

NORTH EAST ELECTRONICS LIMITED

PNC 4 OPERATING MANUAL

REVISED EDITION - NOVEMBER 1987

CONTENTS

REFERENCE

IMPORTANT

INTRODUCTION

1.1	GENERAL REMARKS ABOUT THE PNC 4 SYSTEM
1.2	PNC 4 DRIVE CONTROLS
1.3	PNC 4 UNIT FACIA
1.4	SYSTEM PARAMETERS AND CONVENTIONS
1.5	MAIN SYSTEM FUNCTIONS
1.6	NATURAL STATUS CONDITIONS
1.7	SET UP SYSTEM PARAMETERS
1.8	SYSTEM PARAMETERS
2.1	PRINCIPALS OF OPERATING AND PROGRAMING
2.2	ENTERING A BLOCK
2.3	RUNNING NON -PROGRAM COMMANDS
2.4	A SIMPLE PRACTICAL PROGRAM
2.5	ENTERING A SIMPLE PROGRAM
2.6	RUNNING A PROGRAM
2.7	LOOKING AHEAD
2.8	CONTINUOUS CONSTANT FEED MOVEMENT FACILITY
2.9	BLOCK SEARCH FACILITY
2.10	PROGRAM VALIDATION (TEST KEY)
3.1	CASSETTE FUNCTIONS
4.1	PROGRAMING DIAGONALS AND INT/EXT. PROFILES
4.2	PROGRAMING CIRCULAR ELEMENTS
4.3	EXAMPLES OF CIRCULAR ELEMENTS
4.4	WORKING AND PROGRAMING IN THE INCREMENTAL MODE ..
4.5	CIRCULAR PROGRAMING IN INCREMENTAL MODE

5.1	EDIT FACILITY
6.1	DATA LINK
6.2	INPUT TO PNC 4
6.3	RESPONSES FROM PNC 4 WHEN IN SERIAL LOAD MODE ...
6.4	STORE PROGRAM TO DATA LINK
6.5	HOST COMPUTER SOFTWARE
7.1	RAPID TRAVERSE
7.2	DWELL
7.3	MIRROR IMAGE
7.4	ROTATE
7.5	SCALE FACTORS
7.6	TOOL RADIUS COMPENSATION (CUTTER OFFSETS)
7.7	OFFSETS
7.8	METRIC/IMPERIAL UNITS
7.9	REPEAT/REPEAT-DIVIDE
7.10	ABSOLUTE DATUM
7.11	FLOATING DATUM
7.12	PROGRAM STOP
7.13	CONTACTS 1 AND 2
7.14	TOOL CHANGE
7.15	LABELS
7.16	AUXILIARY FUNCTIONS
7.17	INPUT FACILITIES
7.18	GOTO LABEL (CONDITIONALLY)
7.19	CALL SUBROUTINE
7.20	PLANE SWITCHING
8.1	TOOL OFFSETS
8.2	CUTTER OFFSETS

8.3 EFFECTS OF CUTTER OFFSETS
8.4 PROGRAMMING USING CUTTER OFFSETS
9.1 SUB-ROUTINES
10.1 DIGITAL TO ANALOGUE CONVERTER (OPTION)
10.2 DUAL DIGITAL TO ANALOGUE CONVERTERS (OPTION) ...
11.1 CANNED CYCLE FACILITY (OPTION)
12.1 WAIT FOR SERIAL INPUT FACILITY (OPTION)
13.1 GRAPHICS FACILITY (OPTION)
14.1 LATHE CYCLE FACILITY (OPTION)
15.1 MOVE UNTIL INPUT (OPTION)
16.1 STOP SUBROUTINE (OPTION)

APPENDIX A ERROR MESSAGES

APPENDIX B INTERNAL SWITCH SETTINGS

APPENDIX C G CODE SUMMARY
M CODE SUMMARY

APPENDIX D EXAMPLE PROGRAMS

IMPORTANT

The PNC 4 should be electrically isolated from the supply if its cover is removed.

Before applying power to the system ensure that a) the input power is at the correct voltage and b) the drives/machine switches and auxiliary outputs are correctly connected.

~~The PNC 4 should be properly earthed (grounded) at all times.~~

The stepper motor drive boards in the PNC 4 may sustain severe damage if the motor connections are made or broken with the power still on.

PNC 4's with integral stepping motor drives are equipped with a fan which keeps the drives cool ensuring high reliability. The fan filter must be kept clear. This may be done by using a vacuum cleaner (do not use an air line) to remove the dust etc. The fan filter outer cover may be removed and the filter changed if necessary.

The digital cassette unit and cassettes may be adversely affected by dust/swarf etc. In consequence, care should be taken to keep the cassettes and cassette unit as free from dust as possible. The cassettes should be stored in their cases after first being rewound to the clear leader.

INTRODUCTION

We are pleased to introduce you to this programmable numerical controller, which has been designed, developed, and manufactured by North East Electronics Ltd.

This low cost, reliable, accurate system is ideal for a wide range of applications which require precise positioning, for example on a variety of machine tools, for drilling, milling, engraving, welding and profile burning by laser or conventional methods using rotary and/or x-y tables.

At the heart of this system is a powerful NEE designed computer which gives the PNC 4 the ability to precisely control up to 4 axes of movement with linear and circular interpolation at programmable feedrates. The PNC 4 can also act as a process controller with up to 12 programmable auxiliary outputs and monitoring 8 programmable user assigned input signals.

It is the policy of North East Electronics to continually review and upgrade its products. This PNC 4 is the latest in a line of controllers whose forerunners are working in diverse applications with no small measure of success.

NEE will be pleased to discuss any application on upgrades to existing machines or applications/modifications of the PNC 4 control unit to suit new equipment.

The maximum step rates available from the PNC 4 are:

1 axis	13,000 steps/second
2 axes	12,000 steps/second linear 8,000 steps/second circular
3 axes	8,500 steps/second linear 8,000 steps/second circular
4 axes	7,500 steps/second linear or circular

If the extra cost option High Speed facility is incorporated these figures are increased by 30%.

1.1 GENERAL REMARKS ABOUT THE PNC 4 SYSTEM

To start-up the PNC system, check that the main power supply is on and that the table is well clear, then press the Green button titled 'POWER' (this will illuminate when pressed to switch on). After a few seconds the main title will be displayed.

PNC 4

Designed and Manufactured
by North East Electronics Ltd.

Rotate circular 'STOP' button clockwise to ensure it is fully released, then press the lower Green button titled 'DRIVE'. This powers up the drive section of the control system. You will also hear the Cooling Fan come into operation (if drives are fitted internally).

Now action the prompt 'PRESS <ZERO> TO DATUM MACHINE' by pressing the 'ZERO' key. A prompt will display:- 'SELECT NUMBER OF AXES TO DATUM'. Select the number of axes to seek mechanical (proximity detector) datums i.e. '3' to datum X, Y and Z.

On selecting number of axes, press 'ENTER' and the system will drive the machine to the appropriate datum position(s) at the same time displaying 'moving to datum'. On completion of the datum cycle the PNC 4 is then ready to accept commands.

There is an important point to note concerning the shutting down of the machine. Never switch off or on with a cassette in the Cassette Chamber of the PNC 4 Unit. To do so can result in corruption of the cassette program.

To switch system OFF press circular 'STOP' button to switch off drives then press green 'POWER' button to switch off system.

N.B. Whilst the PNC 4 set up procedure (see 1.8) allows the user to give different names to each axis, this manual assumes that the standard naming convention has been used. i.e.

axis 1 = X
axis 2 = Y
axis 3 = Z
axis 4 = d

1.2 PNC 4 DRIVE CONTROLS

The PNC 4 is operated from the 'PROGRAMME' section of the control panel. These controls will be seen to comprise of STOP and START buttons and a MANUAL/SINGLE STEP/AUTO Key Switch.

START This button instructs the system to commence execution of the program in its memory.

MANUAL/SINGLE STEP/AUTO The execution of the program is controlled in one of two ways. By setting this switch to AUTO, the system will drive the machine through the complete memorised program from beginning to end, without interruption. If the SINGLE STEP setting is used, the program will be executed only one stage at a time (such stages are called Program Blocks, as explained later). The program block executed is displayed on the monitor. Press the START Button for each stage. This facility is useful for trying new programs in order to prove them. The manual setting allows access to the computer programming facilities.

This key switch can be used to allow access to the program and setup functions by authorised keyholders only. Turn the key to the Auto setting and remove it.

To allow access, put the key in and turn it to the MANUAL setting.

STOP - Pressing the STOP button causes program execution to stop.

1.3 PNC 4 UNIT FACIA

The front panel of the PNC Unit is represented in Fig. 1 at the end of this section, where the descriptive terms for the various controls are stated. Study this diagram and become familiar with the terms. It will help in following the remainder of this Manual.

In particular, attention is drawn to the ZERO button at the lower right-hand corner of the MODE section. It is an important command button in operation of the PNC 4 system, and when referred to it should not be confused with the button marked with a nought (0) in the Data section. In what follows, the term 'ZERO Button' always refers to the lower right-hand button in the MODE section.

The Cassette section is basically like that of any ordinary tape recorder, with the actual Cassette being inserted, ejected and removed in the normal way. As stated previously, never switch the PNC 4 on or off with a Cassette inserted, since this can lead to corruption of recorded data.

The large red STOP Button on the PNC facia is the Drive shut down button. Note that it locks 'in'. It is released by a clockwise rotation of the Button, otherwise the drives cannot be restarted. This button will NOT switch off the computer section of the controller.

Other features on the facia will be mentioned later in the context of their use.

1.4 SYSTEM PARAMETERS AND CONVENTIONS (See also 1.7)

The PNC 4 system is programed to work within a specified working area. If the machine is asked to travel outside its programed area an error will be displayed.

When the ZERO key is pressed, the axes will travel towards datum detectors. These can be positioned at the maximum or minimum limits of the table. When each axes in turn has found its datum detectors the machine is now known to be at its Datum Position.

TEST

When used at the pre-datum stage the TEST key enters the system test facilities, normally only used by Service Engineers.

Once past the datum stage the TEST key initiates the testing of the presently loaded program, by effectively doing a "dry run" i.e. ~~executing the program~~ without actually turning the motors. (See 2.10).

1.5 MAIN SYSTEM FUNCTIONS

The computer system is a very flexible and versatile system with a host of functions and modes which provide a powerful control medium. It is not necessary to utilise all of the functions when first learning to use the machine, though an appreciation of what is available will help the operator develop his technique of machine control.

The keyboard is split up into three distinct areas:- MODE, FUNCTION and DATA. See fig. 1.

Let us consider each keypad in turn:-

"MODE" KEYPAD - Reading from Left to Right, Top to Bottom

SUBROUTINE - A subroutine is a sub program which is stored in a separate memory and may be called into the main memory by its subroutine number. (See 9.1).

CASS - This is the Cassette mode which is used to STORE (record) a program on to cassette from the system memory. Also used to LOAD (play back) a program from a cassette into the memory and used for Rewinding or Erasing. (See Section 3.1).

EDIT - Used to enter a program or make changes to a program which is already stored in the system memory. (See Section 5.1)

DATA LINK - Used to communicate with external data sources. Information is passed down an RS232 cable or a centronics parallel interface, which plug into sockets at the rear of the computer. (See 6.1).

TEST - Used to check the correct functioning of the system, (normally only used by Service Engineer) and to test the program in memory (see 2.10).

BLANK KEY - Not used.

BLOCK SEARCH - This key allows you to search for a block of information in memory and execute it. (See 2.9).

HOME - When this key is pressed part way through a program (Keyswitch in Manual) the Z axis will move to its datum position and then the X and Y axes will return to their datum position at 100% feed.

SETUP - Used to alter/display system parameters (see 1.7) and to enter plane switching blocks in EDIT mode.

ABS/INC - ABSOLUTE and INCREMENTAL are two modes in which the computer can operate. This key allows the operator to select the mode of his choice. (See 4.5).

INCH/MM - Allows the Operator to select either Metric or Imperial operating units.

ZERO - Sets or resets the machine to its Datum Position.

"FUNCTION" KEYPAD

RESET - Allows the operator to escape from information entered into the controller, successive presses escape from descending levels of input.

AUX/INPUT - AUXILIARIES allows signals generated by the PNC 4 to be used to control users own devices (see 7.16). INPUT allows signals from external switches to be input to the PNC 4 computer. (See 7.17).

ROTATE - Used to rotate programs through any angle. Extra cost option. (See section 7.4).

RAPID - Provides rapid moves between operations without disturbing the programmed feed rates. (See 7.1).

FEED - When entering blocks which define axis movement (such as linear, circular, repeat and cancel) the FEED key can only be used once an axis has been defined.

If the FEED key is used outside of block input in manual (MDI) mode the feedrate scale value can be altered, the range of feed scale can be from 0.1 to 6500%. The feedrate scale value is a percentage scale value which increases or decreases the feed specified within a program. If for example the feed scale value was set to 200% and the feed within a program was 30%, this would adjust the feed at run time to 60% feed. The feed scale value is also adjusted when in single step mode and the feedrate is overridden as described in the feedrate override section. The feedrate scale value is reset to 100% when a delete all is executed within EDIT or a new program is loaded via CASSETTE or RS232.

X, Y, Z, d- These keys are used to define the axis being input.

MIRROR - This allows programs to be mirror imaged in either X or Y axes. (See 7.3).

LABEL/GOTO - This allows Label or Goto label blocks to be entered in a program. (See 7.15 and 7.18).

FLOAT DATUM - Used to set a user defined Datum Position specified within the program. (See 7.11).

REPEAT - This allows programs to be repeated with offsets in any of the axes. (See 7.9).

CW/CCW - Used to define direction of arc moves during input of data. (See section 4).

SCALE - This function is used to magnify or reduce the finished item size using the same original program. (See 7.5).

DWELL - Programmable DWELL with 0.01 seconds to 999.99 second range. (See 7.2).

TOOL - Allows various size tools to be defined and incorporated into a program. (See section 8).

G - For input of G codes (see Appendix C).

PROG STOP - Used to halt execution of the program for operator intervention. (See 7.12).

OFFSET - Used to run a program from memory displaced from its normal position according to information keyed in at the keyboard. (See Section 7.7).

TOOL COMP - Used for defining the direction of tool compensation applied. (See section 8.1).

M - For input of M codes (see Appendix C).

E.O.B. - END OF BLOCK - This key allows the programmer to define where the end of the command is, before starting a new command.

ENTER - When data or commands are keyed into the computer they will not be accepted until the operator presses this key.

"DATA" KEYPAD

NUMERALS 0-9 - Allows input of data in decimal format. For details of using these keys refer to the Programming Sections.

"MANUAL" KEYPAD

ARROW KEYS - The machine can be jogged into position by depressing the directional arrow keys, which can be found at the top left of the front panel in the manual section. One press will give one step. If a jog key is held, continuous movement is made. More than one axes may be moved at once. The + or - keys allow the Jog Speed to be increased or decreased. If the + or - keys are used before any directional arrow key is depressed, the speed the specified axes will start at is 6% and 1% of full speed respectively.

+, - KEYS - When the machine is executing in MDI or SINGLE STEP mode, the feed at which the machine is moving at can be overridden, using the + or - keys to increase or decrease the feed respectively. If the feed is altered when in SINGLE STEP mode the feedrate scale (as set by the FEED button in the FUNCTION section) will also be altered.

ON, OFF KEYS - When the machine is operating in MDI or SINGLE STEP mode, or in auto with no program running, the ON, OFF keys can be used to switch contacts 1 and 2. When the machine is in Edit mode the ON, OFF keys can be used to input contact ON, OFF blocks into a program.

DIG KEY - Digitise or Teach Mode, will enable the current table position to be stored in memory. Simply begin loading a program as normal but use the jog keys and the <DIG> key to load a linear block to the current position. Whenever the <DIG> key is pressed the PNC compares its current position with the latest position loaded into its program memory and will load a move of only those positions which differ. This facility can also be used in conjunction with the Alter, Replace and Insert commands.

Digitise can also be used to define the end point and centre for circular moves in the following way. First select direction CW or CCW, then select the type of arc input required endpoint/centre, angle/centre, or endpoint/radius (see programming circle blocks 4.2). Pressing <DIG> will load the present position into the circle end point or circle centre, depending on the circle type selected.

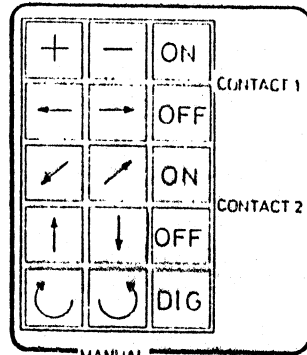
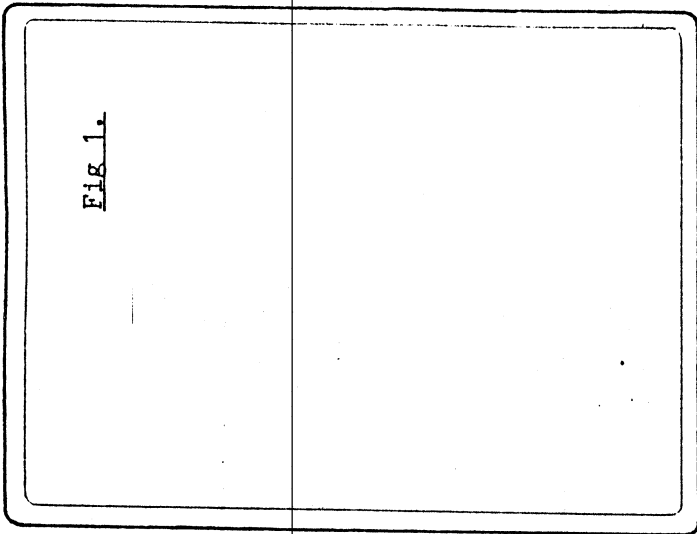
3 Point Digitised Arcs (Option)

Arcs can also be produced entirely by using the digitise key.

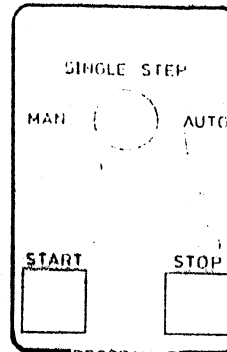
Press the CW/CCW key three times and the block header will change from CIRCULAR CW to CIRCULAR CCW to DIGITISE ARC. When this shows then press ENTER and the block skeleton of an end point centre type arc will be displayed. The jog keys can now be used to position the machine at a mid point along the required arc. If the DIG key is now pressed the present position will be displayed in the block for verification. The jog keys can now be used to position the machine at the arc end point. If the DIG key is now pressed, the PNC 4 will use the two digitised positions to calculate the centre point. The block displayed will change to show the resulting end point centre arc. The block can now be accepted in the normal way by pressing EOB.

If the PNC 4 is unable to calculate the required arc then instead of an end point centre block being displayed, a linear block is displayed showing the digitised end point. Arcs digitised in this way must be less than 360°.

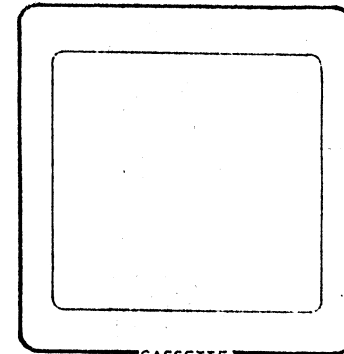
FIG 1.



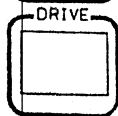
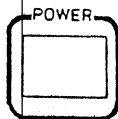
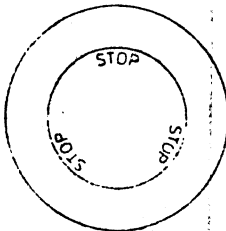
MANUAL



PROGRAMME

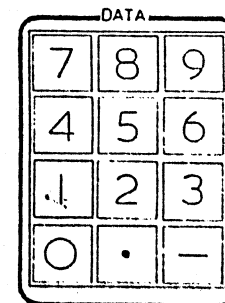
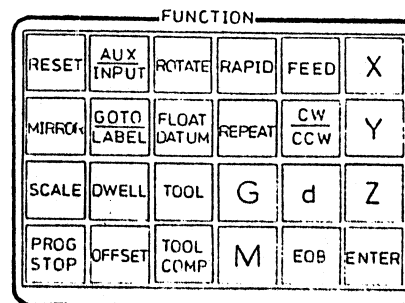
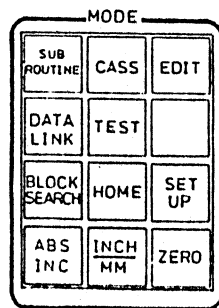


CASSETTE



PNC 4

north east electronics ltd.



1.6 NATURAL STATUS CONDITIONS

It is important to appreciate that when the PNC system is started up, it not only finds datum position as described in Section 1.1, it also sets itself internally to certain Natural Status Conditions, unless commanded otherwise as follows:-

METRIC/IMPERIAL - The system sets itself up according to the switch setting (Interface board DIP Switch 3). ON sets it to metric, OFF sets it to inch mode. The mode can also be switched via the keyboard (see 7.8).

FEEDRATE - Unless instructed otherwise, the system will always apply feed in all Axes at the default feed (see 1.8.4). A command is necessary to change the feed to any other percentage.

ABSOLUTE - The system sets itself for operating in the Absolute mode. This means that positions defined to it will always be interpreted as a distance from the origin. It can, however, be commanded to change to the Incremental mode, and commands will then be interpreted as a distance to be moved from the last defined position.

This is the first time these two modes have been mentioned, so an example will help. Let us say that from a previous command the system was positioned 10mm from the origin in the X Axis. A command is now given to move X 10mm. In its Natural Status the system would remain stationary because, it is already at the command point of 10mm with respect to the origin and it is interpreting in Absolute mode. On the other hand, if the system had been changed over to the Incremental mode, then it would add the newly introduced 10mm to the 10mm already given to it, and would take up a position 20mm from the Origin.

LINEAR MOVEMENTS - Any command defining single movements in the X or Y Axes will normally result in straight line motions along these axes. It is possible, however, to use instructions to generate circular and diagonal movements by combining the two axes.

WHEN THE MACHINE IS SWITCHED ON REMEMBER IT IS IN THESE NATURAL STATUS CONDITIONS, AND, AS INSTRUCTIONS ARE INTRODUCED THE ABOVE RESPONSES WILL BE EXPERIENCED UNLESS SPECIFIC COMMANDS ARE GIVEN TO THE CONTRARY.

1.7 SET UP SYSTEM PARAMETERS

The PNC 4 keeps a set of system parameters in its battery backed RAM. These parameters define such things as number of axes, limits and step sizes. To enable these parameters to be defined first make sure that switch A6 on the interface board is ON (see Appendix B). This enables new system parameters to be stored in the battery backed RAM. With switch A6 ON all other PNC 4 ~~behaviour is normal so that the machine can be thoroughly exercised until the user is certain that the system parameters are correct.~~ Then it is advisable to set switch A6 OFF to prevent any of the parameters being accidentally altered.

The procedure to alter or display the system parameters is as follows:

1. Press the SET UP key to obtain the Machine Set Up menu.
2. Select option 1 EDIT (the other options will be explained later). The PNC 4 will now display a list of all the parameters that can be altered. Select the option required and alter as necessary (see 1.8).
3. Use the RESET key to return to the main menu.
4. Now select option 4 to exit the set up procedure. Remember that switch A6 must be ON otherwise the new values will be ignored.

Select option 2 CASSETTE, allows the system parameters to be stored onto cassette or loaded from cassette. (See section 3 for cassette operation).

Select option 3 Set DEFAULTS, loads a set of default parameters into the system. These default values leave the PNC 4 in a safe state with no movement possible (see 1.8.10 for a complete list).

1.8 SYSTEM PARAMETERS

1.8.1 AXES

Select this option and then enter the number of axes 1 to 4. At this point the PNC 4 will also set the datum order to the default. The display will then show a page of variables for the 1st axis.

Use the → key to move onto the next axis, and the ← key to move back to the previous axis. Use the ↑ and ↓ keys to move the cursor around in the current page. Note that the prompt changes as the cursor moves from one item to another. Use the numeric keypad to alter any values, then press RESET to return to the main Set Up menu.

The values that can be altered are:

- NAME: axis name X, Y, Z or d.
- UNITS: m.m., inch, degrees or mils can be selected per axis. If m.m. or inch is selected then metric/imperial conversion will be possible for that axis.
- STEPS/UNIT: the number of motor steps per unit of travel, e.g. 80 would indicate a step size of $1/80 = 0.0125$ units per step.

+VE LIMIT: the maximum number of units of travel. (Multiply this by the steps/unit to give the total travel in motor steps).

ROTARY: Yes or No. If the axis is defined as rotary then the PNC 4 will assume that its +ve limit is the same as 0 (e.g. 360 degrees is the same as 0).

FIXED DATUM: Yes or No. If fixed datum is set to no, then whenever this axis is sent to datum it will simply take its current position as datum. If set to yes then it will actively seek datum detectors.

DATUM AT +VE LIMIT: Yes or No. If yes then the datum position will be the same as the +ve limit. If no then it will be 0.

1.8.2 DATUM ORDER

This option allows the user to specify the order in which the axes seek datum. E.g. in a milling machine application, it would be normal to make the Z axis move to datum first so that the spindle is in a known safe position before the X and Y axes move to datum. Press RESET to return to main menu.

1.8.3 ACCEL/DECEL RAMP

This option allows the acceleration/deceleration distance to be set. The screen will display a menu of possible settings. Enter a value 1 to 13 to select the required setting. The value selected indicates the total number of motor steps required to accelerate to 100% speed. E.g. a value of 800 means that a 400 step motor will require two full revolutions to reach 100% speed. Press RESET to return to main menu.

1.8.4 FEEDRATE

This option allows the various parameters of the machine feedrate to be set up.

The units value can be set to percentage or units/min. If set to units/min then the units will depend on how the machine is set up. If the first two axes are set to m.m, then this value will represent m.m/min. If the first two axes are set to inches then this value will represent inch/min. The other option is to set the value to percentage where feedrates are defined as a percentage of full speed.

The maximum displayed value will depend on the units selection. If percentage was selected then this value would obviously be set to 100. If units/min was selected then this value would be set to a figure which matched the top speed of the machine.

The default feedrate is the feedrate at which the PNC4 will assume unless otherwise defined. On a high speed positioning machine it would be sensible to set this value to the maximum. All machine movement would be at this value without a feed being specified. On a milling machine this value should be set to a value which is very slow (i.e. 15%), so that if the operator should forget to specify a feedrate, then the machine will not make any sudden high speed moves.

1.8.5 AUXILIARIES

This option allows the user to set up the usage of the auxiliaries 1 to 12. Use the ↓ and ↑ keys to position the cursor against the auxiliary to be changed then use the numeric keypad to set that auxiliary to be on, off, pulsed or momentary. Pulsed auxiliaries close for a certain time (pulsewidth) at the end of each block if they are programmed to be on. ~~Momentary auxiliaries remain closed whilst no axes are moving but they will open just before any axis starts to move.~~

The pulsewidth of the pulsed auxiliaries can be altered by positioning the cursor over the pulsewidth display and entering a new value. The value entered is in milliseconds and can be in the range 1 to 9999. Press RESET to return to main menu.

1.8.6 NAME CONTACTS

This option allows the user to define the names of contacts 1 and 2. Each name can have up to 8 characters. Use the ← and → keys to position the cursor over each character in the name and then use the + and - keys (situated above the direction keys) to alter the character. Each character position can be a space, a digit 0 - 9, or a letter A - Z.

The name for contact 1 will appear in the top display and also whenever an M08 or M09 block is used. The name for contact 2 will appear in the top display and also whenever an M33 or M34 block is used. Press RESET to return to main menu.

1.8.7 SPECIAL FEATURES

When selected this displays another menu from which the following features can be selected:-

1. STOP-START TYPE
2. REPEAT TYPE
3. RS232 BAUD RATE

Press RESET to return to main menu.

1.8.7.1 STOP-START TYPE

This displays the current setting of the stop/start facility and allows the user to change the current setting. Two choices are available. Use the numeric keypad to select the method required.

If restart at same block is selected and the user stops a program in auto using the front panel STOP key or the external STOP, then when START is pressed again the program will restart from the same program block. This facility is subject to the conditions layed out in 2.6.

If restart from block 1 is selected and the user stops a program in auto using the front panel STOP key or the external STOP, then when START is pressed again the program will restart at block 1.

Press RESET to return to the selectable features menu.

1.8.7.2 REPEATS TYPE

This displays the current type of repeat block and also allows the user to change the current type. Two choices are available. Use the numeric keypad to select the required type.

Repeat-divide allows the total repeat offset to be divided by the number of repeats. E.g. to drill 7 equal spaced holes in a distance of 100 mm along the X axis, the repeat block would be something like.

```
G81 Repeat-divide
FROM start TO end 6 TIMES
OFFSET X 100.00 Y 0.0
```

Ordinary Repeats require the offset for each time round the loop to be defined. E.g. to drill 7 holes spaced 14.29 mm apart.

```
G81 Repeat
FROM start TO end 6 TIMES
OFFSET X 14.29 Y 0.0
```

Press RESET to return to main menu.

See section 7.9 for full explanation of Repeats.

1.8.7.3 RS232 BAUD RATE

This displays the current default baudrate. Use the numeric keypad to alter the baudrate. This default baudrate is always assumed at power up but the actual baudrate used can be changed when the DATA LINK key has been pressed. Press RESET to return to main menu.

1.8.8 D TO A SETTINGS (Option)

This option allows the user to define the name of the digital to analogue converter output, the time taken for the D to A to ramp up from 0 to full output, the M03 ident, M04 ident, whether programmable or not and the maximum value displayed.

On selecting this option the cursor will be on the first entry and the user is prompted to "ALTER NAME". Using the same method as 'NAME CONTACTS' (1.8.6) the name of the D to A can be changed.

The manual keys can be used to move the cursor to the next item and the user will be prompted to enter the ramp time in the range 0 to 99 seconds.

The next 2 items allow the user to name the idents for the M03 and M04 blocks in the same way as 'NAME CONTACTS' (1.8.6) with the exception that only 3 characters are allowed. If the D to A were for instance to be controlling a spindle motor, then it would be sensible to name M03 as FWD and M04 as REV.

The next item controls whether the D to A option is fitted or not. If fitted then 'PROGRAMMABLE' would be set to YES otherwise NO.

The next item controls the value the PNC4 will display when the D to A is at full output. If the D to A was controlling a spindle motor capable of 3000 rpm then it would be sensible to set this value to 3000.

Press <RESET> to return to the main setup menu.

1.8.10 DEFAULTS (main menu)

If you select option 3. Set Defaults from the main menu, then the following values will be set.

Two axes named X and Y.

Each axes will be m.m., 80 steps/unit, 0.01 mm +ve limit, not rotary, with fixed datum = no, and +ve limits = no.

The datum order will be axis 1 first and then axis 2.

The accel/decel ramp will be set to its maximum value.

Default feedrate will be 15%.

All the auxiliaries will be defined as on.off and the pulse width will be set to 50mS.

The contacts will be named as CONTACT1 and CONTACT2.

The Repeats type will be Repeat.

The data link baudrate will be 9600.

The D to A name will be ANALOGUE.

The D to A ramp time will be 99 seconds.

2.1 PRINCIPALS OF OPERATING AND PROGRAMMING

This Section and Section 2.2 are included here to help the user in familiarisation with command procedures before starting up the PNC 4 system. It is well to grasp the principles of programming, because whether operating the machine under manual control or composing a full string of sequences (for what we usually mean by 'program') the procedures are the same. Manual commands (outside a program) are entered and then immediately executed whereas a series of commands, strung together so they operate one after the other when run, is a program.

There is one simple rule, and once this is understood then there should be no difficulty at all in progressing to even the most complex programs.

A program step or BLOCK consists of one G code or one M code only.

2.2 ENTERING A BLOCK

The commands in each Block must be given in a specific order, bearing in mind that certain commands can be skipped if they relate to information that is not to be changed.

Each command must be separately keyed in at the Keyboard and then 'accepted' into the system when the 'ENTER' key is pressed. To indicate the End Of a Block of information the E.O.B. key must be pressed. The first group of commands will automatically be denoted by the system as Block 1, the second as Block 2, and so on. No entry is necessary for this numbering; the system does it.

The commands for each positioning type Block must deal with the following and in this stated sequence:-

X Axis position: Y Axis position: X/Y Axis feedrate:
Z Axis position and feedrate: d axis position and
feedrate:

Before proceeding any further it would be well to detail exactly how the sequence is entered. Commands are keyed in at the keyboard prior to acceptance into the system and the information entered appears on the display screen for checking. If an error is made, pressing the RESET Button will delete the entry and allow it to be keyed in again. Repeated pressing of RESET will delete previous layers of command that have not been accepted with an EOB.

E.g. The actual method of entry of a G01 linear block is:-

At the Function Pad of the Keyboard press the X Button. This says you are 'talking to the X Axis'. Now at the Data Pad, key in the position in this Axis that is required. Then press the ENTER Button.

Next press the Y Button and key in the position for the Y Axis, followed by depression of the ENTER Button (the ENTER Button MUST be pressed after each statement).

Now press the FEED Button then key in the feedrate required for travel in the X/Y Axes, then press the ENTER Button.

Press the Z Button and key in the position for the Z Axis, then press the ENTER Button.

Press the FEED Button and key in the Feedrate required during this Z Axis movement, then press the ENTER Button.

~~Finally, press the E.O.B. Button to signify the end of the sequence. The PNC 4 will then move all axes to the defined positions.~~

Any practical program will of course consist of a number of Blocks. In composing the program it is simply a matter of entering one Block after another using this same procedure, each time instructing the machine to go from one position to another using specified feedrates.

In practice it will not be necessary to change positions in all axes for each block, nor the feedrates. If the entry sequence requires a command which will stay exactly as it was in the previous block, simply ignore that entry. For example, if no change is to be made in the X/Y axes feedrate from the previous block, then ignore this entry and move straight onto the Z axis input by pressing the Z button (assuming there is a change in the Z axis position).

It is only necessary to enter changes from the previous Block when compiling the next Block.

Only the rudiments of programming have been dealt with so far. The user should not be tempted at this stage to rush to the machine and try operational programming. It is advisable however, to get the 'feel' of the system by testing the response to a few simple commands before moving on to programming properly.

2.3 RUNNING NON-PROGRAM COMMANDS

The PNC 4 can be commanded to make the machine respond under operator control. This can be done in manual mode without needing to compose a program. In fact, even if there is already a program in the system memory, any commands used in this way will not affect that program.

The operator can reposition X and Y axes for instance, or try out feedrates with a selected Spindle Speed in order to make cutting tests on a particular material before starting to compile a program for machining it.

A typical sequence to move the X and Y axes in this way would be:-

Press the X Button, key in the X axis position required, then press the ENTER Button.

Press the Y Button, key in the Y axis position required, then press the ENTER Button.

Press the EOB Button to indicate the End Of the Block.

The X and Y axes will now move to the commanded position.

From the information given in Section 2.2 there should be no difficulty in implementing various commands in this way for the purpose of familiarisation.

The user is advised to spend some time on this familiarisation exercise until there is a full appreciation of how to control the various movements on the machine and how to compile basic command Blocks. Practical programing, which is dealt with next, will then present no difficulties. Progress should then quickly be made through to full utilisation of the PNC 4 System capabilities.

2.4 A SIMPLE PRACTICAL PROGRAM

Practical programing can now be considered. At the outset it is strongly recommended the user obtains a supply of blank Programing Sheets similar to the one reproduced in Fig. 2 at the end of this Section. This allows a program to be written out and checked over before committing it to the PNC 4 memory. This way mistakes can be avoided. One master blank sheet can be drawn and then duplicated as required. Individual Users may prefer a different detailed layout to suit their own requirements.

The program in fig.2 is for cutting the 'L' shape shown in the upper position of Fig. 3A.

Since we shall not instruct the PNC 4 to the contrary, it will automatically operate in the absolute and metric modes (Section 1.6).

In order to define the 'start' point, the first Block of the program must tell the X and Y Axes where to go and at what feedrate. Wherever possible only specify the XY feedrate in the first Block of a program. It is then easy for manual editing to change it throughout the program if a change is required at a later date.

The first two commands in the first Block tell the Machine to move 30mm in the X Axis and, 20mm in the Y Axis. This defines the 'start' point. It is good practice to write in the remaining commands for this block, although this particular shape contains no circular elements so the circular direction, and centres can be ignored.

Next specify the feedrate, which in this case is 100%. The Z Axis position must now be considered together with the Z Axis feedrate.

Although the Z Axis is not required to move at this stage, it is good practice to enter the position and the feedrate of all axes at the start of a program. It is worth emphasising again that until specific commands to the contrary are given, Natural Status Conditions will apply. Always write in values for every function in the first Block on one of these planning sheets - it could avoid ommisions or mistakes.

The second Block brings the Z Axis down to cutting depth. Since no other axes need to be changed in this Block, only 5mm needs to be entered for the Z Axis position.

The Z Axis will remain constant for the next six Blocks.

The third Block specifies the movement for the next position in order to machine the top of the 'L' and the selected feedrate is 20%. This position will be X40, which is the original 30mm plus another 10mm.

Blocks four to eight define the remaining movements, and reference to Fig.3A will enable the sequence to be followed. Finally in the ninth Block the Z Axis is raised back to its Zero Position by instructing it to move to Z0, and as rapidly as possible i.e. 100%.

This completes the 'mapping out' of the program. It can now be entered into the system.

2.5 ENTERING A SIMPLE PROGRAM

To load a program you must enter EDIT mode by pressing the EDIT Button, and then select option 5.

The program can then be entered Block by Block. The keys to press and the sequence of entry for each Block are given below.

Press the E.O.B. Button to end a Block (if Blocks are not ended they will be cancelled by commands in later Blocks and will be lost).

The following notation has been used -

[ED]	represents	EDIT on the MODE KEYS
[E]	represents	Enter on the FUNCTION KEYS
[E.O.B.]	represents	E.O.B. on the FUNCTION KEYS
[F]	represents	FEED on the MODE KEYS
[R]	represents	RESET on the FUNCTION KEYS
[C]	represents	CW/CCW on the FUNCTION KEYS

If a mistake is made while composing a Block press the RESET Button to delete and make the entry again.

[ED] 4 3 (to delete any previous program)
5 (to insert new data)

1. X30[E] Y20[E] F100[E] Z0[E] F20[E] [EOB]
2. Z5[E] [EOB]
3. X40[E] F75[E] [EOB]
4. Y60[E] [EOB]
5. X80[E] [EOB]
6. Y80[E] [EOB]
7. X30[E] [EOB]
8. Y20[E] [EOB]
9. Z0[E] F100[E] [EOB]

[R] [R]

It is not necessary to key in the Block numbers when entering a program. The system computer does this automatically.

The last command ([R] [R]) is not part of the program. It resets out of Insert mode and then resets from EDIT. The BLOCK SEARCH 1 command can then be entered, to run to Block 1.

2.6 RUNNING A PROGRAM

Before progressing on to more complex routines, this simple program can be used to demonstrate further system capabilities.

With the program still in the memory it can be executed (run) in one of two modes, single step, or, sequentially from beginning to end. It is recommended that until there is complete confidence, all new programs should be run first time in the Single Step mode to allow each step to be checked.

Make sure the keyswitch on the front panel is set to SINGLE STEP. Press the START button once and block 1 of the program will be executed. The Motors will move to position in the X and Y axes. Press the START button again and the next block will be executed. This procedure can be repeated to execute all other blocks. Take care! Any further pressing of the START Button will start the execution of the program again!

To run the program in the sequential mode, turn the key switch to AUTO position and press the START button once. The full program will be executed through to the end, whereupon a buzzer will sound. Turn the key switch to MANUAL and press the RESET Button. The program is now ready to run again.

In single step mode, execution of the current program block can be interrupted by pressing the <STOP> button. To restart execution from the same program block, simply press the <START> button.

In auto mode, execution of the current program can be interrupted by pressing the <STOP> button. On PNC 4 controls before version 4.054, the interrupted program can only be restarted from the same block by using the block search facility. If this is not done and the <START> button is pressed, then the program will restart from block 1.

On PNC 4 controls after version 4.053, the interrupted program can be restarted from the same block by pressing the <START> button. However, if after pressing <STOP>, the operator carries out any of the following functions, then the program will restart from block 1 when the <START> button is pressed.

1. Use the cassette functions.
2. Use the edit functions.
3. Use the set up functions.
4. Use data link.
5. Use any tool functions.
6. Use any subroutine functions.
7. Execute a block search.
8. Change a machine scale.
9. Change a machine offset.
10. Execute a tool comp block.

After pressing the <STOP> button, the operator can drive the machine to another position and switch on contacts or auxiliaries if required. When the <START> button is pressed the machine will drive to the stopped position in X and Y, then drive the Z axis to its stopped position. The auxiliaries and contacts will also be restored to their correct state.

2.7 LOOKING AHEAD

When running in single step mode, each time the START button is pressed the PNC 4 takes one program block out of its memory, calculates the required movement if necessary and starts the motors moving. It then waits until that movement is complete. In AUTO the procedure is different. In effect the PNC 4 is taking program blocks out of memory, calculating and at the same time moving the motors. It does this by looking ahead through the program memory, calculating the necessary movements, and putting them into a buffer. As soon as one movement is complete the next can be started with a minimum of delay (200 microseconds).

The buffer can hold up to 8 movement blocks. When the buffer is full the PNC 4 waits until one movement block has finished. Then it looks for the next movement block in the program, performs the calculation and puts that block into the buffer.

In cases where the time to do the movement is short compared with the time to perform the calculation, then the buffer will gradually become empty. This will happen if the program contains lots of short moves and extra calculations are needed because of scaling or cutter radius compensation.

The program may contain blocks which require some action to take place when the axes are in a certain position (e.g. inputs, pulsed or momentary auxiliaries and manual tool change blocks). In this case the PNC 4 will stop looking ahead, wait until the buffer is empty and all axis movement has stopped before carrying out the required action. As soon as the action is complete it will start looking ahead.

When the PNC 4 starts the looking ahead process, (either when the START button is first pressed or after an action as described above) then it looks ahead until the buffer is half full before starting the motors moving. This may be noticeable as a slight pause after the START button is pressed in cases where there are a lot of calculations to be done on each block.

If the PNC 4 is switched from AUTO into S.STEP or MDI then it will actually remain in AUTO until all the moves have been emptied out of the buffer.

2.8 CONTINUOUS CONSTANT FEED MOVEMENT FACILITY

The PNC4 has a facility selectable by SWA 4 on the interface board to make continuous movements at a constant feed.

The maximum continuous feed possible is dependent upon

- i) which option of 8031 processor system fitted
- ii) the length of the current and next movement
- iii) whether pulsed or momentary auxiliaries are operated at the end of the current movement
- iv) the movement angle between the end of the current move and the start of the next move

) One of two options of 8031 processor system can be fitted i.e.

a) Standard speed

b) High Speed (extra cost option)

a) The standard speed 8031 processor system is capable of 2 axis continuous XY movement in a 4 axis system of up to 3000 motor steps per second.

b) The high speed option increases the speed in a) above by 33%

ii) The maximum feed possible between 2 consecutive blocks is dependent upon the movement length in both the current and next block. The current block movement length controls the amount of machine acceleration possible. The acceleration itself is a function of the acceleration parameters built into the PNC4. In general 'light' machines can accelerate more quickly than 'heavy' machines. The next block movement length affects the maximum feed possible in that the PNC4 requires that it must be able to decelerate during this movement to STOP even though in reality continuous movement is to continue after the next block. This fact is a requirement of the PNC4 design because when 90 degree ARC's are being executed one axis has to decelerate to STOP and the other accelerate. During program execution the PNC4 recalculates the maximum feed possible between each consecutive pair of blocks. If the component program comprises of some short movements the machine feed will be slowed at these points.

iii) If pulsed or momentary auxiliaries are operated between consecutive blocks machine movement stops whilst these auxiliaries are operated. For early PNC4's this is also true for ON/OFF auxiliaries.

iv) For continuous constant feed movement the angle of machine movement at the end of the current block must be within 2 degrees to the angle machine movement at the start of the next block.

If machine feed is greater than the maximum continuous feed possible between two consecutive blocks is programmed, machine feed between the two blocks will be decelerated to the maximum value at the junction of the two blocks then accelerated (provided conditions ii) through to iv) above are met).

2.9 BLOCK SEARCH facility

A program in the PNC 4 memory may be run i.e. executed, either from the start of the program i.e. block 1, or from any point in the program, except from within a repeat using the block search facility.

This facility is useful for continuing a program after some interruption e.g. replacing a broken cutter. To use, press the Block Search key and enter the desired block number. This "pre-run" enables the PNC 4 to calculate the correct position and also the status of the auxiliaries and contacts. If 3 or more axes are defined, the PNC 4 will also raise the Z axis. When the correct position and status of the block before the desired block has been found the PNC 4 will move to that position. Note that in the case of a 3 axis machine it will move the correct X,Y position first and then bring the Z axis to the correct depth. It will then execute the desired block.

2.10 PROGRAM VALIDATION (TEST KEY)

If a program is in memory and the PNC 4 has been datumed, then pressing the TEST key will cause the program to be validated. Most simple errors in the program will be detected this way with considerable saving in time as opposed to actually executing the program. Program validation works by executing every block in the program but without actually moving motors or switching relays. The first error found will cause an error message to appear on the screen and the buzzer will sound on pressing reset a message will be displayed to indicate in which block the error occurred.

3.1 CASSETTE FUNCTIONS

Any program entered into the system via the Keyboard can be stored on Cassette whether the program has been run or not.

In fact it is strongly recommended that all programs are recorded immediately after they have been entered. Long programs should be recorded in stages as they are being compiled using a 'working' tape prior to recording on the final 'master' tape.

To enter cassette mode press 'CASS' in the MODE section (the key switch must be set at 'MANUAL') and the Cassette Menu will display as shown below.

CASSETTE

- 1) REWIND
- 2) LOAD PROGRAM
- 3) STORE PROGRAM
- 4) FIND END OF DATA
- 5) CONTINUE PROGRAM LOAD
- 6) SUBROUTINES
- 7) ERASE

SELECT

The Cassette Menu offers a selection of seven Cassette functions, as follows -

Note: If a function is selected with no cassette in position the display will show the message 'NO TAPE LOADED', in which case press the 'RESET' Button and insert a cassette.

3.2 REWIND

Press key '1', rewinding is immediately activated

Display reads: 'REWINDING'

Prompt reads: 'PRESS (STOP) TO ABORT'

Whenever a cassette is inserted, even if it is a new cassette, and certainly always before a cassette is removed, tape rewind should be used.

3.3 LOAD PROGRAM

Press key '2'

- a) Display reads: 'BYTES AVAILABLE nnnnn'.
Prompt reads: 'ENTER PROGRAM NUMBER'.

Select number from DATA section and press 'ENTER'. (E.g. 42)

- b) Display reads: 'SEARCHING FOR PROGRAM NUMBER 42'
Prompt reads: 'PRESS (STOP) TO ABORT'.

Alternatively if the program number is not known, instead of entering a number press 'ENTER' it will then search and display the first program number it finds, press 'ENTER' to load program or press 'CASS' to continue search.

On finding the next or selected program number it will display -

- c) Display reads: 'PRESS (ENTER) TO LOAD PROGRAM'.
Prompt reads: 'PRESS (CASS) TO CONTINUE SEARCH'

On pressing key 'CASS' it will continue search and display will revert to 'b'.

On pressing key 'ENTER' it will display

- d) Display reads: 'READING PROGRAM'
Prompt reads: 'PRESS (STOP) TO ABORT'

On completion of loading it will briefly display for example -

- e) Display reads: 'PROGRAM NO. 42
 BLOCKS LOADED 13
 BYTES AVAILABLE nnnnn'

and then display automatically returns to Cassette Menu.

3.4 STORE PROGRAM

Press key '3'

Prompt reads: 'ENTER PROGRAM NUMBER'.

Select a number consisting of one to six figures and press 'ENTER'.

Prompt reads: 'ENTER NUMBER OF COPIES 1 TO 9, DEFAULT IS 1.'

Select how many recordings you require by selecting, say 3. On pressing '3' in the 'DATA' section followed by 'ENTER' it will store your program three times in succession.

Display reads: 'PROGRAM BEING STORED', however if key 'ENTER' is pressed without selecting a value the program will only be recorded once. On completing the storing operation it will automatically return to Cassette menu.

BEFORE STORING ALWAYS USE FUNCTION 4 TO FIND END OF PREVIOUS PROGRAM DATA THIS WILL ENSURE PROGRAM CONTINUITY WITH NO FALSE DATA ENDS.

3.5 FIND END OF DATA

Press key '4'

Display reads: 'SEARCHING FOR END OF DATA'.

Prompt reads: 'PRESS (STOP) TO ABORT'.

While finding the end of program data it will list any program numbers. When at the end of the program data the prompt reads 'END OF DATA FOUND'. Press 'RESET' once to return to the Cassette Menu to proceed.

Note: When searching for the end of the data, if there is a space between the end of one program and the start of another program, it will display 'End of data Found' (called False Ends) at the end of the first program and any remaining programs could be accidentally erased by over-recording.

3.6 CONTINUE PROGRAM LOAD

Press key '5' and proceed as per cassette function 2 - Load Program. This facility permits a number of short programs to be loaded, one behind the other.

3.7 SUBROUTINES

Press key '6'

- a) Display reads: '1. LOAD SUBROUTINES
'2. LOAD A BATCH OF SUBROUTINES'
Prompt reads: 'SELECT'

On selecting '1'

Prompt will read: ENTER SUBROUTINE I.D. (1 to 255)

after selecting Sub Number press 'ENTER'.

Display reads: 'SEARCHING FOR PROGRAM NUMBER'
Prompts reads: 'PRESS (STOP) TO ABORT'

On finding the sub program

Display reads: 'FOUND PROGRAM NUMBER SUB 1.

Prompt reads: 'PRESS (ENTER) TO LOAD PROGRAM'
'PRESS (CASS) TO CONTINUE SEARCH'

On pressing 'ENTER'.

Display reads: 'READING PROGRAM'

Prompt reads: 'PRESS (STOP) TO ABORT'

On selecting '2' of display an automatic sequence of loading subroutines is triggered. The system will immediately search and load from the first subroutine until it detects a program or end of data. It will then revert to the Cassette Menu.

- b) If some subroutines are already in memory:

Display '3. STORE ALL SUBROUTINES'
'4. STORE SUBROUTINE'

Prompt reads: 'SELECT'.

On selecting 3 the PNC 4 will store all subroutines in its memory onto cassette.

Display reads: 'PROGRAM BEING STORED'

Prompt reads: 'PRESS <STOP> TO ABORT'

On selecting 4

Prompt will read 'ENTER SUBROUTINE ID (1 to 255)'

Enter the subroutine number and press 'ENTER'.

Prompt reads: 'ENTER NUMBER OF COPIES 1 TO 9 DEFAULT IS 1'.

Select the number of times and then press ENTER. The subroutine will then be stored onto cassette.

3.8 ERASE

Press key '7'.

Prompt will read: 'PRESS (ENTER) TO ERASE CASSETTE'

On pressing 'ENTER'

Display reads: 'ERASING'

Prompt reads: 'PRESS (STOP) TO ABORT'

On completion system reverts to Cassette Menu.

Note: If an incorrect entry is made on selecting a cassette facility press 'RESET' it will then revert to Cassette Menu.

MAKE IT A PRACTICE TO ALWAYS REWIND THE TAPE CASSETTE AFTER USE.

4.1 PROGRAMMING DIAGONALS AND INT/EXT. PROFILES

The shape in Fig.3B introduces two new programming elements - diagonal lines, and a 2-part profile consisting of one shape inside another.

Whereas to produce each leg of the 'L' shape in the previous example the program commanded movement in the X and Y Axes separately from each other, ~~commands for diagonals give movements in both these Axes simultaneously~~ between defined start and finish points. The system then automatically computes the path for the Cutter to produce a straight line between these points.

To produce one shape inside another it is merely a matter of programming the inner shape first; raising the cutter; repositioning it at the start point for the external shape; and then programming the latter shape.

The following would be the program for the combined shape, and there should be no difficulty in following it -

```
[ED] 4 3 5
1. X60[E] Y110[E] F100[E] ZO[E] F20[E] [EOB] *(Start Internal)
2. Z5[E] [EOB]
3. Y90[E] F15[E] [EOB]
4. X80[E] [EOB]           The new elements referred to above can be seen to
                           affect those Blocks which have been identified thus
                           ""

5. Y110[E] [EOB]
6. X60[E] [EOB]
7. ZO[E] F100[E] [EOB]
8. X40[E] Y100[E] F100[E] ZO[E] [EOB] *(Start External)
9. Z5[E] F20[E] [EOB]
10. X70[E] Y130[E] F15[E] *
11. X100[E] Y100[E] [EOB] * Note that reposition to the start point for cutting
                           external profile, which is commanded in Block 8, can
                           be programmed to take place at maximum feed. In fact
                           any positioning movements (as distinct from cutting
                           movements) can be at maximum feed. This can minimize
                           non-cutting time, especially with long movements.

12. X70[E] Y70[E] [EOB] *
13. X40[E] Y100[E] [EOB] *
14. ZO[E] F100[E] [EOB] *
    [R] [R]
```

4.2 PROGRAMMING CIRCULAR ELEMENTS

The outline produced by program 7 (Appendix D) shows a simple shape consisting of an internal and an external profile which contain arc movements. The programming of such profiles can now be dealt with and should present no difficulties in view of what has been learned so far.

Programming circle blocks

There are three types of circle programming available on the PNC 4, namely endpoint/centre arcs, angle/centre arcs and endpoint/radius arcs. To program an arc block, press the <CW/CCW> key until the desired direction (clockwise or anti-clockwise) is obtained, then press <ENTER>. The outline of an endpoint/centre arc block is then displayed. If the <CW/CCW> key is pressed again, the outline of an angle/centre arc block is displayed. Another press displays the outline of an endpoint/radius arc block and another press still takes the display back to the endpoint/centre arc. To select the type of arc required, just press the <ENTER> key when the appropriate display appears.

4.2.1 Endpoint/centre arcs

a. Keying-in

The X, Y, Z, d coordinates and their feeds may be entered in the same way as for a linear block i.e. select the axis, key in a value and press <ENTER>. To program the X and Y centre coordinates, first press the <CW/CCW> key. The prompt now changes to 'ENTER CIRCLE CENTRE'. The X and Y keys now select the input of the X and Y centre coordinates respectively. When the centre coordinates have been keyed in (or completed by PNC 4 - see (b) below), press <RESET> to resume normal operation of the X and Y keys.

b. Calculation of missing coordinates

Four parameters are required for endpoint/centre arcs i.e. X endpoint coordinate, Y endpoint coordinate, X centre coordinate and Y centre coordinate. However, when any three of these parameters have been supplied, the PNC 4 attempts to calculate the remaining parameter. If it is successful, the remaining parameter is displayed in the block and it will not need to be keyed in by the user (although it can of course, be changed by simply keying in a new value). The PNC 4 will not be able to supply the remaining parameter if it finds that there are no solutions or an infinite number of solutions.

c. Validation

When <EOB> is pressed:-

- i. If the X, Y, X centre and Y centre parameters are not all provided, an 'INCOMPLETE BLOCK' error results.
- ii. If the centre point is the same as either the start point or the end point, an 'ERROR IN CIRCLE' condition arises.

To clear the above errors, press <RESET> and correct the errors.

4.2.2 Angle/centre arcs

a. Keying-in

The Z and d coordinates and their feeds may be entered in the same way as for a linear move. The X and Y keys, however, immediately select input of the X and Y centre coordinates. The <CW/CCW> key selects input of the angle which may be in the range 0.01 to 360 degrees. The XY feed is selected by pressing <FEED> after keying in the angle.

b. Validation

Angle input will be ignored if it is not in the range 0.01 to 360 degrees.

When <EOB> is pressed:-

- i. If the angle, X centre and Y centre parameters are not all provided, an 'INCOMPLETE BLOCK' error results.
- ii. If the centre point is the same as the start point, an 'ERROR IN CIRCLE' condition arises.

To clear the above errors, press <RESET> and correct the error.

4.2.3 Endpoint/radius arcs

a. Keying-in

The X, Y, Z, d coordinates and their feeds may be entered in the same way as for a linear block. To select input of the radius, press the <CW/CCW> key. The value keyed in for the radius must always be greater than zero.

b. Validation

Radius input will be ignored if it is not greater than zero.

When <EOB> is pressed:-

- i. If the X, Y and radius parameters are not all provided, an 'INCOMPLETE BLOCK' error results.
- ii. If the end point is the same as the start point, an 'ERROR IN CIRCLE' condition arises.

To clear the above errors, press <RESET> and correct the error.

4.3 Execution of circle blocks

4.3.1 Validation

An 'ERROR IN CIRCLE' condition arises if :-

- the centre point is the same as either the start point or the end point (for endpoint/centre arcs).

- the centre point is the same as the start point (for angle/centre arcs).

- the end point is the same as the start point (for endpoint/radius arcs).

4.3.2 Full circular movement

Full circular movement will be performed when -

- the end point is the same as the start point (for endpoint/centre arcs).

- the requested angle of movement is 360° (for angle/centre arcs).

Note that full circular movements are not possible with endpoint/radius arcs.

4.3.3 Breaking-up of circles

When executing arc movements, angle/centre and endpoint/radius arcs are first converted to their endpoint/centre equivalent. The arc is then split into sections each of which is entirely contained within a quadrant. This means that circles could be executed in several distinct movements.

e.g. try running the program below.

```
[ED] 4 3 5
1. X10 [E] Y10 [E] [EOB]
2. [C] [E] [C] [E] [C] 350 [E] X15 [E] Y12 [E] [EOB]
[R] [R]
```

It will be seen that the 350° arc in block 2 is split into five separate smaller arcs of 21.8° , 90° , 90° , 90° , and 58.2° , each of which is executed separately. However, if the continuous constant feed movement facility (see 2.8) is used, the complete arc will be profiled in a single continuous movement.

4.3.4 Re-calculation of centre point

When the PNC 4 executes an arc movement, it first calculates the radius of the arc using the start position and nominal centre point and uses it to adjust the centre point such that the arc end point will always be reached.

e.g. try the following program

```
[ED] 4 3 5
1. XO [E] YO [E] [EOB]
2. [C] [E] [E] X10 [E] Y5 [E] [C] X10 [E] YO [E] [EOB]
```

Although the centre was given as (10,0) the PNC 4 will recalculate it to be (8.71,-4.92) thereby profiling an arc of radius 10.

4.4 Examples of circular elements

The profile of program 7 has been programmed using each of the three types of arc described above. The best type to use for a particular application is dependant on the context in which the arc is being used. However, in general, endpoint/radius arcs are not suitable for arc movements of more than 180° , and ~~angle/centre arcs are best suited to full circular movements (i.e. 360°) as~~ they are unaffected by changes to the start point.

4.4.1 Endpoint/centre arcs

Program 7 uses endpoint/centre arcs to produce the profile. It is shown below together with some explanatory notes.

[ED] 4 3 5

1. X20 [E] Y20 [E] F100 [E] [EOB]
2. Z0 [E] F20 [E] [EOB]
3. X50 [E] F20 [E] [EOB]
4. [C] [C] [E] [E] X50 [E] Y60 [E]
[C] X50 [E] [EOB]
5. X20 [E] [EOB]
6. Y20 [E] [EOB]
7. Z10 [E] F100 [E] [EOB]
8. X30 [E] Y40 [E] F100 [E] [EOB]
9. Z0 [E] F20 [E] [EOB]
10. [C] [E] [E] X30 [E] Y40 [E]
[C] X40 [E] Y40 [E] [R]
F20 [E] [EOB]
11. Z100 [E] F100 [E] [EOB]
12. X0 [E] Y0 [E] F100 [E] [EOB]

Block

Summary

- 1 Drive to initial position before cutting commences.
- 2 Drive cutter down to workpiece.
- 3 Cut along bottom of external profile.

- 4 Cut around arc.
- Note:-
- i) [C] [C] [E] selects anti-clockwise arc.
 - ii) Next [E] selects endpoint/centre radius.
 - iii) The Y centre coordinate is automatically calculated.
- 5 Cut along top of external profile.
- 6 Cut down side of external profile.
- 7 Lift cutter.
- 8 Start position for internal position.
- 9 Drive cutter down to workpiece.
- 10 Cut around internal profile:-
- i) [C] [E] selects clockwise arc.
 - ii) Next [E] selects end point/centre radius.
 - iii) The Y centre coordinate is not automatically calculated because there are no infinite number of solutions.
- 11 [R] returns to normal if XY coordinate input.

4.4.2 Angle/centre arcs

Program 8 uses angle/centre arcs to produce the profile. It is shown below together with some explanatory notes.

[ED] 4 3 5

1. X20 [E] Y20 [E] F100 [E] [EOB] - Drive to initial position before cutting commences.
2. Z0 [E] F20 [E] [EOB]
3. X50 [E] F20 [E] [EOB]
4. [C] [C] [E] [C] [E] [C] 180 [E]
X50 [E] Y40 [E] [EOB]
5. X20 [E] [EOB]
6. Y20 [E] [EOB]
7. Z10 [E] F100 [E] [EOB]
8. X30 [E] Y40 [E] F100 [E] [EOB]


```

9. Z0 [E] F20 [E] [EOB]
10. [C] [E] [C] [E] [C] 360 [E]
    F20 [E] X40 [E] Y40 [E] [EOB]
11. Z100 [E] F100 [E] [EOB]
12. X0 [E] Y0 [E] F100 [E] [EOB]

```

<u>Block</u>	<u>Remarks</u>
1	Drive to initial position before cutting commences.
2	Drive cutter down to workpiece.
3	Cut along bottom of external profile.
4	Cut around arc.
	Note:-
	i) [C] [C] [E] selects anti-clockwise arc.
	ii) Next [C] [E] selects angle/centre arc.
	iii) Next [C] selects angle.
5	Cut along top of external profile.
6	Cut down side of external profile.
7	Lift cutter.
8	Start position for internal profile.
9	Drive cutter down to workpiece.
10	Cut around internal profile.

Note:-

- i) [C] [E] selects clockwise arc.
- ii) Next [C] [E] selects angle/centre arc.
- iii) Next [C] selects angle.

4.4.3 Endpoint/radius arcs

Program 9 uses endpoint/radius arcs to produce the profile. It is shown below together with some explanatory notes.

[ED] 4 3 5

```

1. X20 [E] Y20 [E] F100 [E] [EOB]
2. Z0 [E] F20 [E] [EOB]

```

3. X50 [E] F20 [E] [EOB]
 4. [C] [C] [E] [C] [C] [E]
 X50 [E] Y60 [E] [C] 20 [E] [EOB]
 5. X20 [E] [EOB]
 6. Y20 [E] [EOB]

7. Z10 [E] F100 [E] [EOB]
 8. X30 [E] Y40 [E] F100 [E] [EOB]
 9. Z0 [E] F20 [E] [EOB]
 10. [C] [E] [C] [C] [E] X50 [E] Y40 [E]
 F20 [E] [C] 10 [E] [EOB]
 11. [C] [E] [C] [C] [E] X30 [E] Y40 [E]
 [C] 10 [E] [EOB]
 12. Z100 [E] F100 [E] [EOB]
 13. X0 [E] Y0 [E] F100 [E] [EOB] - Return to origin.

<u>Block</u>	<u>Remarks</u>
1	Drive to initial position before cutter commences.
2	Drive cutter down to workpiece.
3	Cut along bottom of external profile.
4	Cut around arc.
	Note:-
	i) [C] [C] [E] selects anti-clockwise arc.
	ii) Next [C] [C] [E] selects endpoint/radius arc.
	iii) Next [C] selects radius.
5	Cut along top of external profile.
6	Cut down side of external profile.
7	Lift cutter.
8	Start position for internal profile.
9	Drive cutter down to workpiece.
10	Cut around top of internal profile.

- i) [C] [E] selects clockwise arc.
 - ii) Next [C] [C] [E] selects endpoint/radius arc.
 - iii) Next [C] selects radius.
- 11 Cut around bottom of internal profile.
-
- 12 Lift cutter.
- 13 Return to origin.

4.5 WORKING AND PROGRAMMING IN THE INCREMENTAL MODE

The difference between the Absolute and Incremental modes of operation has been dealt with earlier. Absolute is when the destination points defined by commands for movements of the system are every time expressed as a distance from the Datum Point. Incremental is when each position is defined as movements from the previous position.

The system is always in the Absolute mode unless commanded otherwise. To operate in the Incremental mode, an incremental block must be stored into the the start of a program, thereafter all co-ordinates programed will be incremental until the command is cancelled by inserting an absolute block.

Apart from statements to control movement distances, the principles of direct control or programming are the same as when working in the Absolute mode, with one important exception.

In the Absolute mode, commands take the axes from one point to another, but normally the points are always the same side of the origin. Therefore the system accepts all commands as being positive because all points are the '+' side of the origin in all axes. Exceptions are when Datum Offset or Floating Datum commands have been given, and these special cases will be dealt with in later Sections.

In the Incremental mode, the last commanded point becomes in effect the datum point for movement to the next. Thus the system needs to be told the direction in which to move. Left or right, forward or backward, from the last point.

This is done by making positive (+) commands for movements away from the system origin, or negative (-) commands for movements toward the origin.

Unless commanded to the contrary, the system will interpret all comands as positive. Thus only negative commands needs special input and this is done by pressing the '-' Button on the Data Keyboard before every distance value. The display will then be shown 'X-33.5' for example.

To revert the system to the Absolute mode after operating in the Incremental mode, an incremental block must be inserted. If the system is switched off while still in Incremental it will automatically adopt the Absolute mode when switched on again (Natural Status Conditions - Section 1.6)

Taking now the program for the 'L' shape in Fig.3, it can be re-written for Incremental working as shown below. Compare this with the program written in the Absolute mode (Section-2.5).

[ED] [E]

1. X30[E] Y20[E] F100[E] Z0[E] F20[E] [EOB]
2. INCREMENTAL UNITS
3. Z5[E] [EOB]
4. X10[E] F20[E] [EOB]
5. Y40[E] [EOB]
6. X40[E] [EOB]
7. Y20[E] [EOB]
8. X-50[E] [EOB]
9. Y-60 [E] [EOB]
10. Z-5[E] F100[E] [EOB]

4.6 Circular Programing in Incremental Mode

When programing circular movements in the Incremental mode it should be remembered that the arc centre must each time be defined with reference to the last defined point. In the Absolute mode, it will be recalled, the centre point was defined with reference to the origin in programs 7 to 9. The difference can be illustrated by rewriting program 7 this time in incremental mode. (See program 9a).

1. X20[E] Y20[E] F100[E] Z100[E] F100[E] [EOB]
2. INCREMENTAL UNITS
3. Z-100[E] F20[E] [EOB]
4. X30[E] F20[E] [EOB]
5. [C] [C] [E] [E] X0[E] Y40[E] [C] X0[E] Y20[E] [EOB]
6. X-30[E] [EOB]
7. Y-40[E] [EOB]
8. Z10[E] F100[E] [EOB]
9. X10[E] Y20[E] F100[E] [EOB]
10. Z-10[E] F20[E] [EOB]
11. [C] [E] [E] X0[E] Y0[E] [C] X10[E] Y0[E] [EOB]
12. Z100[E] F100[E] [EOB]
13. X-30[E] Y-40[E] F100[E] [EOB]

5.1 Edit Facility

Programs can be edited (amended) while in the system memory. There are four edit functions - Insert, Delete, Replace and Alter - and these are applied to the program blocks as required.

To access the EDITOR use the EDIT key which is located on the MODE key Pad.

The screen will display the 1st line of program at the bottom - this is known as the ACTION line.

A menu with the following selections will be located at the bottom of the screen.

1. Previous 2. Next 3. Replace 4. Delete
5. Insert 6. Alter 7. Search

Selection of 2 will increment the program forward and can continue to be pressed until the required Block of information is displayed.

Similarly, selection of key 1 will step the program back a Block at a time.

If you wish to move to a Block well into the program, the selection of 7 will allow you to enter the Block number required.

Once in the Edit mode, the four possible functions can be used as required according to the following explanations -

Insert Select 5 and this will allow you to add new Blocks in front of the line which was displayed on the action line. When this line is input use the same procedure as outlined in section 2.2 and end each statement with the E.O.B. key. Press RESET to return to the EDIT menu.

Delete Select 4 and this will give 3 options 1) deleting the line shown on the action line or 2) deletion of a batch of lines by specifying the start block and the end block. Entering these selections will automatically remove them from memory. All remaining Blocks will be automatically renumbered or 3) provides a convenient way of deleting the entire program (it should be noted that on selecting this option machine offsets, scales, feed scales and offsets are all reset).

Replace Locate the Block to be replaced and select 3 from the menu enter the revised Block in full and key in [E], [EOB]. The new Block will replace the old one.

Alter Select 6 and this will give you the opportunity of changing one or more of the parameters within the Block of information, without "Replacing" the full line. To confirm the change press [E] and to end the command press [EOB].

Always after using Insert, Delete, Replace or Alter, check that the changes have been carried out.

E.g.

G00X0Y0
G00X25Y56

M08
G01X100FX30
G01Y100
G01X50
G01Y56
G01X75
M09
G00X0Y0

6.1 DATA LINK

This facility permits the PNC 4 to receive programs transmitted direct via the RS232 interface link from any other host computer with RS232 capability, and appropriate software.

The Data Link menu also provides for the transfer of program data from the PNC 4 to a suitable interfaced printer such as the Epson RX80, this is available as an accessory.

This is the Data Link menu, obtained by pressing the key marked "DATA LINK" to access.

DATA LINK

- 1) CHANGE BAUD RATE
- 2) LOAD PROGRAM
- 3) CONTINUE LOADING PROGRAM
- 4) STORE PROGRAM TO DATA LINK
- 5) PRINT PROGRAM

- 1) CHANGE BAUD RATE - Press key "1" display reads

DATA LINK

CURRENT BAUD RATE = 4800

- 1) 300 2) 1200 3) 4800 4) 9600

The BAUD RATE will default depending on the system parameters (see 1.8) Other BAUD RATES are for aligning with other external CAD/CAM terminals that have different communication rates.

- 2) LOAD PROGRAM - Press key "2"

The display will then show a menu of 2 choices:-

1. HOST COMPUTER
2. PAPER TAPE

Select 1 or 2 - the difference is explained later.

Enter program number followed by pressing the "ENTER".

Display will then change to read -

DATA LINK

LOADING FROM RS232c SERIAL INTERFACE DATA LINK

BAUD RATE = 4800
STOP BITS = 2

DATA BITS = 7
PARITY = EVEN

On completion the display will confirm loading by displaying program number, blocks loaded and bytes available.

Display will then revert to the full DATA LINK menu.

3) CONTINUE LOADING PROGRAM - Press key "3"

This allows continuation of programs to be loaded into the PNC 4 within the memory capacity. Method of loading is as previous instruction given.

4) STORE PROGRAM TO DATA LINK - Press key "4".

5) PRINT PROGRAM - Press key "5".

The prompt will display (if not previously established since switching unit on) -

"ENTER DATE" - (this should comprise of six figures) - press key "ENTER". Display will read

PRINTER

- 1) COMPLETE PROGRAM
- 2) PART PROGRAM
- 3) PRINT SUBROUTINE
- 4) PRINT ALL SUBROUTINES

on pressing key "1" prompt reads - "ENTER PROGRAM NUMBER" select and press "ENTER".

Display reads - 'PRINTING PROGRAM'

Prompt reads - 'PRESS (STOP) TO ABORT'

On completion of printing, display will revert to DATA LINK menu.

On pressing key "2" prompt reads - 'ENTER START BLOCK 1 TO 66

From the example, the start block could be nominated as "20" and end at 41.

Press keys "20" press "ENTER"

Press keys "41" press "ENTER"

Display reads - 'PRINTING PROGRAM'

Prompts reads - 'PRESS (STOP) TO ABORT OPERATION'.

On completion display reverts to DATA link menu.

To exit from DATA LINK mode press "RESET".

6.2 Input to PNC 4

The enhanced RS232 interface allows a host computer to use all the facilities of the PNC 4. The enhanced specification includes comprehensive error message transmission. Each block must start with <STX> and end with <CR> but these characters have been omitted from the following text for clarity.

(N.B. STX = 02H, CR = 0DH)

Important: The PNC 4 will only accept data with a 7 bit word length, 2 stop bit and even parity. If a parity error occurs, an error 01 will be sent from the PNC 4.

The PNC 4 ~~only needs 3 wires to implement an RS232 link.~~ They are Data Transmit, Data Receive and Signal Ground. The connection to the PNC 4 is via a 7 pin DIN socket (fitted to the unit rear) using pins 7, 6 and 2 for TxDa, RxDa and Ground respectively. TxDa is the output from the PNC 4 and RxDa is the input to the PNC 4.

All information on the RS232 link is in the form of ASCII characters.

The maximum amount of data transmitted to the PNC 4 in one sequence will be one program block.

General operation is as follows:-

- a) Operator selects serial load from host computer.
- b) Host sends 1 block e.g. <STX>G01X23<CR>
- c) PNC 4 executes that block
- d) Host must wait for a response from PNC 4 (e.g. <ACK>) before sending next block.
- e) Repeat b) c) d) until done.
- f) Host may optionally send <STX>M02L<CR> to cause PNC 4 to exit RS232 mode and return to manual operation.

The following paragraphs define the exact data format required by PNC 4.

The PNC 4 will ignore the following characters:

Null	00H
Tab	09H
Space	20H
Delete	7FH

Also if the first character after the <STX> is N, the PNC 4 will ignore any digits following the N. This is to allow users to number the blocks being transmitted. Note that the numbers are ignored and the PNC 4 will automatically number its blocks during a load sequence.

Blocks with "L" as the last character before the <CR> will not be executed directly but will be loaded into the PNC 4's internal memory.

Rapid Moves

G00 Xnnnn Ynnnn Znnnn dnnnn

Linear Moves

G01 Xnnnn Ynnnn FXnnnn Znnnn FZnnnn dnnnn Fdnnnn

Clockwise circular Moves

End point centre type

G02 Xnnnn Ynnnn FXffff Znnnn FZffff CXcccc Ycccc dnnnn Fdffff

Angle centre type

G02 Aaaaa CXcccc Ycccc FXffff Znnnn FZffff dnnnn Fdffff

End point radius type

G02 Xnnnn Ynnnn FXffff Znnnn FZffff Rrrrrr dnnnn Fdffff

Counter clockwise circular moves

As clockwise except use G03.

nnnn = X,Y, Z or d position in mm or inches
ffff = Feedrate in percentage
cccc = XY circle centre point in mm or inches
aaaa = Arc angle in degrees
rrrr = Arc radius in mm or inches

Dwell

G04 Ddddd

dddd = dwell time in the range 0 to 999.99 seconds

Mirror functions

Mirror X

G10L

Cancel Mirror X

G11L

Mirror Y

G12L

Cancel Mirror Y

G13L

Rotate Functions (Note! Rotate is an extra cost option).

Rotate

G14 Xnnnn Ynnnn Aaaaa

aaaa = angle in range 0.01 to 360.00 degrees.

Cancel Rotate

G15L

Plane Switching

X-Y Plane

G17L

X-Z Plane

G18L

Y-Z Plane

G19L

Note that these blocks cannot be executed directly but must be part of a program load.

Scaling function

Program Scale

G20 SssssL

(NOTE! Only available as part of a program load)

Machine Scale

G21 Sssss

(NOTE! Only available outside program load)

ssss = percentage scaling required in the range 0.1% to 6500%

Cutter Radius Compensation

Cancel Tool Compensation

G40

Tool Compensation Right

G41

Tool Compensation Left

G42

Offsets

Program offsets

G54 Xnnnn Ynnnn Znnnn dnnnn L

(NOTE! Only available as part of a program load)

Machine offsets

G55 Xnnnn Ynnnn Znnnn dnnnn

(NOTE! Only available outside program load)

Imperial Units

G70

Metric units

G71

Repeat loops

(NOTE! Only available as part of a program load)

G81R<start label>E<end label>N<no of times>Xnnnn Ynnnn FXffff Znnnn FZffff dnnnn Fdffff L

Where the X, Y, Z and d dimensions are optional incremental offsets and the FX, FZ and Fd values are optional feedrates.

Canned Cycles (option)

Unconditional re-enable
G79L

Circular Cycle re-enable
G79CL

Drilling Cycle re-enable
G79DL

Rectangular Cycle re-enable
G79RL

Deactivate Cycle
G80L

Circular Cycle
G82 Rrrrrr Xffff Zdddd Ccccc Zzzzz

Drilling Cycle
G83 Zdddd FZfff Ccccc Zzzzz

Rect-lar Cycle
G84 Xxxxx Yyyyy FXfff Zdddd Ccccc Zzzzz L

Notes!

rrrr = radius of pocket
ffff = feed value
dddd = depth of pocket
cccc = number of cuts
zzzz = clearance value
xxxx = X dimension of pocket
yyyy = Y dimension of pocket

Absolute units
G90

Incremental units
G91

Absolute Datum
G98

Floating Datum (NOTE! Only available as part of a program load)
G99L

Program Stop (NOTE! Only available as part of a program load)
M00L

ANALOGUE FWD
M03S100L

ANALOGUE REV
M04S78L

ANALOGUE OFF
M05L

Tool change
M06 T<tool number>

Contact 1 on
M08

Contact 1 off
M09

Label

M17L (Label number).

Auxiliary output and Input control

Aux outputs

To turn auxiliary <n> on

M20 A<n>+

To turn auxiliary <n> off

M20 A<n>-

Combinations of different auxiliaries to be on/off can be built up
e.g.

M20 A1+A2-A3+A4-

will turn on Aux 1 and Aux 3 and turn off Aux 2 and Aux 4

Note: The auxiliaries do not necessarily have to be input in order.

Inputs

To wait for input <n> to be high

M21 I<n>+

To wait for input <n> to be low

M21 I<n>-

Combinations of conditions can be tested

e.g.

M21 I1+I2+I3-I4-

will wait until Inputs 1 and 2 are high and Inputs 3 and 4 are low

Note: the inputs do not necessarily have to be input in order.

Serial Input (option)

Unconditional

M22L

Conditional

M22 Cxxx L

Note! xxx = character value 0 to 127

Goto Label if Inputs

M32 L<label number> I<n>+I<n>-L

e.g.

M32 L420L ;goto label 420 (See M17 labels)

M32 L407I2-I3+17+L ;goto label 407 if Input 2 is low and inputs 3 and 7 are high

Contact 2 on
M33

Contact 2 off
M34

Call Subroutine
M98 S<sub number>L

To Run a Program

B will cause the whole program to be executed

B<nnnn> will cause the program to be executed from block nnnn

To Erase a Program (i.e. to clear PNC 4 memory)

E will erase the program

To End the Load Sequence

M02L When this block is seen by the PNC 4 it will exit from the DATA LINK facility and return to MDI mode.

6.3 Responses from PNC 4 when in serial load mode

6.3.1 Host Computer

1. Providing the PNC 4 has received valid data as specified in Section 6.2 it will respond with the character <ACK>
<ACK> = 06H. This signifies that the PNC 4 is ready to receive more data.

2. If an error has occurred, the PNC 4 will respond with the following:

<NAK>nn
where nn is a two digit error code
<NAK> = 15H

The PNC 4 will then be ready to receive more data
The error codes are defined in the following table.

3. If reset is pressed on the PNC 4 while in RS232c mode with the PNC 4 waiting for data then it will transmit a <BEL>(07H) and return to MDI mode.

NOTE: that if some baud rate mismatch or other error prevents PNC 4 from recognizing <STX> and <CR> then no response codes will be sent.

6.3.2 PAPER TAPE

When serial load from tape is selected, the PNC 4 will operate using an Xon/Xoff type protocol as follows.

1) When the PNC 4 is ready to read a block from the serial link it will transmit a <DC1> character (=11H).

- 2) The PNC 4 will read characters until it reads a <CR>, whereupon it will transmit a <DC3> character (=13H).
- 3) Once the block has been decoded, the PNC 4 will transmit either an <ACK> character or a <NAK> followed by an error code number.
- ~~4) The PNC 4 will then continue as in step 1 by transmitting a <DC1> character~~
- 5) The exception to the above sequence is when an M02 block is received by the PNC 4, in which case no <DC1> is transmitted and the PNC 4 returns to MDI mode.

If the PNC 4 is to be used with a Portazip type paper tape reader, the following connections should be used.

<u>PNC 4</u>		<u>PORTAZIP</u>
<u>7 PIN DIN</u>		<u>SIGNAL 25 WAY D TYPE</u>
2	0v COMMON	7
7	DATA FROM PNC 4	3
6	DATA TO PNC 4	2

N.B. These connections are only suitable for reading paper tape using Serial Load. For tape punching using Serial Store see the relevant section of this manual.

6.3.3 PNC 4 RS232 ERROR CODES

<u>Error No.</u>	<u>Meaning</u>
01	Parity error in received character
02	Illegal G code received
03	Illegal M code received
04	Illegal character for this block
05	Move exceeds machine limits
06	Block not completed successfully
07	This block not allowed to execute immediately (Must be ended with L)
08	This block not allowed in a program
09	Attempt to run to non existant block
10	PNC 4 memory full
11	Block too big for input buffer
12	X axis drive system fault/OVER TRAVEL
13	Y axis drive system fault/OVER TRAVEL
14	Z axis drive system fault/OVER TRAVEL
15	W axis drive system fault/OVER TRAVEL
16	Incomplete block received
17	Error in input coordinate
18	Error in input feedrate
19	X and Y moves not present in circular move
20	Position not known machine must be driven to datum
22	Dwell value error
23	Scale value error

- 24 Tool number error
- 25 Auxiliary selection error

- 26 Input selection error
- 27 Repeat start label error
- 28 Repeat end label error
- 29 Number of repeats error
- 30 Nest error in repeat levels

- 31 Error in Repeat offsets
- 32 Error in offset block
- 35 Error when driving to datum
- 37 Error in number of thread starts range is 1 to 255
- 38 Error in thread pitch
- 39 Error in number of cuts
- 40 Subroutine no. out of range
- 43 Repeat within own labels
- 44 End label found before start
- 45 Goto label not found
- 46 Label number error
- 47 Angle error within circular move
- 48 Radius error within circular move
- 49 End point must not be defined
- 50 D to A ramp time error
- 51 Error in cycle length
- 52 Error in cycle depth
- 53 Error in move until input block

6.4 STORE PROGRAM TO DATA LINK.

When function 4 is selected in the DATA LINK menu, the PNC 4 responds with a menu:-

- 1) COMPLETE PROGRAM
- 2) PART OF PROGRAM

If 1 is selected then the whole of the program in memory is transmitted via the RS232c link.

If 2 is selected then the user is requested to enter the start and end blocks. When this has been done, the portion of the program selected is transmitted via the RS232c link.

(N.B. During transmission the message "STORING TO SERIAL DATA LINK" is displayed).

The data transmitted by the PNC 4 is exactly the same as it expects to receive when loading from the RS232c link, including block numbers at the start of each block.

i.e.

<STX> Nnnnnn G----- <CR><LF>
where nnnnn is the block number

During transmission the RxDa line is used as a busy signal thus:-

if RxDa is high (4V to 12V) then the PNC 4 will transmit
if RxDa is low (-12V to 0V) then transmission is inhibited at the end of the current character and the PNC 4 will wait for a high level before continuing to transmit.

~~After the last block in the program has been sent to the serial link, the PNC 4 will transmit an M02 block to signify the end of the program.~~

If the PNC 4 is to be used with a Portazip type paper tape punch, the following connections should be used.

PNC 4		PORTAZIP	
<u>7 PIN DIN</u>		<u>SIGNAL</u>	<u>25 WAY D TYPE</u>
2	0v COMMON	7	
7	DATA FROM PNC 4	3	
6	BUSY LINE	4	

N.B. These connections are only suitable for storing to paper tape.

6.5.1 Host Computer Software

Suppose it is required to cut a rectangle 30 mm by 10 mm using the host computer to control the PNC 4. The required data would be something like:-

```
G00 XOYOZO
G00 X100 Y100.5
G01 Z4FZ10
G01 X130 FX10
G01 Y110.5
G01 X100
G01 Y100.5
G01 Z5
G01 X130
G01 Y110.5
G01 X100
G01 Y100.5
G00 Z0
G00 XOYO
```

The host computer software could (as an example) operate in the following manner:-

- Send <STX><CR> to PNC 4 and wait for response.
- Response should be <ACK>. If any other character is received then perhaps the baud rate or parity is wrong. If no response is seen at all then the PNC 4 did not even recognize the data sent to it.
- Read the first line of the data (from a disk file perhaps).

- d) Send <STX> followed by the data followed by <CR>.
 - e) Wait for a response.
 - f) If response is not <ACK> then prompt for operator assistance.
 - g) If the response is <ACK> then read the next line of data and repeat steps 4 to 6 until all the data has been sent.
-

Note

The above program could be modified at step 4 to send L<CR> at the end of every block. This will cause the data to be directed into the PNC 4's memory. Once in memory it can be edited, printed, stored to cassette just like any program.

If the suggested structure is used, it has the advantage that other part programs can be generated just by altering the data.

If the program has been loaded into the PNC 4's memory then it can be executed under host computer control (providing the PNC 4 has been switched into AUTO) by sending:-

<STX>B<CR>

6.5.2 Operation of PNC 4 under host computer control

The following is a simplified description of how the PNC 4 operates when under host computer control.

1. When serial load from the host computer is selected via the front panel, the response character is set to <ACK>.
2. PNC 4 sends the response character to the Host computer.
3. PNC 4 enters a loop waiting for an <STX> character. Any other characters will be ignored.
4. Once the <STX> has been received, the PNC 4 then enters a loop storing all received characters into a temporary buffer until a <CR> is received.
5. The PNC 4 then decodes the temporary buffer.
6. If there was no error decoding the buffer then the information in the buffer is acted upon.
7. If there is any error then the response character is set to <NAK> and an error code is set up. If there are no errors then the response character is set to <ACK>.
8. Steps 2 to 7 are repeated until RESET is pressed on the PNC 4 front panel or an "M02L" block is decoded in step 5.

6.5.3 The following is an example program for sending data to the PNC 4 via the serial link. The program is written in BASIC and is suitable for running on an IBM PC or PC-AT using BASICA.

```
10 OPEN "com1:9600,E,7,2" AS #2
20 ON ERROR GOTO 220
30 OPEN "txdata" FOR INPUT AS #1
50 REM
55 ON ERROR GOTO 230
60 IF EOF(1) THEN GOTO 210
70 LINE INPUT #1,A$ :REM read string from file
80 PRINT A$
90 A$ = CHR$(2) + A$ + CHR$(13)
95 ON ERROR GOTO 240
100 PRINT#2,A$ :REM send string to pnc
110 R$ = INPUT$(1,#2) :REM wait for response from pnc
120 IF R$ = CHR$(6) THEN GOTO 55
130 REM
140 IF R$ = CHR$(21) THEN GOTO 160
150 PRINT "bad response from pnc"CHR$(13)"baudrate mismatch?" :GOTO 180
160 R$ = INPUT$(2,#2) :REM INPUT PNC ERROR CODE
170 PRINT "pnc error code" ;R$
180 PRINT "press any key to retry"
190 K$ = INKEY$:IF K$ = "" THEN 190
200 GOTO 100
210 PRINT "all data sent to pnc" :END
220 PRINT "error opening txdata file" :END
230 PRINT "error reading txdata file" :END
240 PRINT "serial input error" :END
```

The above program also requires a disc file called txdata which contains data blocks suitable for sending to the PNC 4.

7.1 RAPID TRAVERSE

RAPID TRAVERSE is a command that sets all feed rates to 100% without entering the feed values, it can also be inserted in the middle of a program without disturbing the programed feed rate.

To action this command, during input or edit modes, press the RAPID key or select GO and enter the parameters which you wish to drive.

NOTE: This command will NOT lift the Z axis automatically before moving, if a Z value is programed and you tell it to RAPID to another part without lifting the Z it will leave the Z in the programed position.

7.2 DWELL

This type of block simply causes the PNC 4 to wait for a certain amount of time before executing the next block. To input this block press the dwell key or enter G4. The PNC 4 will then prompt for the dwell time to be entered. The dwell time can be in the range 0 to 999.99 seconds.

7.3 MIRROR IMAGE

This function, which can only be used within a program, causes all information after the command to be reversed, or "mirrored", in the direction specified in the command.

To program a Mirror Image along the X or Y axis, the command is placed directly in front of the position you wish to mirror about. It must be remembered that this command will continue to mirror all co-ordinates until a cancel command is given within the program.

See Appendix D for an example program.

7.4 ROTATE (Option)

This is used to rotate programs about a specified point by any angle in units of 0.01 degrees.

The function is used inside a program by inserting a command at the appropriate point. This command specifies the point about which rotation has to take place and the angle through which it has to be turned.

NOTE: With the Rotate command angles are incremented, the cancel rotate must be used between rotate statements or the effective angle will be the sum of the angles input.

The key presses required to rotate a program through 180° about a point X100, Y100 would be -

Rotate, [E], X100[E], Y100[E] Rotate, 180[E], [EOB]

To cancel the rotate function this command will be -

Rotate, Rotate, [E], [EOB]

See Appendix D for some example programs using rotate.

7.5 SCALE FACTORS

There are two useful commands whereby all distance dimensions can be scaled up or down after a special command has been given. They are known as MACHINE SCALE and PROGRAM SCALE, the latter being used to scale sections within the program.

The MACHINE SCALE command is obtained when the computer is in its MDI MODE by pressing the SCALE key on the function pad and entering the value required. Note that a Machine scale is not part of the program and will not be stored onto cassette or serial link.

i.e. 200% Enter 200
 25% Enter 25

The range of scale is from 0.1 to 6500%.

i.e. 1/1000 of program size to 65 times program size.

The PROGRAM SCALE is used in the same manner, the only difference is when the SCALE key is pressed during INPUT or EDIT MODE a program scale can be applied.

The Natural Status condition for scale factors is 100%.

NB Machine Scale will also scale program scales!

7.6 TOOL RADIUS COMPENSATION (CUTTER OFFSETS)

If 2 or more axes have been defined during the set up procedure then Tool radius compensation becomes available. Press TOOL COMP or enter G40, G41 or G42. See section 8 for more details on how to use the cutter offsets.

7.7 OFFSET

The offset key allows 2 types of offset to be implemented, MACHINE OFFSET and PROGRAM OFFSET. Program offset will cause all succeeding blocks in a program to be offset. Machine offset will offset the entire program. A program offset block can only be entered as part of a program (using insert or replace in EDIT), a machine offset can only be entered outside a program i.e. when the prompt line is showing "Select function". Note that a Machine offset will add a further amount to any Program Offset value.

Press OFFSET and then X, Y, Z, or d followed by a dimension. Offsets values for each axis can be entered. Press EOB when all the values are correct. Note that a machine offset will further offset any program offsets. (See Appendix D for example programs).

7.8 METRIC/IMPERIAL UNITS

G70 and G71 blocks can be inserted in the program at any point to facilitate input of coordinates. Also, pressing the Inch/m.m. key will cause the units to change from one type to the other. Note that only those axes defined as mm or inch in the set up procedure will be affected by those blocks.

7.9 REPEAT/REPEAT-DIVIDE

Repeat can only be performed within a program. It is a way of repeating a shape already defined in the program so that it is reproduced again in one or more further positions within the available working area, each positioned being stepped along from the previous positions.

The function is activated by a command which tells the system which program Blocks it must repeat, and how many times, then tells it how far to step along the four axes before starting to repeat those blocks.

To use the Repeat facility you enter it in as a block of information via the REPEAT key located on the function key pad, or selecting G81.

Once selected the prompt line asks the following questions -

- a) ENTER START LABEL enter your selection and press ENTER.
- b) ENTER END LABEL number and press ENTER.
- c) ENTER LOOP COUNT 1 TO 65535 and press ENTER.
- d) ENTER REPEAT OFFSETS in terms of X, Y, Z, or d i.e. X, 15, ENTER, EOB.

(See Appendix D for example programs).

7.10 ABSOLUTE DATUM

This type of block can be entered into a program by pressing the ZERO key or entering G98. When this block is executed it will cause all the defined axes to go to their datum position as defined in the Set Up procedure.

Each axis can have the following attributes for datum:-

- 1) datum position either 0 or its maximum +ve limit.
- 2) Fixed datum or not - if not fixed then that axis will simply take its present position as its new datum point. If fixed then it will actively seek datum markers.

The order in which each axis takes up its datum position can be specified by the Set Up procedure.

At power up (or whenever the datum procedure is stopped by pressing the STOP button) the user can select the number of axes to be datumed. Thereafter, until the PNC 4 is switched off it will remember that number and use it whenever <ENTER> is pressed without a number being entered.

7.11 FLOATING DATUM

The Floating Datum facility differs from the Offset described in Section 7.7. Whereas the latter effectively shifts the System Datum Point by adding specified amounts to all distance commands in the identified axes, the Floating Datum can be established anywhere within the working area. The Floating Datum is established using its present position at execution and unlike the Offset command it can only be used within a program.

In practice, the Floating Datum facility is often used where it is necessary to set a specific datum point within a shape in order to facilitate the remainder of the program. (See Appendix D for an example program).

7.12 PROGRAM STOP

This type of block causes program execution to halt until the START button is pressed. Either the front panel START button or the external start switch can be used to continue program execution.

7.13 CONTACTS 1 AND 2

Contacts 1 and 2 behave exactly like on.off auxiliaries. The only difference is that there is a separate key for each contact and special M codes; M8 to turn contact 1 ON, M9 to turn it off, M33 to turn contact 2 ON, M34 to turn it off. The separate keys (marked ON or OFF, next to the jog keys) can be used at any time to switch the contacts directly. These keys are not active whilst the jog keys are in use.

A further advantage of using these contacts instead of ordinary auxiliaries is that they can be given names which are meaningful to the operator. E.g. if contact 1 is being used to control coolant flow then it can be renamed to 'COOLANT' (see 1.8.6).

7.14 TOOL CHANGE

Tool change blocks can be inserted into a program by pressing the TOOL key or by entering M6. The PNC 4 will then ask the user to 'ENTER TOOL NUMBER'. Numbers 1 to 16 refer to the tool offsets stored in the tool offset library (see 8.0).

If 3 or more axes have been defined as active in the Set Up procedure then a tool change block will cause the 3rd axis to move to its datum position before waiting for the START button. However if the tool has been defined as an auto tool then it will not wait for the start button or move the 3rd axis to datum.

Tool 0 will cause the PNC 4 to move to the tool change position but will not actually change from the current tool.

If the Stop button is pressed at any time contact 2 and auxiliary 10 will be turned off (see also 7.16).

7.15 LABELS

Labels can be inserted anywhere in a program by pressing the LABEL/GOTO key followed by ENTER or by entering M17. A label number in the range 1 to 65535 can then be entered. Labels are used by the Repeat function (G81) and the Goto function (M32).

7.16 AUXILIARY functions

Auxiliary functions allow user assigned devices to be controlled i.e. switches on and off the 12 integral PNC 4 auxiliary relays. Three types of auxiliary functions are supplied.

The three types are:-

- a) ON/OFF
- b) MOMENTARY
- c) PULSED

(See also 1.8.5)

a) ON/OFF auxiliaries are set when programed. If the auxiliary is programed ON it will remain ON until programed OFF. Such auxiliaries could be used, for example, to control lubricant.

b) MOMENTARY auxiliaries are switched ON (if programed to be on) only when the machine is at a programed position. When the axes are moving, momentary auxiliaries are always OFF. This type of auxiliary can be used, for example, to provide a table locking signal or to activate the main drilling head on a drilling machine.

c) PULSED auxiliaries provide a pulse output (if programed ON) each time the machine completes a program block.

To program auxiliaries press the AUX/INPUT key, and press ENTER. The PNC 4 will prompt the user to select the auxiliaries that are to be programed ON.

On entering the specific auxiliary number pressing [ENTER] will select that auxiliary, all subsequent auxiliaries can be selected in the same way.

Pressing [ENTER] without an auxiliary specified will cause the prompt to invite the user to select auxiliaries that are to be programed OFF, these auxiliaries are programed in the same way as described above. Pressing [EOB] will end that block of information.

If the stop button is pressed at any time, contact 2 and auxiliary 10 will be turned off. This is a safety feature implemented to ensure that the PNC 4 returns to a "safe" state in an emergency. The user should therefore ensure that the equipment controlled by the auxiliaries is inactive (safe) whenever the auxiliary contacts are open.

7.17 INPUT facilities

The PNC 4 is equipped to monitor 8 user assigned input signals from external switches. The condition of the switches may be checked to see if they are open or closed during program execution. If the switches are not in the state, the PNC 4 waits until the switch signals are as programed before proceeding. Switch levels may be programed to be closed (ON) or open (OFF). These inputs could be used for example to check, if safety guards are in the correct position before movement, or to check the position of an auxiliary controlled hydraulic ram, or to make the PNC 4 wait for an external signal from a robot before proceeding.

To program inputs press the AUX/INPUT key twice and then press ENTER. The inputs are entered in the same way as auxiliaries.

The auxiliary outputs and the inputs enable the PNC 4 to function not only as a precise positioning control system but also as a sequence controller.

7.18 GOTO label

A GOTO block can be inserted into a program by pressing the LABEL/GOTO key twice followed by ENTER or by entering M32. A label number in the range 1 to 65535 can then be entered. At this point the display will show an empty input block and request the user to select inputs to be on. Pressing <EOB> at this point will cause an unconditional GOTO block to be entered. Alternatively the user can specify which inputs have to be ON or OFF to make the PNC 4 go to the specified label.

At execution time the PNC 4 will check the status of its inputs and if they match those specified in the block then it will goto the specified label. If the inputs do not match, then it will simply go on to the next block in the program.

This feature can be used to selectively execute different parts of the program
E.g.

```
N1      XO      YO
N2      GOTO LABEL 1 IF INPUT ON 1
N3      CALL SUBROUTINE 1
N4      GOTO LABEL 99
N5      LABEL 1
N6      CALL SUBROUTINE 2
N7      LABEL 99
N8      END OF PROGRAM
```

This program will execute subroutine 1 if input 1 is OFF or subroutine 2 if input 1 is ON.

Another feature of the GOTO block is that if it goes to a label which is the 1st block in the program then it effectively causes the program to restart. I.e. the program offsets will be cleared, the program scale will be set to 100%, the run counter will be incremented, all repeat offsets will be cleared and cutter diameter compensation will be reinitialised.

N.B. When validating a program or using Block search past a GOTO block then the PNC 4 will always assume that the input condition fails and will drop through to the next block. However, in the case of validating it will also check that the label number specified does exist in the program somewhere.

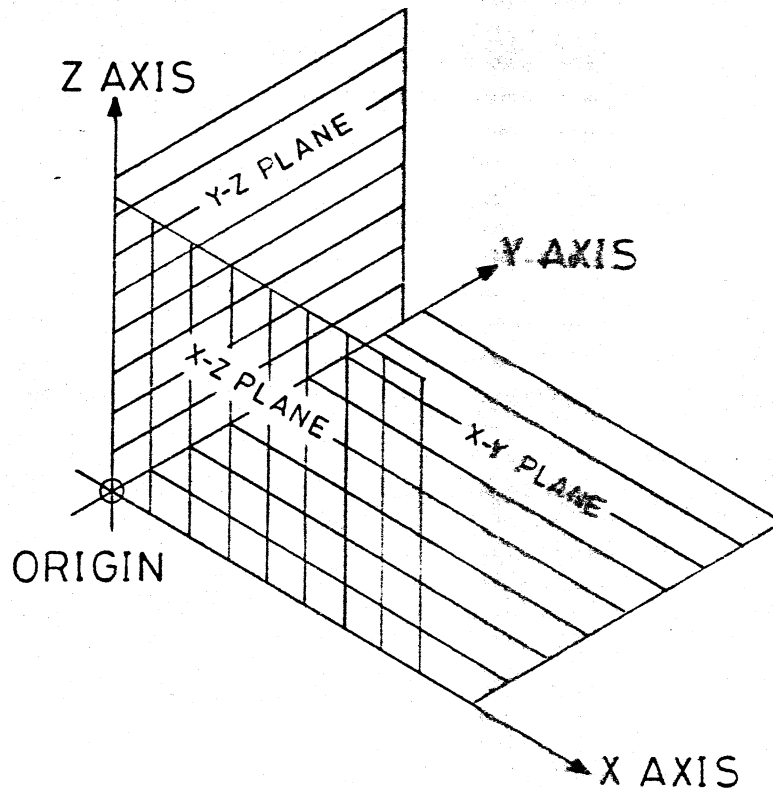
GOTO blocks can also be used inside subroutines in exactly the same way as in a program. A GOTO block in a subroutine will only search inside the subroutine and not in the main program. This means that a LABEL block inside a subroutine can have the same label number as inside a program.

7.19 CALL SUBROUTINE

Press the SUBROUTINE key or enter M98 to specify a call to a subroutine. The PNC 4 will request a subroutine number in the range 1 to 255. Enter a value and press <EOB> to end the block. See section 9.1 for more information on the use of subroutines.

7.20 PLANE SWITCHING (Note only active if PNC 4 has three or more axes selected)

Plane switching allows the operator to selectively switch the contouring axes from the XY axes to either the XZ axes or the YZ axes. This enables linear and circular interpolation to be executed within any of the three planes shown below.



G17 XY PLANE, G18 XZ PLANE and G19 YZ PLANE are the blocks used to select in which plane you wish to contour. These blocks are only available within a program, they can be selected using their relevant G codes or by pressing the SETUP key which will display a menu of the three planes available,

- 1) G17 XY PLANE
- 2) G18 XZ PLANE
- 3) G19 YZ PLANE

then by selecting one of the options 1 to 3 the relevant plane switching block will be inserted within the program.

It is advisable and good programming practice, that the block immediately after a plane switch block should have all three switchable axes defined.

Examples of the plane switch blocks being used are at the rear of the manual, programs 11 to 13. In program 11 you will notice that the XY plane has not been specified within the program, this is because the start of program sets the XY plane. When the plane switch blocks are executed the plane status is displayed in the righthand corner of the screen, on the same line as the inputs, the status is blank when the XY plane is in operation.

Because of physical characteristics plane switching is not active during the execution of some blocks.

1) An absolute datum block will always datum the axes in the same order and direction as in the XY plane.

2) A tool change block will regardless of the plane selected, move the Z axis to its datum point in order for the tool change operation to take place.

3) When executing moves directly it should be noted that the plane will be switched to the XY plane before the move is executed.

IMPORTANT for plane switching to work correctly the stepsize of the three switchable axes should be all the same.

8.1 TOOL OFFSETS

When the program is loaded into the PNC 4 the following procedure should be used to activate the cutter offsets.

Press the TOOL key on the function keyboard.

A menu will appear:-

1. DISPLAY & EDIT TOOL OFFSETS
2. SET TOOL OFFSETS
3. CHANGE CURRENT TOOL
4. CLEAR ALL TOOL OFFSETS

8.1.1 Selection of 1 will display a table of the 16 tools available.

Prompt will read 'ENTER TOOL NUMBER'.

Select the required tool number 1 to 16 then press <ENTER>.

If you have 3 or more active axis defined in the SET UP procedure then the PNC 4 will ask:-

ENTER OFFSET CHANGES Z

NOTE: If you do not require tool length compensation a value of zero should be entered or alternatively press the ENTER key without a value.

ENTER OFFSET CHANGES D - Enter your cutter diameter values.

Repeat this section for each tool you wish to set up.

When complete press RESET key twice to escape from the menu.

8.1.2 Selection of 2 Tool Setting (Non Lathe).

ENTER TOOL NUMBER

Select -

1. MANUAL CHANGE
2. AUTO CHANGE

Selection of 1 will instruct the computer to stop at this tool change block and wait for the tool to be changed.

Selection of 2 will not stop the machine to change the tool when this tool number is seen. (This can be used to advantage when regular shapes require clearing out. The same tool is left in the machine and various cutter diameters are selected).

Enter your selection.

WARNING!! Beware of the Z offset.

This selection allows you to set the tool length with the Jog Keys. Position the cutter to the surface required and press Z, or simply press ENTER if no length offset is required.

Key in the cutter diameter required and press ENTER then press EOB to enter into the computer memory.

Selection of 2 Tool Setting (Lathe)

Prompt will be 'ENTER TOOL NUMBER'

Select the required tool number 1 to 16 then press <ENTER>.

Display will read:-

- 1) FRONT TOOL/MANUAL CHANGE
- 2) REAR TOOL/MANUAL CHANGE
- 3) REAR TOOL/AUTO CHANGE
- 4) FRONT TOOL/AUTO CHANGE

Selection of 1 or 2 will instruct the PNC4 to stop at this tool change and wait for the tool to be changed. The operator must then press <START> to continue program execution.

Selection of 3 or 4 will instruct the PNC4 not to stop at this tool change but to simply pick up the new tool offsets and continue executing the program.

After selecting the tool type the display will invite the operator to use the jog keys to face the end of the workpiece. When this is done the operator should keep the Z position the same then press <Z> to fix the Z offset. This will cause the PNC4 to use this position as Z0.

The operator is now invited to use the jog keys to turn the workpiece. When this is done, the operator should keep the X position the same, stop the spindle, measure the turned diameter then key this diameter in to the PNC4. This will cause the PNC4 to fix the X offset for this tool such that the centre line of the workpiece becomes 0.

It should be noted that the PNC4 uses the fact that the tool could be selected for front or back and automatically compensates for this.

The operator must then press <EOB> to accept the tool offsets.

8.1.3 Selection of 3 Tool Change

Enter a tool number and the PNC 4 will execute an M06 Tool change block.

8.1.4 Selection of 4 Clear all Tool Offsets

This is a handy way of removing all tool offset compensation. Simply press 4 and all tool offsets will be set to 0. (Use selection 1 to confirm that this is the case).

8.2 Cutter Offsets (Option)

Tool radius compensation (or automatic cutter offset calculation) is available on PNC 4's configured to control two or more axes. It allows the machine programmer to compensate for the diameter of the current cutter. This means that the PNC 4's program need only specify the true dimensions of the desired finished product. This facility is obtained by pressing the TOOL COMP key either inside or outside a Program loading sequence.

1. G40 CANCEL TOOL COMP
2. G41 TOOL COMP LEFT
3. G42 TOOL COMP RIGHT

The amount of compensation is determined by the diameter of the current tool (see toolsetting). If the current tool has been set to have zero diameter then no cutter offset will be calculated.

When the cutter offset is activated, the PNC 4 will look ahead to the next two X, Y positions in the program and use them to calculate the first offset position. The PNC 4 will then move from its present position to the offset when it executes the 1st of these two X, Y blocks.

8.3 Effects of Cutter Offsets

In the following section all possible combinations of 2 adjacent blocks will be considered. For the purpose of the examples the current block will be known as block 1 moving from point P_1 to point P_2 and the next block will be known as block 2 moving from point P_2 to point P_3 . The user must appreciate that in order for the PNC 4 to calculate the correct offset path it must "look ahead" from its current block to the next block. In fact the PNC 4 looks ahead until it finds a block defining an X,Y position which is different from the X,Y position in its current block. This feature allows Input and Auxiliary blocks to be effectively ignored when calculating the offset path. Note that if scaling factors, mirror imaging or repeats (or any similar function) are currently in use, the cutter offset calculations will be carried out first, acting on the raw program data and then the other functions will be performed.

This is particularly important when using mirror imaging or rotation because they use the PNC 4's present X,Y position to define their axes of symmetry or point of rotation. It is necessary therefore to de-activate cutter offsets before including the mirror imaging or rotation block. The cutter offsets can then be re-activated in the normal way.

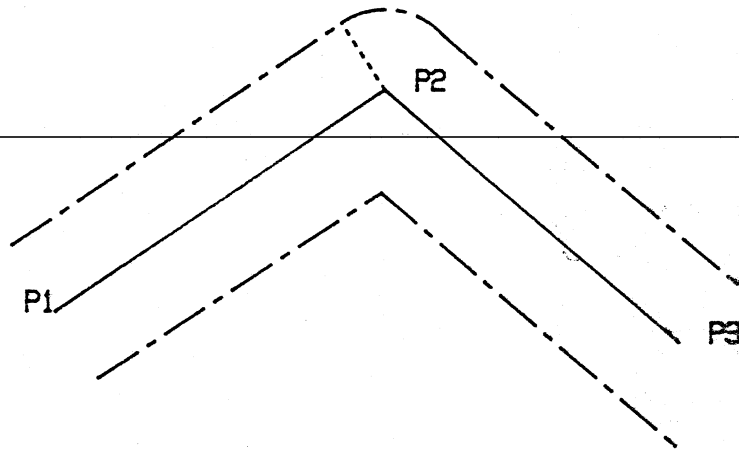
In the following examples the terms internal and external corner will be used. They are defined as follows:

External Corner - when the angle between the two blocks is greater than 180° on the side of the material being worked, i.e. making a left turn when working on the right-hand side of the material, or making a right turn when working on the left-hand side of the material.

Internal Corner - when the angle between the blocks is less than 180° i.e. making a right turn when working on the right hand side of the material or making a left turn when working on the left hand side of the material.

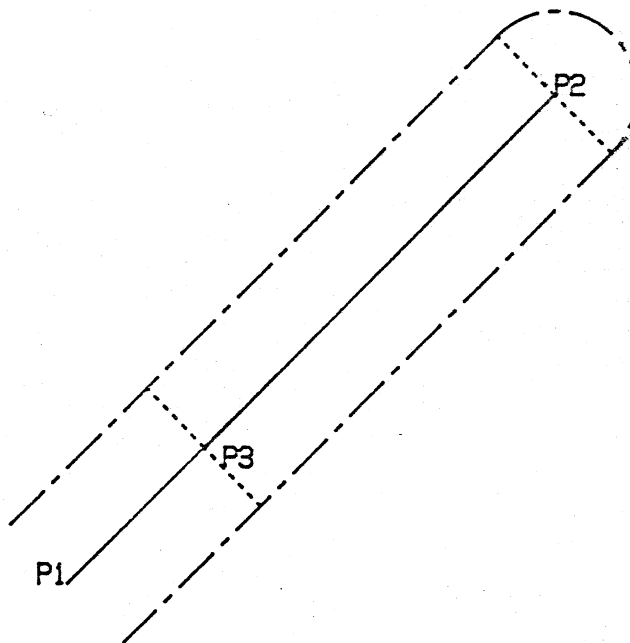
As will be seen from the following examples, the PNC 4's action on encountering an external corner is to move around it in an arc of radius equal to the cutter radius about a centre located at point P_2 . During this circular motion the tool will always be in contact with point P_2 thus creating a burr-free corner.

Straight Line to Straight Line Intersection

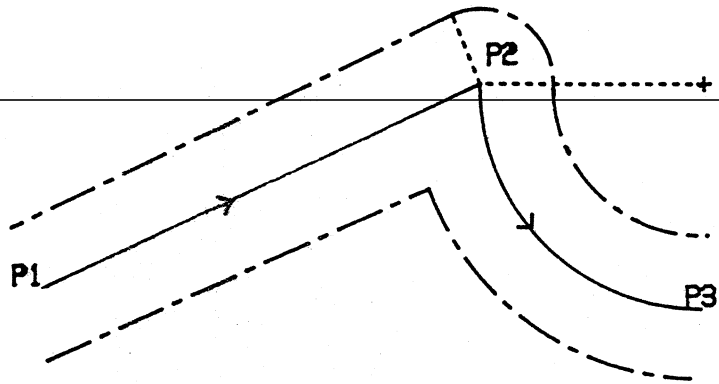


In the extreme case where block 2 actually doubles back onto itself this will always be taken as an external corner independent of whether a right hand or a left hand offset is being used.

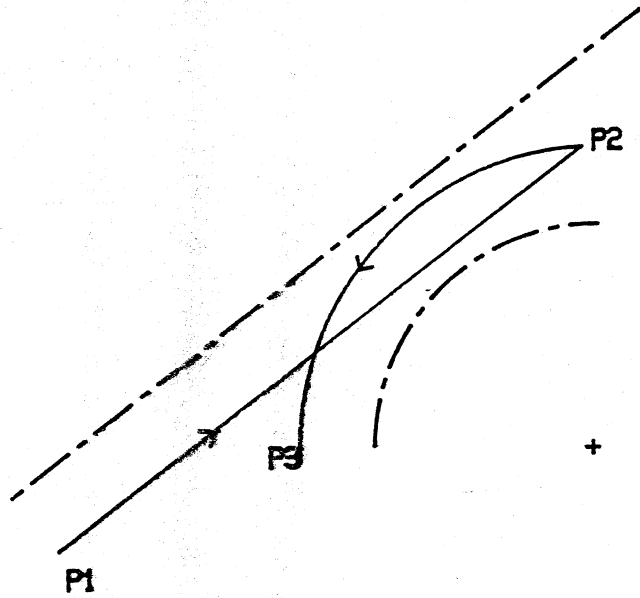
E.g.



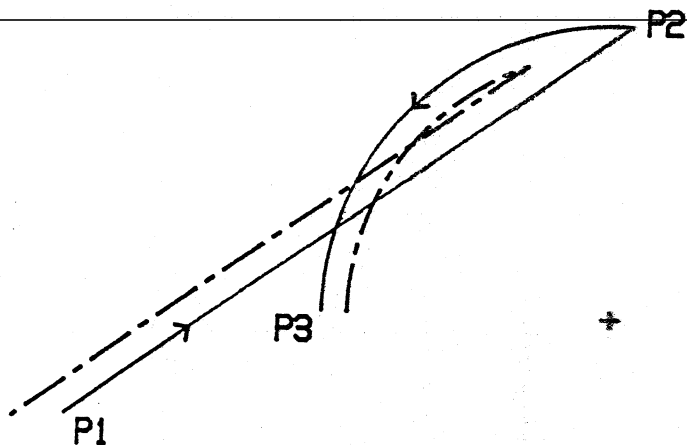
Straight Line to Arc Intersection



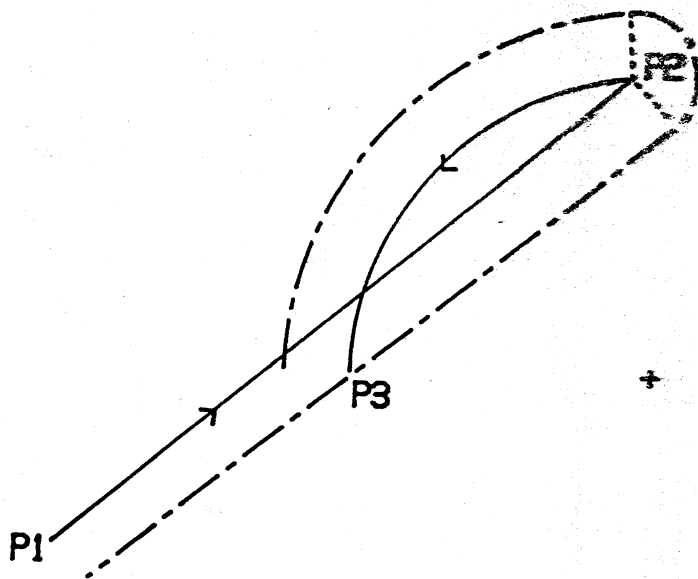
The PNC 4 will cope with any kind of Line to Arc intersection except where no intersection is possible E.g.



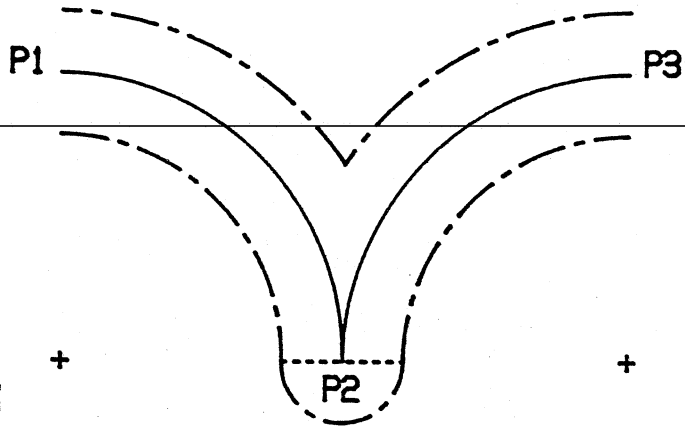
In which case there will be a pause during the calculation period, followed by an error message on the display indicating that perhaps the wrong tool diameter has been input. Note that in this case if the cutter diameter is small enough an intersection may be possible E.g.



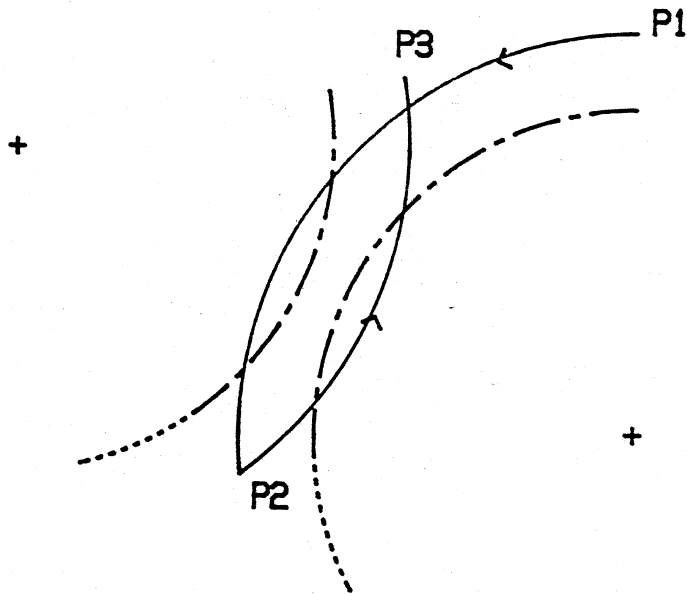
Note that in both the previous examples if a right hand (or positive) offset is applied then P₂ will be taken as an external corner and the following will result:



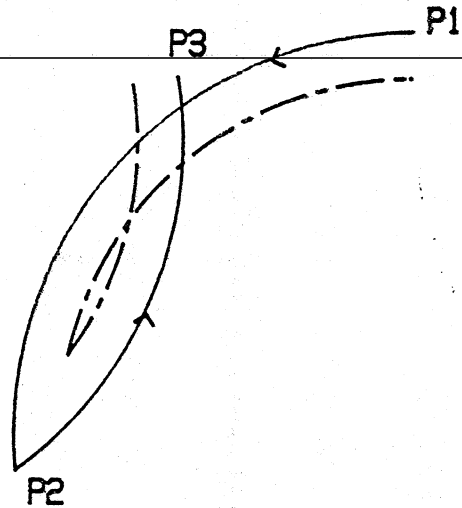
Arc to Arc Intersection



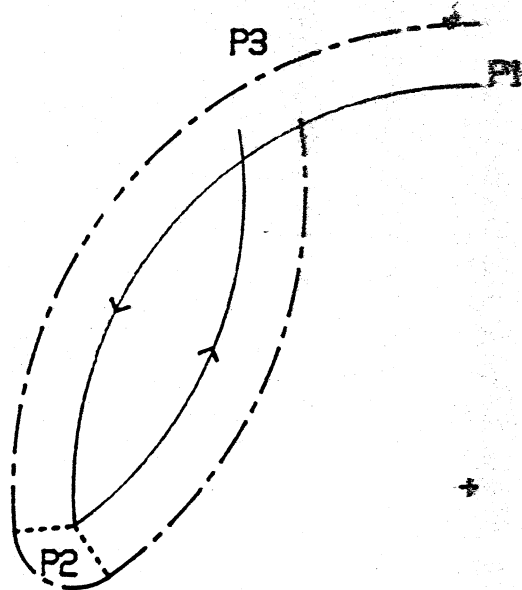
The PNC 4 will cope with any kind of Arc to Arc intersection except where no intersection is possible E.g.



In which case there will be a pause while the PNC 4 tries to find an intersect followed by an error message on the display indicating that perhaps the wrong tool diameter has been input. Note that in this case if the cutter diameter is small enough an intersection may be possible E.g.



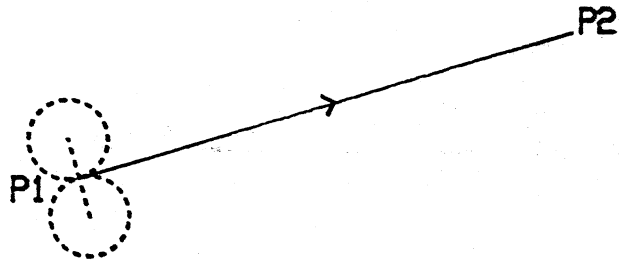
Note that in both the previous examples if a right hand (or positive) offset is applied then P₂ will be taken as an external corner and the following will result.



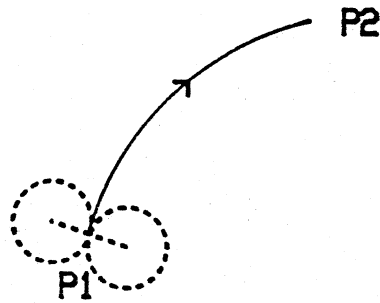
First Offset after Activation

When the cutter offset has just been activated it will take up a position perpendicular to the point P_1 at a distance of the cutter radius away from it

E.g.



or if the line P_1 to P_2 is circular:-



Last Offset when De-activated

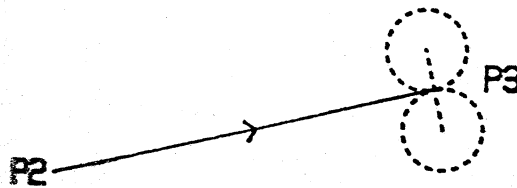
Any block apart from linear/circular blocks and those listed below will stop the look-ahead process and deactivate cutter offsets.

M3/M4	Contact 1 ON/OFF
M8/M9	Contact 2 ON/OFF
M00	Program stop
M20	Auxiliaries
M21	Inputs
M17	Label
G04	Dwell
G70/71	Imperial/Metric units
G79	Re-enable cycle
G80	Deactivate cycle
G82	Circular cycle
G83	Drilling cycle
G84	Rect-lar cycle

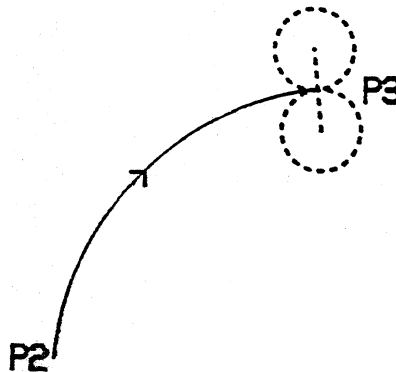
Also the last block in any program will automatically de-activate cutter offsets.

When the cutter offset has just been de-activated it will take up a position perpendicular to the point P₃ at a distance of the cutter radius away from it.

E.g.



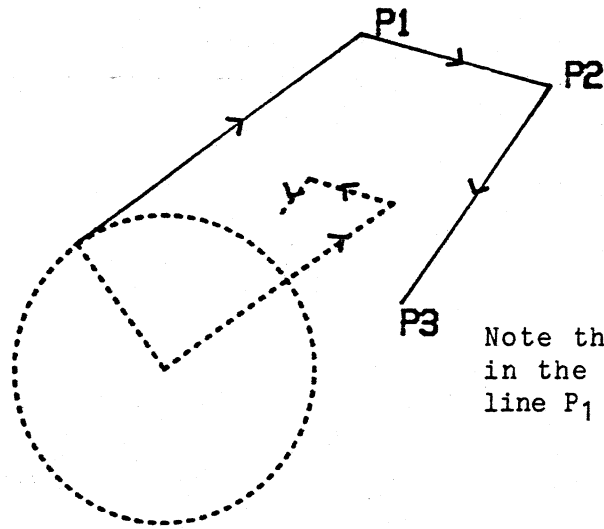
or if the line P₂ to P₃ is circular



Error Condition

In addition to the condition mentioned in the examples where an intersect is called for which is impossible to calculate, the error display may result if ever the PNC 4 calculates an offset path which moves in the opposite direction to the programmed line. This condition will only arise if the user is trying to mill out an internal corner which is too small for the cutter diameter.

E.g.



Note the calculated offset line is in the opposite direction to the line P₁ to P₂

In all cases where the error display occurs, the program can be successfully re-run with a smaller diameter cutter being specified.

8.4 Programming Using Cutter Offsets

This section describes some useful tips on programming the PNC 4 using cutter radius compensation (G40, G41, G42). At the end of this section there is an example program.

- a. When cutter compensation is active the word COMP will be highlighted in the top right area of the display underneath the "abs" or "incr". This will only appear if a G41 or G42 has been executed and the current tool's diameter is not zero.
- b. Neither a G41 nor a G42 block will cause any axis to move. They are used to activate the compensation process so that the next linear, rapid or circular block will take up the radius offset.
- c. The usual format for a part program is to take up a position away from the work, enable cutter compensation and move round the profile, then disable compensation and move away from the work. This format makes the assumption that cutter compensation is not active at the program start. The assumption is not always true, for instance if a program is stopped while compensation is active and then restarted - compensation will be active from the first block in the program. The safest solution is to include a G40 block at the start of the part program.

8.4.1 Enabling Cutter Offsets

- a) Consider the following section of PNC 4 program:-

```
G40 CANCEL TOOL COMP
G01 X50 Y50
G41 TOOL COMP LEFT
G01 X50 Y50
G01 X100 Y50
G40 CANCEL TOOL COMP
```

By having the same XY position after the G41 as before it then a move distance equal to the cutter radius will occur. I.e. assuming a cutter diameter of 10 then the PNC 4 will move to X50, Y50 then X50, Y55 then X100, Y55.

- b) Consider the following section of PNC 4 program:-

```
G40 CANCEL TOOL COMP
G01 X50 Y50
G41 TOOL COMP LEFT
G01 X100 Y50
G01 X200 Y50
G40 CANCEL TOOL COMP
```

In this case the cutter compensation will be taken up on the move to 100, 50. I.e. the PNC 4 will move to X50, Y50 then X100, Y55 then X200, Y55.

Block 15 cuts the final bit of the square to the correct depth. (Remember that in block 8 the Z axis is moving down as the Y axis moves to 150 therefore a small amount of material will be left uncut).

Block 16 to 18 cancel tool compensation lift out of the material and then move to the centre of the circle.

Block 19 brings the Z axis down into the material again. In this case there is no simultaneous X or Y axis movement. (The reader may like to predict the effect of deleting this block, and then try it out).

Blocks 20 to 24 enable cutter compensation and define a complete circle.

Block 25 Repeats the circle with a Z axis offset producing a final depth of 20mm.

Block 26 and 27 disable compensation and lift the cutter out of the material.

Blocks 28 to 36 change to a different tool and cause 4 holes to be drilled.

Block 37 is a dummy tool change (to tool 0) which has the effect of fully retracting the Z axis.

Block 38 moves to a 'safe' position out of the way to allow the workpiece to be inspected/removed.

c) Now consider the effect of including Z moves in the previous examples. Adopting the convention that Z0 is the workpiece surface and negative Z values cut into the workpiece.

```
G40 CANCEL TOOL COMP
G01 X50 Y50 Z10
G41 TOOL COMP LEFT
G01 X50 Y50 Z-5
G01 X100 Y50
G40 CANCEL TOOL COMP
```

The PNC 4 will move to X50, Y50, Z10 then X50, Y55, Z-5 and then to X100, Y55, Z-5. In other words the Z axis will move down into the workpiece as the cutter radius offset is taken up.

d) Consider the following:-

```
G40 CANCEL TOOL COMP
G01 X50 Y50 Z10
G41 TOOL COMP LEFT
G01 X100 Y50 Z-5
G01 X200 Y50
G40 CANCEL TOOL COMP
```

In this case the PNC 4 will move to X50, Y50, Z10 then to X100, Y55, Z-5 then X200, Y55, Z-5. In this case the Z axis will move down into the workpiece while the XY axes are moving to X100 Y55.

8.4.2 If you are using cutter offsets to mill a closed profile then be sure to start the profile away from a corner. If you start on a corner then the cutter offset path will not close properly. That is, the position where cutter offsets are deactivated will not be the same as the position where they are activated. This effect will make repeating the profile impossible.

In the example program note that square profile is started at the mid point of one side.

8.4.3 This example program is designed to bring out the points discussed earlier. If you run this program on your PNC 4 remember to set a tool length offset of at least 20mm on tool 1 and 2. Tool 1 should also have a diameter specified.

Blocks 1 to 3 make sure the PNC 4 will be in the correct state for the next part of the program.

Block 4 is a rapid move to a point just above the work.

Blocks 5 to 7 enable the cutter compensation using the method described in 4.1.

Block 8 starts to move up the left hand side of the square whilst simultaneously bringing the Z axis down to cutting depth.

Blocks 9 to 13 complete the square and block 14 repeats it with a Z offset cutting down to a final depth of 20mm.

9.1 SUB-ROUTINES

Sub-Routines are relatively short, independent programs for frequently-used sequences. They can be stored in a special part of the PNC 4 memory and then called up by simple commands in the main program as required. The sequence specified in the sub-routine will be performed, and then the system will continue with the main program.

Sub-routines do occupy memory space which would otherwise be available for the main programs, so their use should be restricted to routines which are in frequent use. On the other hand, their use allows the main program to be shorter, so to some extent one effect offsets the other.

Sub-routines can be compiled using positive and negative (+ or -) moves about a datum point, thus allowing symmetrical shapes to be positioned about their true centre. The datum point can be located at any convenient position within the sub-routine sequence.

For example, the following would be the program for a Sub-routine to produce a 10mm diameter circle -

```
SUB 1  
  
N1 G01 LINEAR      X 0      Y 0  
N2 G01 LINEAR      Y -5  
N3 G02 CIRCULAR CLW X 0      Y -5  
      CENTRE X 0      Y 0  
N4 M99 SUBROUTINE END
```

The instruction within the main program to call up this Sub-routine will contain instructions to position the circle.

For example:-

```
N1 G01 LINEAR      X 150    Y 100    F 100.0  
N2 M98 CALL SUBROUTINE 1  
N3 M02 END OF PROGRAM
```

The above statement will result in a 10mm circle being produced with its centre (its datum point) at X150 Y100.

Alternatively, the same Sub-routine could be used to produce a circle of different size by using a Program Scale Factor (see section 7.5). For instance to produce a 4mm diameter circle the instruction would be -

```
N1 G01 LINEAR      X 150    Y 100    F 100.0  
N2 G20 PROGRAM SCALE 40.0%  
N3 M98 CALL SUBROUTINE 1  
N4 G20 PROGRAM SCALE 100.0%  
N5 M02 END OF PROGRAM
```

The last statement returns the Scale Factor to normal for continuing the remainder of the main program.

Sub-routine Identifiers (numbers) can be allocated as required, within the range of 1 to 255. Storage onto tape can be selected as all the Sub-routines in memory or as an individual Sub-routine.

Sub-routines can be edited in the same way as the main program.

~~Access to the edit facility is through the Sub-routine key on the Function Key Pad.~~

Printing of Sub-routine programs can be obtained through the DATA LINK mode selection 4 see section 6.1.

NOTE: Repeats and other special functions, including cutter offset, cannot be contained in Sub-routines. Also, the use of Sub-routines is not confined to circles, but can also embrace rectangles, triangles and any other definable geometric shape or any frequently used section of program.

10.1 DIGITAL TO ANALOGUE CONVERTER (Option)

The PNC4 D to A converter output can be varied between 0 and 10V and can be altered manually or from programmed blocks.

There are three different blocks to control the D to A output. These are M03-FORWARD, M04-REVERSE and M05-OFF. These blocks can only be entered by keying in their M code. ~~If M03 or M04 is selected then the user is prompted to enter the required speed. A value between 1 and 100 can then be entered which represents the percentage of full output required.~~

If M05 is selected, the user can only press EOB to enter the block.

The current state of the D to A output is displayed on the top line of the PNC4 screen e.g. D to A F75, D to A R100 or D to A OFF.

Note! F denotes FORWARD, R denotes REVERSE.

If a tool change block is encountered then the D to A output will ramp down to 0. When the tool has been changed and <START> is pressed, the D to A output will ramp back up to its previous setting.

In MDI mode when <CASS>, <EDIT>, <SETUP> or <ZERO> is pressed, the D to A output will ramp down to zero.

The D to A output can be given a name which is meaningful to the operator. (See 1.8.9).

10.2 Dual Digital to Analogue Converters (option)

The PNC 4 dual D to A converters can be individually varied between 0 and 10v and can be altered manually or from programmed blocks.

There are two blocks to control the D to A outputs. These are M23 for output 1 and M24 for output 2. These blocks can only be entered by keying in their M codes.

If one of these blocks is selected then the user is prompted to enter the output level required. A value between 0 and 100.00 which represents the percentage of full output required.

The current state of the D to A outputs are displayed on the top line of the PNC 4 screen e.g. OUTPUT A 75.25 OUTPUT B 0.00.

If a tool change block is encountered then both D to A outputs will ramp down to 0. When the tool has been changed and <START> is pressed, then both outputs will ramp back to their previous levels.

In MDI mode when <CASS>, <EDIT>, <SETUP> or <ZERO> is pressed, both outputs will ramp down to 0.

In SET UP the outputs can be given names which are meaningful to the operator. The time taken to ramp each output from 0 to 100% can also be set.

The user is prompted to select which D to A parameters to set. When an output is selected the user is prompted to "ALTER NAME". Using the same method as 'NAME CONTACTS' (1.8.6) the name of the D to A output (up to 8 characters) can be changed.

The ↓ key can be used to advance to the range time setting facility. The user will then be prompted to enter the ramp time required in the range 0 to 99.9 seconds. The ↑ key can be used to return to the name D to A facility.

10.3 Digital to Analogue Converters on the Fly

The Digital to Analogue Converter (DAC) can be made to alter its value whilst one or more of the axes are moving. Thus for example, in a laser application where the DAC controls the power to the laser, the laser power can be varied whilst the XY axes are moving around a contour.

This facility is enabled quite simply by entering DAC control blocks into the program just before the block which defines the new axis position.

E.g.

```
N1 G01 X0
N2 M4 ANALOGUE 40%
N3 G01 X50
```

In this example the DAC output will change to 40% whilst the X axis moves from 0 to 50.

NOTES:-

- a) To enable this function correctly the block defining the new DAC value must be immediately preceeded by a movement type block and must be immediately followed by a movement type block. E.g. in the following program the DAC will not change its value whilst the X axis is moving.

```
N1 G01 X0
N2 M04 ANALOGUE 40%
N3 G04 DWELL 0.01 seconds
N4 G01 X50
```

- b) The feedrate and distance of the movement block must be adjusted to suit both the DAC ramp time and the change in DAC output, so that the DAC can achieve its new output before the movement ends. If this is not the case then a "MOVE NOT READY" error will be displayed and all movement will stop dead.

E.g.

```
N1 M05 ANALOGUE OFF
N2 G01 X0
N3 M04 ANALOGUE 100%
N4 G01 X100 FX100
```

In this example assume that 100% feed is equal to 100mm per minute, then the time taken to move the X axis to X = 100 will be 60 seconds. Therefore the DAC ramp time must be set to a value of less than 60 seconds for the program to work correctly.

- c) In some cases the user may not want the DAC output to change whilst the axes are moving. If so the simplest way of disabling this feature is to follow each DAC block with a dwell block of 0.01 seconds, as shown in note a) above.
-

11.1 Canned Cycles Facility (option)

The canned cycle facilities on the PNC 4 are only available within a programmed sequence.

The following canned cycles are available:-

```
00 05 15
      G79 RE-ENABLE CYCLE
      G80 DEACTIVATE CYCLE
      G82 CIRCULAR CYCLE
      G83 DRILLING CYCLE
      G84 RECT-LAR CYCLE
```

Important Notes Concerning Canned Cycles

There are no dedicated keys on the PNC 4 for the canned cycles. The only method of entering a canned cycle is by using its G code.

Canned cycles when executed are centred about the previous X and Y axes coordinates of the program. The Z clearance and Z depth are incrementally negative from the previous Z axis coordinate of the program.

Once entered a canned cycle remains active until it is shut off with a G80 DEACTIVATE CYCLE or another cycle is enabled with a G79 RE-ENABLE CYCLE or another canned cycle block is encountered.

When a cycle is enabled it will be automatically executed at the end of all subsequent programmed moves.

The PNC 4 'remembers' which canned cycles have been entered. Only the most recent of each type of canned cycle is 'remembered'. The most recent canned cycle encountered is the 'active cycle'.

Cutter compensation is used in circular and rectangular cycle execution. The current tool diameter is used. If the tool diameter is not set, or is too large then an error will be given.

11.2 G79 RE-ENABLE CYCLE

The re-enable cycle block is available in two forms:-

- a) Unconditional
- b) Conditional

An unconditional re-enable cycle block when executed will simply re-enable the current 'active cycle'. A conditional re-enable cycle block when executed will make the cycle selected, the 'active cycle' and enable it. An attempt to re-enable a cycle which has not been entered and 'remembered' by the PNC 4 will give an error.

The data required to specify a re-enable cycle is:-

- a) The G code only for unconditional type.
- b) The G code and cycle type selection for conditional type.

During input the user is prompted to select the type of cycle to enable. If the EOB key is pressed at this point then an unconditional re-enable cycle block is specified.

11.3 G80 DEACTIVATE CYCLE

The deactivate cycle block when executed will prevent the execution of the current 'active cycle' after subsequent programmed moves.

The data required to specify a deactivate cycle is the G code only.

11.4 G82 CIRCULAR CYCLE

The circular cycle when encountered in a program becomes the current 'active cycle' and is enabled.

The data required to specify a circular cycle is:-

- a) The radius of the circular pocket.
- b) The feed rate of the X and Y axes when in cycle.
- c) The total depth of the pocket.
- d) The number of cuts required to reach the required depth.
- e) The clearance above the workpiece at the start of the cycle.

11.5 G83 DRILLING CYCLE

The drilling cycle when encountered in a program becomes the current 'active cycle' and is enabled.

The data required to specify a drilling cycle is:-

- a) The total depth of the hole.
- b) The feedrate of the Z axis when in cycle.
- c) The number of cuts required to reach the required depth.
- d) The clearance above the workpiece at the start of the cycle.

11.6 G84 RECT-LAR CYCLE

The rectangular cycle when encountered in a program becomes the current 'active cycle' and is enabled.

11.7 RS232 Format

The PNC 4 expects to receive and will transmit the blocks as follows:-

(STX)G79L(CR) unconditional re-enable
(STX)G79CL(CR) circular cycle re-enable
(STX)G79DL(CR) drilling cycle re-enable
(STX)G79RL(CR) rectangular cycle re-enable

(STX)G80L(CR)
(STX)G82RrrrrF(XorY)ffff(Z)ddddCcccf(Z)zzzzL(CR)
(STX)G83(Z)dddF(Z)ffffCcccc(Z)zzzz(CR)
(STX)G84(X)xxxx(Y)yyyyF(XorY)ffff(Z)ddddCcccc(Z)zzzzL(CR)

Notes:-

rrrr = radius of pocket
ffff = feed value
dddd = depth of pocket
cccc = number of cuts
zzzz = clearance value
xxxx = X dimension of pocket
yyyy = Y dimension of pocket

Axis identifiers shown in parenthesis depend on set up. If a 3rd axis feed is present in G82 and G84 then it will be ignored by the PNC 4.

If no clearance is present then the PNC 4 will default to 1 unit (depending on set up).

11.8 Sample Program

<u>Block</u>	<u>Remarks</u>
1	Moves to X50 Y50 Z52.
2	Executes a circular pocket of radius 25 and depth 25 with a clearance of 5, taking 2 cuts to reach depth.
3	Ensures that the circular pocket is not executed at the end of the next move.
4	Moves to X100 Y100.
5	Executes a rectangular pocket of X50, Y50, depth 25, clearance 5, taking 3 cuts to reach depth.
6	Ensures that the rectangular cycle is not executed at the end of the next move.
7	Moves to X150 Y150.
8	Executes a drilling cycle of depth 25 at Z feed of 29.9%, clearance 5, taking 5 cuts to reach depth.
9	Changes current 'active cycle' to the circular cycle defined in block 2.

- 10 Moves to X100 then executes the circular pocket cycle.
- 11 Changes current 'active cycle' to the rectangular cycle defined in block 5.
- 12 Moves to X50 then executes the rectangular pocket cycle.

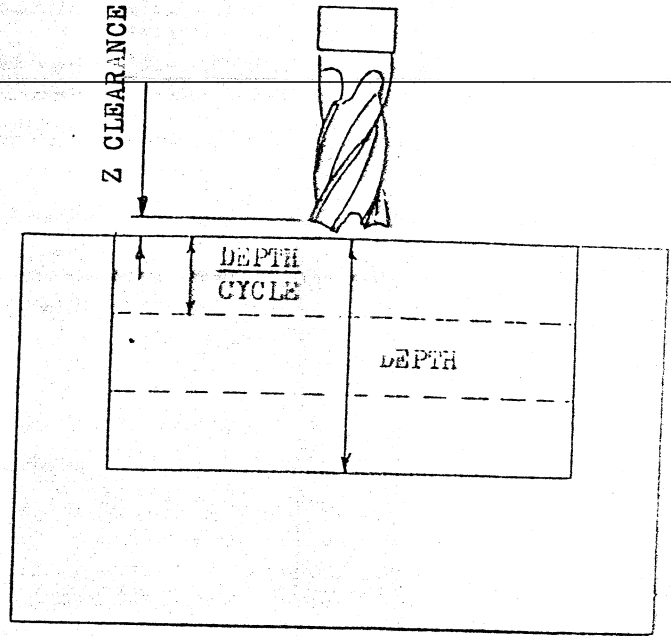
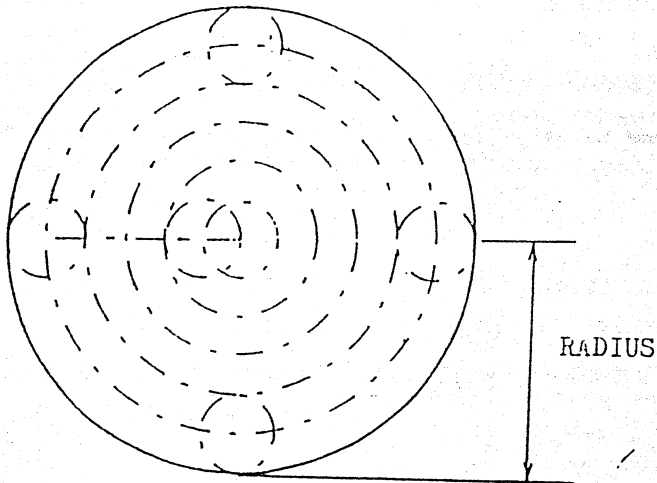
- 13 Changes current 'active cycle' to the drilling cycle defined in block 8.
- 14 Moves to X0 then executes the drilling cycle.
- 15 Prevents execution of the 'active cycle' after all subsequent moves.
- 16 Moves to X0 Y0.

The data required to specify a rectangular cycle is:-

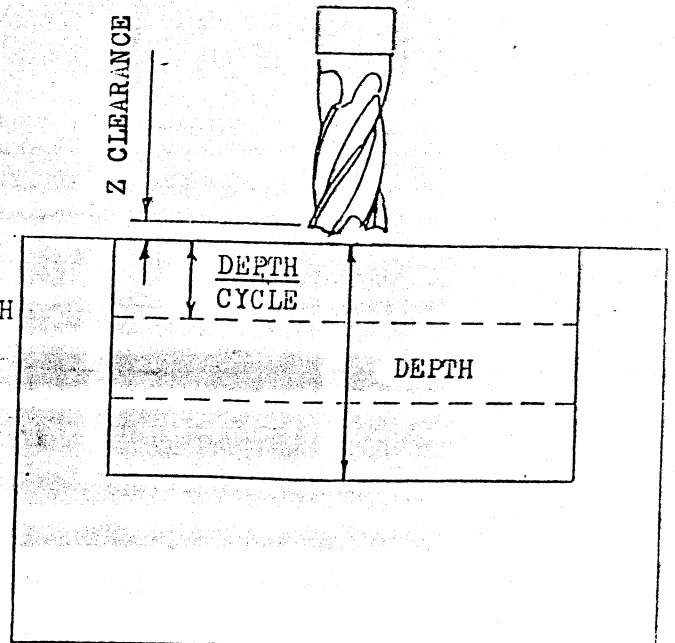
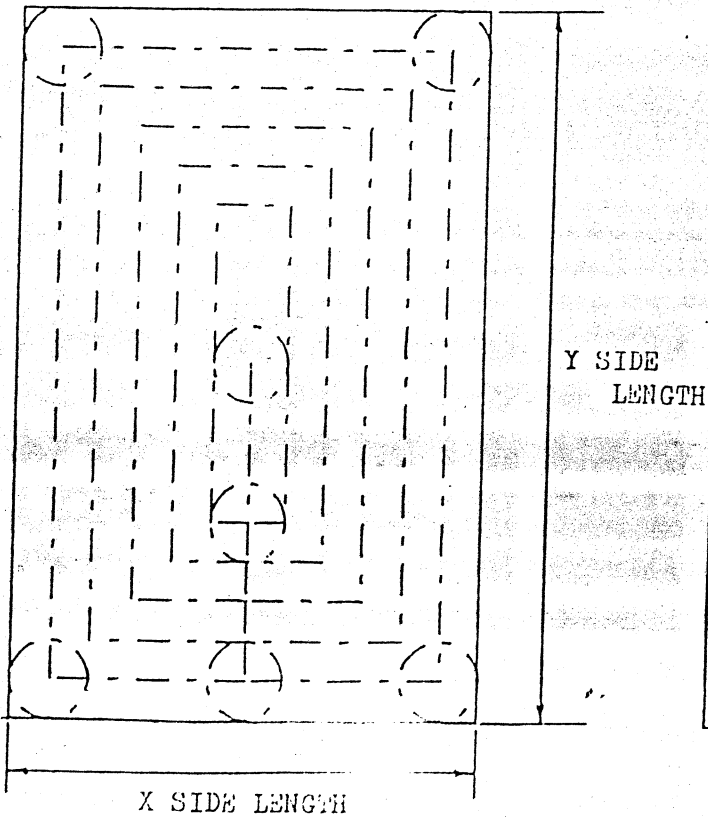
- a) The length of the pocket along the X axis.
 - b) The length of the pocket along the Y axis.
 - c) The feed rate of the X and Y axes when in cycle.
 - d) The total depth of the pocket.
 - e) The number of cuts required to reach the required depth.
 - f) The clearance above the workpiece at the start of the cycle.
-

G82 CIRCULAR CYCLE

----- CUTTER PATH



G84 RECT-LAR CYCLE



12.1 Wait for Serial Input Facility (Option)

The wait for serial input facility on the PNC 4 is only available within a programmed sequence.

There is no dedicated key on the PNC 4 for this facility. The only method of entering a wait for serial block is by using its M code.

12.2 M22 SERIAL INPUT

The wait for serial input block is available in two forms:-

- a) Unconditional
- b) Conditional

The unconditional wait for serial input block when executed will halt program execution until any character is received from the PNC 4 RS232 data line. The PNC 4 will then continue program execution.

The conditional wait for serial input block when executed will halt program execution until the specified character is received from the PNC 4 RS232 data line. The PNC 4 will then continue program execution.

The data required to specify a wait for serial input block is:-

- a) The M code only for unconditional type.
- b) The M code and the required character for conditional type.

During input the user is prompted for the character required. If the EOB key is pressed at this point then an unconditional wait for serial input block is specified.

The PNC 4 will only accept character values between 0 and 127. If the character selected conforms to a printable ascii code (32 to 122) then the PNC 4 will display the ascii value in parenthesis after the numeric value.

12.3 PRINT FORMAT

The PNC 4 will print the block as follows:-

Unconditional

Nnnnnn M22 SERIAL INPUT

Conditional

Nnnnnn M22 SERIAL INPUT xxx (a)

Note! xxx = character value 0 to 127
a = ascii equivalent if possible

12.4 Sample Program

<u>Block</u>	<u>Remarks</u>
1	Wait until the character (decimal 49, ascii 1) is received on the RS232 line.
2 to 6	Moves in X axis.
7	Wait until any character is received on the RS232 line.
8 to 12	Moves in X axis.
13	Wait until the character (decimal 52, ascii 4) is received on the RS232 line.
19 to 24	Moves in X axis.

N1	M22 SERIAL INPUT			49 (1)				
N2	G01 LINEAR	X	10.00Y		F	Z	F	m. m.
N3	G01 LINEAR	X	20.00Y		F	Z	F	m. m.
N4	G01 LINEAR	X	30.00Y		F	Z	F	m. m.
N5	G01 LINEAR	X	40.00Y		F	Z	F	m. m.
N6	G01 LINEAR	X	50.00Y		F	Z	F	m. m.
N7	M22 SERIAL INPUT							
N8	G01 LINEAR	X	60.00Y		F	Z	F	m. m.
N9	G01 LINEAR	X	70.00Y		F	Z	F	m. m.
N10	G01 LINEAR	X	80.00Y		F	Z	F	m. m.
N11	G01 LINEAR	X	90.00Y		F	Z	F	m. m.
N12	G01 LINEAR	X	100.00Y		F	Z	F	m. m.
N13	M22 SERIAL INPUT			20				
N14	G01 LINEAR	X	90.00Y		F	Z	F	m. m.
N15	G01 LINEAR	X	80.00Y		F	Z	F	m. m.
N16	G01 LINEAR	X	70.00Y		F	Z	F	m. m.
N17	G01 LINEAR	X	60.00Y		F	Z	F	m. m.
N18	G01 LINEAR	X	50.00Y		F	Z	F	m. m.
N19	M22 SERIAL INPUT			52 (4)				
N20	G01 LINEAR	X	40.00Y		F	Z	F	m. m.
N21	G01 LINEAR	X	30.00Y		F	Z	F	m. m.
N22	G01 LINEAR	X	20.00Y		F	Z	F	m. m.
N23	G01 LINEAR	X	10.00Y		F	Z	F	m. m.
N24	G01 LINEAR	X	0.00Y		F	Z	F	m. m.

13.1 GRAPHICS FACILITY (TPG key)

When you enter Tool Path Graphics mode you will notice that the display will now show different information.

The Graphics display has a machine status area similar to that of the standard PNC4 display showing the status of Tool, Contacts, Auxiliaries and Inputs, also the present program number is displayed. The extra information shown on the Graphics display is the direction of view the magnify factor and the cutter path switch setting, all of which are described in this section.

Tool Path Graphics enables the program in memory to be tested prior to cutting metal. This pre-run will give full error detection, identical to that given at run time. A graphical representation of the programmed profile can be given in four views - plan, side, front and isometric also with an added feature of drawing the cutter and the cutter path.

Any point on the graphics display may be enlarged to bring out the detail of the programmed profile. When in the tool path graphics mode a copy of the graphics display may be produced on a printer.

CURSOR MOVEMENT

The graphics display has a cursor which is moved around the screen using the MANUAL keys. The position of the cursor is displayed at the bottom of the graphic screen, the axes in which the cursor position is displayed is dependant upon the view, (no cursor position is given in Isometric). The cursor is also used in magnifying, for the display is magnified about the cursor position.

1. INITIALISE

Returns the cursor to the centre of the graphics display and the cutter option is turned off. The magnification factor is reset to 1, which allows all of the table area to be displayed on the graphics screen. The loaded program is then plotted to the graphics display starting from the Absolute Datum position.

2. MAGNIFY

Magnify allows selected parts of the plotted program to be enlarged by a factor between 1 to 99. To select an area to be enlarged the cursor must be placed in the centre of this area. Then by selecting the magnify option and entering the required factor of enlargement, the screen is firstly centralized around the cursor and magnified about the centre of the display area. The loaded program is then plotted to the graphics display.

3. VIEW

View displays a menu of viewing directions of the programmed profile:-

- 1) PLAN VIEW
- 2) SIDE VIEW
- 3) FRONT VIEW
- 4) ISOMETRIC
- 5) CUTTER

When an option is selected the relevant view is displayed in the machine status area of the graphics screen.

CUTTER

Cutter activates the cutter facility. When this facility is not active only the programmed profile is plotted on the screen.

Cutter is displayed in the machine status area, when this facility is activated. With cutter activated the program is always shown by a two pass plot. The first pass being the tool path, which is always shown by a chained line. A circle is drawn after each plot of a linear or a circular block which is to represent the cutter diameter. The plot on the second pass represents the programmed profile.

4. RAPID

Rapid regenerates the graphics screen as quickly as possible without displaying the program blocks.

5. CONTINUOUS

Continuous continues the program plot from the present block, each block is displayed on the Triac screen and the path is plotted on the graphics display. The plot can be stopped by pressing the <RESET> key, and restarted by either a single step or another continuous.

6. SINGLE STEP

Single Step executes the same as continuous but stopping after each block, and waiting for either another single step or a continuous.

7. PRINTER

Printer gives the facility to produce a copy of the graphics screen on an Epson RX80 or RX100 printer.

8. GRAPHICS SET-UP

This allows the user to select the type of line drawn ie. solid and dashed or solid and hidden depending on the state of an auxiliary or contact or Z position.

When entering graphics set-up, a menu is displayed

```
SOLID LINES AND DASHED LINES
SOLID LINE DEFINED BY AUX
SOLID LINE DEFINED BY CONTACT
SOLID LINE DEFINED BY Z AXIS
```

The cursor will be positioned on then first menu selection. The user can select dashed or hidden lines by use of the 1 & 2 keys. If the cursor is moved on to the next menu entry, then the user can select solid lines to be drawn if a selected auxiliary is on or off. If this is not required then the user can press <ENTER> to disable the feature. In the same way, solid lines can be drawn depending on the state of either contact 1 or contact 2 by moving the cursor to the next menu entry. If the cursor is moved to the next menu entry, then the user can select solid lines depending on the position of the Z axis. The choices are (=) equal to, <> not equal to, < less than or > than.

The user must select one of these options then enter the Z position required,

e.g. If < 5.00 is selected, then solid lines would be drawn whenever the Z axis position in the program is lower than 5.00.

Pressing <RESET> exits from the graphics set up function.

To exit graphics mode and return to standard PNC4 display, press <TPG>.

14.1 LATHE CYCLE FACILITY

There are three new functions on the PNC 4 if the lathe option is specified.

1) G85 TURNING CYCLE

This function allows the user to carry out stock removal in the bed direction by entering a single block. Stock removal can be carried out at angles of between 0° and 45° from the bed centre line.

2) G86 FACING CYCLE

This function allows the user to carry out stock removal in the cross slide direction by entering a single block. Stock removal can be carried out at angles of between 0° and 45° from the normal cross slide axis.

3) G33 THREAD

This function allows the user to cut single or multiple start threads. By entering a single block threads can be cut at angles of between 0° and 90° from the bed centre line, therefore scroll threads are possible.

Turning Cycle

There is only one block (G85) which covers straight and taper turning. Any taper angle between -45° and $+45^\circ$ is allowed. (0° being straight turning). If an angle of greater than 45° is required then a facing cycle would be used. A turning cycle is executed relative to the current position i.e. the user must ensure that the previous blocks result in the machine being at the correct position. To enter a turning cycle the user must:-

- a) Select G85.
- b) Enter the length +ve or -ve with respect to the Z start position.
- c) Enter the taper angle (see fig.1 for sense of angle).
- d) Enter the overall depth +ve or -ve with respect to the X start position.
- e) Enter the number of cuts 1 to 99 that are required to reach the required depth.

See examples attached for sample turning cycles and their associated blocks.

Facing Cycle

There is only one block (G86) which covers straight taper facing. Any taper angle between -45° and $+45^\circ$ is allowed. (0° being straight facing). If an angle greater than 45° is required then a turning cycle would be used. A facing cycle is executed relative to the current position. i.e. the user must ensure that the previous blocks result in the machine being at the correct position. To enter a facing cycle the user must:-

- a) Select G86.
- b) Enter the length +ve or -ve with respect to the X start position.
- c) Enter the taper angle. (See fig.2 for sense of angle)
- d) Enter the overall depth +ve or -ve with respect to the Z start position.
- e) Enter the number of cuts 1 to 99 required to reach required depth.

See examples attached for sample facing cycles and their associated blocks.

Threading

~~There is only one block which covers straight, tapered, single and multistart threads. There is no distinction required between external and internal threads. Any compound thread angle between 0° and 90° is possible. Any angle between -90° and 90° is available, giving the possibility of scroll threads. To enter a screwcutting block the user must:-~~

- a) Select G33.
- b) Select the compound thread angle required.
- c) Enter the X start position. E.g for an external thread using a front tool then the start of the thread would be used. For an internal thread using a rear tool then the start radius minus the thread depth would be used. This method avoids the need to distinguish between external and internal threads (as other controls).
- d) Enter the Z start position.
- e) Enter the number of starts required 1 to 99.
- f) Enter the length +ve or -ve with respect to the Z start position.
- g) Enter the taper angle -90° to $+90^{\circ}$ to a resolution of 0.01° . (See fig.1 for sense of angle)
- h) Enter the depth, +ve or -ve, with respect to the X start position.
- i) Enter the number of cuts to reach required depth 1 to 99.
- j) Enter the pitch.

It should be noted that the X and Z positions programmed should be the actual start position of the thread. The PNC 4 will use this position to calculate how far away it has to position itself from the thread to ensure that the machine acceleration is completed before the cut starts.

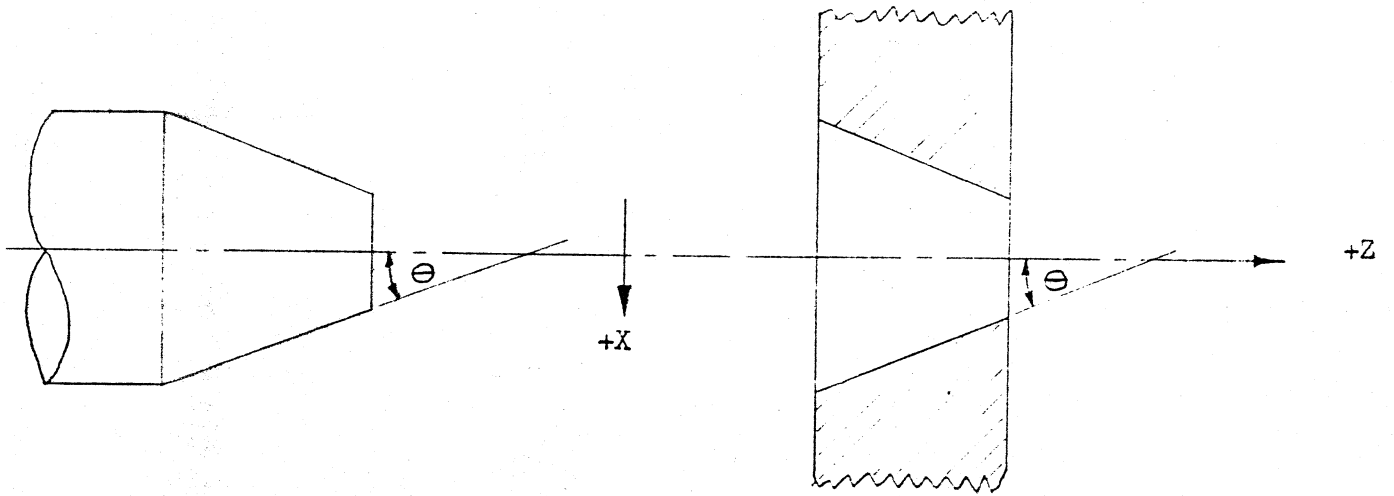
Fig. 1.

Sense of angle for turning cycles and screwcutting.

Negative angle

EXTERNAL

INTERNAL



Positive angle

EXTERNAL

INTERNAL

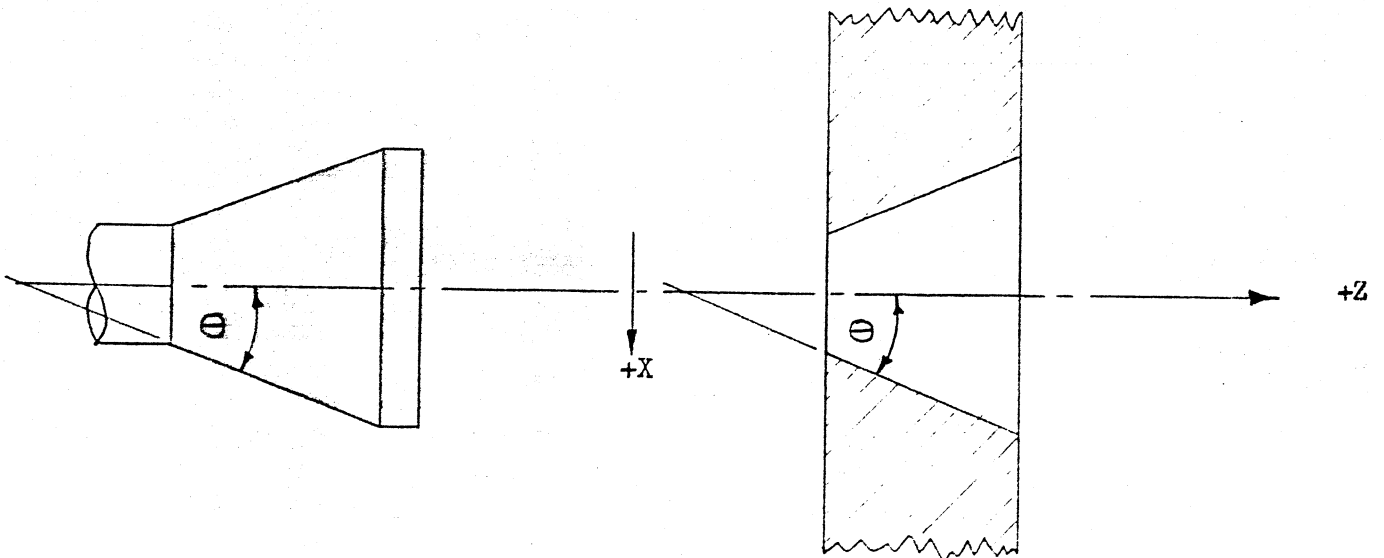
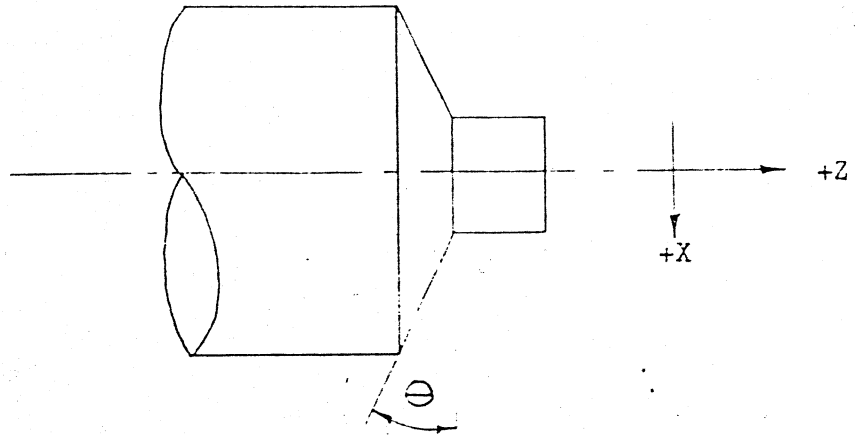


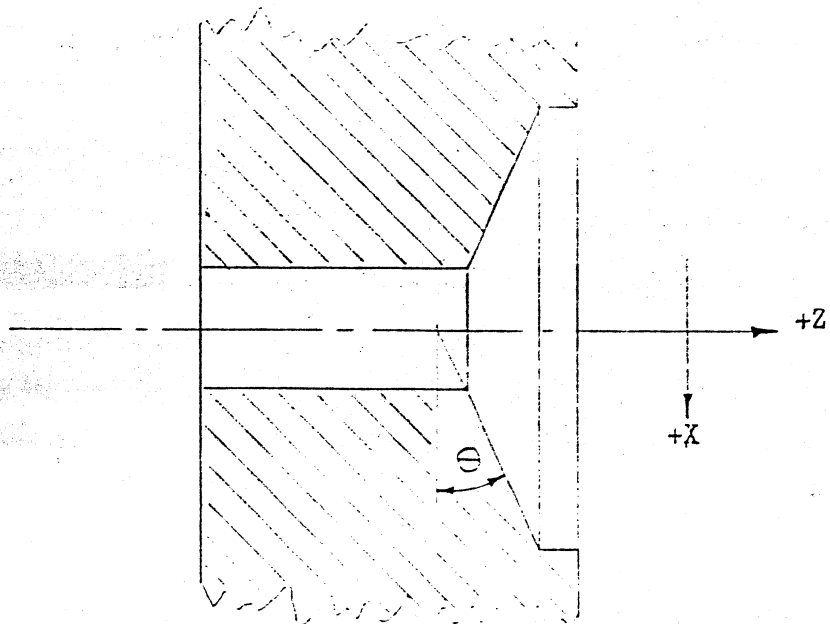
Fig. 2.

Sense of angle for facing cycles.

Negative angle

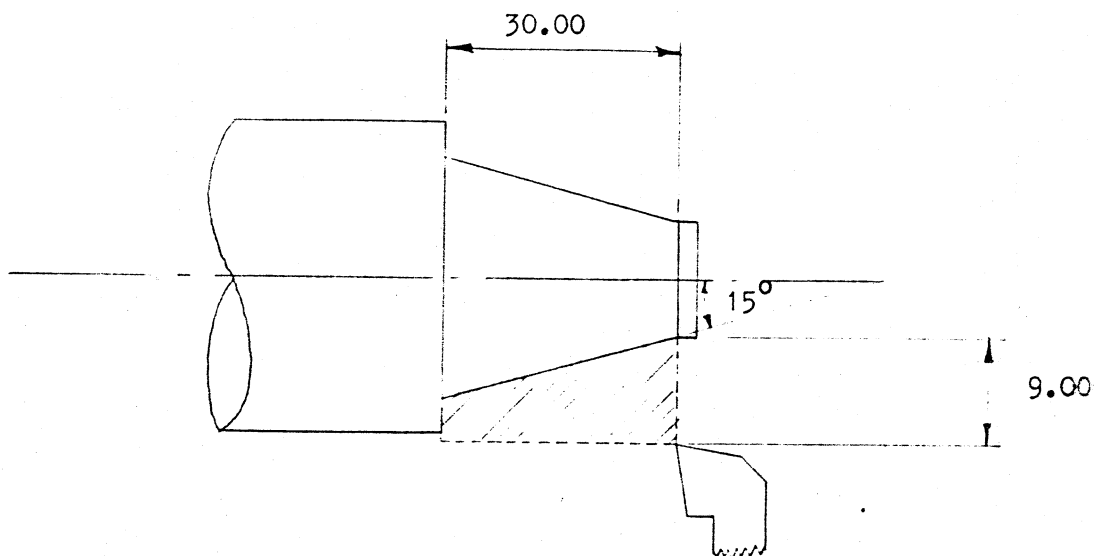


Positive angle



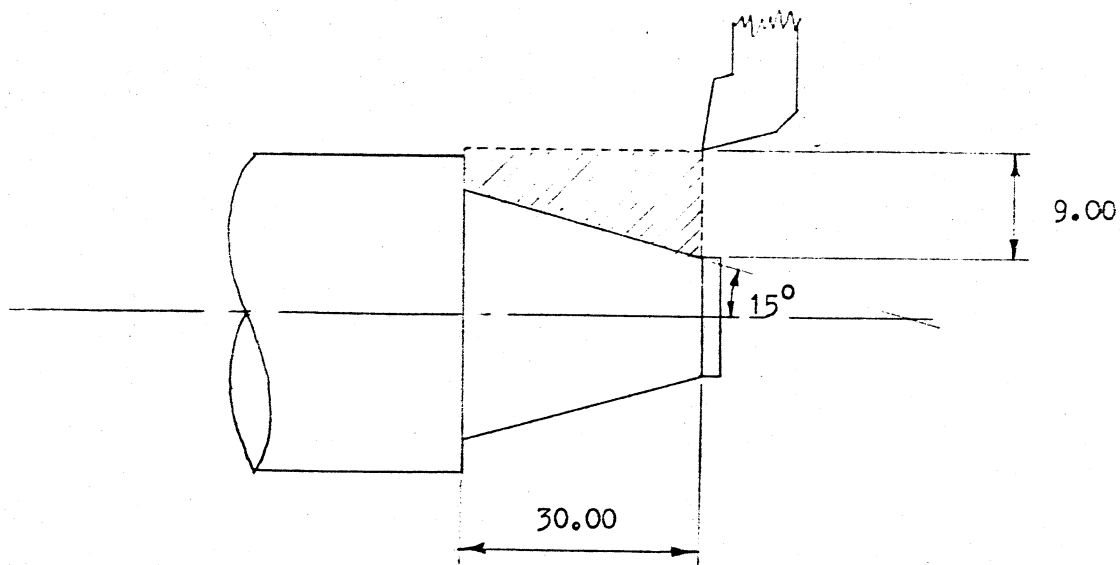
Turning Cycle Examples

External (front tool)



Nnnnnn G85 TURNING CYCLE
LENGTH -30.00 TAPER ANGLE -15.00
DEPTH -9.00 CUTS 4

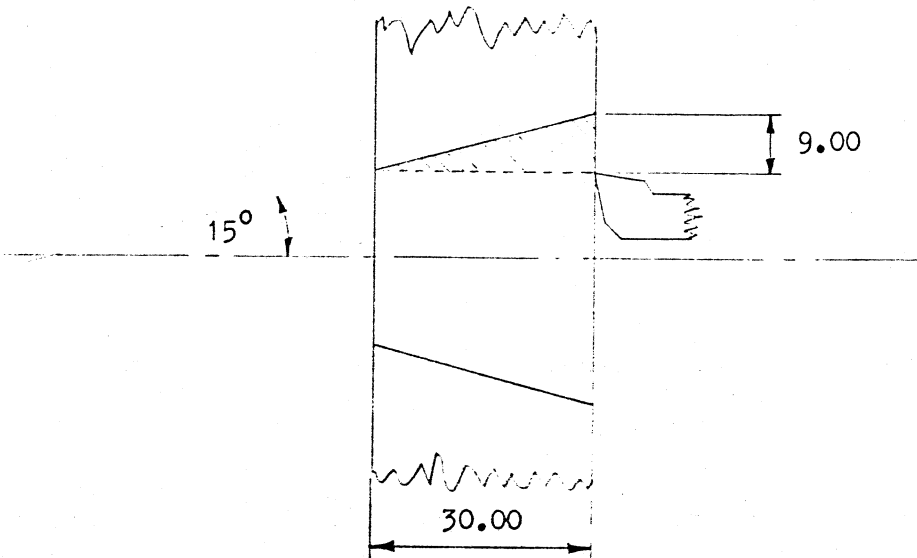
External (rear tool)



Nnnnnn G85 TURNING CYCLE
LENGTH -30.00 TAPER ANGLE 15.00
DEPTH 9.00 CUTS 4

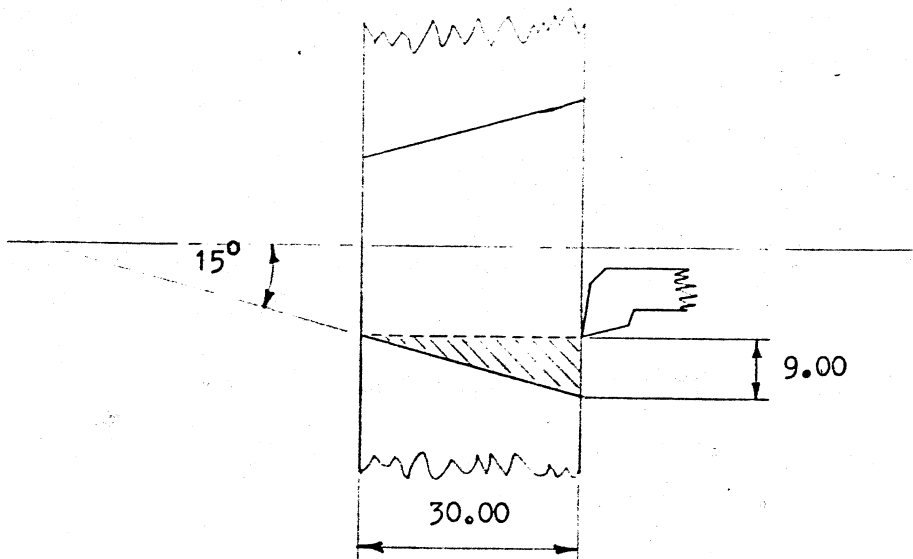
Turning Cycle Examples

Internal (front tool)



Nnnnnn G85 TURNING CYCLE
LENGTH -30.00 TAPER ANGLE-15.00
DEPTH -9.00 CUTS 4

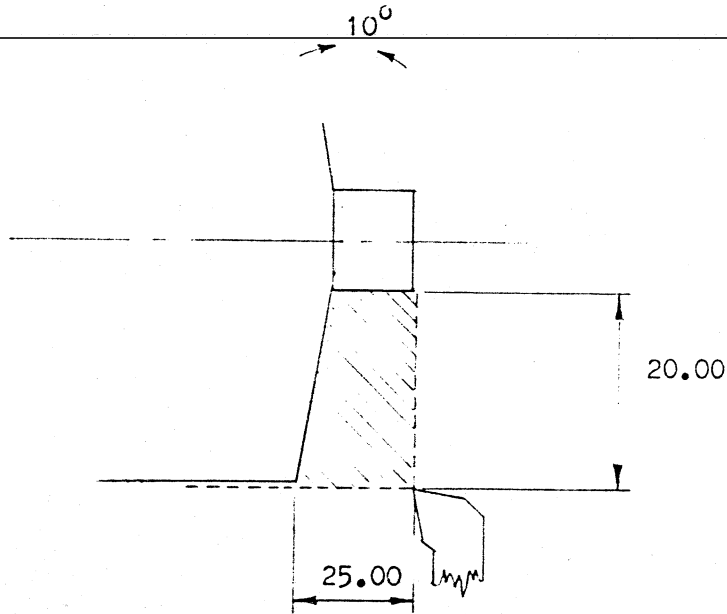
Internal (rear tool)



Nnnnnn G85 TURNING CYCLE
LENGTH -30.00 TAPER ANGLE 15.00
DEPTH 9.00 CUTS 4

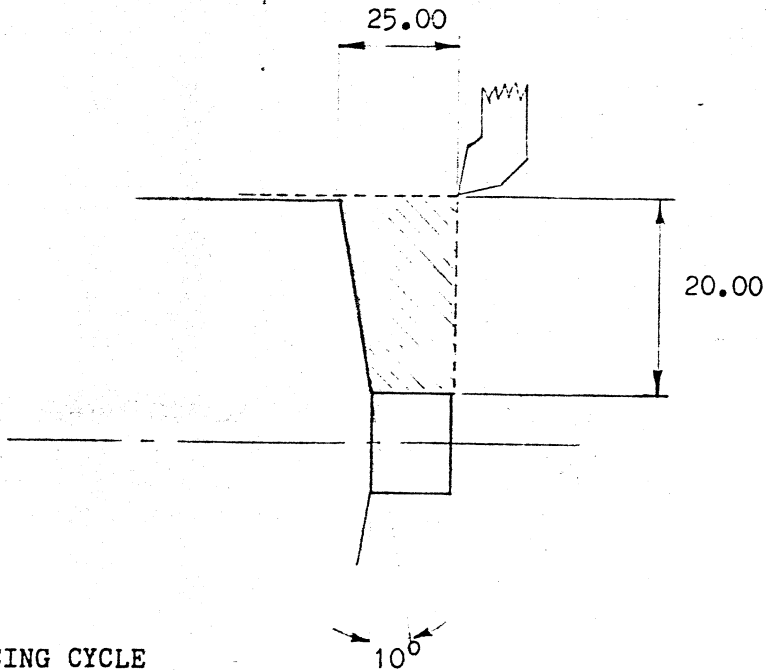
Facing Cycle Examples

Front Tool



Nnnnnn G86 FACING CYCLE
LENGTH -20.00 TAPER ANGLE -10.00
DEPTH -25.00 CUTS 7

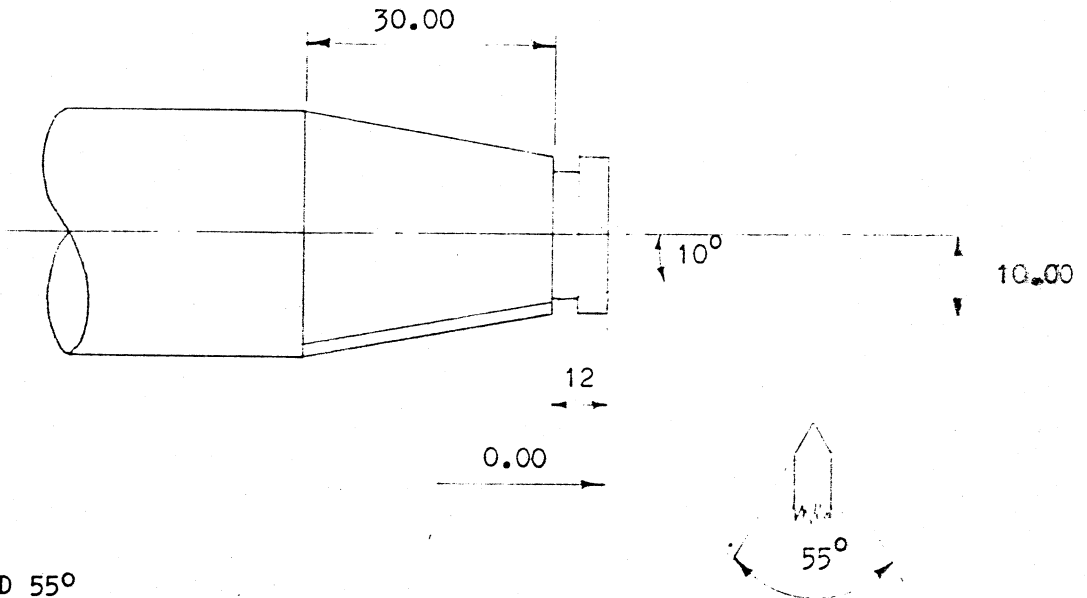
Rear Tool



Nnnnnn G86 FACING CYCLE
LENGTH 20.00 TAPER ANGLE 10.00
DEPTH -25.00 CUTS 9

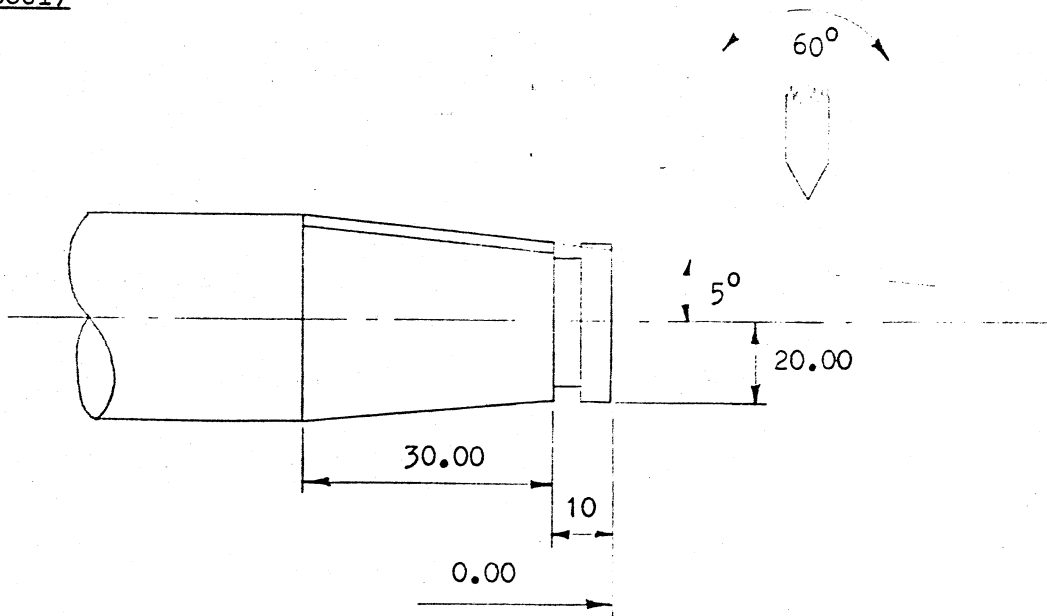
Threading Examples

External (front tool)



Nnnnnn G33 THREAD 55°
 X 10.00 Z -12.00 STARTS 1
 LENGTH -30.00 TAPER ANGLE-10.00
 DEPTH -0.92 CUTS 5 PITCH 1.00

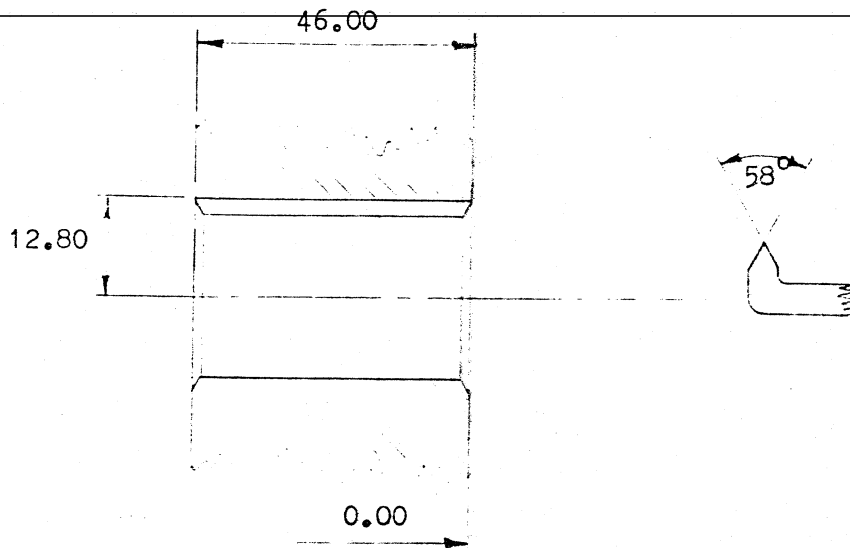
External (rear tool)



Nnnnnn G33 THREAD 60°
 X-20.00 Z -10.00 STARTS 2
 LENGTH -30.00 TAPER ANGLE 5.00
 DEPTH 1.60 CUTS 7 PITCH 3.80

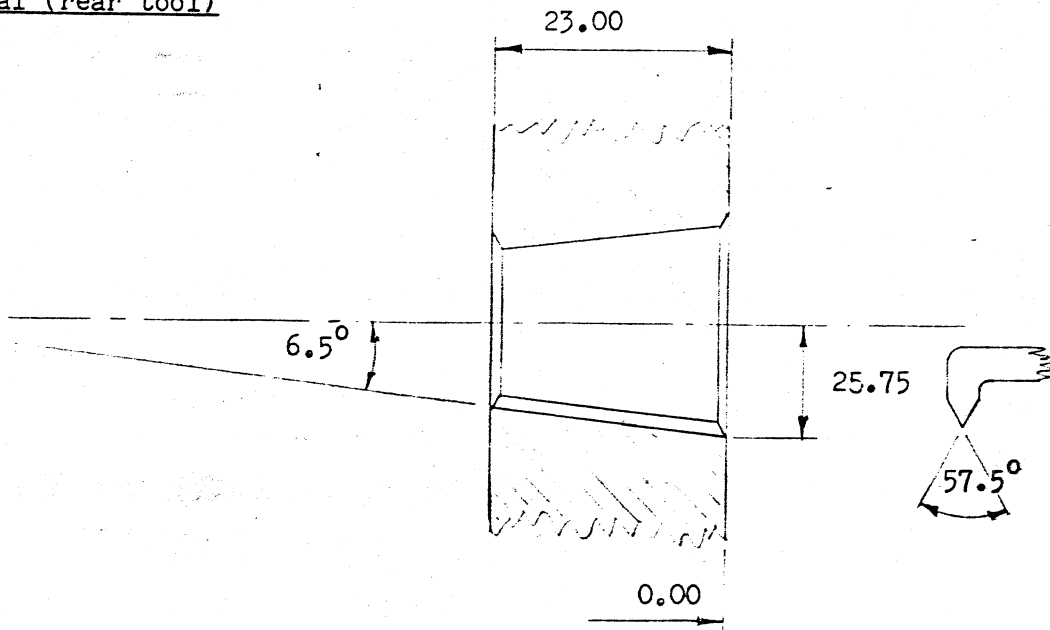
Threading Example

Internal (front tool)



Nnnnnn G33 THREAD 58°
X -12.00 Z 0.00 STARTS 3
LENGTH -50.00 TAPER ANGLE 0.00
DEPTH -0.80 CUTS 4 PITCH 3.00

Internal (rear tool)



Nnnnnn G33 THREAD 57.50°
X 25.00 Z 0.00 STARTS 1
LENGTH -26.00 TAPER ANGLE 6.50
DEPTH 0.75 CUTS 6 PITCH 0.90

15.1 MOVE UNTIL INPUT (G05)

Move until input is a command which can be used move a single axis at a specified feedrate until an input or combination of inputs becomes true. If the input condition never becomes true then the programmed move will stop when the selected position is reached.

The feedrate is limited to a maximum value of 50% of full speed.

This command is not affected by mirror, scale, offset or rotate functions.

To action this command during input or edit modes:-

Select G05

Select an axis and enter the required coordinate.

Enter the required feedrate (max 50%)

Enter input conditions (see 7.17)

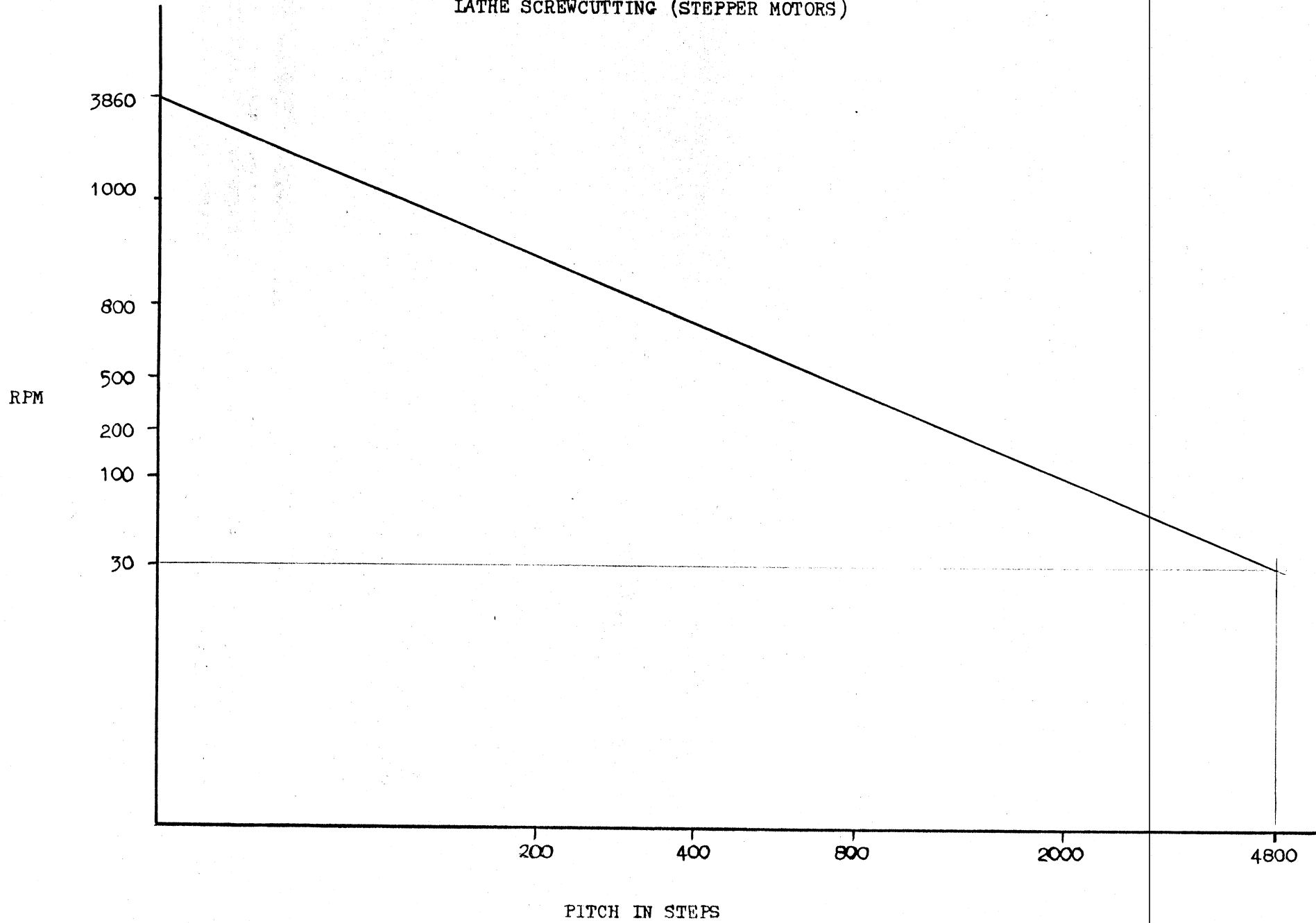
Press EOB

To enter this command via the RS232 link the data must be as follows:-

<STX> G05Xnnnn.nnnnFnnn.nnnIn+-----In-<CR>

This would result in a move in the X axis. To use another axis simply replace the X with Y, Z or d as required.

LATHE SCREWCUTTING (STEPPER MOTORS)



16.1 STOP SUBROUTINE

A 'stop subroutine' is a predetermined set of actions the user requires to happen automatically whenever an external stop signal is received by the PNC4 when executing in AUTO mode. This could be used as a safety feature, for instance to clear a tool head out of the way of an hydraulic ram to prevent damage. The 'stop subroutine' must be entered into the PNC4 as part of the machine setup parameters.

To enter a stop subroutine into the PNC4, use the following sequence.

- 1) Press <SETUP> to enter SETUP mode
- 2) Select 1) EDIT from the menu
- 3) Select 7) SELECTABLE FEATURES from the edit menu
- 4) Select 4) SPECIAL SUBROUTINES from the features menu
- 5) Select 9) STOP SUBROUTINE from the special subroutines menu

The stop subroutine can then be entered or altered using:-

- 1) EDIT or removed using
- 2) DELETE

After entering the stop subroutine using the above method, the user must keep pressing <RESET> to get back to the main setup menu. Setup can then be exited as normal and the changes saved.

RULES

- 1) Only the external stop signal can cause execution of the stop subroutine. The front panel stop button and the remote keyboard stop button will not cause the stop subroutine to execute.
- 2) The stop subroutine is only active when a program is executing in AUTO.
- 3) The external stop signal must latch on, because if the signal ceases, the stop subroutine will abort.
- 4) If the PNC4 is switched out of AUTO into MDI whilst the stop subroutine is executing:-
 - a) The stop subroutine will halt.
 - b) A warning will be displayed.

If <RESET> is pressed, the stop subroutine will be aborted. If any other key is pressed, the stop subroutine will continue.

- 5) If the PNC4 is switched to S.STEP then the subroutine can be single stepped. This could be used in order to test the operation of the stop subroutine when first entered.
- 6) When <START> is pressed to resume the program, a block will be displayed showing the position the machine was in when stopped. When <START> is pressed again the PNC4 will execute a move to that position then carry on with the program.
- 7) The front panel and remote keyboard STOP buttons cannot halt the stop subroutine.

Appendix A

ERROR MESSAGES

This section gives a list of the PNC 4's error messages, a description of why the error occurred and, where applicable, a possible solution to the error.

NO PROGRAM LOADED

CAUSE: The PNC 4 displays this message if an attempt is made to enter Single Step or Auto mode without a program being loaded into memory.

SOLUTION: Switch the PNC 4 to MDI mode and load a program into memory using either the keyboard, cassette or RS232 data link.

MEMORY FULL

This error message is displayed if an attempt is made to enter more blocks than the PNC 4's memory can hold.

MOVE EXCEEDS LIMIT

CAUSE: A move which takes the PNC 4 outside the machine limits will cause this message to be displayed.

SOLUTION: Make sure that all moves are within the machine limits (remembering to add on offsets, compensate for scaling, etc.).

LIMIT EXCEEDED IN X/Y/Z

The PNC 4 has detected an overtravel condition in the specified axis i.e. it has driven past the machine limits. The only action that can be taken after an overtravel fault is to datum the machine.

DRIVE FAULT IN X/Y/Z

CAUSE: This message is displayed if the motor drive unit appears to be faulty.

The error could be due to several factors:-

- 1) There is no power supply to the motor drive
- 2) The motor drive has been overloaded. In this case, the top red LED on the specified drive unit will be illuminated.
- 3) There has been a supply failure. This will cause the second red LED to be illuminated.
- 4) The motor drive unit has overheated. This causes the third red LED to be illuminated.
- 5) The motor drive has recently been powered down and powered up in quick succession.

TAPE PROTECTED

CAUSE: If the "write enable" tab on the tape currently in the cassette unit has been removed, it is impossible for the PNC 4 to store data on the tape.

SOLUTION: Put a "write enable" tab on the tape housing.

e.g. by placing a small piece of sticky tape over the tab position.

NO TAPE LOADED

CAUSE: This message is displayed if an attempt is made to access the PNC 4's cassette functions while there is no tape in the cassette unit.

SOLUTION: Put a tape in the cassette unit before using the PNC 4's cassette functions.

TAPE ERROR

CAUSE: When reading a paper tape and a parity error occurs or an invalid block is decoded. Or when reading a cassette the PNC 4 cannot get into synchronization with the program header data on the tape. This could be caused by tape stretching or data corruption due to magnetic fields etc.

SOLUTION: Try reading the tape again. Remember always to store the program 3 times on the the cassette.

SCALED MOVE OUT OF RANGE

If a scaling operation causes the move to exceed 8388608 steps, the above message is displayed.

DIAMETER WARNING BLOCK

If the PNC 4's cutter radius compensation facility is being used and it cannot calculate an exact position, then it will use an approximate position instead. This will only happen if the block concerned defines an intersect between 2 axes which are very nearly, but not quite, blended. If this situation occurs then the above message is displayed, but the PNC 4 will continue executing the rest of the program.

SUBROUTINE NOT IN MEMORY

This error message is displayed when there has been an attempt to delete a subroutine or execute a subroutine call, to a subroutine which has not been loaded into memory.

SOLUTION: If more than one of the red LEDs is illuminated, the drive is almost certainly faulty. If the supply failure LED is illuminated and the supply to the motor drive is correct, the drive is faulty. If the overload LED is illuminated, disconnect the motor and switch on again. If the overload LED still comes on, the drive is faulty. If not, check for short circuits or crossed connections on the motor leads. If the overtemperature LED is on, switch off the motor drive unit and allow plenty of time for it to cool down.

REPEAT NEST ERROR

CAUSE: The PNC 4 will only implement nine nested repeat loops at any time (A "nested" repeat is a repeat of a repeat). This error indicates that a tenth level of nesting has been attempted.

SOLUTION: Simplify the program.

START LABEL NOT FOUND

CAUSE: A Repeat Block in the program has specified a Start Label which does not exist.

SOLUTION: Use EDIT to insert a label at the correct position in the program.

END LABEL NOT FOUND

CAUSE: A repeat block in the program has specified an End Label which does not exist.

SOLUTION: Use EDIT to insert a label at the correct position in the program.

END LABEL FOUND BEFORE START

CAUSE: A Repeat block in the program has specified an End Label which occurs earlier in the program than the start label.

SOLUTION: Use EDIT to put the labels in the correct place.

REPEAT WITHIN OWN LABELS

CAUSE: The Repeat block is within the part of the program to be repeated.

SOLUTION: Reposition the labels or the Repeat block. Remember, the area of program between the start and end label can occur after or before the Repeat block itself. But the Repeat block must not be positioned within its own start and end labels.

SUBROUTINE CALL ERROR

This error message is displayed when a subroutine call is executed within a subroutine.

SUBROUTINE RETURN ERROR

This error will occur if a subroutine return is executed before a subroutine call.

MOVE DATA > 1048575 STEPS

This error message is displayed if a move is entered or scaled above the internal limits of the controller.

ERROR - DEFAULT SET-UP LOADED

This occurs at power on if the battery backed RAM has failed. Use SET UP to re-store the correct values. If the problem persists then contact your equipment supplier.

SET-UP PROTECTED

This occurs when the user has tried to set the systems parameters when switch A6 = OFF. RESET and EXIT set up mode without saving the new values.

ERROR - TOOL OFFSETS CLEARED

This occurs at power on if the battery backed RAM containing the tool offset library has failed.

ROM ERROR

This occurs at power on if the PNC 4 ROM contents have been lost. Refer this problem to your equipment supplier.

RAM ERROR

This error is displayed at power on if the RAM is found to be permanently faulty. Refer this problem to your equipment supplier.

8031 IS IN TEST MODE

CAUSE: 8031 TEST MODE has been enabled.

SOLUTION: Turn switch A4 on the processor board off. (See section 11.1).

INVALID BLOCK

CAUSE: An invalid block has been executed or displayed in EDIT. An invalid block could be for example MIRROR Y when only 1 axis is defined by the SET UP procedure.

SOLUTION: Use EDIT to delete the block, or use SET UP to alter the system parameters.

INCOMPLETE BLOCK

CAUSE: This error will occur on input of a circular move or a Rotate block where not all the parameters have been defined. E.g. on input of a rotate block where the X Y centre has been defined but not the angle of rotation.

SOLUTION: Make sure all necessary values have been input.

ERROR IN CIRCLE

CAUSE: The following conditions will cause this error message to be displayed on execution or input of circular blocks.

- 1) Start point and centre the same.
- 2) End point and centre the same.
- 3) Radius too small (only occurs if block is end point + radius type)
- 4) Start point and end point the same (only occurs if block is end point + radius type)

SOLUTION: Make sure the parameters are correctly defined.

FAULT IN BLOCK n

Pressing the test key when the PNC 4 has been datumed causes a program validation sequence to be entered. This error message occurs if there has been a fault in the program.

MOVE DATA NOT READY

CAUSE: This error indicates that the no stopping routines do not have enough time to work properly.

SOLUTION: Try reducing the speed of axis motion either by turning down the speed pot on the processor board or entering lower feedrates into the program. Alternatively the no stopping process can be disabled by turning switch 4 on the interface board ON. If the problem persists refer it to your equipment supplier.

8085 TO 8031 DATA ERROR

This error message indicates a fault on the main processor board and the problem should be referred to your equipment supplier.

KM ENDED

CAUSE: This message can appear for 2 reasons.

- 1) The PNC 4 has tried to execute an incorrectly programmed circular block.
- 2) The speed pot is turned up too high.

SOLUTION: If the error occurred whilst executing a circular block, then you should check the data in that block and the preceding block. A common mistake is to repeat a circular block without including the block defining the start position. Example:-

```
N1 G01 X50 Y100
N2 M17 LABEL 1
N3 G02 X50 Y100 CX50 Y50
N4 M17 LABEL 2
N5 G01 Y90
N6 G12 MIRROR Y
N7 G81 REPEAT 1 TO 2 REPEATS 1
```

This program will give a KM ENDED error message. The solution in this case is to swap block N1 with N2.

If you are sure that the data in the program is correct then you should consider the possibility that the speed pot is turned up too high. In this case the error signifies that the PNC 4 has lost position. This error is more likely to occur at high feedrates. Try running the program again with a lower feedrate scale. If this cures the problem then you should turn the speed pot down to a lower setting.

If the problem persists then you should contact your equipment supplier.

KM DATA ERROR

This error message indicates a fault on the main processor board and the problem should be referred to your equipment supplier.

8031 NOT READY

This error message indicates a fault on the main processor board and the problem should be referred to your equipment supplier.

TIME-OUT ERROR

This error message indicates a fault on the main processor board and the problem should be referred to your equipment supplier.

AXIS DUPLICATED

CAUSE: This message will be displayed if the same axis name is used more than once when specifying the order in which the axes are to go to datum.

SOLUTION: Make sure each axis name is used only once.

DATUM POSITION ERROR

CAUSE: When an axis which is not specified as Floating Datum is sent to its datum position, it measures the distance taken to move to the datum detector and compares that with the distance it assumed it needed to move. If there is a difference of more than 4 steps then this error will occur.

This situation can occur if the motor has stalled out or the axis has hit an end stop.

SOLUTION: Press RESET and try again. If the problem persists try a longer acceleration ramp by using the SET UP procedure.

PRINTER ERROR

CAUSE: The PNC 4 has timed out waiting for the printer to be ready.

SOLUTION: Make sure that the printer is on-line and working. Also check that it is properly connected to the PNC 4 with a known, good working printer connection lead.

NO SUCH CYCLE

CAUSE: This error message can occur if a re-enable cycle block is encountered for a cycle which has not been entered into the PNC 4.

SOLUTION: Ensure cycle block is entered before re-enable cycle block is executed.

CUTTER DIA. NOT SET

CAUSE: Attempt to execute a circular or rectangular pocket cycle when the current tool diameter is 0.

SOLUTION: Ensure that the tool diameter is set.

Appendix B

INTERNAL SWITCH settings

There are 2 sets of 6 switches in dual-in-line (d.i.l.) packages that are used to configure a number of options inside the PNC 4. These switches are mounted on the interface board and processor board, they are only accessible when the PNC 4's cover has been removed. When the cover has been removed by undoing the fixing screws and sliding it forward, the processor board is between the keyboard and the interface board which is at the rear. One set of dil switches is mounted on the processor board just below the RS232 connector, the other dil switch is mounted on the interface board at the top next to the printer cable. The meanings of each switch are as follows:-

On Processor Board

<u>DIP SW</u> <u>PIN NO.</u>	<u>FUNCTION</u>	<u>FACTORY SET</u> <u>CONDITION</u>
SWA-1	NOT USED	OFF
SWA-2	NOT USED	OFF
SWA-3	ON: 8085 to 8031 transfers check enabled OFF: 8085 to 8031 transfers check disabled	OFF
SWA-4	ON: 8031 test mode enabled OFF: 8031 test mode disabled	OFF
SWA-5	ON = SLOW 8031 OFF = FAST 8031	OFF
SWA-6	NOT USED	OFF

On Interface Board

<u>DIP SW</u> <u>PIN NO.</u>	<u>FUNCTION</u>	<u>FACTORY SET</u> <u>CONDITION</u>
SWA-1	NOT USED	OFF
SWA-2	ON: No stopping limit=15° OFF: No stopping limit=2°	OFF
SWA-3	ON: Default units 'm.m' OFF: Default units are inches	ON
SWA-4	ON: None stopping between axes movement OFF: Stop between axes movement	ON
SWA-5	ON: Single axis on 3rd axis hardware OFF: Single axis on 1st axis hardware	OFF
SWA-6	ON: Setup enable OFF: Setup disabled	ON

NOTE

If interface board SWA-2 is ON, then the PNC 4 will attempt to perform continuous movement on the contouring axes even if consecutive blocks are as much as 15° away from being truly tangential. Using this facility can cause harsh acceleration so it should be used with care. The best solution is always to ensure that your programs blend to within 2°. (See section 2.8 Continuous Constant Feed Movement facility).

Appendix C

G Codes

0	Rapid Traverse
1	Linear
2	Circular CLW
3	Circular CCLW
4	Dwell
5	Move until input
10	Mirror X
11	Cancel Mirror X
12	Mirror Y
13	Cancel Mirror Y
14	Rotate
15	Cancel Rotate
17	X-Y Plane
18	X-Z Plane
19	Y-Z Plane
20	Program Scale
21	Machine Scale
33	Threading Cycle
40	Cancel Tool Comp
41	Tool Comp Left
42	Tool Comp Right
54	Program Offset
55	Machine Offset
70	Imperial Units
71	Metric Units
79	Re-enable cycle
80	Deactivate cycle
81	Repeat/Repeat-Divide
82	Circular cycle
83	Drilling cycle
84	Rect-lar cycle
85	Turning cycle
86	Facing cycle
90	Absolute Units
91	Incremental Units
98	Absolute Datum
99	Floating Datum

M Codes

0	Program Stop
2	End of Program
3	Analogue FWD
4	Analogue REV
5	Analogue OFF
6	Tool Change
8	Contact 1 ON
9	Contact 1 OFF
17	Label
20	Auxiliaries
21	Inputs
22	Serial Input
23	Output A
24	Output B
32	Goto Label
33	Contact 2 ON
34	Contact 2 OFF
98	Call Subroutine
99	Subroutine End

Appendix D

Example Programs

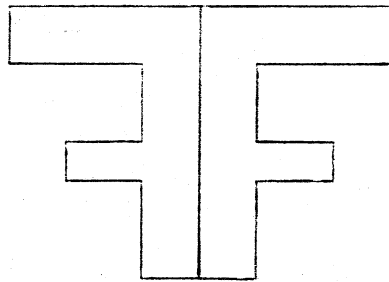
The following pages show some example programs. Each example has been chosen to illustrate a particular feature of the PNC 4. The later examples are more complex than the earlier ones so it is suggested that you work through from the first example to the last only moving on when you fully understand any new features that have been introduced.

The examples were produced by a PNC 4 connected to a 3 axis table with X0, Y0 being the bottom left corner and Z0 being the pen touching the paper. If the mechanics of your system are different or if the table limits are smaller then you will have to alter the programs accordingly.

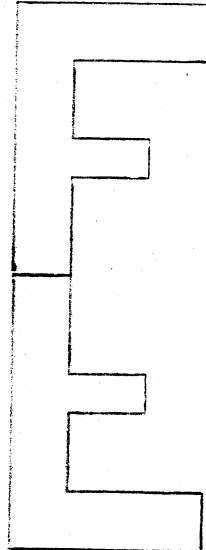
N1	M17 LABEL 1									
N2	G01 LINEAR	X	50.00Y	50.00F	100.0	Z	10.00F	100.0	m.m	
N3	G01 LINEAR	X	Y	F		Z	0.00F	10.0	m.m	
N4	G01 LINEAR	X	Y	85.00F		Z	F		m.m	
N5	G01 LINEAR	X	75.00Y	F		Z	F		m.m	
N6	G01 LINEAR	X	Y	77.50F		Z	F		m.m	
N7	G01 LINEAR	X	57.50Y	F		Z	F		m.m	
N8	G01 LINEAR	X	Y	67.50F		Z	F		m.m	
N9	G01 LINEAR	X	67.50Y	F		Z	F		m.m	
N10	G01 LINEAR	X	Y	62.50F		Z	F		m.m	
N11	G01 LINEAR	X	57.50Y	F		Z	F		m.m	
N12	G01 LINEAR	X	Y	50.00F		Z	F		m.m	
N13	G01 LINEAR	X	50.00Y	F		Z	F		m.m	
N14	G01 LINEAR	X	Y	F		Z	10.00F		m.m	
N15	M17 LABEL 2									
N16	G81 REPEAT-DIVIDE LABEL 1 TO 2			REPEATS 1						
	OFFSETS :-	X	30.00Y	F		Z	F		m.m	
N17	G01 LINEAR	X	0.00Y	0.00F		Z	F		m.m	

FF

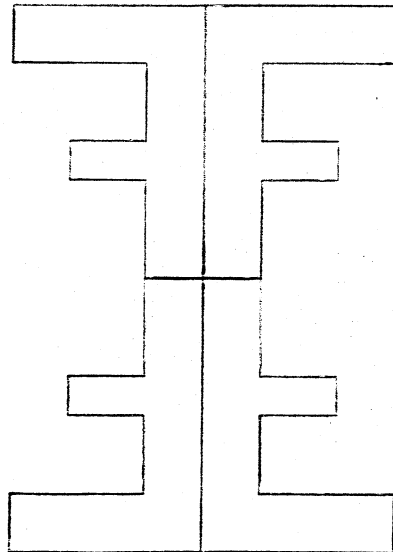
N1	M17 LABEL 1									
N2	G01 LINEAR	X	50.00Y	50.00F	100.0	Z	10.00F	100.0	m.m	
N3	G01 LINEAR	X	Y	F		Z	0.00F	10.0	m.m	
N4	G01 LINEAR	X	Y	85.00F		Z	F		m.m	
N5	G01 LINEAR	X	75.00Y	F		Z	F		m.m	
N6	G01 LINEAR	X	Y	77.50F		Z	F		m.m	
N7	G01 LINEAR	X	57.50Y	F		Z	F		m.m	
N8	G01 LINEAR	X	Y	67.50F		Z	F		m.m	
N9	G01 LINEAR	X	67.50Y	F		Z	F		m.m	
N10	G01 LINEAR	X	Y	62.50F		Z	F		m.m	
N11	G01 LINEAR	X	57.50Y	F		Z	F		m.m	
N12	G01 LINEAR	X	Y	50.00F		Z	F		m.m	
N13	G01 LINEAR	X	50.00Y	F		Z	F		m.m	
N14	G01 LINEAR	X	Y	F		Z	10.00F		m.m	
N15	M17 LABEL 2									
N16	G10 MIRROR X									
N17	G81 REPEAT-DIVIDE LABEL 1 TO 2			REPEATS 1						
	OFFSETS :-	X	Y	F		Z	F		m.m	
N18	G11 CANCEL MIRROR X									
N19	G01 LINEAR	X	0.00Y	0.00F		Z	F		m.m	



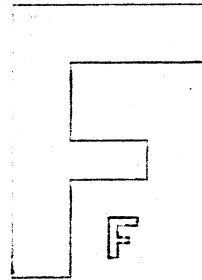
N1	M17 LABEL 1									
N2	G01 LINEAR	X	50.00Y	50.00F	100.0	Z	10.00F	100.0	m.m	
N3	G01 LINEAR	X	Y	F		Z	0.00F	10.0	m.m	
N4	G01 LINEAR	X	Y	85.00F		Z	F		m.m	
N5	G01 LINEAR	X	75.00Y	F		Z	F		m.m	
N6	G01 LINEAR	X	Y	77.50F		Z	F		m.m	
N7	G01 LINEAR	X	57.50Y	F		Z	F		m.m	
N8	G01 LINEAR	X	Y	67.50F		Z	F		m.m	
N9	G01 LINEAR	X	67.50Y	F		Z	F		m.m	
N10	G01 LINEAR	X	Y	62.50F		Z	F		m.m	
N11	G01 LINEAR	X	57.50Y	F		Z	F		m.m	
N12	G01 LINEAR	X	Y	50.00F		Z	F		m.m	
N13	G01 LINEAR	X	50.00Y	F		Z	F		m.m	
N14	G01 LINEAR	X	Y	F		Z	10.00F		m.m	
N15	M17 LABEL 2									
N16	G12 MIRROR Y									
N17	G81 REPEAT-DIVIDE LABEL 1 TO 2			REPEATS 1						
	OFFSETS :-	X	Y	F		Z	F		m.m	
N18	G13 CANCEL MIRROR Y									
N19	G01 LINEAR	X	0.00Y	0.00F		Z	F		m.m	



N1	M17 LABEL 1									
N2	G01 LINEAR	X	50.00Y	50.00F	100.0	Z	10.00F	100.0		m.m
N3	G01 LINEAR	X	Y	F		Z	0.00F	10.0		m.m
N4	G01 LINEAR	X	Y	85.00F		Z	F			m.m
N5	G01 LINEAR	X	75.00Y	F		Z	F			m.m
N6	G01 LINEAR	X	Y	77.50F		Z	F			m.m
N7	G01 LINEAR	X	57.50Y	F		Z	F			m.m
N8	G01 LINEAR	X	Y	67.50F		Z	F			m.m
N9	G01 LINEAR	X	67.50Y	F		Z	F			m.m
N10	G01 LINEAR	X	Y	62.50F		Z	F			m.m
N11	G01 LINEAR	X	57.50Y	F		Z	F			m.m
N12	G01 LINEAR	X	Y	50.00F		Z	F			m.m
N13	G01 LINEAR	X	50.00Y	F		Z	F			m.m
N14	G01 LINEAR	X	Y	F		Z	10.00F			m.m
N15	M17 LABEL 2									
N16	G12 MIRROR Y									
N17	G81 REPEAT-DIVIDE LABEL 1 TO 2			REPEATS 1						
	OFFSETS :-	X	Y	F		Z	F			m.m
N18	G13 CANCEL MIRROR Y									
N19	M17 LABEL 3									
N20	G10 MIRROR X									
N21	G81 REPEAT-DIVIDE LABEL 1 TO 3			REPEATS 1						
	OFFSETS :-	X	Y	F		Z	F			m.m
N22	G11 CANCEL MIRROR X									
N23	G01 LINEAR	X	0.00Y	0.00F		Z	F			m.m



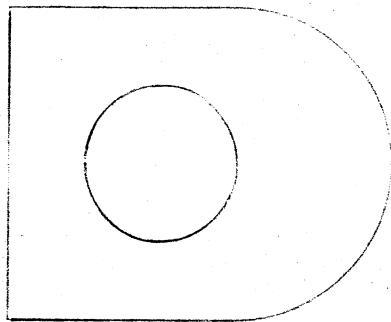
N1	M17 LABEL 1								
N2	G01 LINEAR	X	50.00Y	50.00F	100.0	Z	10.00F	100.0	m.m
N3	G01 LINEAR	X	Y	F		Z	0.00F	10.0	m.m
N4	G01 LINEAR	X	Y	85.00F		Z	F		m.m
N5	G01 LINEAR	X	75.00Y	F		Z	F		m.m
N6	G01 LINEAR	X	Y	77.50F		Z	F		m.m
N7	G01 LINEAR	X	57.50Y	F		Z	F		m.m
N8	G01 LINEAR	X	Y	67.50F		Z	F		m.m
N9	G01 LINEAR	X	67.50Y	F		Z	F		m.m
N10	G01 LINEAR	X	Y	62.50F		Z	F		m.m
N11	G01 LINEAR	X	57.50Y	F		Z	F		m.m
N12	G01 LINEAR	X	Y	50.00F		Z	F		m.m
N13	G01 LINEAR	X	50.00Y	F		Z	F		m.m
N14	G01 LINEAR	X	Y	F		Z	10.00F		m.m
N15	M17 LABEL 2								
N16	G54 PROGRAM OFFSET	X	55.00Y	45.00Z					m.m
N17	G20 PROGRAM SCALE		15.0 %						
N18	G81 REPEAT-DIVIDE LABEL 1 TO 2			REPEATS 1					
	OFFSETS :-	X	Y	F		Z	F		m.m
N19	G20 PROGRAM SCALE		100.0 %						
N20	G54 PROGRAM OFFSET	X	-55.00Y	-45.00Z					m.m
N21	G01 LINEAR	X	0.00Y	0.00F		Z	F		m.m



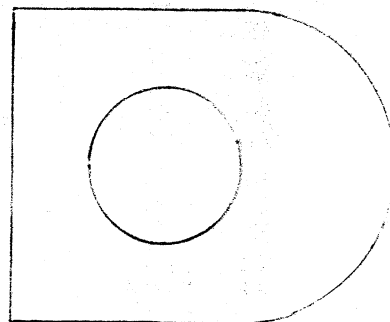
N1	M17 LABEL 1									
N2	G01 LINEAR	X	50.00Y	60.00F	100.0	Z	10.00F	100.0	m.m	
N3	G41 TOOL COMP LEFT									
N4	G01 LINEAR	X	50.00Y	60.00F		Z	F		m.m	
N5	G01 LINEAR	X	Y	F		Z	0.00F	10.0	m.m	
N6	G01 LINEAR	X	Y	85.00F		Z	F		m.m	
N7	G01 LINEAR	X	75.00Y	F		Z	F		m.m	
N8	G01 LINEAR	X	Y	77.50F		Z	F		m.m	
N9	G01 LINEAR	X	57.50Y	F		Z	F		m.m	
N10	G01 LINEAR	X	Y	67.50F		Z	F		m.m	
N11	G01 LINEAR	X	67.50Y	F		Z	F		m.m	
N12	G01 LINEAR	X	Y	62.50F		Z	F		m.m	
N13	G01 LINEAR	X	57.50Y	F		Z	F		m.m	
N14	G01 LINEAR	X	Y	50.00F		Z	F		m.m	
N15	G01 LINEAR	X	50.00Y	F		Z	F		m.m	
N16	G01 LINEAR	X	Y	60.00F		Z	F		m.m	
N17	G40 CANCEL TOOL COMP									
N18	G01 LINEAR	X	Y	F		Z	10.00F		m.m	
N19	M17 LABEL 2									
N20	G81 REPEAT-DIVIDE LABEL 1 TO 2			REPEATS 1						
	OFFSETS :-	X	30.00Y	F		Z	F		m.m	
N21	G01 LINEAR	X	0.00Y	0.00F		Z	F		m.m	

A large, bold, outlined graphic of the letters 'FF' is centered on the page. The letters are thick and have a double-line border, giving them a three-dimensional appearance. The 'F's are slightly overlapping.

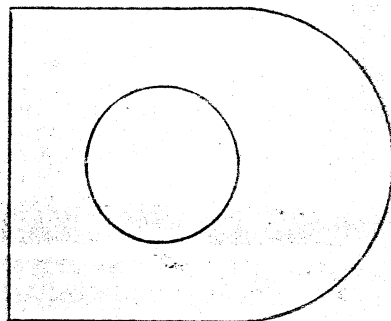
N1	G01 LINEAR	X	20.00Y	20.00F	100.0	Z	F		m.m
						d	F		
N2	G01 LINEAR	X	Y	F		Z	0.00F	20.0	m.m
						d	F		
N3	G01 LINEAR	X	50.00Y	F	20.0	Z	F		m.m
						d	F		
N4	G03 CIRCULAR CCW	X	50.00Y	60.00F		Z	F		m.m
	CENTRE	X	50.00Y	40.00		d	F		
N5	G01 LINEAR	X	20.00Y	F		Z	F		m.m
						d	F		
N6	G01 LINEAR	X	Y	20.00F		Z	F		m.m
						d	F		
N7	G01 LINEAR	X	Y	F		Z	10.00F	100.0	m.m
						d	F		
N8	G01 LINEAR	X	30.00Y	40.00F	100.0	Z	F		m.m
						d	F		
N9	G01 LINEAR	X	Y	F		Z	0.00F	20.0	m.m
						d	F		
N10	G02 CIRCULAR CW	X	30.00Y	40.00F	20.0	Z	F		m.m
	CENTRE	X	40.00Y	40.00		d	F		
N11	G01 LINEAR	X	Y	F		Z	100.00F	100.0	m.m
						d	F		
N12	G01 LINEAR	X	0.00Y	0.00F	100.0	Z	F		m.m
						d	F		



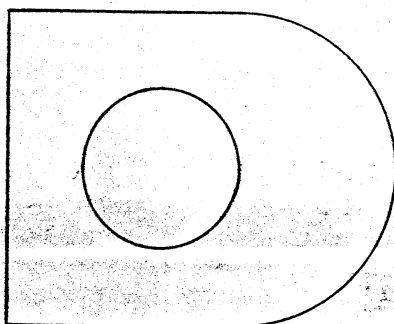
N1	G01 LINEAR	X	20.00Y	20.00F	100.0	Z	F		m.m
		d					F		
N2	G01 LINEAR	X	Y	F		Z	0.00F	20.0	m.m
		d					F		
N3	G01 LINEAR	X	50.00Y	F	20.0	Z	F		m.m
		d					F		
N4	G03 CIRCULAR CCW	ANGLE	180.00	F		Z	F		m.m
	CENTRE	X	50.00Y	40.00		d	F		
N5	G01 LINEAR	X	20.00Y	F		Z	F		m.m
		d					F		
N6	G01 LINEAR	X	Y	20.00F		Z	F		m.m
		d					F		
N7	G01 LINEAR	X	Y	F		Z	10.00F	100.0	m.m
		d					F		
N8	G01 LINEAR	X	30.00Y	40.00F	100.0	Z	F		m.m
		d					F		
N9	G01 LINEAR	X	Y	F		Z	0.00F	20.0	m.m
		d					F		
N10	G02 CIRCULAR CW	ANGLE	360.00	F	20.0	Z	F		m.m
	CENTRE	X	40.00Y	40.00		d	F		
N11	G01 LINEAR	X	Y	F		Z	100.00F	100.0	m.m
		d					F		
N12	G01 LINEAR	X	0.00Y	0.00F	100.0	Z	F		m.m
		d					F		



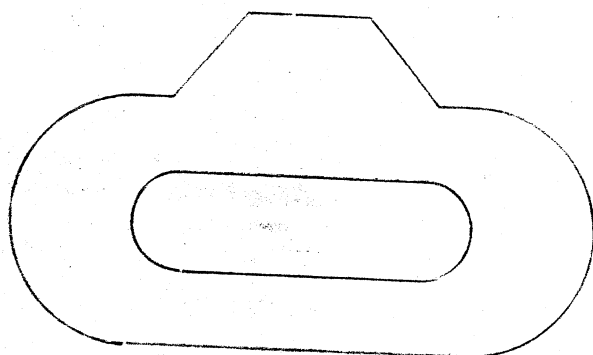
N1	G01 LINEAR	X	20.00Y	20.00F	100.0	Z	F		m.m
						d	F		
N2	G01 LINEAR	X	Y	F		Z	0.00F	20.0	m.m
						d	F		
N3	G01 LINEAR	X	50.00Y	F	20.0	Z	F		m.m
						d	F		
N4	G03 CIRCULAR CCW	X	50.00Y	60.00F		Z	F		m.m
			RADIUS 20.00			d	F		
N5	G01 LINEAR	X	20.00Y	F		Z	F		m.m
						d	F		
N6	G01 LINEAR	X	Y	20.00F		Z	F		m.m
						d	F		
N7	G01 LINEAR	X	Y	F		Z	10.00F	100.0	m.m
						d	F		
N8	G01 LINEAR	X	30.00Y	40.00F	100.0	Z	F		m.m
						d	F		
N9	G01 LINEAR	X	Y	F		Z	0.00F	20.0	m.m
						d	F		
N10	G02 CIRCULAR CW	X	50.00Y	40.00F	20.0	Z	F		m.m
			RADIUS 10.00			d	F		
N11	G02 CIRCULAR CW	X	30.00Y	40.00F		Z	F		m.m
			RADIUS 10.00			d	F		
N12	G01 LINEAR	X	Y	F		Z	100.00F	100.0	m.m
						d	F		
N13	G01 LINEAR	X	0.00Y	0.00F	100.0	Z	F		m.m
						d	F		



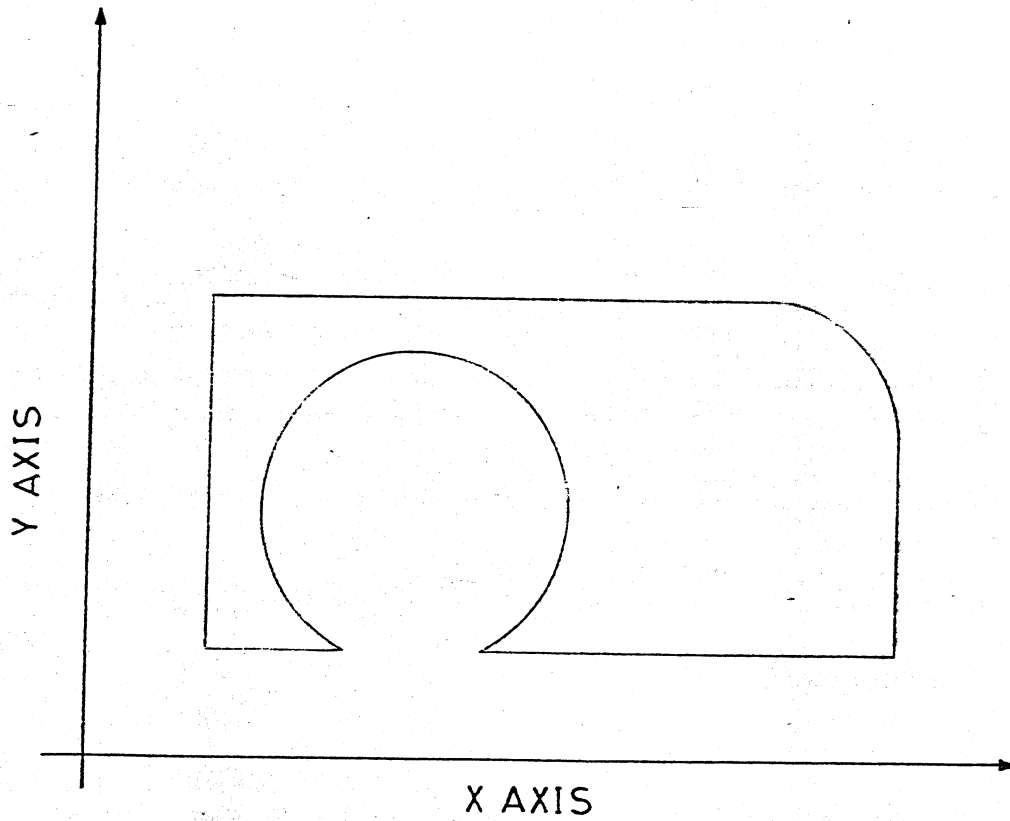
N1	G91 INCR'TAL UNITS										
N2	G01 LINEAR	X	20.00Y	20.00F	100.0	Z		F			m.m
						d		F			
N3	G01 LINEAR	X	Y	F		Z	-100.00F	20.0			m.m
						d		F			
N4	G01 LINEAR	X	30.00Y	F	20.0	Z		F			m.m
						d		F			
N5	G03 CIRCULAR CCW	X	0.00Y	40.00F		Z		F			m.m
	CENTRE	X	0.00Y	20.00		d		F			
N6	G01 LINEAR	X	-30.00Y	F		Z		F			m.m
						d		F			
N7	G01 LINEAR	X	Y	-40.00F		Z		F			m.m
						d		F			
N8	G01 LINEAR	X	Y	F		Z	10.00F	100.0			m.m
						d		F			
N9	G01 LINEAR	X	10.00Y	20.00F	100.0	Z		F			m.m
						d		F			
N10	G01 LINEAR	X	Y	F		Z	-10.00F	20.0			m.m
						d		F			
N11	G02 CIRCULAR CW	X	0.00Y	0.00F		Z		F			m.m
	CENTRE	X	10.00Y	0.00		d		F			
N12	G01 LINEAR	X	Y	F		Z	100.00F	100.0			m.m
						d		F			
N13	G01 LINEAR	X	-30.00Y	-40.00F	100.0	Z		F			m.m
						d		F			



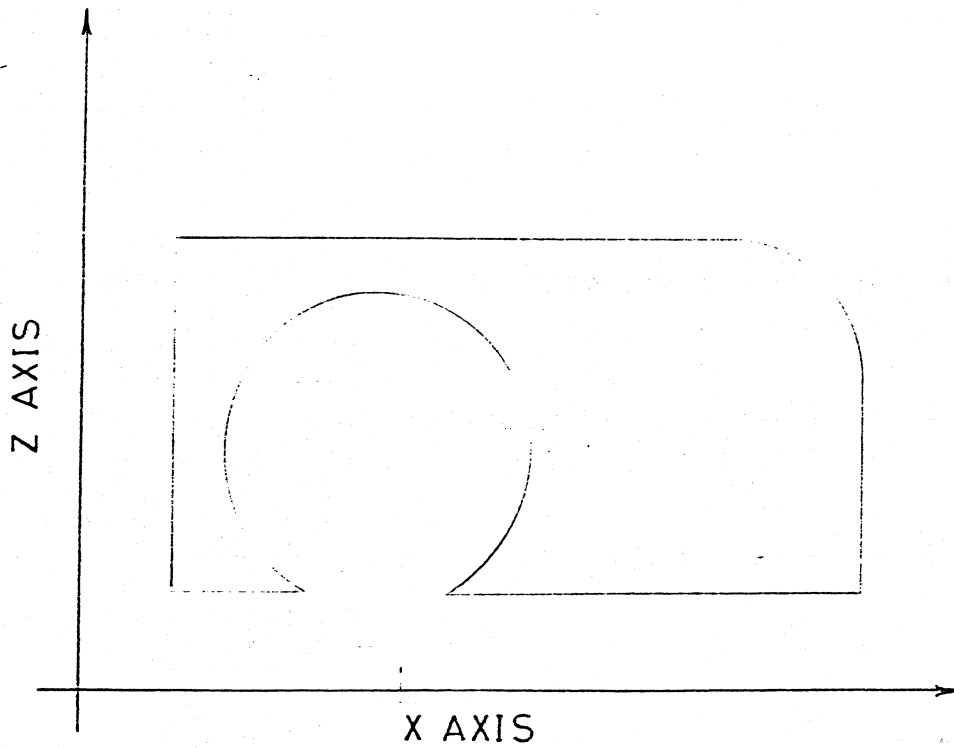
Line	Code	Units	X	Y	Z	Feed	Rate	Offset	Radius	Incl
N1	G70	IMPERIAL UNITS								
N2	G20	PROGRAM SCALE								
N3	M06	TOOL CHANGE		50.0 %						
N4	G00	RAPID TRAVERSE	X	0.000Y	0.000Z					
N5	G00	RAPID TRAVERSE	X	2.500Y	2.750Z					
N6	G00	RAPID TRAVERSE	X	Y	Z					
N7	G01	LINEAR	X	Y	Z	0.250d				
N8	G02	CIRCULAR CW	X	2.500Y	5.250F	50.0		-0.250F	20.0	
		CENTRE	X	2.500Y	4.000			F		
N9	G01	LINEAR	X	2.875Y	F			F		
N10	G01	LINEAR	X	3.625Y	6.125F			F		
N11	G01	LINEAR	X	4.875Y	F			F		
N12	G01	LINEAR	X	5.625Y	5.250F			F		
N13	G01	LINEAR	X	6.000Y	F			F		
N14	G02	CIRCULAR CW	X	6.000Y	2.750F			F		
		CENTRE	X	6.000Y	4.000			F		
N15	G01	LINEAR	X	2.500Y	F			F		
N16	G00	RAPID TRAVERSE	X	Y	Z			F		
N17	G00	RAPID TRAVERSE	X	3.000Y	3.500Z	0.250d				
N18	G01	LINEAR	X	Y	F					
N19	G01	LINEAR	X	5.500Y	F			-0.250F		
N20	G03	CIRCULAR CCW	X	5.500Y	4.500F			F		
		CENTRE	X	5.500Y	4.000			F		
N21	G01	LINEAR	X	3.000Y	F			F		
N22	G03	CIRCULAR CCW	X	3.000Y	3.500F			F		
		CENTRE	X	3.000Y	4.000			F		
N23	M06	TOOL CHANGE						F		
N24	G00	RAPID TRAVERSE	X	0.000Y	0.000Z					



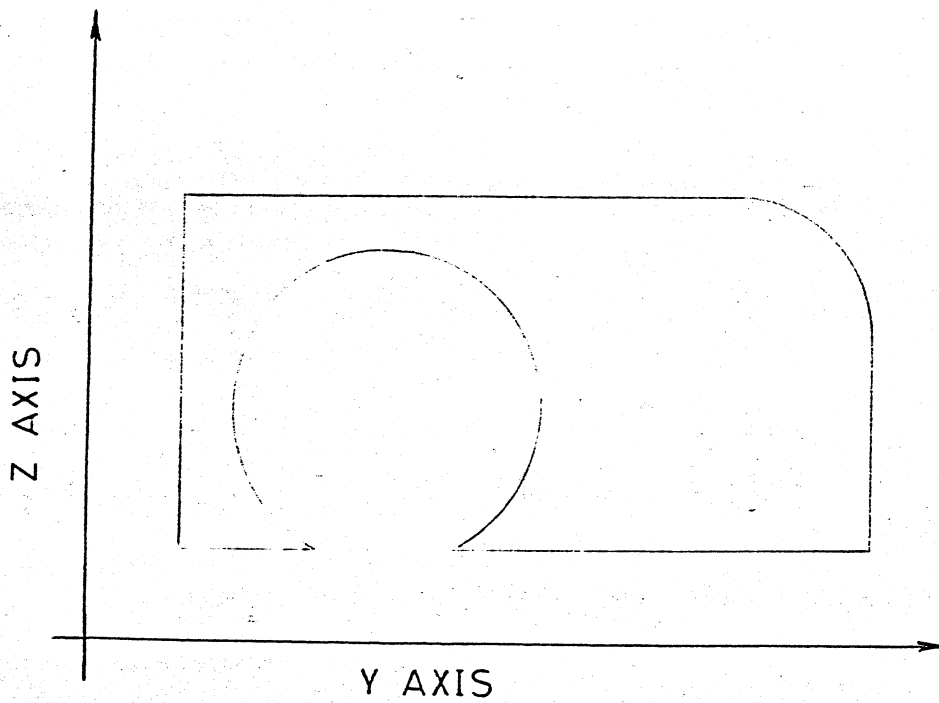
N1	G01 LINEAR	X	50.00Y	50.00F	Z	10.00F	m.m.
N2	G01 LINEAR	X	Y	F	Z	0.00F	m.m.
N3	G01 LINEAR	X	80.00Y	F	Z	F	m.m.
N4	G02 CIRCULAR CW	X	100.00Y	30.00F	Z	F	m.m.
	CENTRE	X	80.00Y	30.00			
N5	G01 LINEAR	X	Y	0.00F	Z	F	m.m.
N6	G01 LINEAR	X	40.00Y	F	Z	F	m.m.
N7	G03 CIRCULAR CCW	ANGLE	306.82	F	Z	F	m.m.
	CENTRE	X	30.00Y	20.00			
N8	G01 LINEAR	X	0.00Y	F	Z	F	m.m.
N9	G01 LINEAR	X	Y	50.00F	Z	F	m.m.
N10	G01 LINEAR	X	50.00Y	F	Z	F	m.m.
N11	G01 LINEAR	X	Y	F	Z	10.00F	m.m.



N1	G19 XZ PLANE						
N2	G01 LINEAR	X	50.00Z	50.00F	Y	10.00F	m.m.
N3	G01 LINEAR	X	Z	F	Y	0.00F	m.m.
N4	G01 LINEAR	X	80.00Z	F	Y	F	m.m.
N5	G02 CIRCULAR CW	X	100.00Z	30.00F	Y	F	m.m.
	CENTRE	X	80.00Z	30.00			
N6	G01 LINEAR	X	Z	0.00F	Y	F	m.m.
N7	G01 LINEAR	X	40.00Z	F	Y	F	m.m.
N8	G03 CIRCULAR CCW	ANGLE	306.82	F	Y	F	m.m.
	CENTRE	X	30.00Z	20.00			
N9	G01 LINEAR	X	0.00Z	F	Y	F	m.m.
N10	G01 LINEAR	X	Z	50.00F	Y	F	m.m.
N11	G01 LINEAR	X	50.00Z	F	Y	F	m.m.
N12	G01 LINEAR	X	Z	F	Y	10.00F	m.m.



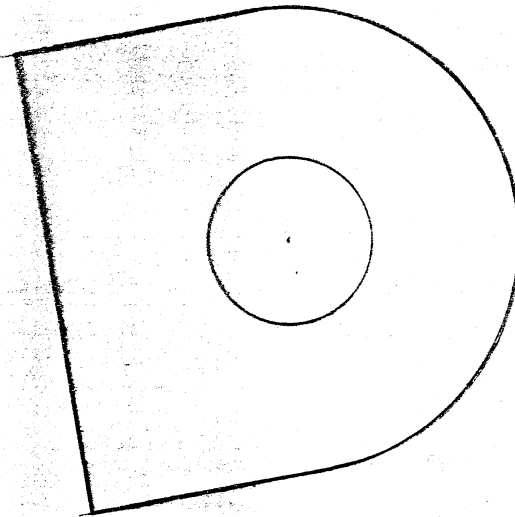
N1	G19 YZ PLANE							
N2	G01 LINEAR	Y	50.00Z	50.00F	X	10.00F		m.m.
N3	G01 LINEAR	Y	Z	F	X	0.00F		m.m.
N4	G01 LINEAR	Y	80.00Z	F	X	F		m.m.
N5	G02 CIRCULAR CW	Y	100.00Z	30.00F	X	F		m.m.
	CENTRE	Y	80.00Z	30.00				
N6	G01 LINEAR	Y	Z	0.00F	X	F		m.m.
N7	G01 LINEAR	Y	40.00Z	F	X	F		m.m.
N8	G03 CIRCULAR CCW	ANGLE	306.82	F	X	F		m.m.
	CENTRE	Y	30.00Z	20.00				
N9	G01 LINEAR	Y	0.00Z	F	X	F		m.m.
N10	G01 LINEAR	Y	Z	50.00F	X	F		m.m.
N11	G01 LINEAR	Y	50.00Z	F	X	F		m.m.
N12	G01 LINEAR	Y	Z	F	X	10.00F		m.m.



DATE 06/07/88

NO 14

N1	G14 ROTATE	X	40.00Y	40.00	ROTATE	10.00			m.m.
N2	G00 RAPID TRAVERSE	X	20.00Y	20.00	Z				m.m.
N3	G01 LINEAR	X	Y	F	Z	0.00F	20.00		m.m.
N4	G01 LINEAR	X	50.00Y	F	20.00	Z	F		m.m.
N5	G03 CIRCULAR CCW	X	50.00Y	60.00F	Z	F			m.m.
			RADIUS	20.00					
N6	G01 LINEAR	X	20.00Y	F	Z	F			m.m.
N7	G01 LINEAR	X	Y	20.00F	Z	F			m.m.
N8	G00 RAPID TRAVERSE	X	Y	Z	10.00				m.m.
N9	G00 RAPID TRAVERSE	X	30.00Y	40.00	Z				m.m.
N10	G01 LINEAR	X	Y	F	Z	0.00F	20.00		m.m.
N11	G02 CIRCULAR CW	ANGLE	360.00	F	20.00	Z	F		m.m.
	CENTRE	X	40.00Y	40.00					
N12	G00 RAPID TRAVERSE	X	Y	Z	10.00				m.m.
N13	G15 CANCEL ROTATE								
N14	G00 RAPID TRAVERSE	X	0.00Y	0.00	Z				m.m.



Appendix E

Tool Subroutines (Option)

Programming Points

1. Before inputting any tool subroutines the correct machine setup must have been accepted (see order for entering tool subroutines).
2. When inputting tool subroutines, the tool subroutine number for tool 1 will be 1 etc.
3. Execution of an 'M06 TOOL CHANGE' block from within a subroutine will come up error if the tool has to execute a tool subroutine.
4. If programming linear moves these moves are incremental from the present position before the tool change was executed.
5. If moves exist in a tool subroutine it is good practice that all axes in the subroutine return to 0 before the subroutine end.

Order for Entering Tool Subroutines

1. Input new machine setup parameters.
2. Accept new changes.
3. Enter setup and enter tool subroutines.

via EDIT then TOOL SUBROUTINES

4. Store machine set up to cassette (this will store machine set up then tool subroutines on cassette).
5. Exit Set up.

Notes:-

1. If moves exist in the subroutines and the step size is changed then the moves will change within the blocks so some editing may be required.
2. If no new machine set up is to be entered but subroutines need to be altered then just do steps 3-5.

Executing Tool Change Blocks

For a tool subroutine to be executed then the specified tool subroutine must be in memory and the relevant tool must be set to automatic tool. See Operating Manual for 'Setting Tool Offsets'.

If a tool change block is part of a program then the tool subroutine can be single stepped or automatically executed.

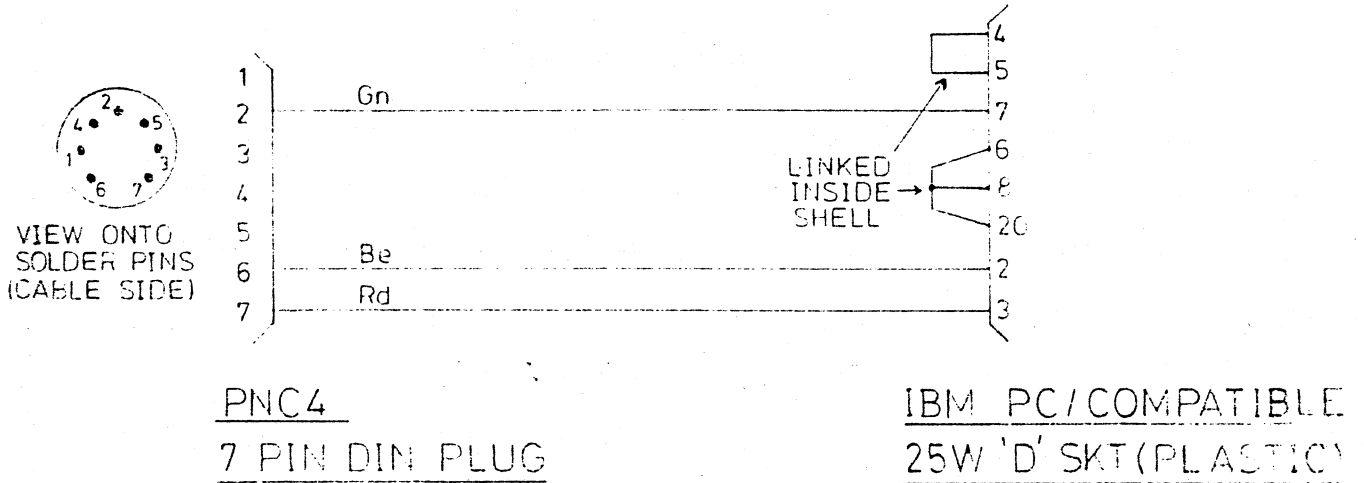
If a tool change block is a single move then the subroutine will be executed as if in auto mode.

Appendix F

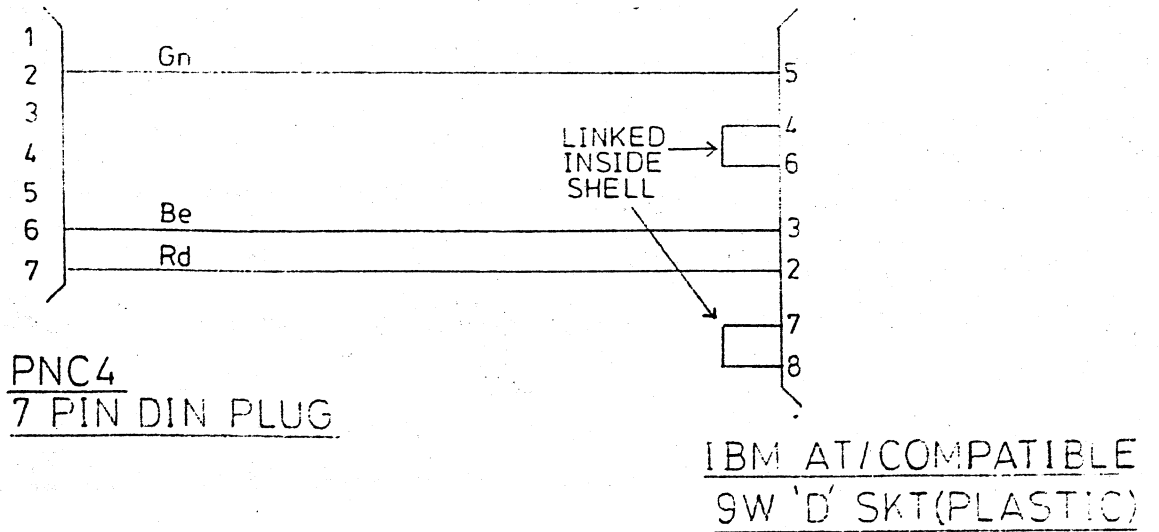
COMPUTER TO PNC4 INTERFACE CABLES

25/2/88

IBM PC/COMPATIBLE TO PNC4 LINK



IBM AT/COMPATIBLE TO PNC4 LINK



Appendix G NOV88

PNC 4 RS232 Buffer

The PNC 4 RS232 Buffer receives RS232 characters at a preset baud rate, parity and protocol as defined by the set-up procedure. By pressing the setup key followed by key 1 for Edit, then key 7 for Special Features a menu with four options will be displayed. By selecting option four, RS232 serial buffer, allows the user to configure the RS232 buffer to different serial parameters.

Pressing the up and down arrows moves the cursor to the next or previous option. A prompt is displayed at the bottom of the screen indicating the options available and the key to press to obtain a particular option.

The RS232 buffer can operate at any one of six baud rates ranging from 110 to 19200. It uses RTS/CTS or XON/XOFF to handshake with the device transmitting data to the PNC 4. If the RS232 buffer external ram has less than 513 bytes free then RTS is reset, or if XON/XOFF is selected, then XOFF is transmitted to the device transmitting data to the PNC 4. When 512 bytes are free RTS is set, or if XON/XOFF is selected, then XON is transmitted. The RS232 buffer is able to accept up to 512 more characters after RTS is reset. XON/XOFF can only be transmitted if CTS is high.

Parity checking can be selected via the setup procedure. If parity checking is enabled, then the parity type may be setup as either odd or even. If a parity error is detected, an error will be displayed and if parity errors transmitted is selected an ASCII 'E' is transmitted together with a decimal character 4 to the Host computer. If parity checking is not selected then parity type and parity error transmitted options will automatically be disabled.

Pressing reset goes back through the menu levels. When the base menu is displayed selecting option 4 exit, then option 1 accept changes, will initialise the RS232 buffer to the current RS232 serial buffer setup.

The PNC 4 has a built in test procedure which allows elements of the RS232 serial buffer to be tested. This option is selected by pressing the test key at the pre-datum screen. A menu with three options available is displayed, by selecting option three, RS232 buffer tests, a menu with six options is displayed indicating the tests available.

Test 1:Checksum/Version

At the conclusion of this test, the version number and checksum of the EPROM is displayed.

Test 2:External Ram

This test, tests the ram chip on the RS232 buffer board. At the conclusion of this test the size of the buffer is displayed. If there is a fault with the ram then the address of the error along with the byte at that address is displayed.

N.B. If data is transmitted to the PNC 4 before option 7 is selected then the data will be ignored.

When the PNC 4 has received 40 blocks or an end of program block, the PNC will prompt the user to press start to execute. Upon pressing the start key the program will start to execute from block one. Before movement starts, the PNC will test for low lubrication. If an error is detected, an error will be displayed. This test also happens when a program is restarted after a stop. At any time whilst executing a program a fault occurs with the laser then the PNC 4 will stop moving, close the laser shutter and display an error. If the stop key is pressed whilst executing a program, then the PNC will close the laser shutter.

At the end of the program the laser shutter will be closed and if the program is less than 28K then the PNC will retain the program in its battery backed memory, otherwise it will clear its battery backed memory.

OPTION 8: Clear RS232 Serial Buffer

Selecting this option will clear the RS232 serial buffer data ram

N.B Selecting either option 6 or 7 automatically clears the RS232 serial buffer. The 28K PNC 4 ram is also automatically cleared.

If at any point during loading from the serial buffer an error occurs the PNC will display the data it received.