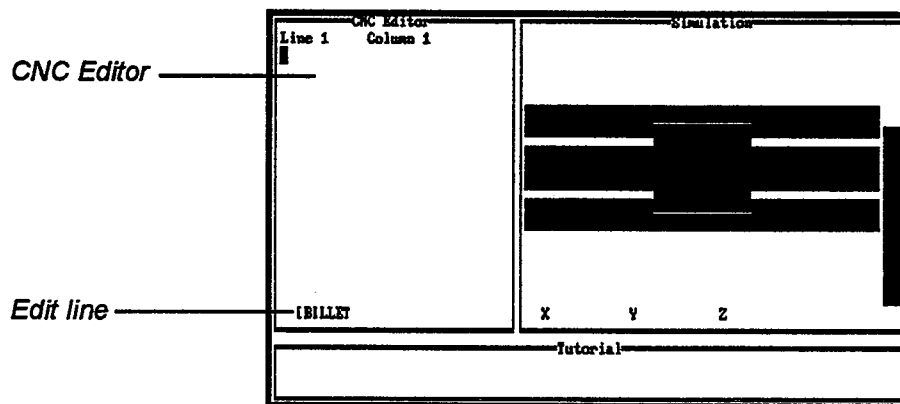
[SUBPROGRAM

Allows a program with a non-numeric name to be called as a subprogram.

For example: *[SUBPROGRAM 2 FRED*

INSERTING DIRECTIVES INTO THE EDITOR

Press the directive key to toggle through the selections. The selections will be displayed on the edit line of the CNC editor screen as shown in the screen below. Press the **Insert** key to apply the directive to your program. Once the directive is entered, you can type all the necessary coordinates and codes.



G CODES, M CODES, AND COORDINATES

Once your directives are set, you can proceed to enter all codes and coordinates.

You must use the Insert key (see page 14-1) to register all data and the Enter key to register the end of a block.

Refer to Appendix 9 for a step-by-step tutorial
on manually creating a part program.

CUTTING A PART

NOTE: Offsets must be set or loaded before running the program.

1. You first need to change modes by pressing the **Alt** + **J** keys. This will put the machine in Jog Mode. Now press the **Alt** + **A** keys. This will put you in Auto Mode. The top of the screen will indicate what mode you are in.
2. Turn the spindle on and adjust the speed.
3. When in the Auto Mode, the milling machine can execute your file. When you are ready to cut the part, press the **F9** key, cursor to **Execute CNC**, and press the **Enter ↵** key.

If you wish to stop the machine, you can press the emergency stop button. Do not open the guard while the machine is running.

The following chart is a quick guide to the complete manufacturing process.

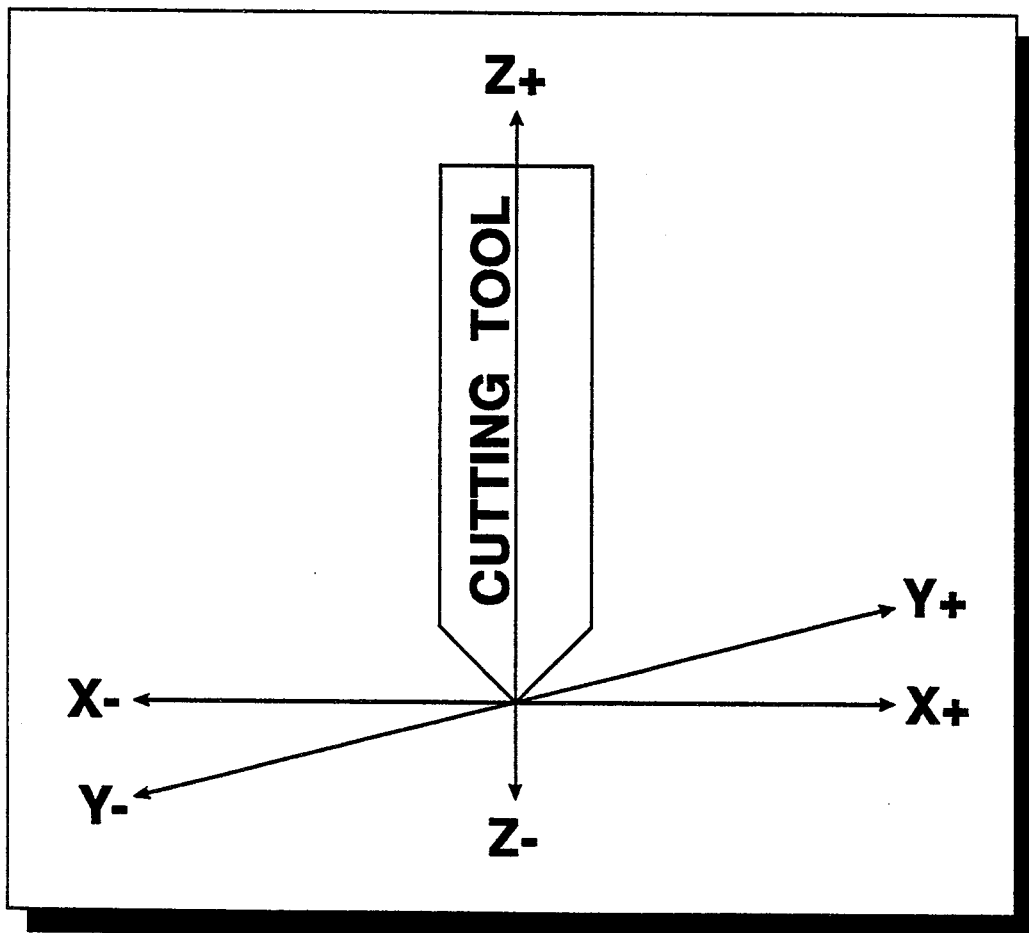
FROM START TO FINISH

Follow these steps	See this chapter for instructions.
1. Load the stock	7
2. Turn on machine	10
3. Open control software.	5
4. Write a new CNC program or load a previously saved file.	12 or 14
5. Always simulate the program.	13
6. Home the machine	6
7. Set the tool offsets or load a previously saved offset file.	9
8. Cut the part.	15

NOTE: The Tool offsets may be set or loaded at any time before cutting a part.

16 BASIC MOVEMENTS

This chapter explains the theory of machine axis movement. The understanding of movements is vital in the CNC manufacturing process.

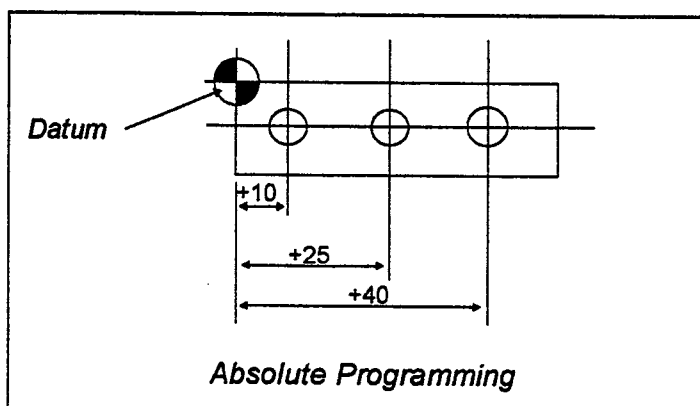


BASIC MOVES

ABSOLUTE PROGRAMMING

In absolute programming, the programmer commands the tool to move to a new position which is relevant to component zero (also known as workpiece datum). The direction of motion is determined by the position of the datum.

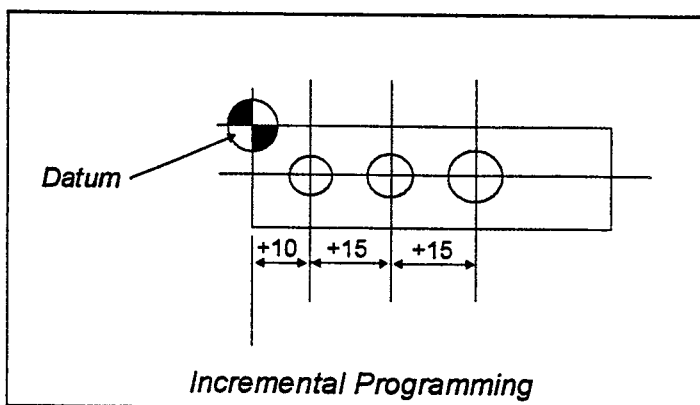
The programmed word, which instructs the control to accept the X, Y, and Z values as absolute, is G90.



INCREMENTAL PROGRAMMING

In incremental programming, the programmer commands the tool to move to a new position which is a specific distance and direction from its previous position. The direction of motion is determined by the sign of the value for the command.

The programmed word, which instructs the control to accept the X, Y, and Z values as incremental, is G91.



CUTTING MOTIONS

STRAIGHT LINE MOTIONS - G00 AND G01

Almost all motion commands are programmed as straight line motion for moving to or cutting along a face. During cutting statements, it is only necessary to change this mode of operation when an arc is to be generated.

G00

This code causes the tool to move to the specified position at rapid traverse. (When using rapid traverse, material is not cut)

For example: *G00 X.5 Y1 Z.1*

Here the tool is moved to X.5", Y1", and Z.1" at a rapid traverse.

G01

This code causes the tool to move to the specified position at a feedrate. (When using feedrates, material is cut)

For example: *G01 X1 Y1.25 Z-.2 F2.5*

Here the tool is moved to X1", Y1.25", and Z-.2 at a feedrate of 2.5" per minute.

CIRCULAR MOTIONS - G02 AND G03

For circular interpolation (G02 and G03) the maximum programmable radius is restricted to 5.1575".

G02

This code causes a clockwise circular motion. When a G02 is programmed, the coordinates of the end point of the arc need to be specified as well as the radius or the center of the arc. If a positive radius is specified, a short arc is cut. Accordingly, if a negative radius is specified, a long arc is cut.

For example: *G02 X1.50 Y1 R.75 F3*

In this example, the tool is moved to X1.5" and Y1". The arc has a radius of .75".

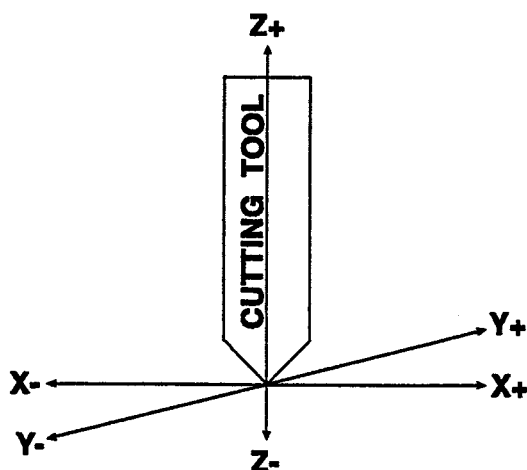
G03

This code causes a counterclockwise circular motion. The example above also applies to this command.

AXIS DEFINITION

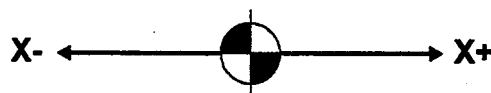
The illustration below shows the positive and negative movement of the axes. This is an important concept to understand because all programming is done with this in mind.

NOTE: All motion programmed is the movement of the tool, not the movement of the machine table.



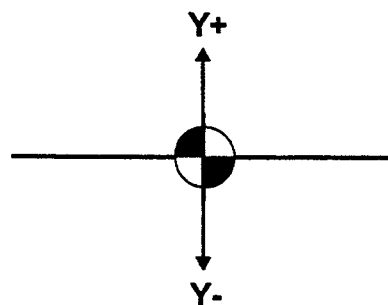
X axis direction from the part datum

- X- Left forward movement of the tool
- X+ Right forward movement of the tool



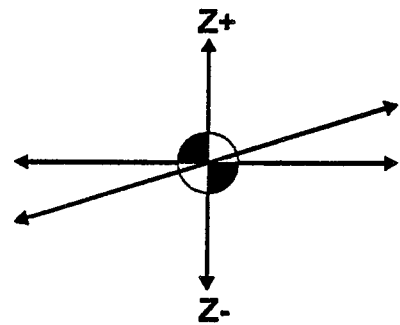
Y axis direction from the part datum

- Y+ Tool moves away from the operator
- Y- Tool moves towards the operator



Z axis direction from the part datum

Z+ Tool moves up, away from the table
Z- Tool moves down, towards the table



This page is intentionally left blank.

17 CUTTER RADIUS COMPENSATION

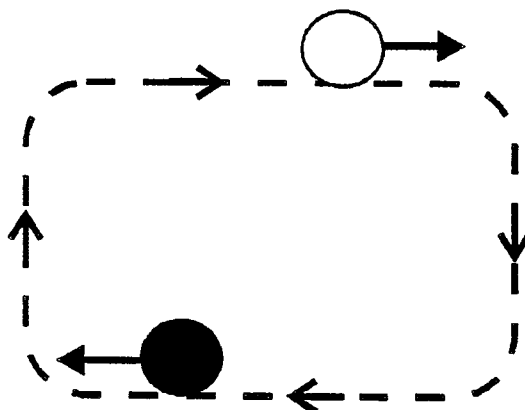
Each tool has a Cutter Radius Compensation associated with it. This is of particular importance for contouring operations. The machine control is capable of modifying the cutter center path to take into account the actual point of cutter contact.

In most cases the "nominal" diameter will provide an appropriate level of accuracy. In some cases, however, the cutter radius offset may have to be adjusted.

G41 calls for compensation on the left hand side of the work looking along the direction of travel.

G42 called for compensation on the right hand side of the work looking along the direction of travel.

G40 cancels cutter compensation.



Direction of travel



Programmed cutter path



G41 - Left hand compensation



G42 - Right hand compensation



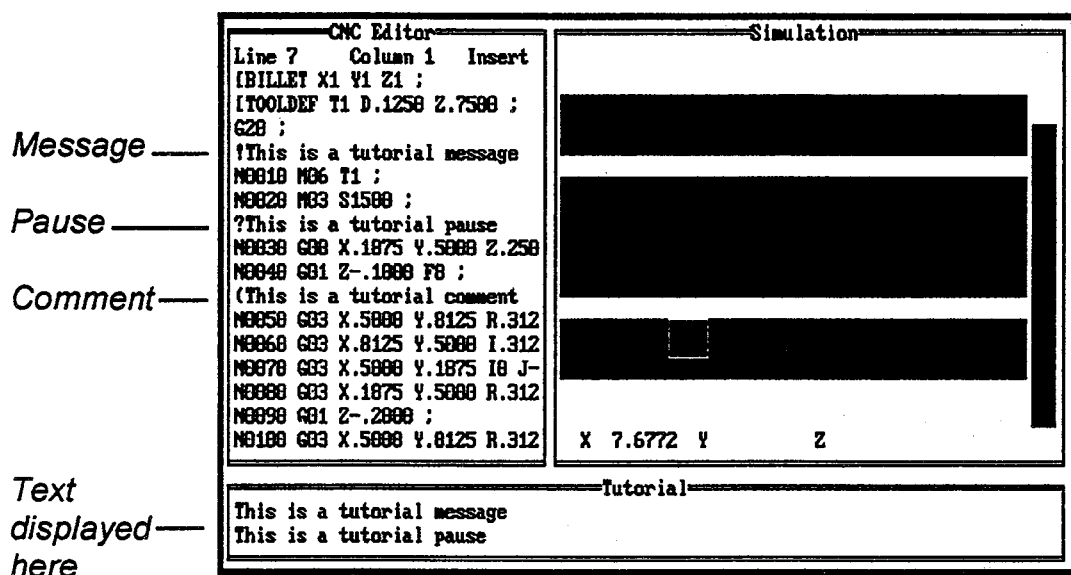
This page is intentionally left blank.

18 TUTORIALS AND COMMENTS

This feature of the control software is intended as a teaching aid.

Messages and comments can be added into the CNC program and displayed in the tutorial window at the bottom of the screen when the program is simulated. This feature can be used to create interactive lessons.

Below is a sample of the 3 different types of tutorials.



The following page explains each type of tutorial.

TUTORIAL MESSAGE

(displays message without stopping)

Messages begin with an explanation mark (!) which is followed by some text.

For example: !Using Tool 2

TUTORIAL PAUSE

(displays message but stops for keypress)

Pauses begin with a question mark (?) which is followed by some text.

For example: ?Check the position

COMMENTS

Comments begin with an open bracket "(" and can be used to annotate a program.

Comments are ignored when the program is executed.

For example: (Entering circular cycle

CLEARING TUTORIAL MESSAGES, PAUSES, AND COMMENTS

Since only two lines of text can be displayed in the Tutorial window at any given moment, you can add the directive [CLEAR to the program. Adding this will clear the text in the tutorial window to allow more messages to be viewed.

19 MACHINE MAINTENANCE

PLANNING

When carrying out maintenance work, pay special attention to the following items.

1. Before starting any maintenance work, define the task and obtain the information relevant to carry out the maintenance. Also define the time period needed to do the necessary work, to obtain the correct tools, and to order spare parts if needed.
2. During the maintenance work period, put up a notice which states that the machine is under maintenance and that it should not be used until the notice is removed.
3. Safety should be a priority when carrying out any maintenance work. Covers and safety guards that are removed during the maintenance period should be replaced after the work is completed.
4. All work should be carried out by suitably qualified personnel.
5. When replacing electrical components, ensure that they are the components specified.
6. All work done on the machine whether progressive or preventive should be logged so that a complete service record can be kept for future referral.

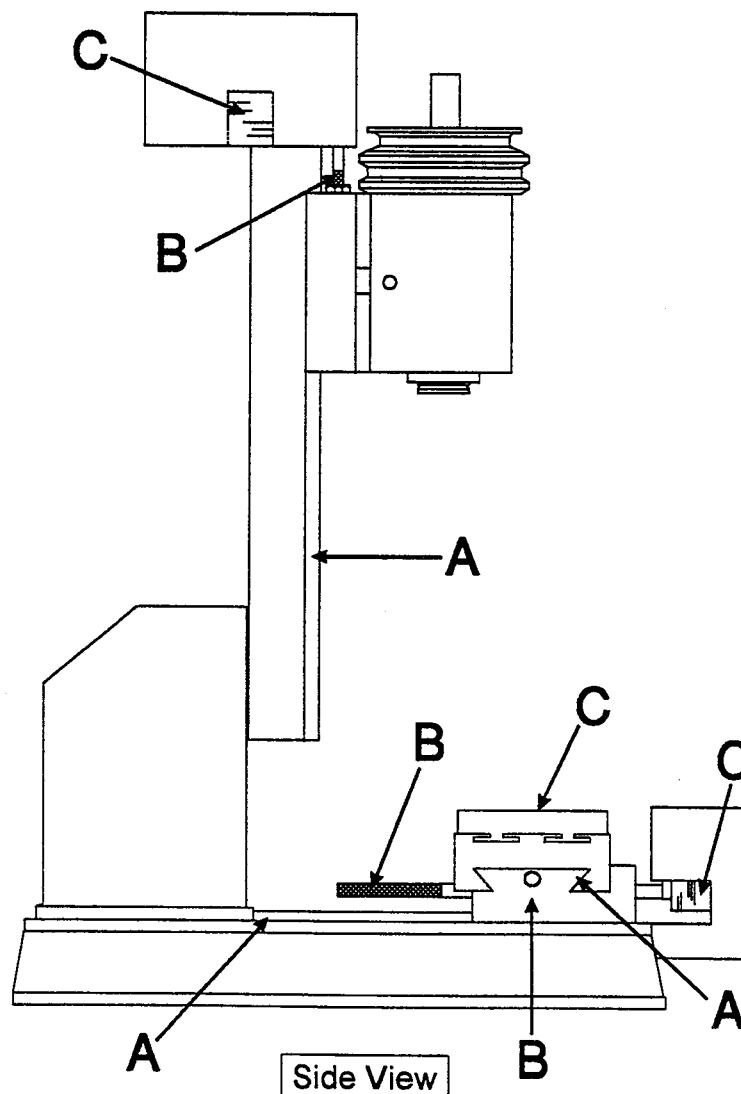
MACHINE LUBRICATION

The following items on the machine should be ***oiled daily***.

A light machine oil should be used.

- A. Slides
- B. Leadscrews
- C. Thrust bearings

The illustration below labels each area which requires daily lubrication.

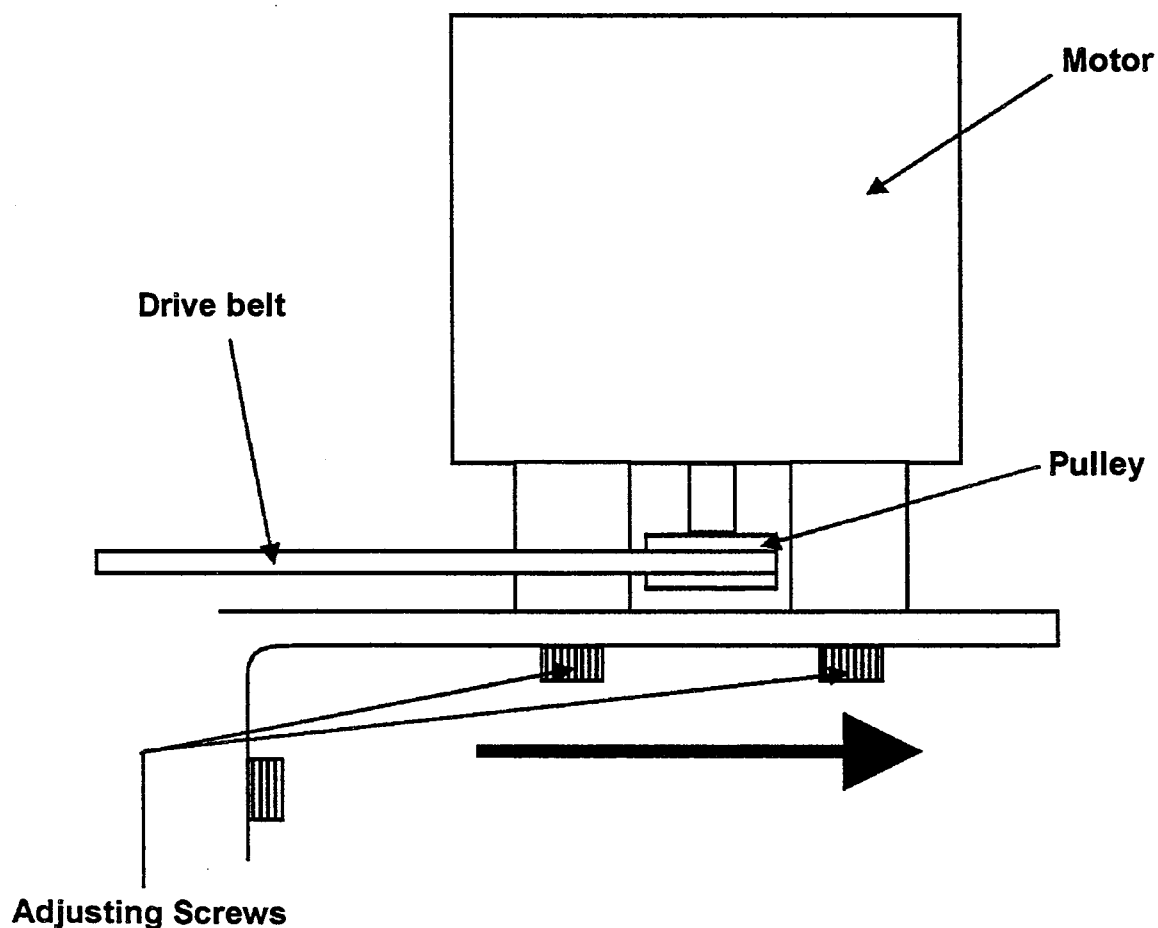


Note: Your mill is encased in a cabinet. For illustration purposes, this diagram shows the mill without the cabinet.

ADJUSTING THE BELT TENSION

The drive belt from the motor to the machine spindle may become slack from time to time due to constant use. To tighten the belt, loosen the two adjusting screws, tighten the belt by moving the motor in the direction of the arrow (see illustration below) then re-tighten the adjusting screws.

DO NOT OVERTIGHTEN THE BELT.



This page is intentionally left blank.

APPENDIX 1 - CUSTOMIZING SETTINGS FILE

Most of the settings are self explanatory. Some settings options can be changed by loading a Settings File and adding extra option to it. Because a control might have changed over the years, it is possible to change various settings to make the software compatible with all controls. The vast majority of end users won't need any modifications to be made.

EDITING THE FANUCSMD.OPT FILE

This file, as with any of Denford's settings files, can be edited by using any text editor.

Below is the typical contents of the FANUCSMD.OPT file.

```
NARROWSIMBOX 0
SHOWTOOL 0
USE3D 0
FANUC_SDEVICE machine
UTILS_MENU_1 Dos Access
UTILS_GO_1 dos
HIGHSPINDLE 2800
EXTCOMM_ENABLE 0
EXTCOMM_ROUTE 1
EXTCOMM_FILE $$DRIP$$CNC
EXTCOMM_CHECKRATE 200
EXTCOMM_SDEVICE COM1
EXTCOMM_BAUD 9600
EXTCOMM_PARITY 0
EXTCOMM_DATABITS 8
EXTCOMM_STOPBITS 1
EXTCOMM_LF 1
EXTCOMM_ES 1
EXTCOMM_EE 2
ATC 0
MOVERANGEX 209.55
MOVERANGEZ 165.1
MOVERANGEY 76.2
highfeedz 500
highfeed 500
datumfeedz 500
METRIC 0
```

CHANGING COM PORT SETTINGS

EDITING THE FANUCSMD.GO FILE

Below is the typical contents of the FANUCSMD.OPT file.

```
IBM.RS2 2
$3F8 4 COM1
$2F8 3 MACHINE
QDENFORD.KBD 1
18.2 0
EGA.CRT 0
FANUCSMD.APP
```

The machine connection for the Micromill is set for the COM2 port from the factory as default. Should you wish to operate the machine from COM1, edit the FANUCSMD.GO file as follows:

From the root directory in DOS

1. Type:

```
C: [Enter]
CD\IDENFORD [Enter]
EDIT FANUCSMD.GO [Enter]
```

2. Change the following line:

```
$2F8 3 MACHINE
      to
$3F8 4 MACHINE
```

3. Press the [Alt] + [F] keys to open the File Menu.

4. Press the S key to save your changes.

APPENDIX 2 - GLOSSARY OF TERMS

ARC - A portion of a circle

ARC CENTER OFFSET - The incremental distance between the programs cutter path and the arc center in the X and/or Z direction. The X and Z are written as the I and K amounts without a direction sign.

ASCII - American Standard for Information Interchange.

AUTOMATIC CYCLE - is a mode of control operation that continuously runs the cycle or stored program until a program stop or end of program word is read.

AUXILIARY FUNCTION - A function of a machine other than the control of the coordinates of a workpiece or cutter. i.e: F00, S1000, T1, M08.

AXIS - X axis: Line perpendicular to the spindle centerline (absolute)
U axis: Line perpendicular to the spindle centerline (incremental)
Z axis: Line parallel with the spindle centerline (absolute)
W axis: Line parallel with the spindle centerline (incremental)
Y axis: Line perpendicular to both the X and Z axes (absolute)
V axis: Line perpendicular to both the U and W axes (incremental)

BINARY - A system for describing numbers using only two digits.

BIT - The smallest programmable unit in machine code. 8 bits = 1 byte.

BLOCK - A word(s) that collectively provide sufficient information for a complete operation. The block is separated from other blocks by an end of block character.

BLOCK DELETE - A feature which provides means for skipping blocks (which have been proceeded by a slash) at the discretion of the operator.

BUFFER STORAGE - An information storing area which is utilized to transfer the stored data to active storage almost instantaneously.

CANNED CYCLE - An automatic motion cycle which is held in buffer storage for the duration of the cycle.

CHARACTER - A number, letter, or symbol as entered into a CNC program.

CIRCULAR INTERPOLATION - A means of programming an arc by providing a few basic statements.

CLOSED LOOP - A system in which the result of the output is measured and fed back for comparison with the input.

CNC - Computer Numerical Control. See **SOFTWARED NC**

COMMAND - A signal or group of signals initiating one step in the execution of a program.

CONTOURING (CONTINUOUS PATH) - Coordinating simultaneous motion of two or more axes.

CORNERING - The effect at the machine due to the difference of electrical commands and the slide positioning while commanding an abrupt change of direction.

C.S.S. - Constant Surface Speed: A feature in a control system which tracks the point of a cutting tool so that the spindle can be constantly monitored to give the correct peripheral speed condition.

C.P.U. - Central Processing Unit: The part of a computer's system which consists of the arithmetic, control units, and the working memory.

DECODE - The translation from tape coded language to control language.

DRY RUN - This function enables the operator to run through the program replacing the programmed feedrates with the maximum jog rates in order to test the program and check tool clearance.

DWELL - A programmed time delay.

EDIT - Blocks of data can be displayed for examining. The faulty blocks can then be erased and correct information put in its place by means of MDI and the EDIT facility.

E.I.A. - Electrical Industries Association: An organization responsible for setting many NC standards, one being tape coding.

ENCODER - A device connected to the leadscrew for comparing the position with the control input instruction.

END OF BLOCK (EOB) - A character inputted in the program which denotes the end of a block of data.

EXECUTIVE - In CNC systems, the executive software determines the manner in which the program data is processed.

FEED HOLD - At any stage in the cycle, the operator can stop the machine movements by using Feed Hold.

FEEDRATE - A word addressed by the letter F and followed by a number, used by the control to drive the slides. The rate of feed may be programmed as feed / min, or feed / rev.

FEEDRATE OVERRIDE - A manual function that can be used by the operator to override the programmed feedrate within certain limits.

FOLLOWING ERROR - The lag distance between the actual machine position and the command position.

FORMAT - The arrangement and overall pattern in which the input data is organized (formatted)

G CODE - A word addressed by the letter G and followed by a numerical code defining preparatory functions.

HARDWARE - The physical components of a control system or computer.

HARDWIRED - A numerical control system which is dedicated to driving a machine through committed circuit connections and requiring the input data to be in a fixed format.

I REGISTER - A value, which when programmed, is used by the control as an X axis arc center offset, or as a velocity rate for feed and threading.

INCREMENTAL PROGRAMMING - A method of programming in which the motion statements relate from the previous programmed position. The signs which accompany the departure commands in this system are directional in meaning.

INPUT - The transfer of external information into the control system.

INTEGRATED CIRCUIT (IC) - A complete circuit constructed within to on the surface of a silicon chip.

INTERFACE - The medium through which the control or computer directs the machine tool.

INTERPOLATION - The method used by the control system to achieve a series of approximations, via straight lines, to enable an acceptable execution for tapers or arcs.

I.S.O - International Standards Organizations.

K REGISTER - A value which, when programmed, is used by the control as a Z axis arc center offset, or as a velocity rate for feed and threading.

M CODE - A word addressed by the letter M and followed by a numerical code defining auxiliary (miscellaneous) functions.

MANUAL DATA INPUT (MDI) - A means of inserting data into the control system manually.

MEMORY - The storage capacity of a system to retain a part program or programs.

MODAL - Information that is entered into the control and retained until changed.

N WORD - A word comprised of a sequence number which is used to identify a complete block of information.

OCTAL NUMBERS - A numbering system which operates to a base of 8. Octal is more readily convertible to the decimal system than is the binary system.

OPEN LOOP - A system in which the output cannot be measured and feed back for comparison with the input.

OPTIONAL STOP - A miscellaneous command that is given by the programmer to stop the spindle and feed at a specific point in the program at the discretion of the operator.

PROGRAM STOP - A miscellaneous command that is given by the programmer to stop the spindle and feed at a specific point in the program.

RESOLVER - A device geared to the leadscrew for comparing the position with the control input instruction.

RECTANGULAR COORDINATES - A component graphically shown as two perpendicular axes (X and Z) along which any point can be described in terms of distance and direction from any other point. The part program is written from this source.

RESOLUTION - The smallest increment of distance that is allowed, by the control, to command a machine movement.

S ADDRESS - A word addressed by the letter S used to command spindle speed.

SEQUENCE NUMBER - See N word

SOFTWARED NC - A control system that has been pre-programmed with software to enable it to drive a machine tool.

STEP-BY-STEP OPERATION - An extension of MDI that permits the machine to operate block by block to permit the checking of each stage of the job.

SUB-ROUTINE - A stored sub-program which is called from the main program.

TAB - Tab characters may be used to space out words in a program manuscript in columns if required.

TOOL OFFSET - A feature that allows the operator to make tool adjustments to compensate for the difference between the actual and programmed settings dimensions.

U WORD - A word addressed by the letter U which is used to command motion perpendicular to the spindle centerline (incremental).

W WORD - A word addressed by the letter W which is used to command motion parallel to the spindle centerline (incremental).

WORD - A combination of the letter address and numerical digits.

X WORD - The X word is used to command motion perpendicular to the spindle centerline (absolute).

Y WORD - The Y word is used to command motion perpendicular to both the X and Z axes (absolute).

Z WORD - The Z word is used to command motion parallel to the spindle centerline (absolute).

ZERO - In absolute programming, zero is the point from which all other points refer to.

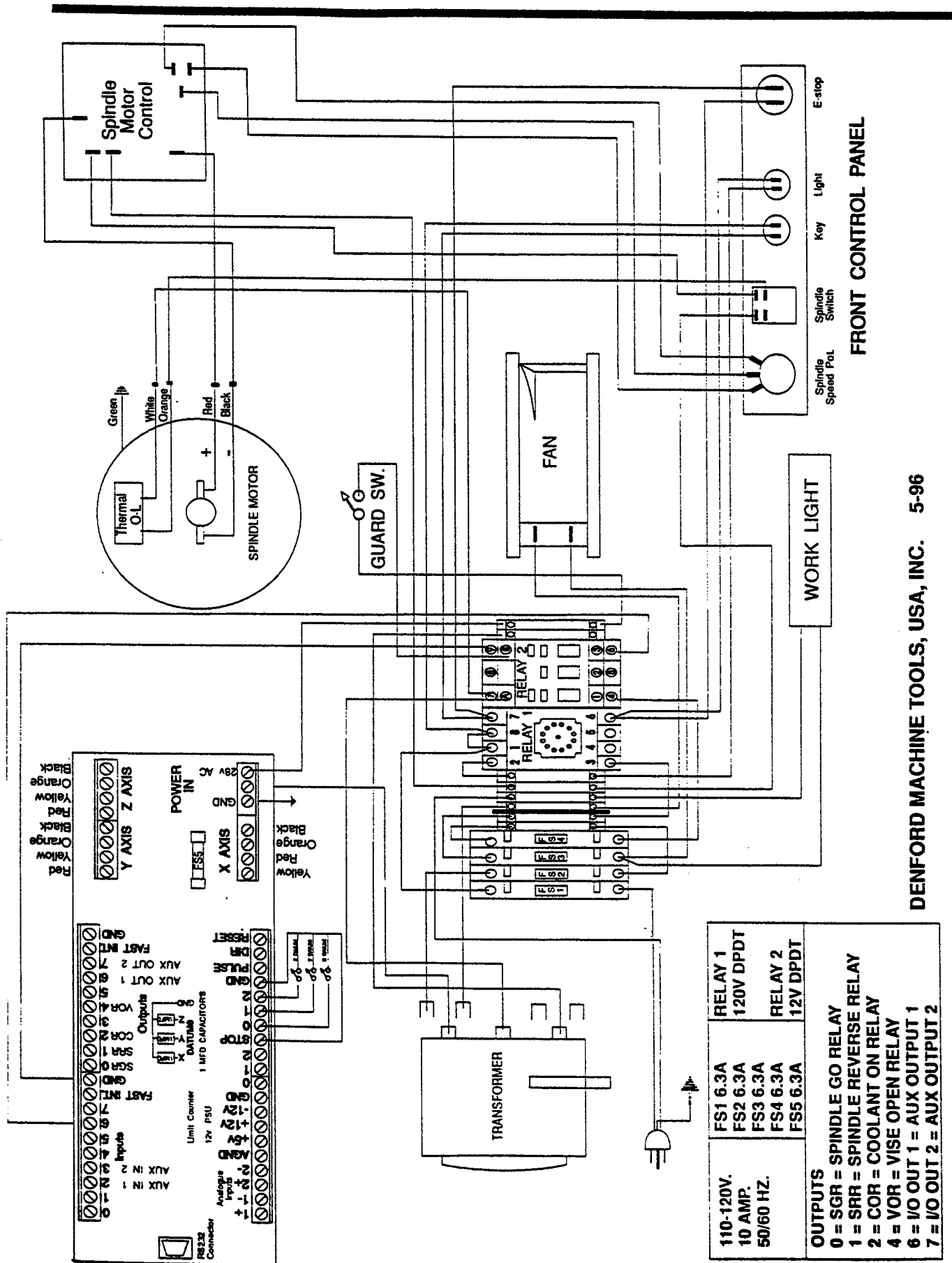
APPENDIX 3 - LIST OF ABBREVIATIONS

BCD	Binary Coded Decimal
DVM	Digital Volt Meter
EPROM	Erasable Programmable Read Only Memory
G.B.T. No.	Group Block Terminal
IC	Input Filter or Conditioner
IP	Initial Position or zero reference point
LED	Light Emitting Diodes
LSI	Large Scale Integration
MCL	Machine Control Logic
MDI	Manual Data Input
MSD	Machine Set-up Data or parameters
OD	Output Driver
PCI	Programmed Controlled Interface
PCL	Programmable Controller Language
POT	Potentiometer or variable resistor
PROM	Programmable Read Only Memory
SCR	Silicone Controlled Rectifier
TRIAC	Solid State Relay
+VE	Positive
-VE	Negative

This page is intentionally left blank.

APPENDIX 4 - ELECTRICAL CONTROL PANEL DIAGRAM

The diagram on the following page illustrates the basic components and wiring configurations for the Micromill.

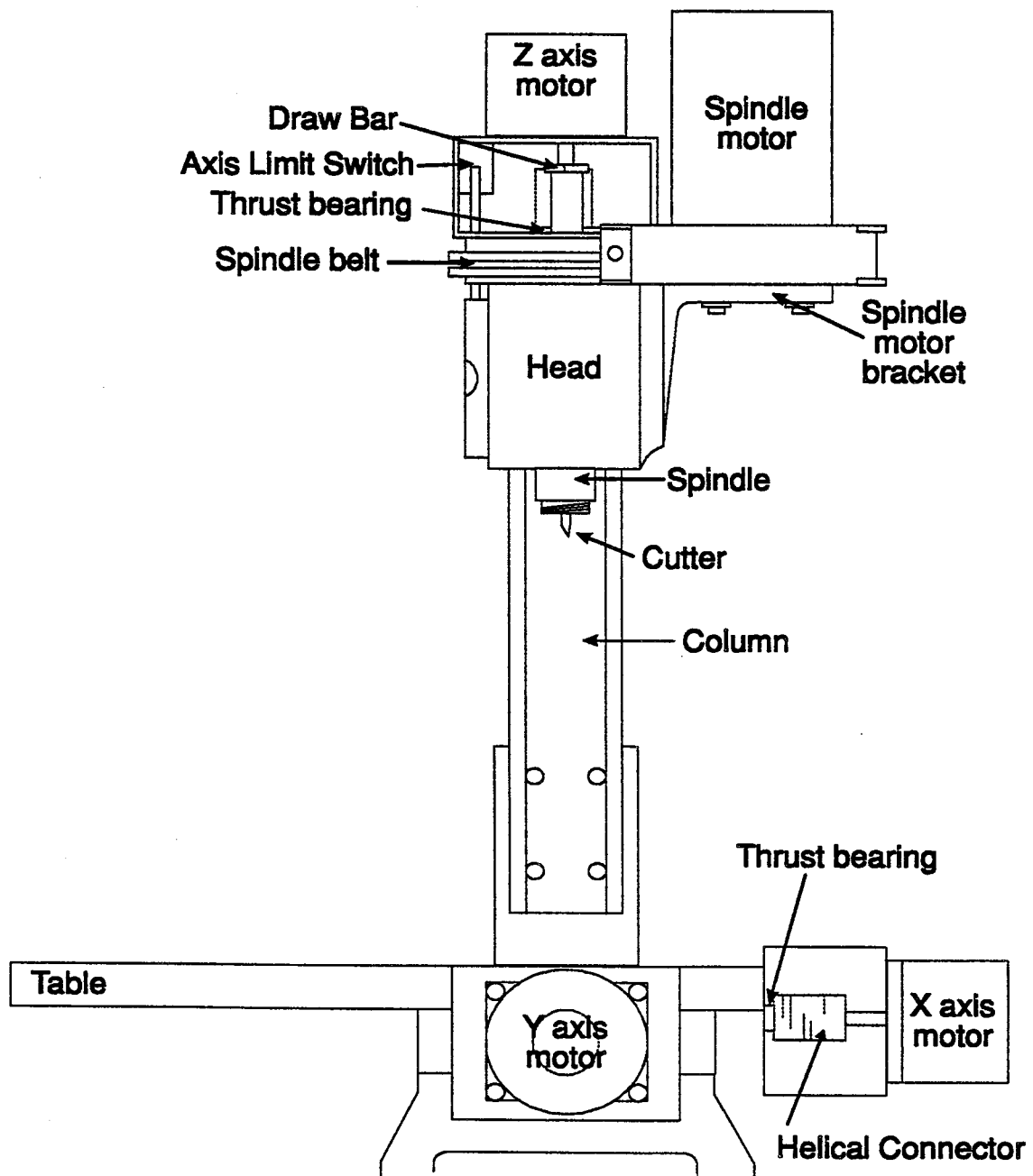


DENFORD MACHINE TOOLS, USA, INC. 5-96

APPENDIX 5 - MICROMILL NOMENCLATURE

The illustration below identifies the major components of the Micromill.

For illustration purposes, the mill is shown without the cabinet.



This page is intentionally left blank.

APPENDIX 6 - SUMMARY OF TYPICAL ADDRESS CHARACTERS

N	Sequence number
X	X axis
Y	Y axis
Z	Z axis
G	Preparatory functions
I	Incremental distance on the X axis
J	Incremental distance on the Y axis
K	Incremental distance on the Z axis
R	Radius
M	Miscellaneous functions
T	Tool numbers
S	Spindle speed
F	Feedrates

This page is intentionally left blank.

APPENDIX 7 - G AND M CODE LISTING

Note: Detailed descriptions of G and M codes are available in the software by pressing the [F1] key, selecting CNC Instructions and then selecting the appropriate codes.

G CODE LISTING - Note: Some codes listed below cannot be used with the Micromill.

G00	Rapid (fast) Traverse, non-cutting movement
G01	Linear Traverse (feedrate, cutting movement)
G02	Circular Interpolation, CW arc
G03	Circular Interpolation, CCW arc
G04	Dwell, Exact stop
G20	Imperial Units (inches)
G21	Metric Units (metric)
G28	Return to Reference Point
G40	Cancel Tool Comp
G41	Left Tool Comp (cutter compensation left)
G42	Right Tool Comp (cutter compensation right)
G73	Pecking Drilling Cycle
G74	Counter Tapping Cycle
G76	Fine Boring
G80	Cancel Canned Cycle
G81	Spot Boring Drilling Cycle
G82	Counter Boring Drilling Cycle
G83	Peck Drilling Cycle
G84	Tapping Cycle
G85	Boring Cycle
G86	Boring Cycle
G87	Back Boring Cycle
G89	Boring Cycle
G90	Absolute Movement
G91	Incremental Movement
G92	Set Datum (absolute zero point)
G94	Feed Per Minute
G95	Feed Per Revolution
G98	Initial Level Return Mode
G99	Return to R Point







M CODE LISTING - Note: Some codes listed below cannot be used with the Micromill.

M00	Program Stop
M02	End of Program
M03	Spindle Forward
M04	Spindle Reverse
M05	Spindle Stop
M06	Tool Change
M08	Coolant On
M09	Coolant Off
M10/11	Work Clamp
M13	Coolant, Spindle Forward
M14	Coolant, Spindle Rev
M19	Orientate Spindle
M20/21	ATC Arm In/Out
M22/23	ATC Arm Down/Up
M24/25	
M62	Aux. 1 On
M63	Aux. 2 On
M64	Aux. 1 Off
M65	Aux. 2 Off
M66	Wait, Aux. 1
M67	Wait, Aux 2
M70	Mirror X On
M71	Mirror Y On
M76	Wait for Aux. 1 to go low
M77	Wait for Aux 2 to go low
M80	Mirror X Off
M81	Mirror Y Off
M98	Subprogram Call
M99	Subprogram Exit

APPENDIX 8 - Fast Keys / Help Screens










To help with software operations, there are several Fast Keys that can be used. They are displayed at the bottom of the screen on the Fast Key Strip.

The fast keys change from menu to menu and are constantly updated by the key strip. The following is a list of fast keys available.

	Get Help
	Quick Save CNC Program
	Quick Load CNC Program
	Get Information
	Check / Run CNC Program
	Get Main Menu

HELP SCREENS

Context sensitive help is available at all times. The following keys are used:

	To select Help
 + 	To select G and M code Help
	To select Sub-Related Help
 	To select from Help Options
 	To display help pages
	To exit

CONTEXT SENSITIVE HELP

HELP SCREENS

All Help Screens which appear in the software can be altered by the user.

All the help information is stored in two files:

1. FANUCSMD.TXT (the text version)
2. FANUCSMD (the compiled version)

The text file can be loaded into any word processor, or even the CNC Editor. Changes can be made and the new file can be saved. NOTE: If changes are going to be made to the FANUCSMD.TXT file, make a back-up copy of the original file before proceeding with any additions or alterations.

Once the changes have been made and a new text file has been saved, you must compile the file. Simply creating a text file alone will not activate your changes.

IMPORTANT NOTE: It is vital that the information in the text file relating to error messages, fast key strips, and text which is directly used by the software, are not altered. The software relies on certain lines being in sequence. Help screens that are activated by pressing the F1 key can be placed anywhere within the text file, however we suggest to put them at the end of the file.

There are two general steps to follow when editing help screens:

1. Alter the text file
2. Compile the new file

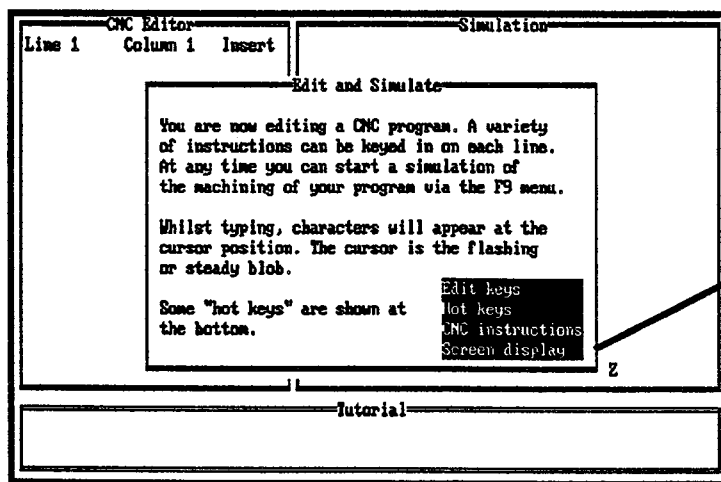
Refer to the following pages for a step-by-step tutorial on creating custom help screens.

Using the Context Sensitive Help Step-by-Step Tutorial

1. The first step is to decide first, what type of help to add and second, where in the program the help should be placed.

In this tutorial, we are going to create a lesson on CNC programming to be available by pressing the F1 key. Before going any further, make a back-up copy of the original FANUCSMD.TXT file which is located in the Denford directory.

The marked area in the screen below illustrates where we are going to add the new selection. To view this screen, execute the Micromill Control Software. Once the software is loaded, press the F1 key.



This is where the new entry will appear.

2. The next step is to decide what the menu option should be called. In this case, we will call it CNC LESSONS.
3. Now you must choose a title for the new help page. In this case, LESSON 1.
4. It is now time to load the FANUCSMD.TXT file in order to edit. To do this, go to the DOS prompt. Now type CD\DENFORD and press Enter. Next, type EDIT FANUCSMD.TXT and press Enter. The file will load.

-
5. Once the text file is loaded, you must locate the area you want to edit. Scroll down until you find the text shown on the screen on page A8-3.
 6. Find the line that says `#SCREEN DISPLAY#DISPLAY A`
 7. Position the cursor after this line and press Enter. You should now have a blank line inserted. This is where we are going to add the new help menu entry.
 8. Now type `#CNC LESSONS#LESSON1 A`
 9. Next, scroll to the very end of the file and position the cursor on a blank line.
 10. Type `|LESSON1 A|` and press Enter. This is the start of Lesson 1 and the page number is A. Always use capital letters for page starts.
 11. Now you can add any text you would like. Be sure that the lines are no longer than 47 characters. If you want more room for the CNC lesson, you can start a new page by typing `|LESSON1 B|` and adding more text.
 12. If you want sub-menus to be displayed on your new help page, simply place the sub-menus at the bottom of your help page.
For example, `#G AND M CODES#HELP PAGE1 A`
 13. Once you are done adding the text, save the file by first pressing the Alt + F keys and then selecting Save. Exit the Editor when the file is saved.
 14. You must now compile the file. To do this follow these steps.
 1. Go to the DOS prompt.
 2. Type `CD\IDENFORD` and press Enter.
 3. Type `MESSAGES FANUCSMD` and press Enter.
 4. The file should compile.

15. After the file is compiled, execute the Micromill Control Software. Once the software is loaded, press the F1 key. If your changes were successful, you will see your new entry on the screen. If your new entry does not appear, look back at the text file and see if you entered everything correctly. If all text appears to be correctly entered, try this:

1. Go to the DOS prompt.
2. Type CD\IDENFORD and press Enter.
3. Type MESSAGES FANUCSMD /N and press Enter.
4. Now go back and run the Micromill Control Software and see if the new entry appears.

Again, caution must be exercised when editing the FANUCSMD text file. Always make a back-up copy of the original file.

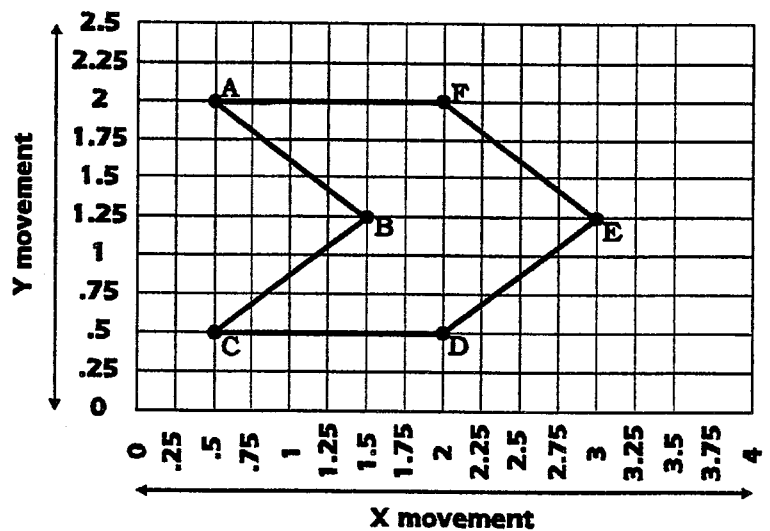
This page is intentionally left blank.

APPENDIX 9 - SAMPLE PROGRAMS

SAMPLE PROGRAM 1

The following program is intended to demonstrate the general procedure used to manually write a simple part program using linear interpolation.

Billet Material: plexiglass
Billet size: 4" x 2.75" x .25"
Tool: engraving cutter



```
[BILLET X4 Y2.75 X.25;  
[TOOLDEF T1 D.0625;  
G20;  
N10 M06 T1;  
N20 M03 S1500;  
N30 G00 X.5 Y2 Z.25;  
N40 G01 Z-.02 F8;  
N50 G01 X1.5 Y1.25 F8;  
N60 G01 X.5 Y.5;  
N70 G01 X2 Y.5;  
N80 G01 X3 Y1.25;  
N90 G01 X2 Y2;  
N100 G01 X.5 Y2;  
N110 G00 Z.25;  
N120 G28 X4 Z1;  
M02;
```


← This is what
the completed
G-code file
looks like.

The following pages provide step-by-step
instructions for writing the program for the part
shown above.

SAMPLE PROGRAM 1

Follow the proceeding instructions to manually write a program. You must be in the Edit and Simulate screen of the Micromill Control Software.

1. Define the size of the stock.

- a.) Press the directive key  until the word **[BILLET** appears in the lower left corner of the CNC Editor. Now press the *Insert* key.
- b.) Type **X4**, press the *Insert* key, type **Y2.75**, press *Insert*, type **Z.25**, press *Insert*.
- c.) Press *Enter* then *Insert*.
- d.) The line should look like this: **[BILLET X4 Y2.75 Z.25;**

2. Define the tool being used.

- a.) Press the directive key until the word **[TOOLDEF** appears. Press *Insert*.
- b.) Type **T1** press *Insert* type **D.0625** press *Insert*.
(This line defines Tool 1 as having a diameter of .0625)
- c.) Press *Enter* then *Insert*.
- d.) The line should look like this: **[TOOLDEF T1 D.0625;**

3. Set the units of measurement.

- a.) Type **G20** *Insert*. (This sets the units of measurement to inches)
- b.) Press *Enter* then *Insert*.
- c.) The line should look like this: **G20;**

4. Tell the program that you are going to use Tool 1.

- a.) Type **N10** *Insert* (From this point on line numbers are going to be used)
- b.) Type **M06** *Insert*.
- c.) Type **T1** *Insert*.
- d.) Press *Enter* then *Insert*.
- e.) The line should look like this: **N10 M06 T1;**

5. Set spindle speed.

- a.) Type **N20** *Insert*
- b.) Type **M03** *Insert*, **S1500** *Insert*.
(This turns on the spindle and sets its speed to 1500 RPM.)
- c.) Press *Enter* then *Insert*.
- d.) The line should look like this: **N20 M03 S1500;**

6. Move the tool into position to get ready to make the first cut.
 - a.) Type **N30 Insert**
 - b.) Type **G00 Insert, X.5 Insert, Y2 Insert, Z.25 Insert.**
(Rapid traverse to point A keeping the cutter .25" above the part)
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **N30 G00 X.5 Y2 Z.25;**

7. Move the tool into the billet and specify the feedrate.
 - a.) Type **N40 Insert.**
 - b.) Type **G01 Insert, Z-.02 Insert, F8 Insert.**
(Move the tool -.02" into the billet. Use a feedrate of 8" per minute)
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **N40 G01 Z-.02 F8;**

8. Begin to cut the shape. Move the tool, at a feedrate, to point B.
 - a.) Type **N50 Insert.**
 - b.) Type **G01 Insert, X1.5 Insert, Y1.25 Insert, F8 Insert.**
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **N50 G01 X1.5 Y1.25 F8;**

9. Move the the tool, at a feedrate, to point C.
 - a.) Type **N60 Insert.**
 - b.) Type **G01 Insert, X.5 Insert, Y.5 Insert.**
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **N60 G01 X.5 Y.5;**

10. Move the tool, at a feedrate, to point D.
 - a.) Type **N70 Insert.**
 - b.) Type **G01 Insert, X2 Insert, Y.5 Insert.**
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **N70 G01 X2 Y.5;**

SAMPLE PROGRAM 1

11. Move the tool, at a feedrate, to point E.
 - a.) Type **N80 Insert**.
 - b.) Type **G01 Insert, X3 Insert, Y1.25 Insert**.
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: N80 G01 X3 Y1.25;

12. Move the tool, at a feedrate. to point F.
 - a.) Type **N90 Insert**.
 - b.) Type **G01 Insert, X2 Insert, Y2 Insert**.
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: N90 G01 X2 Y2;

13. Move the tool, at a feedrate, to point A again to complete the profile.
 - a.) Type **N100 Insert**.
 - b.) Type **G01 Insert, X.5 Insert, Y2 Insert**.
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: N100 G01 X.5 Y2;

14. Now that the profile is complete, retract the tool from the billet.
 - a.) Type **N110 Insert**.
 - b.) Type **G00 Insert, Z.25 Insert**.
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: N110 G00 Z.25;

15. Move the tool away from the billet to allow enough clearance to remove the part.
 - a.) Type **N120 Insert**.
 - b.) Type **G28 Insert, X4 Insert, Z1 Insert**.
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: N120 G28 X4 Z1;

16. End of the program.
 - a.) Type **M02 Insert**.
 - b.) Press *Enter* then *Insert*.
 - c.) The line should look like this: M02;

You have just manually written a simple CNC program.

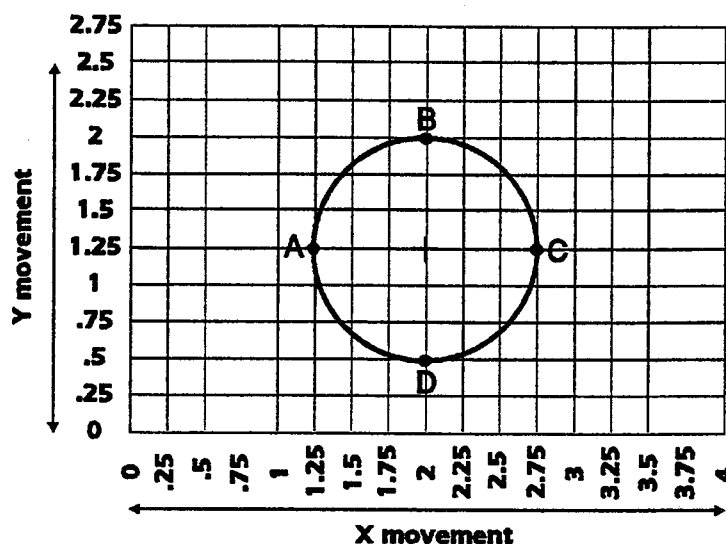
17. You should now save your file. Follow these steps.
 - a.) Press the **F10** key.
 - b.) Highlight **CNC Files** and press *Enter*.
 - c.) Highlight **Save As** and press *Enter*.
 - d.) Type in a filename and press *Enter*.
18. You can also simulate the program. Follow these steps.
 - a.) Press the **Page Up** key.
 - b.) Press the **F9** key.
 - c.) Highlight **Run Program** and press *Enter*.

Refer to Chapter 15 for instructions on manufacturing the part.

SAMPLE PROGRAM 2

The following program is intended to demonstrate the general procedure used to create a simple part using circular interpolation.

Billet Material: plexiglass
Billet size: 4" x 2.75" x .25"
Tool: engraving cutter




```
[BILLET X4 Y2.75 X.25;  
[TOOLDEF T1 D.0625;  
G20;  
N10 M06 T1;  
N20 M03 S1500;  
N30 G00 X1.25 Y1.25 Z.25;  
N40 G01 Z-.02 F8;  
N50 G02 X2 Y2 R.75 F8;  
N60 G02 X2.75 Y1.25 R.75;  
N70 G02 X2 Y.5 R.75;  
N80 G02 X1.25 Y1.25 R.75;  
N90 G00 Z.25;  
N100 G28 X4 Z1;  
M02;
```

← This is what
the completed
G-code file
looks like.

The following pages provide step-by-step instructions for writing the program for the part shown above.

SAMPLE PROGRAM 2

Follow the proceeding instructions to manually write a program. You must be in the Edit and Simulate screen of the Micromill Control Software.

1. Define the size of the stock.
 - a.) Press the directive key  until the word **[BILLET** appears in the lower left corner of the CNC Editor. Now press the *Insert* key.
 - b.) Type **X4**, press the *Insert* key, type **Y2.75**, press *Insert*, type **Z.25**, press *Insert*.
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **[BILLET X4 Y2.75 Z.25;**
2. Define the tool being used.
 - a.) Press the directive key until the word **[TOOLDEF** appears. Press *Insert*.
 - b.) Type **T1** press *Insert* type **D.0625** press *Insert*.
(This line defines Tool 1 as having a diameter of .0625)
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **[TOOLDEF T1 D.0625;**
3. Set the units of measurement.
 - a.) Type **G20** *Insert*. (This sets the units of measurement to inches)
 - b.) Press *Enter* then *Insert*.
 - c.) The line should look like this: **G20;**
4. Tell the program that you are going to use Tool 1.
 - a.) Type **N10** *Insert* (From this point on line numbers are going to be used)
 - b.) Type **M06** *Insert*.
 - c.) Type **T1** *Insert*.
 - d.) Press *Enter* then *Insert*.
 - e.) The line should look like this: **N10 M06 T1;**
5. Set spindle speed.
 - a.) Type **N20** *Insert*
 - b.) Type **M03** *Insert*, **S1500** *Insert*.
(This turns on the spindle and sets its speed to 1500 RPM.)
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **N20 M03 S1500;**

SAMPLE PROGRAM 2

6. Move the tool into position to get ready to make the first cut.
 - a.) Type **N30 Insert**
 - b.) Type **G00 Insert, X.1.25 Insert, Y1.25 Insert, Z.25 Insert.**
(Rapid traverse to point A keeping the cutter .25" above the part)
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **N30 G00 X1.25 Y1.25 Z.25;**

7. Move the tool into the billet and specify the feedrate.
 - a.) Type **N40 Insert.**
 - b.) Type **G01 Insert, Z-.02 Insert, F8 Insert.**
(Move the tool -.02" into the billet. Use a feedrate of 8" per minute)
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **N40 G01 Z-.02 F8;**

8. Machine arc to point B using a radius of .75.
 - a.) Type **N50 Insert.**
 - b.) Type **G02 Insert, X2 Insert, Y2 Insert, R.75 Insert, F8 Insert.**
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **N50 G02 X2 Y2 R.75 F8;**

9. Machine arc to point C.
 - a.) Type **N60 Insert.**
 - b.) Type **G02 Insert, X2.75 Insert, Y1.25 Insert, R.75 Insert.**
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **N60 G02 X2.75 Y1.25 R.75;**

10. Machine arc to point D.
 - a.) Type **N70 Insert.**
 - b.) Type **G02 Insert, X2 Insert, Y.5 Insert, R.75 Insert.**
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: **N70 G02 X2 Y.5 R.75;**

SAMPLE PROGRAM 2

11. Machine arc to point A to complete the profile.
 - a.) Type **N80 Insert**.
 - b.) Type **G02 Insert, X1.25 Insert, Y1.25 Insert, R.75 Insert**.
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: N80 G02 X1.25 Y1.25 R.75;

12. Now that the profile is complete, retract the tool from the billet.
 - a.) Type **N90 Insert**.
 - b.) Type **G00 Insert, Z.25 Insert**.
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: N90 G00 Z.25;

13. Move the tool away from the billet to allow enough clearance to remove the part.
 - a.) Type **N100 Insert**.
 - b.) Type **G28 Insert, X4 Insert, Z1 Insert**.
 - c.) Press *Enter* then *Insert*.
 - d.) The line should look like this: N100 G28 X4 Z1;

14. End of the program.
 - a.) Type **M02 Insert**.
 - b.) Press *Enter* then *Insert*.
 - c.) The line should look like this: M02;

You have just manually written a simple CNC program using circular interpolation.

See page A9-5 for instructions on saving and simulating the part.

This page is intentionally left blank.

APPENDIX 10 - TECH TIPS

This chapter consists of tips and techniques aimed to assist the user with the operations of the Micromill and its software.

If you require technical support, call Denford at 1-800-886-9750 Monday - Friday.
8:30AM to 5:00PM EST.

Micromill Control Software

1. If the user happens to experience a flashing screen while using the software changes should be made in the Control Panel in Windows. Refer to page 5-2 for instructions on making the correct alterations.
2. In some cases, communication errors may occur in the control software. The following are a few possible solutions to this problem.
 - a. The machine must be on before executing the control software.
 - b. Be sure you are totally free of Windows before executing the software.
3. Be sure to add the SET DENFORD_DATA=C:\DENFORD\DATA statement to the users AUTOEXEC.BAT file. See page 5-1.
Adding this statement allows all CNC files to be saved to the same directory thus avoiding problems locating saved files on your hard drive.
4. Whenever saving files on a computer, do not include any spaces in the filename. If you do happen to have a space in a filename (ex: TEST 1.FNC) the computer will not recognize the file.
5. The keyboard driver supplied with the Micromill Control Software alters the key assignments for your keyboard when using the program. Therefore, when saving tool offset files or CNC files created via Manual Data Input (MDI) only *numbers* can be used in the filename. However, if you are importing a CNC file which was created with an outside software package (such as Mill CAM Designer) that has letters in its filename, the control software will recognize the CNC file.

6. Setting Tool Offsets.

There are a few important points to remember when setting and saving offsets.

- a. When setting the tool offsets, an "R" value does not have to be specified if you are using an engraving cutter. Any other cutters, such as an endmill, require the R value to be entered. The R value is the radius of the cutter.
 - b. When moving each axis to set its position, hold down the 5 key on the numeric keypad while also holding down an axis direction key to make the axis move faster.
 - c. When positioning the Z axis, be sure the cutter is lightly touching the top of the stock. Setting the Z axis too high or too low will prevent the part from properly being manufactured because all negative and positive Z coordinates originate from the Z offset point. It is also a good idea to turn the spindle on when setting the Z axis. This is a precautionary measure to prevent the cutter from breaking when setting the Tool Length Offset (TLO).
 - d. When saving tool offsets, only numerical characters can be used for the filename.
7. If your control software will not execute properly, make sure the machine is on and connected properly to the computer.
 8. Do not start the control software in Windows or by using the DOS prompt icon in Windows. The control software must be totally free of Windows.

Micromill CNC Milling Machine

1. Be sure to properly oil the machine. See page 19-2.
2. After continual use of the platen, plexiglass, and double faced tape, residue may build up on the platen thus preventing the stock to properly adhere to the platen. All residue should routinely be cleaned off of the platen to prevent this problem from occurring.
3. If multiple tools are going to be used to machine a particular part, offsets must be set for each tool being used.
4. When the operator goes to turn on the machine after the emergency stop button was used, the button must be in the release position for the machine to turn on again. To release the button push the button in while turning it to the right.
5. A key is required to turn the machine on. Be sure to place the spare key in a safe place.
6. If the guard is open, the program will be interrupted. Always close the guard before machining.

This page is intentionally left blank.

INDEX

A

Abbreviations, list of, A3-1
Absolute programming, 16-2
Addresses
 typical characters, A6-1
 G codes, listing, A7-1
 M codes, listing, A7-2

Axes

 definitions of, 16-4
 diagram of, 16-1
Axis direction keys, 11-4
Autoexec.bat, 5-1

B

Belt tension, 19-3

C

Cables, 4-1
Com ports, changing, A1-2
Control board, A4-1
Control Software
 manual data input, 14-1
 menus:
 Main, 12-1
 Simulation, 13-1
Customizing settings files, A1-1
Cutter radius compensation, 17-1
Cutting, a part, 15-1

D

Datum
 machine, 6-1 (see also homing the machine)
 workpiece, 9-1, 9-2
Drawbar, 8-1

E

Editing
 a cnc program, 11-5
 autoexec.bat, 5-1
 settings files, A1-1
 keys, 11-1
Editor, 11-5

Equipment

 standard, 1-1
 extra, 2-2
Emergency stop button, 10-1
Executing the control software, 5-2
Execution keys, 11-5

F

Fanucsmc.opt, editing, A1-1
Fanucsmc.go, editing, A1-2
Fast keys, A8-1
Features of
 control software, 2-3, 2-4
 safety, 3-1
Fixture assembly, 7-1

G

G codes, A7-1
Graphic simulation, 13-2, 13-3

H

Homing the machine, 6-1
Help, fast keys, A8-1

I

Incremental programming, 16-2
Installation
 of control software, 5-1
 of machine, 4-1

J

Jog Mode, 11-3

K

Keys
 operation, 11-1
 selection, 11-1
 editing, 11-1
 operation select, 11-3
 execution, 11-4
 axis directional, 11-4
 fast keys, A8-1

L

Lubrication, of machine (see Maintenance)

M

Machine

connections, 4-1

datum (see homing the machine)

Manual Data Input, 14-1

Manufacturing, parts, 15-1, 15-2

Main menu, 12-1

Main power, 10-1

Maintenance

planning, 19-1

lubrication, 19-2

belt tension, 19-3

M codes, A7-2

MDI (see Manual Data Input)

Menus, 12-1 to 13-4

Micromill diagram, parts, A5-1

Mounting tooling, 8-1, 8-2

Movements

basic, 16-2

cutting, 16-3

O

Offsets, (see Tool offsets)

Operating panel

spindle speed knob, 10-1

spindle on/off, 10-1

main power, 10-1

power indicator light, 10-1

emergency stop button, 10-1

Operation keys, 11-1

Operation select keys, 11-3

P

Precautions, safety, 3-2

Power indicator light, 10-1

Programming

absolute, 16-2

incremental, 16-2

Manual Data Input, 14-1 to 14-4

Program Samples, A9-1

R

Removing tooling, 8-1

Running the control software, 5-2

S

Safety

features, 3-1

precautions, 3-2

Schematics, electrical control, A4-1

Selection keys, 11-1

Settings files, A1-1

Setting up the machine, 4-1

Shortcut keys, A8-1

Simulation, 13-2

menu, 13-1

3-D, 13-2

Specifications

cabinet, 2-1, 4-1

electrical, 2-1

mechanical, 2-1

Stock, loading, 7-1

T

Tech Tips, A10-1

Terms, A2-1

Tooling

removing, 8-1, 8-2

mounting, 8-1, 8-2

Tool offsets, 9-1, 9-2

Troubleshooting, A10-1

Tutorials & comments, 18-1

W

Word address characters, A6-1