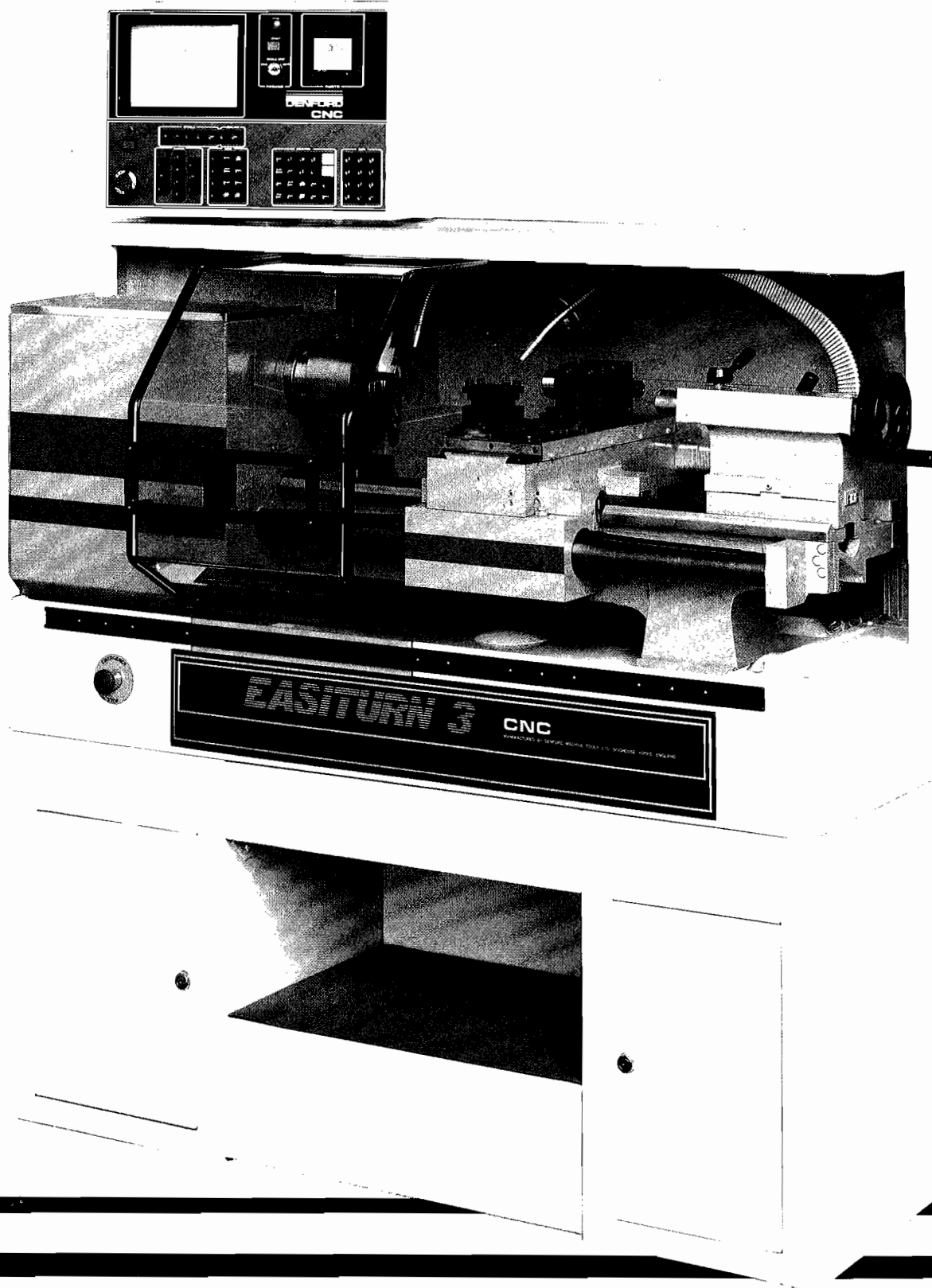


EASITURN 3



DENFORD

MACHINE TOOLS

PNC 3

IMPORTANT

The PNC 3 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 connectors are correctly connected.

The PNC 3 should be properly earthed (grounded) at all times.

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

PNC 3'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 leaders.

Index.

Introduction

PNC 3 Description

General information.

MACHINE codes (M function and G codes)

MOVEMENTS and associated FEED data input

CIRCULAR movements

AUXILIARY functions (Use of the 4 programmable auxiliary functions)

INPUT facilities (Use of the 7 programmable inputs)

BLOCK SEARCH (To run to any block of the program that is in PNC 3's memory)

DATA LINK (To use the interface to a computer or printer)

STANDARD RS232C INTERFACE specification

ENHANCED RS232C INTERFACE specification

LOAD facility (To load PNC 3 memory from the PNC 3 Keyboard)

CASSETTE operations (Use of the integral cassette program storage system)

EDIT facility (To edit a program that is in PNC 3 memory)

JOG facility (Single step movement or feed override of each axes)

SCALE facility (Overall and program scaling)

MIRROR facility (Mirroring of axes)

OFFSET facility (Use of the PNC 3 offset facility to 'offset' program dimensions)

REPEAT facility (Use of the repeat facility to repeat sections of a program a number of times)

DWELL facility (Use of the dwell facility to introduce variable time delays)

PROG. STOP (Program Stop or Inspection Pause)

FLOAT DATUM facility

ABSOLUTE DATUM facility

INCH/MM Selection

ABSOLUTE/INCREMENTAL Selection

SPINDLE speed control

COOLANT control

THREAD facility

TOOL SELECTION

DISPLAY & EDIT tool offsets

TOOL SETTING facility

TOOL OFFSETS

CUTTER OFFSETS (MILL)

EFFECTS OF CUTTER OFFSETS

AUTO/SINGLE STEP Positional error correction

INTERNAL SWITCH SETTINGS (Baud rate selection, metric/imperial default)

ERROR MESSAGES

SELF TEST facility

Introduction.

We are pleased to introduce you to this programmable numerical control, 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 3 the ability to precisely control up to 4 axes of movement with linear and circular interpolation at programmable feedrates. The PNC 3 can also act as a process controller by giving up to 4 programmable auxiliary outputs and monitoring 8 programmable user assigned input signals.

It is North East Electronic's policy to continually review and upgrade its products. This PNC 3 is the latest in a line of controllers whose fore runners 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 3 control unit to suit new equipment.

PNC 3 Description.

The PNC 3 is an extremely versatile continuous path, computer based programmable numerical control unit designed to control via stepper motors a wide variety of machinery where precise control and positioning is required. Related processes and functions can also be controlled by the PNC 3. The programming of stepper motor movements and the process control element is explained fully in this manual. We are confident the PNC 3 will be found to be very easy to operate.

From the front panel total control is obtained by the following features:

An easy to use keyboard for the input of data and commands by a) the keyword system b) standard G and M code programming.

The 9" Display provides the user with :-

- i) A display of the complete machine status
- ii) Prompts to assist the user in using the control system
- iii) Sections of the program during program loading, editing and execution
- iv) Machine, Control Unit and Program error information.

Integral control unit memory stores typically 500 blocks.

Programming facilities include repeat loops, fixed/floating datums, dwells, program offsets, imperial/metric and absolute/incremental programming in any mix, screwcutting (for lathes), program scaling, mirror imaging and cutter diameter and length compensation (for mills).

Integral spindle and coolant control.

The integral fast magnetic tape system provides unlimited program storage space, with each cassette side storing up to 3000 program blocks.

Keyswitch to give manual programming/control, or single step program execution or automatic program operation. Program START and STOP switches.

Jog system giving manual control in all axes with plus and minus keys for feedrate override control.

Integral high power stepper motor drives.

General Information.

The PNC 3 has been designed with the user in mind. It is a versatile control system which is very easy to use.

PNC 3 VDU display

The PNC 3 display is used to display:-

- a) Current Machine Status
- b) Manual Instructions
- c) Function Selection
- d) User Program Data
- e) Operator Prompts
- f) Error Messages

The display area above the horizontal line is used to display the current machine status. The top line shows the current tool number, the spindle status (and r.p.m. if the programmable spindle control option is fitted), and the current controller mode. The possible modes are:-

- M.D.I. Manual Data Input. This mode enables the system to be controller from the PNC 3 keyboard.
- S.STEP Single Step. This mode enables a user program that is in PNC 3 memory to be executed 1 block at a time by pressing the START switch.
- AUTO Automatic mode. This mode enables a user program to be executed from the current block to its end by pressing the START switch.
- CASS. Cassette unit operating.
- EDIT Controller is in Edit mode.
- PRINT The user program is being printed.
- RS232c RS232c serial link is operating.

The second status show the COOLANT status either ON or OFF, the current program number, and the RUN number. The RUN number indicates how many times the current program has been started.

The third status line shows the current block number, the controller status, the coordinates of the current displayed position, and the manual data input format as either absolute or incremental. The controller status is either in position (if the controller has completed an instruction), or in PROGRESS (if the controller is currently executing an instruction), or STOPPED (if the controller was stopped whilst executing an instruction).

The fourth and fifth line of the machine status display indicate the current status of the auxiliary relays and input switches respectively. The auxiliaries that are On are shown, and the input switches that are closed are displayed. NOTE: The input displays are only updated during the execution of input blocks, or when the controller is in either manual or singles step mode and not "IN PROGRESS".

The sixth (and seventh) line of the status display shows the current machine position together with the programd feed. The displayed machines position is corrected for tool offsets and floating datum instructions i.e. The display position is shown with respect to the work piece datum in the case of tool length offsets, and with respect to the floating datum in the case of floating datums.

The display area beneath the horizontal line is used to display keyed in manual instructions which are echoed on to the display. Also this area is used to display function selection menus for e.g. for cassette operations. Additionally this area is used to display up to three user program blocks during edit and either one or 2 program blocks during program execution.

The bottom two lines of the display either show operator prompts or error messages in reverse video. Error messages may be cleared by pressing the RESET key whilst in MANUAL program mode.

The PNC 3 has numerous facilities all accessed by the keyboard in M.D.I. (Manual Data Input) mode. The majority of keys are self explanatory and being dedicated keys their functions are engraved on the key.

A list of keys and their functions follows:-

RESET - to reset from or end the current mode, or cancel the most recent entry.

T 1) Upon switching the PNC 3 on pressing the T key gives a system test display enabling the PNC 3 input signals and machine mounted switches/wiring to be checked. (See the PNC 3 Maintenance Manual for details)

2) After the machine has been datumed the T key is used for tool selection, tool setting and tool offset editing.

M For machine code M function selection (By pressing M then the Enter key a list of the M functions will be displayed).

G For machine code G function selection (By pressing G then the Enter key a list of G codes will be displayed).

(AXES KEYS, Fitted to suit application)

X for X axis moves
Y for Y axis moves
Z for Z axis moves
W for W axis moves

BLOCK

SEARCH To enable program execution to commence from a specific point in the program.

EDIT Permits full editing facilities to be used on the programd sequence of instructions in memory

S Subroutines (OPTION)

ZERO To move machine to zero i.e. machine datum position.

DATA

LINK Selects a data link to external equipment e.g. an external computer or printer.

AUX

INPUT The PNC 3 can act as a process controller having 4 relay closure outputs (Auxiliary function) and can monitor 8 external switches (Input functions).

FEED Input key for axes feeds.

PROG

STOP Operation of 'Program Stop' will stop execution of the current block. Additionally this key is used to program a STOP.

CASS Permits use of the cassette system.

LOAD

END To start loading or complete loading a program.

EOB End of block or end of line of instructions.

ENTER When keying in information it is echoed first on the display. After verification the data is then accepted by pressing "Enter".

The group of 10 keys to the bottom left of the control select the JOG function. The arrow keys indication direction, the circular arrow key with a W is for the fourth axis if fitted and the + and - keys are used either for JOG speed control, as 'feedrate override' keys.

MIRROR (Mill only) The X and Y axes can be mirrored in the program. When the mirror key is pressed, a menu of 4 options is displayed and one of the four options may be selected.

REPEAT Any number of blocks in program can be repeated up to 99 times (with specified offsets and feeds) to 4 levels.

CW

CCW For circular movements

SCALE (Mill only) Movements inside or outside the program can be scaled in the range from 0.01 to 650%.

FLOAT

DATUM Enables a selected position to be used as zero in all axes.

DWELL Selects programmable dwell in the range of 0.1 to 9999.9 seconds

OFFSET Selects machine offset or program offset.

COMP Selects tool radius compensation.

THREAD (Lathe only) Selects screwcutting facility with pitch in the range 0.1mm to 6.4mm.

INCH

MM Selects input of data in imperial or metric units.

ABS

INC Selects input of data in absolute or incremental format.

The group of 7 keys immediately under the display are used for spindle speed control and coolant control. The keys from left to right will be as follows:-

+	to manually increase the spindle speed
-	to manually decrease the spindle speed
FWD	(option) selects forward rotation of spindle
HIGH	(option) selects high spindle speed range
REV	(option) selects reverse rotation of spindle
LOW	(option) selects low spindle speed range
OFF	selects spindle off
ON	Sets coolant on
OFF	Sets coolant off

The spindle rotations and coolant controls can be used directly or during programming.

Program control switches.

There are 3 program control switches namely:

- 1) STOP
- 2) START
- 3) MANUAL/SINGLE STEP/AUTO (MODE)

The third switch is a 3-position key switch: in MANUAL it enables manual data input (MDI). In SINGLE STEP position, one depression of the START switch will cause the next block of the program to be executed. In AUTO one depression of the START switch will give automatic sequential execution of the program in memory, from the current position to the end of program.

The status of this 3-position switch is indicated on the top right hand corner of the display.

By depressing STOP (or PROG STOP), execution of the current block will stop. Execution may be resumed using the START key.

See also the section in this manual entitled "AUTO/S.STEP Positional error correction".

MACHINE codes (M Functions and G codes)

The PNC 3 can be programd by using both M and G codes or programd directly using the dedicated keys (keyword system). A complete list of M and G codes follows, some of which are option dependant.

M functions for use outside the program

- M03 Spindle Forward
- M04 Spindle Reverse
- M05 Spindle Stop
- M06 Tool Change
- M08 Coolant On
- M09 Coolant Off
- M20 Auxiliaries
- M21 Inputs

M functions available inside the program

- M00 Program Stop
- M02 End Of Program
- M03 Spindle Forward
- M04 Spindle Reverse
- M05 Spindle Stop
- M06 Tool Change
- M08 Coolant On
- M09 Coolant Off
- M20 Auxiliaries
- M21 Inputs

G codes for use outside the program

G01 Linear
G02 Circular CLW
G03 Circular CCLW
G04 Dwell
G21 Machine Scale
G33 Thread
G40 Cancel Tool Comp
G41 Cutter Comp Left
G42 Cutter Comp Right
G55 Machine Offset
G70 Imperial Units
G71 Metric Units
G90 Absolute Input
G91 Incremental Input
G98 Absolute Datum

G codes for use inside the program

G01 Linear
G02 Circular CLW
G03 Circular CCLW
G04 Dwell
G10 Mirror X
G11 Cancel Mirror X
G12 Mirror Y
G13 Cancel Mirror Y
G20 Program Scale (replaces G21)
G33 Thread
G40 Cancel Tool Comp
G41 Cutter Comp Left
G42 Cutter Comp Right
G54 Program Offset (replaces G55)
G70 Imperial Units
G71 Metric Units
G81 Repeat Function
G90 Absolute Input
G91 Incremental Input
G98 Absolute Datum
G99 Floating Datum

MOVEMENTS AND ASSOCIATED FEED INPUT

A movement in one or more axes can be input by pressing the desired axis key followed by the required dimensions. These coordinate dimensions and associated FEEDS may be input either as a single block of data which is to be executed immediately or as blocks of data which form part of a programd sequence.

Before coordinate dimensions keyed into the PNC 3 are executed a check is made to ensure that the maximum machine movements are not exceeded. Should any be exceeded a warning message is displayed. RESET restores normal operation

The required FEED is entered in millimetres per minute or inches per minute. If no feed is programd the default feed of 234mm/min is assumed.

CIRCULAR movements

Circular movements are defined by using the CW/CCW key, (one press for clockwise, two presses for counterclockwise), defining the end points of the circular movement then pressing CW/CCW and inputting the circle centre coordinates.

Circular movements are limited to 1 quadrant per block.

When circular movements of less than one quadrant are being made it is not necessary to precisely know the coordinates of the end point of the movement in both axes. The PNC 3 can 'find' one of the coordinates. If this facility is to be used it must be arranged that the movement passes through the coordinate point of the unknown axis dimension before reaching the coordinate point of the known axis dimension at which point movement ceases:-
e.g. If the present position is X=0,Y=0 and we wish to make a circular clockwise movement to X=1 when Y is unknown, centred on X=3,Y=0 the Y end point programd should be between Y=0.1 and Y=2.236 the true end point.

If incorrect i.e. impossible end points, are programd the PNC 3 will move the table continuously in a circle attempting to find the programd end point. The movement may be stopped by depressing either the smaller red STOP switch or the PROG STOP key on the PNC 3 keyboard.

AUXILIARY functions

Auxiliary functions allow user assigned devices to be controlled i.e. switched on and off by the 4 integral PNC 3 auxiliary relays. Three types of auxiliary functions are supplied.

The three types are:

- a) ON/OFF Auxiliaries 3 and 4
- b) MOMENTARY Auxiliary 2
- c) PULSED Auxiliary 1

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

b) MOMENTARY auxiliaries are switched ON (if programd to be on) only when the machine is at a programd 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 of 50 milliseconds (if programd ON) each time the machine completes a program block.

To program auxiliaries, key in M20 or press the AUX/INPUT key once followed by <ENTER>. The PNC 3 will prompt the user to select the auxiliaries that are to be programd ON. If more than one auxiliary is to be on, the "." (decimal point) key can be used to separate the numbers being input. Pressing <ENTER> will cause the prompt to change to invite the user to select auxiliaries that are to be programd OFF. More than one auxiliary can be programd off by repeatedly entering numbrs. When the auxiliaries have been set to ON or OFF as desired pressing <EOB> will end that block of information.

INPUT facilities

The PNC 3 is equipped to monitor 7 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 program state, the PNC 3 waits until the switch signals are as program before proceeding. Switch levels may be program 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 wait for some external signal from a robot before proceeding.

To program inputs, key in M21 or press the AUX/INPUT key twice followed by <ENTER>. The inputs to be ON are entered first, in the same way as auxiliaries, using the "." key as a separator if necessary. When <ENTER> is pressed the inputs to be OFF can be entered in the same way. Pressing <EOB> will end that block of information.

The auxiliary outputs and the inputs enable the PNC 3 to function not only as a precise positioning control system but also as a sequence controller.

BLOCK SEARCH facility

A program that is in PNC 3 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 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. The PNC 3 will display all blocks from block 1 up to the block before the desired block. This "pre-run" enables the PNC 3 to calculate the correct position and also the status of the spindle and coolant. In the case of a Mill the PNC 3 will also raise the cutter out of the job. When the correct position and status of the block before the desired block has been found the PNC 3 will move to that position and bring on the spindle and coolant if necessary. Note that in the case of a Mill it will move to the correct X, Y position first and then bring the cutter to the correct depth. It will then execute the desired block.

DATA LINK facility

Four operations are possible using the Data Link. They are:

- 1) Load Program from Data Link. (RS232C Serial Link)
- 2) Continue Program load from Data Link. (RS232C Serial Link)
- 3) Transmit Program to Data Link. (Enhanced RS232C option only)
- 4) Print Program i.e. transmit program to printer. (Centronics compatible parallel link)

Note 2, 3 and 4 are possible only if the PNC 3 Memory is loaded.

- 1) Enables a program to be loaded into PNC 3 memory from an external device either one block at a time or as a full program. Any program previously contained in PNC 3 memory is overwritten i.e. destroyed. The format of the program data is shown in the RS232c interface specification below.
- 2) Enables an additional program from an external device to be added to a program that already exists in PNC 3 memory. The format of the program data is shown in the RS232c interface specification below.
- 3) Enables the contents of PNC 3 memory to be transmitted to an external device. The memory contents are transmitted as 'ASCII' characters in a similar format to that used by 1) and 2) above.
- 4) Enables the contents of PNC 3 memory to be transmitted to any printer with 80 columns or more which has a standard Centronics parallel interface.
- 5) With PNC 3's version 3.39 onwards, if 1) or 2) is selected, the user will be prompted to specify either Host Computer or Paper Tape. The difference between these two options is explained in the following section entitled "PNC 3 Enhanced RS232c Interface Specification".

PNC 3

Standard PNC 3 RS232 Interface Specification

This interface is only implemented on PNC 3's prior to version 3.39 29/02/84.

The PNC 3 only needs 3 wires to implement an RS232 link (at 4800 baud unless specified differently at ordering). They are Data Transmit, Data Receive and Signal Ground. The connection to the PNC 3 is via a 7 pin DIN socket (fitted to the unit rear) using pins 6, 7 and 2 for TxDa, RxDa and Ground respectively. TxDa is the output from the PNC 3 and RxDa is the input to the PNC 3.

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

The PNC 3 will recognize the following characters:-

<STX>	02	start of transmission
<ETX>	03	end of transmission
0 to 9	30H to 39H	
A	41H.	
C	43H	
D	44H	
E	45H	
F	46H	
I	49H	
L	4CH	
O	4FH	
P	50H	
Q	51H	
R	52H	
S	53H	
T	45H	
X	58H	
Y	59H	
Z	5AH	
+	2BH	
-	2DH	
.	2EH	

The PNC 3 will use the following ASCII characters when transmitting:-

<ACK>	06H	acknowledge
<NAK>	15H	not acknowledge
?	3FH	

Any information i.e. each block, transmitted to the PNC 3 must be preceded by <STX> and terminated by <ETX>.

When the PNC 3 has processed the data and carried out the instructions contained therein, it will respond with either <ACK> (if everything was okay) or <NAK> if some error occurred.

If the PNC 3 has responded with <ACK>, <NAK> or ? then this also indicates that it is waiting for more data from the interface.

If an unrecognised character is received by the PNC 3 then it will cause the PNC 3 to transmit a "?" once the <ETX> has been received. In this case the

PNC 3 will not take any action on the received data but will wait for more data from the interface.

The PNC 3 will respond with (NAK) if the data received constitutes a move to a position outside the table limits.

The maximum amount of data transmitted to the PNC 3 in one go will be one program block.

A linear move program block can contain any of the following characters (to a maximum of 60 for 2 axis or 80 for 3 axis): -

<u>Data</u>	<u>Meaning</u>
X<d>	Move in X axis to position <d>
Y<d>	Move in Y axis to position <d>
FX<p>	Set XY feedrate to <p>
Z<d>	Move in Z axis to position <d>
FZ<p>	Set Z feedrate to <p>

where <d> is a decimal number

<p> is a decimal number in range 0.1 to 1536

Leading and trailing zeroes are not necessary when specifying <d> and <p>.

If the program block contains L as the last character (i.e. the character before the <ETX>) then that program block will not be directly executed but will be loaded into the next available slot in the PNC 3's memory. By sending a succession of blocks with L as the last character a complete program can be built up. Each successive block will be automatically loaded into the next available slot in the PNC's memory. If there are no more free slots in memory then the PNC will respond with <NAK>.

If the transmitted data consists of <STX> R <ETX> then the PNC 3 will execute the current program held in its memory starting at the first block and will not respond with <ACK> until the program has been successfully run. The PNC 3 will respond with <NAK> if any error occurs during program execution

N.B. The PNC 3 must be switched to AUTO for the complete program to be run.

If the program block transmitted to the PNC 3 does not contain the character L, then that block will be directly executed and the PNC 3 will respond with <ACK> on successful completion of the block or <NAK> if any error occurred. Directly executable blocks can be interspersed with loadable blocks if so desired.

If the transmitted data consists of <STX> E <ETX> then the PNC 3 will clear out any program currently in its memory and respond with <ACK>.

Example

Suppose the following data was transmitted to the PNC 3

<STX>X12.94Y15FX25.7Z14FZ100<ETX>

This would result in the PNC 3 moving from its current position to

X = 12.94

Y = 15.0

Z = 14.0

The XY axis would move at a feedrate of 25.7mm/min and the Z axis would move at a feedrate of 100mm/min. When the last axis reaches position the PNC 3 would respond by transmitting <ACK> to the interface.

To specify a circular move the following basic format must be adhered to:-

(N.B. if the PNC 3 is controlling a lathe then Z replaces Y

X<d> Y<d> CCX<XCENTRE> Y<YCENTRE> for Clock Wise Moves

or X<d> Y<d> CAX<XCENTRE> Y<YCENTRE> for Anti C.W. Moves

This basic format can be followed by feedrate specifiers, Z axis moves, etc in the normal way.

If the PNC 3 is controlling a lathe then the following format can be used to specify a screwcutting move.

<STX> S*<diameter> P<pitch> D<depth> C<no. of cuts> L<length> Z<start posn>
<ETX>

Where * can be either E or I for External or Internal threads.

The following format can be used to specify a tool change block.

<STX> T<tool No. 1 to 16> <ETX>

Auxiliary outputs can be controlled using the following format:

<u>Data</u>	<u>Meaning</u>
A<n>+	Turn auxiliary <n> on
A<n>-	Turn auxiliary <n> off
	where <n> is in the range 1 to 4

Input switches can be tested using the following format:

<u>Data</u>	<u>Meaning</u>
I<n>+	Wait for input <n> to be on
I<n>-	Wait for input <n> to be off

where <n> is in the range 0 to 7

Note the PNC 3 will not respond with <ACK> until the desired input has achieved the correct level. The PNC 3 can be forced out of waiting indefinitely for the input by pressing <RESET>.

The following block can be used to reset the PNC 3 to a known position. It actually causes the PNC 3 to act as if the <ZERO> button had been pressed, i.e. each axis in turn will drive to its datum position. The format is as follows:

<STX>Q<ETX>

Program stop (or inspection pause) blocks can be inserted into a program by the following

<STX>P<ETX>

If the PNC 3 is fitted with programmable spindle speed control, the following types of block can be used to control the spindle.

<u>Data</u>	<u>Meaning</u>
FSR<n>	Spindle reverse direction, <n> r.p.m.
FSF<n>	Spindle forward direction, <n> r.p.m.
FSO	Spindle off

where <n> is in the allowable range for that particular machine.

If the PNC 3 is fitted with spindle High/Low speed control, the following types of block can be used to control the spindle.

<u>Data</u>	<u>Meaning</u>
FSF	Spindle high speed
FSS	Spindle low speed
FSO	Spindle off

PNC 3 Enhanced RS232 Interface Specification

Section A Input to PNC 3

The enhanced interface is fitted as standard to PNC 3's after and including version 3.39 dated 29/02/84.

The enhanced RS232 interface allows a host computer to use all the facilities of the PNC 3. 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 3 will only accept data with even parity. If a parity error occurs, an error 01 will be sent from the PNC 3.

The PNC will ignore the following characters: General comments of the STANDARD RS232 Interface Specification apply.

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

Also if the first character after the <STX> is N, the PNC 3 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 3 will automatically number its blocks during a load sequence.

As with the standard interface, blocks with "L" as the last character before the <CR> will not be executed directly but will be loaded into the PNC 3's internal memory.

16 different baud rates are available with the PNC 3. These range from 75 to 9600 and are selected by DIL switches on the interface board (see the section on "switch settings" in this manual). To assist the user, the current baud rate is displayed on the PNC 3's screen during the loading and storing operations.

1. Linear Moves

Mill

G01 Xnnnn Ynnnn FXnnnn Znnnn FZnnnn

Lathe

G01 Xnnnn Znnnn FXnnnn

2. Circular Moves

Mill Clockwise

G02 Xnnnn Ynnnn FXffff Znnnn FZffff CXcccc Ycccc

Counter Clockwise

G03 Xnnnn Ynnnn FXffff Znnnn FZffff CXcccc Ycccc

Lathe Clockwise

G02 Xnnnn Znnnn FXffff CXcccc Zcccc

Counter Clockwise

G03 Xnnnn Znnnn FXffff CXcccc Zcccc

nnnn = X,Y or Z position in mm or inches

ffff = Feedrate in mm/min or ins/min

cccc = XY or XZ circle centre point in mm or inches

3. Dwell

G04 Ddddd

where dddd - dwell time in the range 0.1 to 9999.9 seconds

4. Mirror functions

Mirror X

G10L

Cancel Mirror X

G11L

Mirror Y

G12L

Cancel Mirror Y

G13L

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

5. Scaling function

Program Scale

G20 SssssL

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

Machine Scale

G21 Sssss

(NOTE! Not available as part of a program load)

where ssss is the percentage scaling required in the range 0.01% to 650%

6. Thread cutting move
(Only on lathes)

Internal Thread

G33 I<Diameter>P<pitch>D<depth>C<no. of cuts>L<length>Z<start posn>

External Thread

G33 E<diameter>P<pitch>D<depth>C<no. of cuts>L<length>Z<start posn>

7. Cutter Radius Compensation
(Only on Mills)

Cancel Tool Compensation

G40

Tool Compensation Right

G41

Tool Compensation Left

G42

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

Program offsets

Mill

G54 Xnnnn Ynnnn Znnnn L

Lathe

G54 Xnnnn ZnnnnL

Machine offsets (NOTE! Not available as part of a program load)

Mill

G55 Xnnnn Ynnnn Znnnn

Lathe

G55 Xnnnn Znnnn

9. Imperial Units

G70

10. Metric units

G71

11. Repeat loops (NOTE! Only available as part of a program load)

Mill

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

Lathe

G81R<start blk.>E<end blk.>N<no. of times> Xnnnn Znnnn FXffff L

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

12. Absolute units

G90

13. Incremental units

G91

14. Absolute Datum

G98

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

G99L

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

M00L

17. Spindle Speed Control

For units with programmable spindle speed

Spindle Forward

M03 S<spindle speed forward rpm>

Spindle Reverse

M04 S<spindle speed reverse rpm>

For units with 2 speed settings

High Speed

M03

Low speed

M04

18. Spindle stop

M05

19. Tool change

M06 T<tool number>

20. Coolant control

Coolant on

M08

Coolant off

M09

21. 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

N21 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.

22. 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

23. To Erase a Program (i.e. to clear PNC 3 memory)

E will erase the program

24. To End the Load Sequence

M02 When this block is seen by the PNC 3 it will exit from the DATA LINK facility and return to MDI mode.

(NOTE! only occurs when PNC 3 is receiving data from paper tape).

Section B Output from PNC 3

Host Computer

1. Providing the PNC 3 has received valid data as specified in Section A it will respond with the character <ACK>
<ACK> = 06H

This signifies that the PNC 3 is ready to receive more data.

2. If some error has occurred, the PNC 3 will respond with the following:

<NAK>nn

where nn is a two digit error code

<NAK> = 15H

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

PAPER TAPE

When serial load from tape is selected, the PNC 3 will operate using an Xon/Xoff type protocol as follows.

- 1) When the PNC 3 is ready to read a block from the serial link it will transmit a <DC1> character (=11H).
- 2) The PNC 3 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 3 will transmit either an <ACK> character or a <NAK> followed by a error code number.
- 4) The PNC 3 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 3, in which case no <DC1> is transmitted and the PNC 3 returns to MDI mode.

PNC 3 RS232 ERROR CODES (Enhanced specification only)

<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 memory full
11	Block too big for input buffer
12	X axis drive system fault
13	Y axis drive system fault
14	Z axis drive system fault
15	W axis drive system fault
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
21	Circular move not within a quadrant
22	Dwell value error
23	Scale value error
24	Tool number error
25	Auxiliary selection error
26	Input selection error
27	Repeat start block error
28	Repeat end block error
29	Number of repeats error
30	Nest error in repeat levels
31	Error in Repeat offsets
32	Error in offset block
33	Spindle speed input exceeds limits
34	Spindle direction is opposite to present direction (Stop spindle first)
35	Error when driving to datum
36	Threading input error
37	Spindle speed wrong for threading
38	Spindle drive system error

TRANSMIT PROGRAM TO DATA LINK.

When function 4 is selected in the DATA LINK menu, the PNC 3 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 RS232c Serial Data Link" is displayed).

The data transmitted by the PNC 3 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 low (-12V to 0V) then the PNC 3 will transmit

if RxDa is high (4V to 12V) then transmission is inhibited at the end of the current character and the PNC 3 will wait for a low level before continuing to transmit.

After the last block in the program has been sent to the serial link, the PNC 3 will transmit an M02 block to signify the end of the program.

LOAD facility (LOAD)key
END

This enables PNC 3 memory to be 'loaded' using the PNC 3 Keyboard. The task to be accomplished by the machine is broken down into a sequence of blocks which are then executed consecutively. A block may consist of a point to which the machine should move which is defined using 1,2,3 or 4 coordinates (dependant upon the PNC 3 options fitted), or the auxiliary setting required or the level(s) of input signals required. (Auxiliary functions and the input facility are described elsewhere). A block may also consist of a REPEAT instruction, or a FLOAT DATUM instruction, or an OFFSET instruction, or a SUBROUTINE call (Option), or a SCALE instruction, or in fact any G or M code facility.

Two different LOAD operations are possible:

- 1) LOAD memory from keyboard
- 2) CONTINUE memory load from keyboard

1) LOAD is used to enter a new program into PNC 3 memory. Any previously loaded program is overwritten i.e. destroyed.

2) CONTINUE memory load from keyboard enables an existing program to be continued i.e. extended.

Upon completion of a LOAD or a CONTINUE memory load the PNC 3 displays the program number, how many blocks there are in memory and how much memory remains for a period of 6 seconds after which the PNC 3 displays show normal data. (or until a key is pressed)

Should too much program data be keyed into the PNC 3 such that the memory becomes full, "MEMORY FULL" is displayed and no more data can be entered. Normal operation can be restored by using the RESET Key.

CASSETTE operations

The integral magnetic cassette recorder enables programs to be permanently stored for future use

Six different cassette operations are possible:

- 1) Rewind cassette
- 2) Erase cassette
- 3) Find the end of cassette data
- 4) Load program from cassette
- 5) Continue program load from cassette
- 6) Store program to cassette.

When the CASSETTE facility is initially selected a check is made to see if there is a cassette in the unit. If not, 'NO TAPE LOADED' is displayed and depression of RESET restores normal operation. If the cassette tape 'clear leader' is detected when a cassette operation is selected the PNC 3 runs the cassette for 5 seconds and if the clear leader is still detected, "TAPE ERROR" is displayed. Depression of RESET restores normal operation. If the clear leader has passed the cassette read head the selected cassette operation continues. Some cassette tapes have very long clear leaders and it may be necessary to reselect the cassette operation required thus giving the cassette tape a further 5 seconds to pass the clear leader. If the end of tape clear leader is detected during a cassette operation e.g. during a cassette load "TAPE ERROR" is displayed. Depression of RESET restores normal operation.

1) REWIND CASSETTE enables a cassette to be rewound to the start, i.e. to the clear leader. This operation should be performed prior to recording onto a new cassette and it should be performed before a cassette program is loaded into PNC 3 memory. The rewind operation may be stopped by pressing the RESET key.

If important data is stored which must not be overwritten, cassettes can be protected by punching out two holes at the top of the cassette.

If a cassette having had the two holes made is placed in the PNC 3 and an attempt is made to record a program, the message "CASSETTE IS WRITE PROTECTED" will be displayed.

2) ERASE CASSETTE enables a cassette to be erased i.e. cleared of programs. The cassette should first be rewound. When a cassette is erased 'PROGRAM END' is recorded at the start of the cassette to indicate that there is no further data on the cassette. The cassette erase operation takes approximately 3 minutes for a 50 ft long cassette tape.

3) FIND END OF CASSETTE DATA. This command brings the tape to the end of the recorded programs i.e. to the message "PROGRAM END". The cassette is then ready for other programs to be stored.

4) LOAD PROGRAM FROM CASSETTE enables a program which is on the cassette tape to be loaded from the cassette into PNC 3 memory. The operator may now look for the next cassette program identifier located by depressing Key 4. The program number is requested and by giving the program number and pressing the ENTER key the PNC 3 will search for the number, displaying in turn the numbers on tape which are found, until the program required is found or until the tape end is found.

If the program number is not known, by pressing key 4 followed by ENTER the first program on tape will be found and its program number displayed. Press ENTER key to load into memory or press any other key to proceed to the next program on tape. This procedure may be carried on until tape end is found. Depression of RESET restores the cassette menu.

When data is loaded from the cassette unit into PNC 3 memory a check is made on the validity of the data and if an error was detected during the load process "TAPE ERROR" is displayed, and the memory will not be loaded. If RESET is pressed normal operation is resumed.

Cassette data is validated as follows: when a program or an identifier is stored onto the cassette tape an algorithm is computed the result of which is dependant upon the precise data stored. The numerical result of this algorithm is recorded at the end of the data. When the program or identifier data is subsequently loaded into PNC 3 memory the same algorithm is computed and the numerical result is compared with the prerecorded value, if a difference is detected "TAPE ERROR" is displayed.

5) CONTINUE PROGRAM LOAD FROM CASSETTE.

This facility enables program data contained in PNC 3 memory to be continued i.e. extended, by a program previously recorded onto tape. This facility enables programs to be 'merged' to form larger programs.

6) STORE PROGRAM TO CASSETTE.

This facility enables program data contained in PNC 3 memory to be stored using the integral cassette recorder onto cassette tape. The program is stored after a cassette identifier has been keyed in. The cassette identifier (Program number) can be from 1 to 6 numerals.

Each program is stored as 4 elements separated by blank tape

- i) The cassette program identifier
- ii) The program
- iii) The tool offsets associated with the program
- iiii) A cassette end 'END'

The cassette end is stored to enable the end of the recorded tape to be found when additional programs are to be stored, as each cassette tape can contain many programs. When a program is stored the cassette tape is initially rewound for a short time and then the 3 elements are recorded, this removes any previously recorded cassette END.

It is strongly recommended that more than one recording of the program is made in case one copy becomes corrupted.

EDIT facility

This facility enables a program that is in PNC 3 memory to be edited. When the edit mode is selected the operator may choose to display any block of data.

Seven operations are possible in Edit mode, they are:

- 1) Previous
- 2) Next
- 3) Replace
- 4) Delete
- 5) Add
- 6) Alter
- 7) Search

During edit operations generally 3 blocks are displayed. The selected operation is performed on the bottom block displayed.

- 1) Previous - Depression of key 1 decrements the block numbers displayed by 1
- 2) Next - Depressing key 2 increments the block numbers displayed by one
- 3) Replace - The bottom block displayed may be replaced. This facility differs from the Alter facility in that the replaced block is completely deleted and the new block is inserted.
- 4) Delete - The bottom block displayed may be deleted by using the delete
- 5) Add - A new block of data may be added to a programmed sequence. The data is inserted into the sequence immediately before the bottom block displayed, before ADD was depressed, and all subsequent block numbers in the programmed sequence will be incremented by 1.

Note: If it is desired to add a block or a number of blocks to the end of a programmed sequence the 'Continue memory load from keyboard' facility should be used.

- 6) Alter - The data programmed in the bottom displayed block can be altered using the ALTER facility. The data to be replaced is Keyed in together with its identifier, other data contained in the block will remain unchanged.

- 7) Search - The search facility enables any block in the program to be displayed.

JOG facility

The machine can be jogged into position by depressing the directional arrow in the jog button cluster found bottom left on the panel. 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.

SCALE facility

Program Scaling can be accomplished outside of a program i.e. overall scaling, or inside a program i.e. program scale. Program scale allows the user to key in a value in the range of 0.01% to 650% as a scaling factor for all subsequent X & Y dimensions, Z axis dimensions remain unchanged. Its effect can be cancelled by inserting a Program Scale of 100%.

Overall scale is permitted outside program loading sequence. It allows the user to key in a value in the range 0.01% to 650% as a scaling factor for the complete program. It is automatically set to 100% whenever a new program is loaded.

NOTE Any program scales active in the program will be further scaled by this overall scale function so dimensions can be reduced by up to 0.01 x 0.01% i.e. 100,000,000 times or increased by up to 6.5 x 6.5 i.e. 42.25 times.

MIRROR facility

The mirror program menu includes:

- 1) Mirror X
- 2) Cancel Mirror X
- 3) Mirror Y
- 4) Cancel Mirror Y

Mirror X: This function is only permitted as part of a program loading sequence. It causes the PNC to establish an axis of symmetry at its current X position. All succeeding blocks are then reflected about this axis.

Note:- This function uses the actual X position that it has attained as its axis of symmetry taking into account all scaling factors, any previous X negates and all offsets including repeat offsets.

Mirror Y: This function is identical to 1) Mirror X except that it acts about the Y axis.

OFFSET facility

The PNC 3 offset facility enables programmed dimensions to be offset to any point within the machine movement limits. Separate offsets within and outside the programmed sequence are permitted. These are known as program and machine offsets respectively.

Program offsets (G54) can be used to offset parts of the program. They are incremental in operation i.e. if block 1 is X offset 10mm and block 2 is 5mm, programmed moves from block 3 onwards will be offset by a total of 15mm. Further more if a program offset appears as a block within a repeat loop then the offset will be added into the total offset every time the repeat loop is executed. Program offsets are reset to zero every time the program is executed from the start.

The machine offset (G55) can be used to offset the entire program.

REPEAT facility

The repeat facility enables specified sections of a programmed sequence to be repeated with specified offsets. The repeat facility is only available within a programmed sequence. The data required to specify a repeat is:

- 1) The start block number to be repeated
- 2) The end block number to be repeated
- 3) The number of repeats required
- 4) The required offset dimensions with required FEED changes if any.

Repeats may be programmed up to a nested level of 3, should this level be exceeded "Nest error in repeat levels" is displayed. RESET restores normal operation.

DWELL facility

Variable time delays from 0.1 to 9999.9 seconds can be programmed with the DWELL facility. Simply press the DWELL key and enter the required amount of delay in seconds.

PROG. STOP

The PROG. STOP key can be used in two ways. Firstly it acts as an extra stop button operating in the same way as the stop button to the right of the screen i.e. it can be used to stop axis motion and to stop DWELL blocks before they have timed out.

Secondly it can be used during a program load sequence to enter a Program Stop block (M00). When a program stop is executed by the PNC3, it causes the PNC3 to wait until the START button is pressed before it will execute the next block.

FLOAT DATUM

A floating datum block (G99) will cause the PNC3 to establish a datum position (all axes set to zero) at its current position. All subsequent blocks will then be relative to this datum position. Also the displayed position will be relative to this datum position.

ABSOLUTE DATUM

An Absolute Datum block (G98) can be programmed which will cause the PNC3 to seek its machine datum in each axis in turn. Pressing the <ZERO> key during a program load sequence will cause a G98 Absolute Datum block to be entered.

INCH/MM

The working units of the machine can be altered to and from inches and millimetres by pressing the INCH/MM key. If this key is used during a program load sequence then a G70 or G71 block will be inserted into the program. When this type of block is executed, the axis position readouts will change to the appropriate units.

ABSOLUTE/INCREMENTAL

The mode of the machine can be altered to and from absolute and incremental by pressing the ABS/INC key. The current mode of the machine is displayed in the top right hand area of the screen along with the current units (INCH/MM). In absolute mode all coordinates refer to absolute positions. In incremental mode any keyed in coordinates is added to the current position (or the position in the previous block if entered during program load).

SPINDLE SPEED control

On PNC3 's with programmable spindle speed control, the following codes can be used.

M03 Spindle Forward
M04 Spindle Reverse
M05 Spindle Stop

Additionally these features can be selected by using the dedicated spindle keys FWD, REV, OFF situated directly below the PNC3 screen.

When FWD or REV or M03 or M04 is selected the PNC3 will prompt the user to enter a spindle speed in RPM. It will also display the allowable speed range as part of the prompt. If the user tries to execute a spindle forward block while the spindle is already turning in reverse (or vice-versa) an error message will be displayed indicating that the spindle must be stopped first. Spindle speed changes can be executed at any time providing the direction of rotation is kept the same.

Also the spindle speed override keys (marked +, -) can be used at any time to increase or decrease the spindle speed. If the + key is used to start the spindle from rest, the direction will be the same as when it was last rotating, (with a default to forward when the PNC3 is first powered up).

Some PNC3's have a simple two speed spindle speed system, high speed or low speed, in which case M03 and M04 have different meanings as follows.

M03 High Speed
M04 Low Speed

When these codes are used no actual spindle speed in RPM is required. M05 is used in the normal way to stop the spindle. With this system it is possible to change from one speed to the other without stopping the spindle.

COOLANT CONTROL

Coolant can be switched on or off as required using the dedicated coolant on/off keys (situated just below the PNC3's screen) or using the following codes.

M08 Coolant on
M09 Coolant off

Additionally the same keys can be used during a program load sequence to enter a program block which turns the coolant on or off, in which case the coolant will automatically be turned on or off by the PNC3 when that block in the program is executed.

THREAD (Lathe only)

Threading blocks can be programmed either by keying G33 and pressing <ENTER> or by pressing the <THREAD> key. The PNC3 will then display the following menu:-

- 1) External
- 2) Internal

The user should press 1 or 2 depending on whether an External or Internal thread is to be cut. There are 6 parameters needed to define a thread.

1) The thread diameter must be entered. This diameter always refers to the largest diameter of the thread (see diagram) whether the thread is external or internal.

2) The pitch must be specified. The pitch must be kept in the range 0.1mm to 6.4mm.

3) The depth of thread must be specified.

4) The number of cuts i.e the number of passes along the thread length that the tool will make to reach the desired depth. The amount of material taken in each pass is therefore equal to the depth divided by number of cuts. To ensure a good finish to the thread, the PNC3 will automatically add 3 finishing passes at the full depth. The number of cuts must be in the range 1 to 99.

5) The length of the thread must be specified. The length can either be a positive or a negative value, defining it to be either to the right or to the left of the start position.

6) And finally the start position in the Z axis must be defined. Note that the PNC3 will actually position the tool 3mm away from the start position at the beginning of a thread block. This ensures that the correct feedrate for the thread can be achieved before the actual start point is reached.

For a thread block to execute properly, the spindle must be rotating at the correct r.p.m. There is a minimum spindle speed of 100 r.p.m. fixed by the fact that the spindle motor does not rotate smoothly below this figure. The maximum spindle speed varies with the desired pitch according to the formula:-

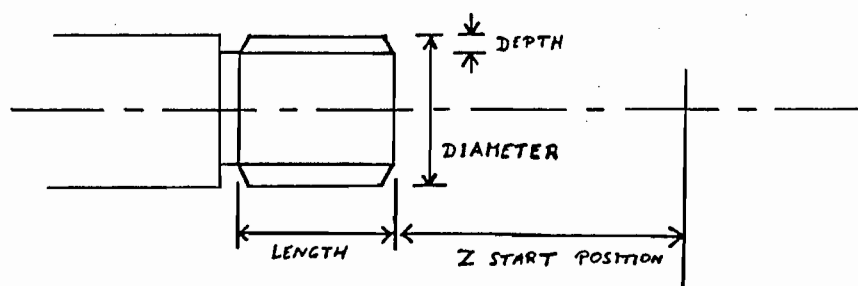
$$\text{Maximum Speed} = 750/\text{Pitch m.m. r.p.m.} - 10\%$$

But this is subject to a fixed overall maximum of 330 r.p.m.

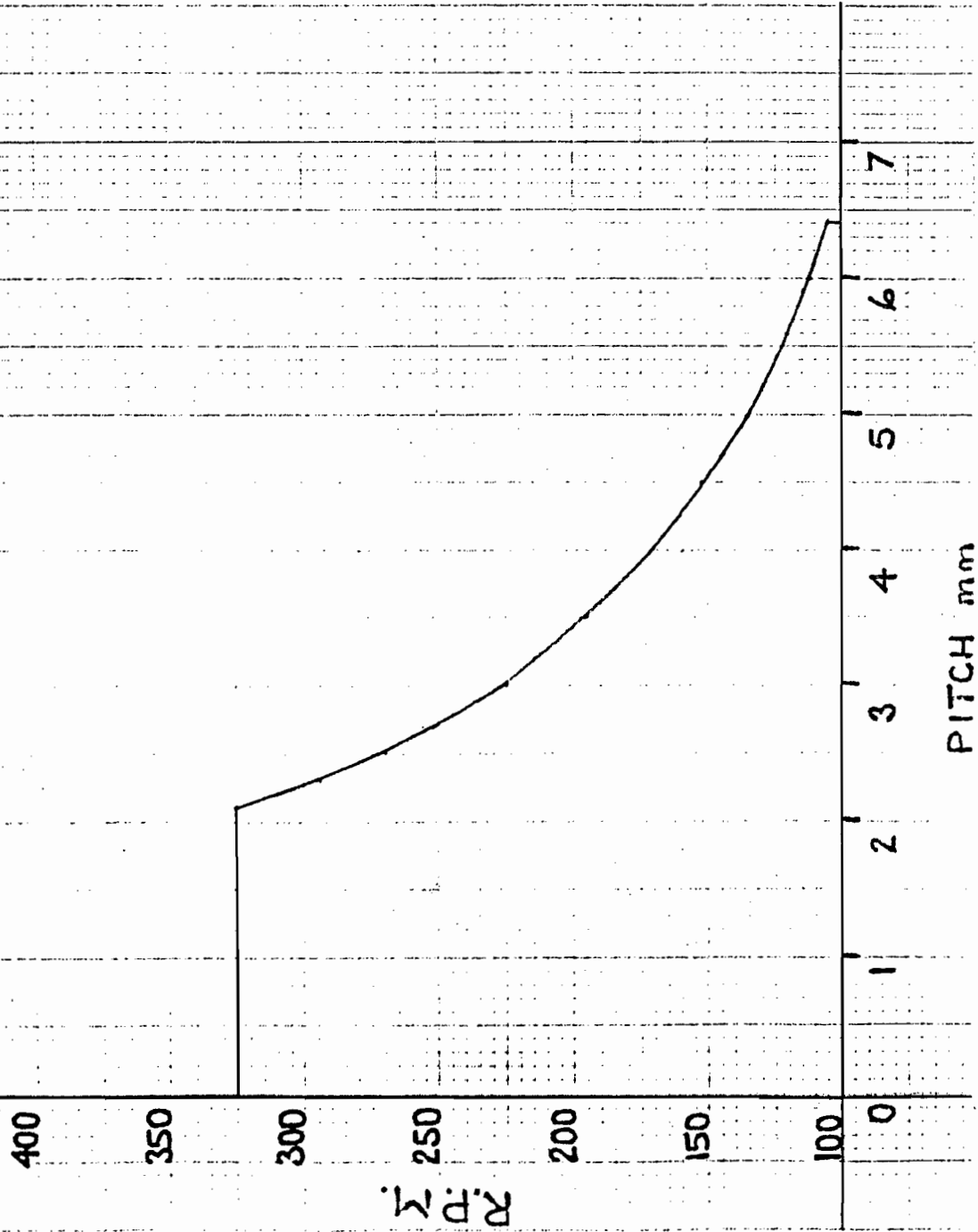
i.e. pitches of less than 2.1m.m. cannot be cut at more than 330 r.p.m.

When a thread block is executed, the PNC3 will move the tool from its present position to a point 3mm away from the start point in a straight line at the maximum feedrate. It will then make the screwcutting pass, stop, bring the tool out to a clearance point 0.5mm away from the work, in X axis then move back to the start position in Z before repeating the screwcutting pass at an increased depth. All moves except each screwcutting pass will be carried out at the maximum feedrate. The clearance position will of course vary depending on whether a front or back tool is used or whether an internal or external thread is being cut.

The following diagram shows the relationship of the depth, length, diameter and start position of the thread block.



PNC 3. LATHE SCREWCUTTING. PERMISSIBLE SPINDLE SPEEDS.



Cutter Offsets

Tool radius compensation (or automatic cutter offset calculation) is available on PNC3's configured to control Milling machines. It allows the machine programmer to compensate for the diameter of the current cutter. This means that the PNC3's program need only specify the true dimensions of the desired finished product. This facility is obtained by pressing the COMP key either inside or outside a Program loading sequence. Alternatively the appropriate G code can be used as follows.

G40 Cancel Tool Compensation
G41 Tool Compensation Left
G42 Tool Compensation 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 (with a G41 or G42 block), the PNC3 will look ahead to the next 2 X, Y positions in the program and use them to calculate the first offset position. The PNC3 will then move from its present position to the offset when it executes the 1st of these 2 X, Y blocks.

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 to calculate the correct offset path it must "look ahead" from its current block to the next block. In fact the PNC 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'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.

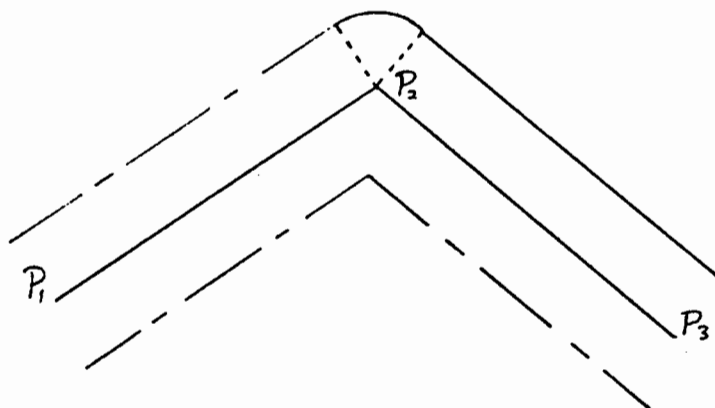
In the following examples the terms internal and external corner will be used. They are defined as follows:

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.

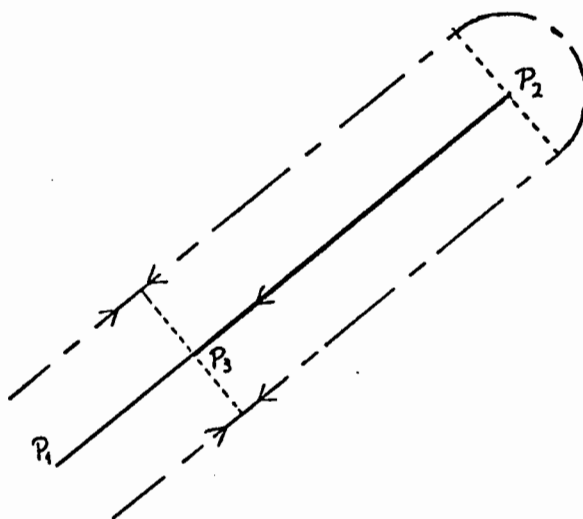
As will be seen from the following examples, the PNC's action on encountering an external corner is to move around it in an arc of radius equal to the cutter radius about a centre located at point P_2 . During this circular motion the tool will always be in contact with point P_2 thus creating a burr-free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

Straight Line to Straight Line Intersection

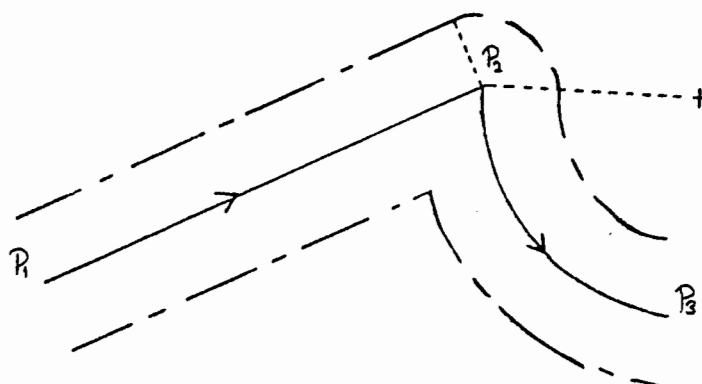


In the extreme case where block 2 actually doubles back onto itself this will always be taken as an external corner independent of whether a right hand or a left hand offset is being used.

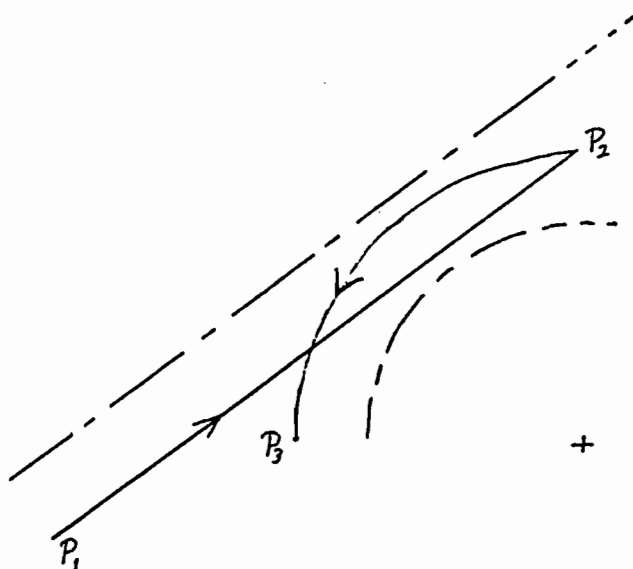
E.g.



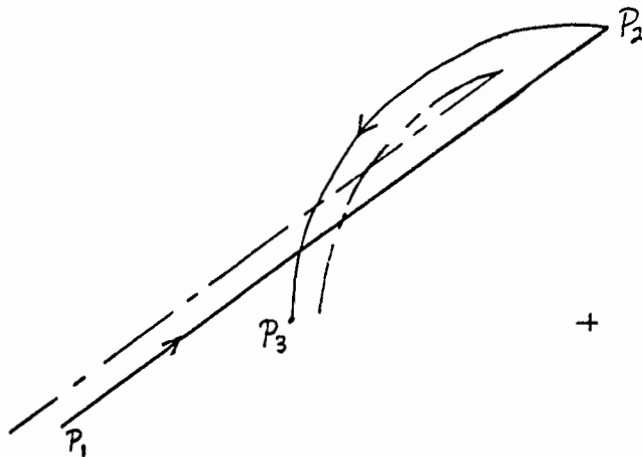
Straight Line to Arc Intersection



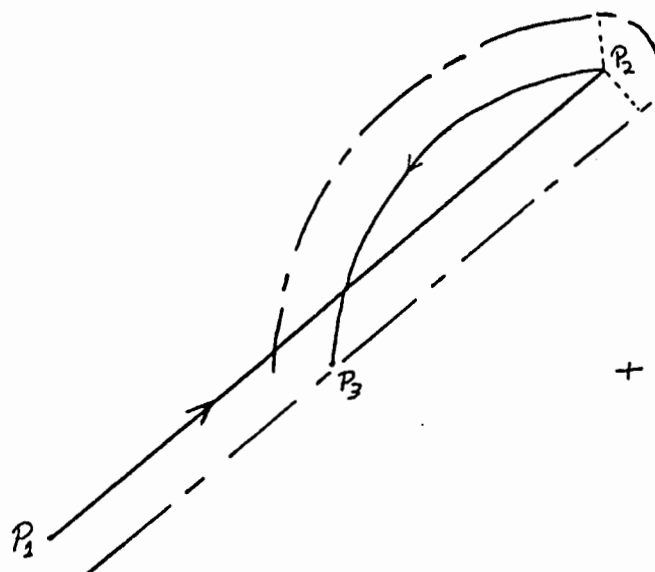
The PNC will cope with any kind of Line to Arc intersection except where no intersection is possible E.g.



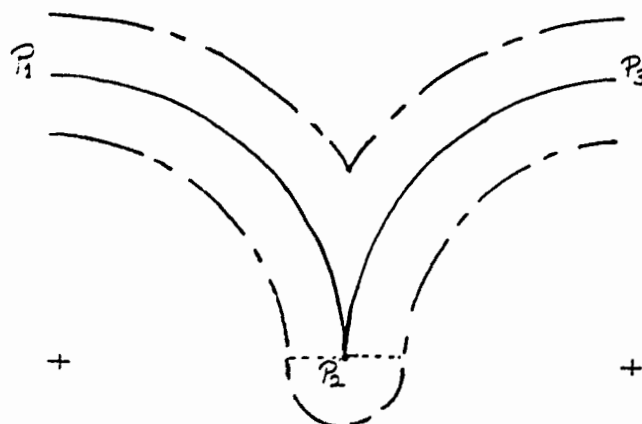
In which case there will be a pause during the calculation period, followed by an error message on the display indicating that perhaps the wrong tool diameter has been input. Note that in this case if the cutter diameter is small enough an intersection may be possible E.g.



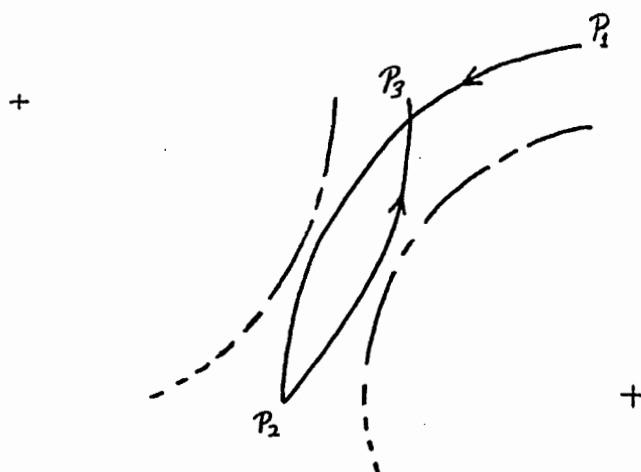
Note that in both the previous examples if a right hand (or positive) offset is applied the P_2 will be taken as an external corner and the following will result:



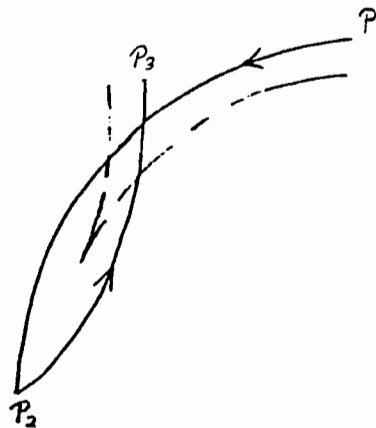
Arc to Arc Intersection



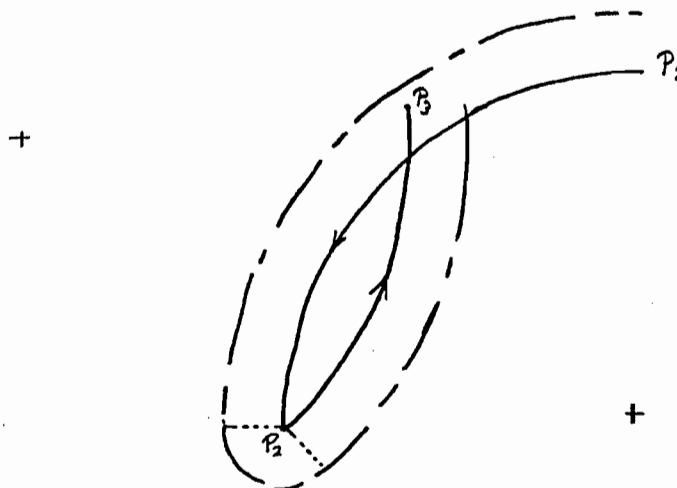
The PNC will cope with any kind of Arc to Arc intersection except where no intersection is possible E.g.



In which case there will be a pause while the PNC 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.



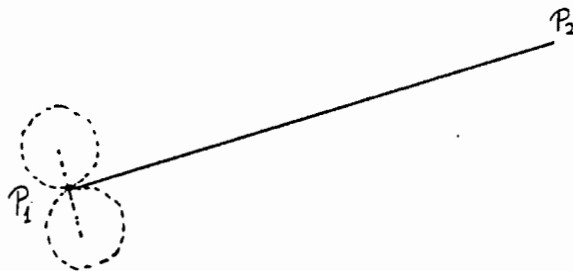
Note that in both the previous examples if a right hand (or positive) offset is applied then P_2 will be taken as an external corner and the following will result.



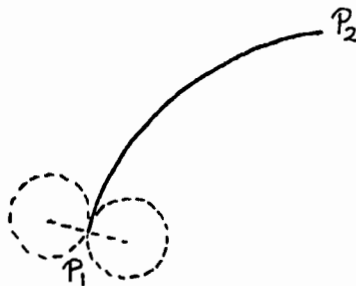
First Offset after Activation

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

Apart from a G40 Cancel Tool Comp block the following types of block will all de-activate the cutter offset.

M06 Tool Change
G10 Mirror X
G11 Cancel Mirror X
G12 Mirror Y
G13 Cancel Mirror Y
G20 Program Scale
G54 Program Offset

G81 Repeat block (although the Start block of the repeat may re-activate)
G98 Absolute Datum
G99 Floating Datum

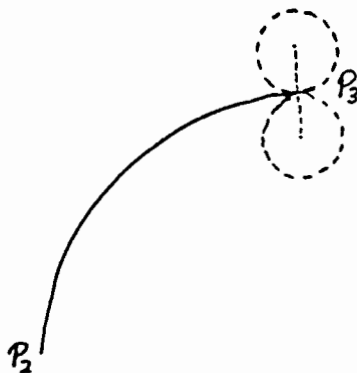
Also the last block in any program will automatically de-activate cutter offsets.

When the cutter offset has just been de-activated it will take up a position perpendicular to the point P_3 at a distance of the cutter radius away from it.

E.g.



or if the line P_2 to P_3 is circular

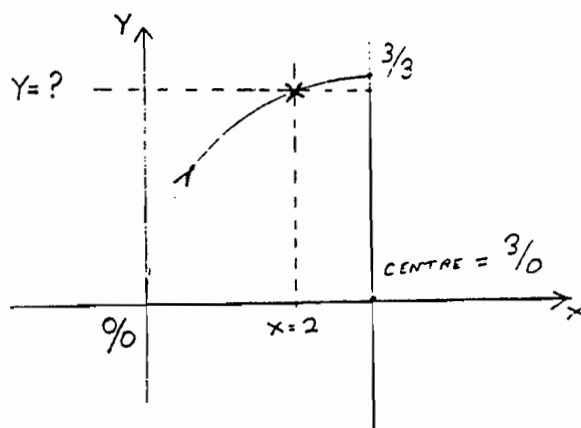


AUTO/S.STEP Positional Error Correction

For linear moves the PNC3 will always get to the desired X,Y (or for lathe X,Z) position. However in the case of circular moves, unless the circle centre is very accurately defined, it may not be possible for the PNC3 to achieve the programmed end position. This situation is particularly prone to occur when the circle end point and centre has been calculated from another computer. If the circle centre is known and the end point needs to be found, the best way of finding it is to use the method described in the section of this manual entitled "Circular movements", however this method is obviously not always convenient.

To help users who are driving the PNC3 from a computer, the PNC3 will do a certain amount of positional error correction. The amount of correction (or error tolerance) available to the user is selectable from + - 1 step in MDI or S.Step mode to + - 0.1mm in AUTO. (1 step = 0.005mm for a Lathe and 0.00625mm for a Mill). The error correction works in the following way, when the PNC3 has completed a circular move it compares its current position with the desired end point specified in the program block. If the error is less than or equal to 0.1mm in both axes the PNC3 will set up and execute a linear move from its current position to the desired end point. If the error is greater than 0.1mm then the PNC3 will display the error message "Machine did not get correct position".

This can be demonstrated with the following example



The desired end point is unknown in Y. by using the method described in the section of this manual entitled "Circular Movements", the actual end point can be found using this program.

Block

1 G01 X 0 Y 0

2 G02 X 2 Y 2

CENTRE 3 Y 0

3 M02 End of program

When this program is executed (in S.Step or AUTO) the PNC3 will give the error display after block 2 "Machine did not get to correct position". Set the switch to MDI and press the <RESET> button. Note the correct position of X =2, Y =2.83 and edit these values in as the end point for block 2. Run the program again and see that the correct position is attained and no error message results.

Now we shall introduce an error of -0.02mm by altering the end point of block 2 to be X=2, Y=2.81. Now run the program in S.Step mode and note that the PNC3 gets to position. X = 2, Y = 2.83 and displays the error message. Set the switch to AUTO and run the program again. This time the PNC3 will get to position X = 2, Y = 2.83 and then do a straight line correction move to X = 2, Y = 2.81 and will not give an error message.

Now alter the end point of block 2 to be X = 2, Y = 2.85. Again we have introduced an error of 0.02mm but in the opposite direction. If the program is run in S.Step mode it will be seen that the actual position attained is X = 2.055, Y = 2.85. If the program is run in AUTO, the PNC3 will self correct from this position the desired position.

Now alter block 2 to read

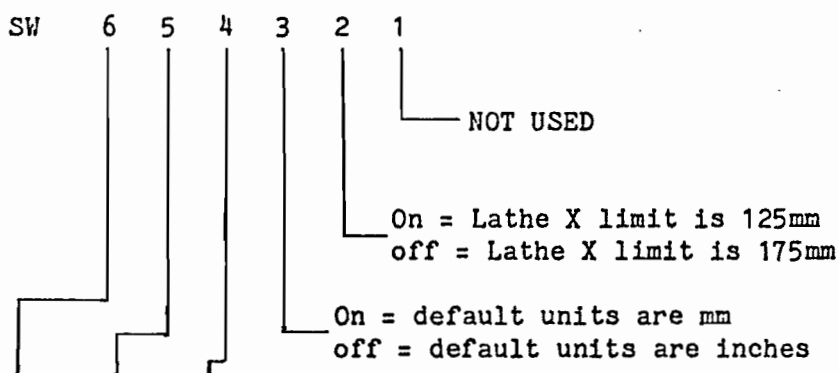
```
G02 X 2.625 Y 2.98  
CENTRE X 3.0 Y 0
```

Run the program in S.Step to prove that the end point is valid and that no error message will occur. If we now introduce an error of +0.02mm in Y axis by altering the end point to X = 2.855, Y = 3.0, It will be found that the actual position attained is X = 2.855, Y = 3.0. Note that the error in the X axis is now 0.23mm which is outside the tolerance band even when switched to AUTO. And this total error was introduced by an error of only 0.02mm in the end point specification.

In general the user should make sure that the radius at the start point is equal to the radius at the end point to an accuracy of ± 0.5 motor steps (i.e. ± 0.003125 mm for Mill, 0.0025mm for Lathe). Programmers who are involved in writing post-processors for the PNC3 are particularly advised to be aware of this fact.

SWITCH settings

There are 6 switches in a dual-in-line (d.i.l.) package that are used to configure a number of options inside the PNC3 at power on. These switches are mounted on the interface board and are only accessible when the PNC3's cover has been removed. When the cover has been removed by undoing the fixing screws and sliding it forward, the interface board can be seen at the right hand side of the PNC3. The dil switches are mounted in the bottom right hand area of the interface board. The meanings of each switch are as follows:-



Buad rate selection

On	On	On	=	75 Baud
On	On	Off	=	110 Baud
On	Off	On	=	150 Baud
On	Off	Off	=	300 Baud
Off	On	On	=	1200 Baud
Off	On	Off	=	2400 Baud
Off	Off	On	=	4800 Baud
Off	Off	Off	=	9600 Baud

N.B. On = Switched closed
Off = Switch open

ERROR MESSAGES

This section gives a list of the PNC3's error messages, a description of why the error occurred and, where applicable, a possible solution to the error.

No Program Loaded

CAUSE: The PNC3 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 PNC3 to MD1 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 PNC3's memory can hold.

Machine did not get to correct position

CAUSE: This error message is displayed when the PNC3 executes a circular move which reaches the desired position in only one of the X and Y axes.

SOLUTION: Re-calculate the circle end-points and amend the block which caused the error.

Move Exceeds Machine Limit

CAUSE: A move which takes the PNC3 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.).

Machine Limit Exceeded in X/Y/Z

The PNC3 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.

Motor 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.

SOLUTION: If more than one of the red LEDs are 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.

Spindle Drive System Error

This error message is displayed if the spindle does not reach the programmed speed.

Spindle direction cannot be changed whilst spindle moving

CAUSE: This message will be displayed if an attempt is made to execute a SPINDLE FWD block while the spindle is moving in reverse, or a SPINDLE REV block while the spindle is moving forward.

SOLUTION: Execute a SPINDLE OFF block before the required spindle move block.

Spindle Speed too High/Low

CAUSE: The PNC3 will display this message if a screwcutting block is executed while the spindle speed is too high or low for the programmed pitch.

SOLUTION: Reduce or increase the spindle speed before executing the screwcutting block.

Repeat Start block must be linear with all axes defined

CAUSE: During the input of a repeat block, if an attempt is made to specify a start block which is either non-linear or in which all axes are not defined, the above message is displayed. This error can also occur while running a program if the program has been edited in such a way as to cause the start block to become non-linear or not to have all axes defined.

SOLUTION: Make sure that the start blocks of any repeat blocks used are linear with all axes defined. During editing, if any blocks are added or deleted before the start of the repeat loop, amend the start and end block parameters to reflect the new block numbers.

Nest Error in Repeat Levels

CAUSE: The PNC3 will only implement four nested repeat loops at any time (A "nested" repeat is a repeat of a repeat). This error indicates that a fifth level of nesting has been attempted.

SOLUTION: Simplify the program.

Tape is Write Protected

CAUSE: If the "write enable" tab on the tape currently in the cassette unit has been removed, it is impossible for the PNC3 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 PNC3's cassette functions while there is no tape in the cassette unit.

SOLUTION: Put a tape in the cassette unit before using the PNC3'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 PNC3 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.

Data Error

CAUSE:- The data stored on the tape is invalid. This could be caused by tape stretching or data corruption due to magnetic fields.

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

Invalid Block for this Machine

CAUSE:- This message is displayed if an attempt is made to load in a block from the tape which the PNC3 does not recognise.

SOLUTION:- Make sure that the program being loaded is valid for the PNC3 in use.

Lathe Tape

A program which has been stored on a PNC 3 lathe cannot be loaded into a PNC 3 mill.

Mill Tape

A program which has been stored on a PNC 3 mill cannot be loaded into a PNC 3 lathe.

Printer Error

CAUSE: This message is displayed if there is no printer connected to the PNC3 or the printer is off line i.e. not ready to receive data.

SOLUTION: Connect a printer to the PNC3 and make sure it is on line.

Error in Circle Input

CAUSE: All circular moves must be confined to a single quadrant. This error indicates that the start and end points of a circular move are not within the same quadrant as relative to the given centre point.

SOLUTION: If a circular move between two points in different quadrants must be executed, break it up into smaller moves which end on the intervening quadrant boundaries.

X and Y Axes must be Defined

CAUSE: The X and Y axes must be defined before the circle centre may be keyed in , otherwise the above error message is displayed.

SOLUTION: Key in the X and Y coordinates before attempting to input the circle centre coordinates.

Centre must be Defined

CAUSE: This error message is displayed if the EOB key is pressed before the circle centre is defined.

SOLUTION: Define the centre before pressing EOB.

This Type of block cannot be Altered (use replace if necessary)

CAUSE: The alter function (in edit) can only be used to alter linear or circular blocks. If an attempt is made to ALTER any other type of block, the above error message is displayed.

SOLUTION: Use REPLACE instead.

Already Selected

This error message is displayed if an attempt is made to re-select an auxilliary or input which has already been selected during input.

No Changes so Block Ignored

This message is displayed if an M20 AUXILIARIES block is loaded which is identical to the last auxiliaries block in the program or an M21 INPUTS block is entered without any parameters.

Value not in Range

This error occurs if a value which is outside the allowed range is keyed in.

Scaled move out of Range

If a scaling operation causes a subsequent move to exceed the machine limits, the above message is displayed.

Cutter Diameter too Big

If the PNC3's cutter compensation facility is being used and the compensation produces a move which is in the opposite direction to the original move, the above message is displayed. See section entitled "Effects of Cutter Offsets".

TOOL SELECTION

A Tool change block (M06) can be programmed by pressing the T key during a program load sequence. If the T key is pressed when not in a program load sequence, a menu of 3 options will appear on the screen.

- 1) DISPLAY & EDIT TOOL OFFSETS
- 2) SET TOOL OFFSETS
- 3) CHANGE CURRENT TOOL

The third option allows an M06 Tool change block to be executed directly, thus setting the offsets correctly for the chosen current tool. These options vary in their operation according to whether the PNC3 is controlling a Lathe or a Milling machine.

With a Lathe, when a Tool change block is executed from program the PNC3 will wait at that block until the Program Start button is pressed, (even if AUTO is selected) whereupon it will take the new tool's offsets into account for all further moves. If however the new tool has been specified as an autochange tool (see the section of this manual entitled "Tool Setting") then the PNC3 will not wait for the START button to be pressed if switched to AUTO.

With a Milling machine, when a Tool change block is executed from program the PNC3 will first stop the spindle (if it is currently moving) and then wait until the Program Start button is pressed whereupon it will take the new tool's offsets into account for all further moves. Also if the spindle was previously moving then the PNC3 will bring the spindle up to its previous speed before moving on to the next block in program.

DISPLAY & EDIT TOOL OFFSETS

The current Tool offsets can be displayed by pressing the T key followed by selection 1 from the menu. In the case of a Lathe each tool has its own X and Z offset. In the case of a Mill each tool has its own length (Z) and diameter offset. All 16 pairs of offsets will be displayed and the PNC3 will then prompt the user to enter a tool number. If the user does not want to edit any of the tool offsets, then simply pressing RESET at this point will return the PNC3 to its previous menu.

Tool offset editing can be accomplished by entering the tool number to be edited and pressing ENTER. The selected tool's offsets will then appear just above the prompt line, and the prompt will change to "Enter Tool Offset Changes" followed by "X" for Lathe or "Z" for Mill indicating the axis to be changed. Values entered at this point will be added to the current offset. If it is required to reduce the offset then negative values must be entered. When ENTER is pressed the prompt will change to "Z" for Lathe or "D" for Mill and this offset value can be changed in the same way. Note that for a Mill the total diameter offset cannot be negative and if the user tries to enter an offset change value which would result in a negative diameter, the PNC3 will automatically enter a total diameter offset of zero.

N.B. When a tool is edited the PNC3 also assumes that the tool is to become

the current tool and the top left display and the present position display will change accordingly.

TOOL SETTING

By pressing the T key and selecting choice 2 from the menu (See Tool Offsets), the toolsetting mode can be initiated. The precise procedure required to set the tool offsets is dependant upon the type of application, i.e. whether the PNC3 control unit is fitted to a Lathe or a Mill.

MILL TOOL SETTING

First the PNC3 will prompt the user to enter the number of the tool to be used. When this number is entered the current offsets for that tool will be displayed and the prompt will change to "Press <ENTER> to fix offset". The user is now invited to use the axis jog keys to position the cutter so that it just touches the surface of the workpiece. Pressing <ENTER> causes the actual Z position of the cutter to be loaded in as the Z offset for that tool, and consequently the Z axis display will change to zero. The prompt will now change to "Enter tool offset changes D", and the user can enter a value which will be added to the current tool diameter offset. In the same way as tool offset editing works, the resulting diameter cannot be less than zero and if the user tries to enter a change which will result in a negative diameter then the PNC3 will automatically set the diameter offset to zero. When the diameter offset has been entered, the prompt will change to "Press <EOB> to accept data". When <EOB> is pressed the PNC3 comes out of toolsetting mode. <RESET> can be pressed at any time to recover from any mistakes.

LATHE TOOL SETTING

First the PNC3 will prompt the user to enter the number of the tool to be used. When this number is entered, a menu of 4 different types of tool is shown and the user is invited to select the type of tool currently being used. The different types of tool are front or rear and auto or manual change. When the type of tool has been entered, the current offsets for that tool will be displayed just above the prompt line and the prompt will change to "Press <ENTER> to fix offset".

The user is now invited to use the axis jog keys to face the work piece. Of course if the work piece has already been faced or is not to be faced then it is only necessary to bring the tool up to touch the face. If the workpiece is to be faced then the spindle can be brought up to speed using the dedicated spindle control keys situated just below the display. Pressing <ENTER> causes the actual Z position of the tool to be loaded in as the Z offset for that tool, and consequently the Z axis display will change to zero.

The X offset can now be set. The prompt will change to "ENTER measured dia" and the user is requested to use the axis jog keys to turn a short length of the workpiece. Again it is only necessary to bring the tool into contact with a known X position, but the spindle control keys are available if turning is required. With the tool still touching the workpiece, enter the actual diameter of the workpiece by keying X followed by that value. Pressing <ENTER> causes the X offset to be fixed for that tool and consequently the X axis display will change to be half of the entered diameter.

SELF TEST FACILITY

When the PNC 3 has just been powered up there are 4 self test routines that can be performed.

1. S Pressing the S key will cause the PNC 3 to do a self test on its own Microprocessor's program. The display will show "actual" and "correct" values for the results of these tests. Any difference between these values will cause the PNC 3 to behave in an unpredictable manner.

While these self tests are being performed the display will also show the date the program was assembled, the version number of that program and in most cases the correct crystal frequency needed by the software. It is important that the correct crystal is fitted as this controls the data link baud rate, cassette data rate and spindle speed readings.

2. T Pressing the T key will cause the PNC 3 to test and display the conditions of its input ports. These ports are:

INPUTS: Showing the status of the input switches in the order:

I/P7 I/P6 I/P5 I/P4 I/P3 I/P2 I/P1 I/P0

A "1" indicates the switch is open and a "0" indicates that it is closed

PRINTER The 1st digit represents the Printer Acknowledge (0 = ACK) and the second digit is the Printer Busy (1 = BUSY).

CASS: Shows the status of the cassette unit clear leader, cassette present and file protect detectors respectively, the fourth being the data in from the cassette.

M/C SWITCHES Shows the condition of the STOP, AUTO, START & SINGLE STEP switches respectively. A "0" represents the switch is closed.

DATUMS: Shows the conditions of the T Axis, Z Axis, Y Axis and X Axis datum markers respectively. A "1" represents axis at datum.

STEP & DIRN: Shows the step feedback from the stepper motor drives and the axis directions which are all internal to the PNC 3.

DRIVE FAULTS: Shows the conditions of the T axis, Z axis, Y axis and X axis stepper motor drives respectively. A "0" represents a drive fault in that axis.

TOOL OFFSETS Lathe

Tool offsets enable the PNC3 to automatically compensate for variations in distance of the cutting point from the datum point for up to 16 different tools. When the tool offset has been correctly set (see the section entitled "Tool setting" in this manual), the $X = 0$, $Z = 0$ point will refer to the cutting point of the tool being on the lathe centre line at the face of the workpiece.

This type of tool offset is always active and cannot be cancelled except by selecting a tool which has zero offset.

TOOL OFFSETS Mill

In addition to cutter radius compensation, the PNC 3 compensates for variations in tool length. When the tool length offset has been correctly set, the $Z = 0$ point will refer to the cutter being positioned on the surface of the work.

This type of tool offset is always active and cannot be cancelled except by selecting a tool which has zero length offset.

O/T's

Shows the condition of the M/C overtravel switches. E.g. "X+" means the overtravel switch at the upper limit of the table in the X axis is closed.

DIL SWITCHES:

Shows the status of the dil switches. See section on switch settings in this manual. They are displayed in descending order, switch 6 to switch 1. A "0" represents the switch closed.

3. X Pressing the X key will cause the screen to be filled with the letter X. This enables the screen geometry to be set up i.e. the height, width, rotation and linearity. This setting up is done at the factory and should not normally need adjusting.
4. KEY TEST Holding any key in whilst powering the PNC 3 up will cause the key test routine to be carried out. In this test each time a key is pressed, a number will appear on the screen. Each key has a different number. Pressing the stop button will terminate this test. Things to watch for in this test are to make sure every key registers on the screen and that each key has a unique number.