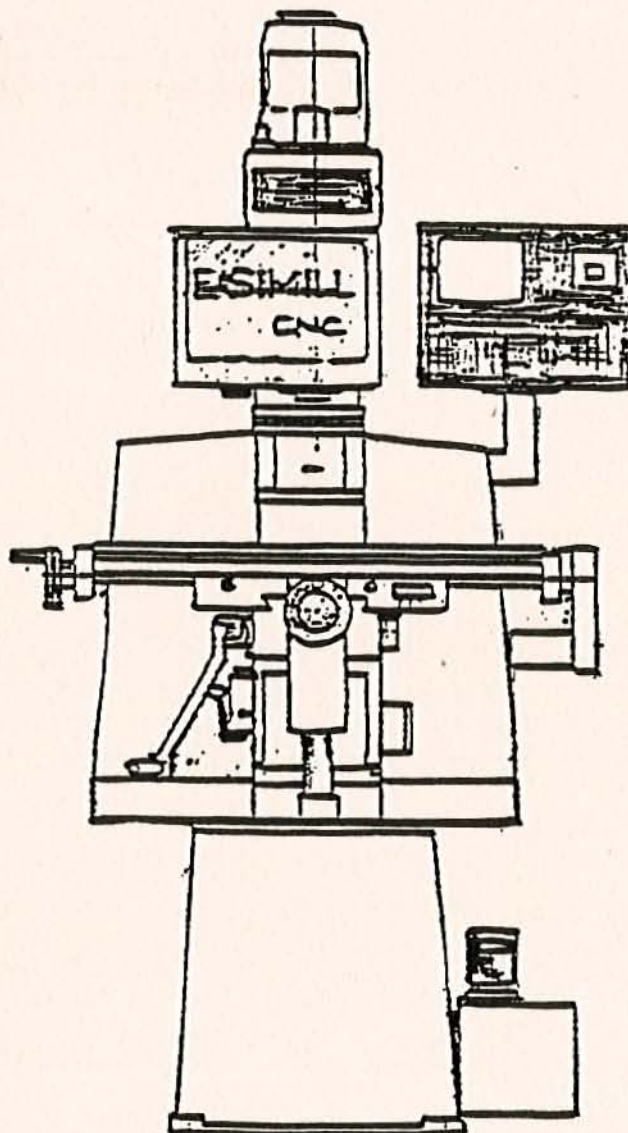


INSTALLATION, PROGRAMMING AND MAINTENANCE MANUAL



DENFORD MACHINE TOOLS LIMITED: BIRDS ROYD: BRIGHOUSE: WEST YORKSHIRE: Phone: 0484 712264: Grams: Dermacto: Telex: 517478

DENFORD

MACHINE TOOLS

INDEX

PAGE

INTRODUCTION

<u>SECTION</u>	1	CONTROL SYSTEMS	1
	2	TECHNICAL SPECIFICATIONS AND STANDARD EQUIPMENT	3
	3	INSTALLATION AND LUBRICATION	6
	4	MAINTENANCE	9
	5	MACHINE AXIS FORMAT	11
	6	MANUAL JOG MODES	15

KEYBOARD

			17
7	AXIS POWER AND EMERGENCY STOP		21
8	CYCLE MODE AND OPERATION		23
9	MANUAL JOG KEYS		25
10	FUNCTION SECTION		28
11	MODE SECTION		31
12	NUMERICAL KEYS		33
13	LIST OF M AND G CODES		34
14	STARTING EASIMILL FROM COLD		40
15	MOVEMENTS AND ASSOCIATED FEED INPUT		43
16	CIRCULAR MOVEMENTS		45
17	AUTO/SINGLE STEP POSITIONAL ERROR CORRECTION		50
18	SETTING AN ABSOLUTE ZERO		53
19	LISTING FLOATING DATUM		57
20	SETTING TOOL LENGTH OFFSETS		58
21	EDITING TOOL LENGTH OFFSET		62
22	SPINDLE AND COOLANT CONTROL		63

PROGRAMMING

			65
23	ABSOLUTE/INCREMENTAL		66
24	ENTERING A NEW PROGRAM		67
25	REPEAT FUNCTION		68
26	MIRROR IMAGE FUNCTION		70
27	SCALING FUNCTION		72
28	OFFSET FUNCTION		74
29	CUTTER COMPENSATION		76
30	EFFECTS OF CUTTER COMPENSATION		79
31	AUXILIARY FUNCTIONS		88
32	INPUT FACILITIES		89

	<u>PAGE</u>
<u>SECTION 33</u> DWELL FUNCTION	90
34 EDITING	91
35 CASSETTE OPERATION	93
36 DATA LINK FACILITY	96
37 EXAMPLES	111

INTRODUCTION

Easimill utilizes the latest advances in microprocessor technology. These advances can be combined with a full backup facility, extra equipment, quick change tooling systems, and work holding accessories to provide a powerful and very versatile machine tool.

Other important features of Easimill include computer software which enables the simulated running of programs for error detection, which can save valuable production time. Inputs and outputs can be monitored to enable Easimill to work with robots in a flexible manufacturing system. The RS232 link allows DNC operation to be a standard feature of the machine, thus allowing direct programming from a host computer. CAD/CAM systems utilize this function to the full.

Easimill is fitted with a parallel printer port, to offer the user full or part program hard copy printouts.

Easimill's programming format is to the International Standard incorporating G and M codes. Alternatively function keys can be used in their place to ease programming. These functions keys offer a simple machine shop language so programming can be accomplished successfully by operators having no CNC experience of G or M code format.

Error messages are built into the system to help the operator to drastically reduce the amount of erroneous data being programmed.

As with any CNC machine tool, the programmer and operator should have a good knowledge of machine shop practice and be familiar with machine shop terminology. The quality of components produced is only a reflection on the programmer.

SECTION 1

THE CNC CONTROL UNIT

PNC 3 DESCRIPTION

The PNC 3 is an extremely versatile continuous path, computer based programmable numerical control unit designed to control up to 4 axis of movement 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. The PNC 3 we are confident 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 the a) keyword system b) ISO G and M code programming. -* Switch a) and b).

The 9" Display provides the user with:-

1. A display of the complete machine status
2. Prompts to assist the user in using the control system
3. Sections of the program during program loading, editing and execution
4. 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, inch/metric and absolute/incremental programming with any mix, program scaling, mirror imaging and tool diameter and length compensation, 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.

CNC CONTROL SYSTEM SPECIFICATION

1. 490x290x335 mm Self Contained Console.
2. Green on Black 9" VDU with Anti-glare screen and outlet to external TV Monitor.
3. Alpha numeric keyboard allowing full manual Data Input.
4. Mini Magnetic Cassette Unit for Multi Program Storage.
5. MDI Single Step and Auto Selector for programs.
6. RS232C Interface 7 Din Pin connection to computers and paper tape punch units.
7. Parallel Printer Interface for obtaining hard copy of programs.
8. ISO Format - allowing 'G' and 'M' Code Programming from DIN 66025 extract.
9. Full 'G' and 'M' Code Listings on VDU when required to assist programming.
10. Single Mode Selection Keys.
11. Axis Jog on All Axes with variable feedrate and 0.01 mm step.
12. Feedrate override from 1 mm/min to 750 mm/min.
13. Programmable Spindle Speed 0-3000 RPM.
Programmable Feedrate 0-1500 mm/min.
14. Linear Interpolation on 2 axes with vectorially correcting feedrates.
15. Circular Interpolation on X-Y Plane.
16. Absolute/Incremental, Inch/mm programming throughout program build-up.
17. Manual and Programmable Program Stops.
18. Repeat facility allowing build-up of canned cycles for drilling and pocket milling cycles.
19. Floating Datum Facility.
20. 500 Block Memory (1000 blocks available).
21. Tool length Offsets for up to 16 tools.
22. Tool diameter compensation.
23. Programmable Coolant.
24. Programmable Dwell from 0.1 to 9999.99 seconds.
25. Four Auxiliary Outputs.
26. Six Programmable Inputs.
27. Mirror Imaging on XY planes simultaneously.
28. Program Scaling and Component Scaling from 0.01%-650%.
29. Program Offset allowing dry run facility.
30. Optional 4th axis control.
31. System resolution 0.01 mm (0.0004").

SECTION 2

TECHNICAL SPECIFICATIONS

AXES TRAVEL

LONGITUDINAL TRAVEL	(X)	375 MM - (14 $\frac{3}{4}$ ")
CROSS TRAVEL	(Y)	175 MM - (7")
VERTICAL TRAVEL	(Z)	70 MM - (2 $\frac{3}{4}$ ")

SPINDLE

SPINDLE BORE - R8 TAPER WITH LOCATING PIN

SPINDLE SPEEDS

- | | | | | | | | |
|----|---|------|-----|-----|------|------|------|
| | | HIGH | 384 | 614 | 1029 | 1675 | 2840 |
| 1. | STANDARD MACHINE - 10 FIXED SPEEDS VIA BELTCHANGE | / | | | | | |
| | | LOW | 183 | 292 | 490 | 796 | 1350 |
| 2. | VARIABLE SPEED (OPTIONAL) RANGE 100-3000 RPM | | | | | | |

SPINDLE DRIVE

- 1.2 KW 2 SPEED MOTOR (HIGH AND LOW) FOR STANDARD 10 SPEED MACHINE
- 1.5 KW FOR VARIABLE SPEED

AXES DRIVE MOTORS

HIGH POWER STEPPER MOTORS

X AXIS STEPPER MOTOR - 200 STEPS/REV

Y AXIS STEPPER MOTOR - 200 STEPS/REV

Z AXIS STEPPER MOTOR - 200 STEPS/REV

FEED RATES

- RAPID TRAVERSE ON ALL AXES G00=1500 MM/MIN (60 INS/MIN)
- FEED RATES INFINATELY VARIABLE ON ALL AXES
- 0-1500 MM/MIN (0-60 IN/MIN)

MECHANICAL RESOLUTION - 0.01 MM (0.0004")

LINEAR INTERPOLATION - ON X, Y AND Z AXIS WITH VECTORICALLY CORRECTED FEED RATES

CIRCULAR INTERPOLATION - ON X AND Y PLANES

FIXED ZERO REFERENCE POSITION

LOOKING AT THE TABLE

X=ZERO WHEN THE TABLE IS AT THE RIGHT HAND SIDE OF THE SPINDLE

Y=ZERO WHEN THE SPINDLE AT THE FRONT EDGE OF THE TABLE

Z=ZERO WHEN THE SPINDLE NOSE IS AT ITS MAXIMUM DISTANCE FROM THE
HEADSTOCK

TABLE

TABLE LENGTH - 760 MM

TABLE WIDTH - 180 MM

SPINDLE TO TABLE - 280 MM

TABLE SLOTS - 5/8" TEE SLOTS

MACHINE DIMENSIONS

LENGTH - 1000 MM (39")

WIDTH - 1320 MM (52")

HEIGHT - 1910 MM (75")

WEIGHT - 550 KILO (1210 LBS)

TOOL LENGTH OFFSETS - 16 TOOLS

TOOL DIAMETER COMPENSATION - 16 TOOLS

STANDARD EQUIPMENT

1 TOOL BOX
1 OIL CAN
2 SCREW DRIVERS
4 ALLEN KEYS 3, 4, 5 AND 6
4 SPANNERS, 12/13, 14/15, 16/17 AND 19 RING
1 MINI CASSETTE
1 POT OF PAINT
1 FUSE 20 MM 250 V 5A
1 FUSE 20 MM 3.15 ANTI SURGE
1 FUSE 20 MM 1.6A

Flood Coolant System
Splash Guard
Table Guards and Swarf Tray
Tool Kit
Halogen Lo Vo Light
Operation and Instruction Manual
One Shot Lubrication System

EXTRA EQUIPMENT

Spray Mist Coolant
Programmable Vari Speed
Quick Change Tooling
Printer
CAD/CAM and Off Line Computer Programming
Desk Top Programmer
Robot

SAFETY FEATURES

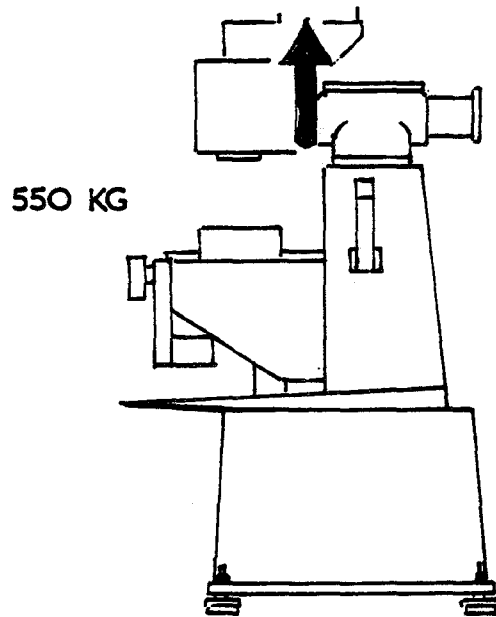
Axis Limit Switches
Diagnostic Fault Finder
Key Operated Program Control

SECTION 3

INSTALLATION

LIFTING

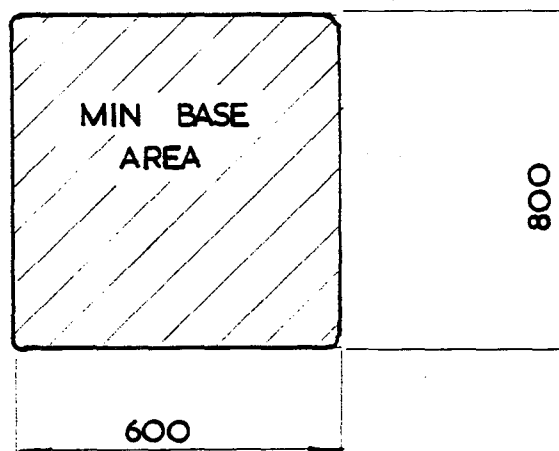
The machine can be lifted from the horizontal head ram at the point between the milling head and top casting. Check to ensure that the lifting slings are of the correct capacity to lift the weight of the machine and that the slings are in good condition.



FOUNDATION

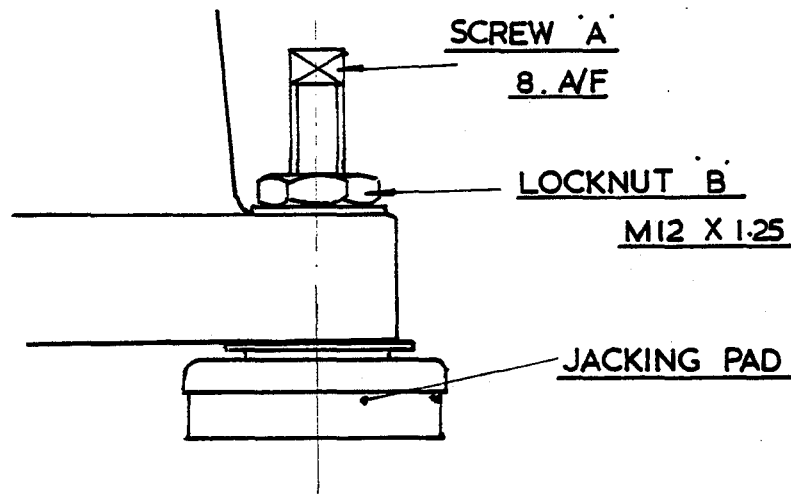
The machine should be set on a firm concrete base and fixed using the four bolting down holes (12 diameter) located at each corner of the base casting.

SEE DRAWING NUMBER SK.727/1



LEVELLING

Four jacking pads must be fitted at each corner of the machine base.



Thoroughly clean the table surface and, using a precision machine level, alternately, in the longitudinal and the transverse positions level the machine using the jacking pads thus:-

With locknut 'B' slack, rotate screw 'A' until the table is level.

Tighten locknut 'B'.




LUBRICATION

The milling machine is fitted with a 'one shot' lubrication system. The reservoir on the system should be filled to the correct level and a check to see if oil is being pumped to the slides and ballscrews before the machine is operated.

Operate the manual pump DAILY.

Use SAE 10 for the lubricating system and a good quality machine grease for the knee bracket gears (readily accessible from the underside of knee).

The spindle and spindle bearings are lubricated via the oil reservoir located at the front of the machine head, to the operator's left of the speed chart.

<u>LUBRICATION</u>		
	<u>SHELL</u>	<u>CASTROL</u>
	VITREA 68	PERFECTO NN
	VITREA 68	PERFECTO NN
	ALVANIA Nº 3	SPHEEROL AP3

CLEANING

The machine on delivery will have the bright surfaces coated with a protective solution. This must be removed by using a kerosene base solvent before any attempt is made to move the slides or operate the machine.

SECTION 4

MAINTENANCE

Routine inspection and maintenance should be carried out to the following schedule:-

<u>PERIOD</u>	<u>MAINTENANCE REQUIRED</u>
DAILY	Check oil reservoir. Operate 'one shot' system. Clean away swarf.
WEEKLY	Clean machine thoroughly. Check coolant tank.
SIX MONTHLY	Check table, cross slide and knee strips. Check spindle drive vee belt.
ANNUALLY	Check machine alignments and accuracy. Check axis drive belts.

ADJUSTMENT OF GIB

1. The table is provided with a full length, hand scraped, tapered gib on the front side of the saddle. There is an adjusting screw at each end. To adjust the gib, tighten the two screws until a slight drag is felt when moving the machine by hand. If the table is not tight enough, loosen the adjusting screw on the small end of the gib and tighten the other end. If the feel is too tight, reverse the procedure.

ADJUSTMENT OF SADDLE AND KNEE GIBS

1. To properly adjust the feel of the cross movement and the vertical movement, follow the same procedure as described above.

CHANGING SPINDLE SPEEDS

1. Loosen the two locking bolts on the sides of the motor. Pull back the tension level.
2. Move the belt to the desired groove.
3. Set belt tension and lock the two locking bolts.

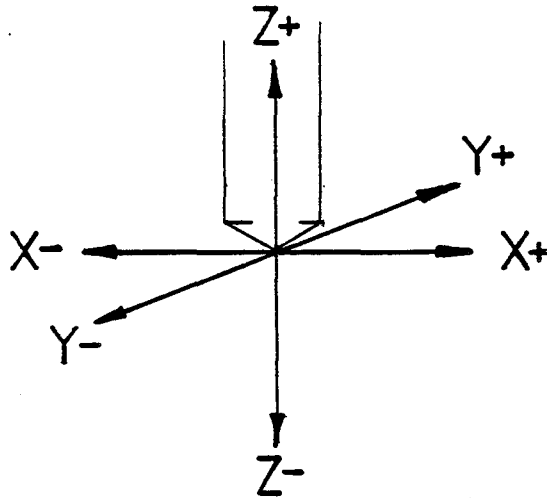
REMOVING MOTOR

1. Proceed to loosen belt according to instructions on previous page.
2. Take off the motor and one of the pulleys.
3. Loosen the eight hexagon socket screws in the upper cover of the belt housing. Lock uniformly the three small hexagon socket screws and take off belt housing cover.
4. Loosen the four hexagon socket screws and take off aluminium cover.

SECTION 5

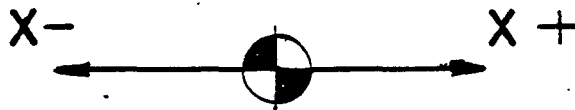
MACHINE AXIS FORMAT

Easimill is a three axis knee type mill. The illustration shows the positive and negative, indicating the direction of the tool movement. This is an important concept to understand. Because all programming is done with this idea in mind, that is all motion programmed is the movement of the tool, not the movement of the table.



ABSOLUTE TOOL MOVEMENT FROM G55 DATUM POSITION

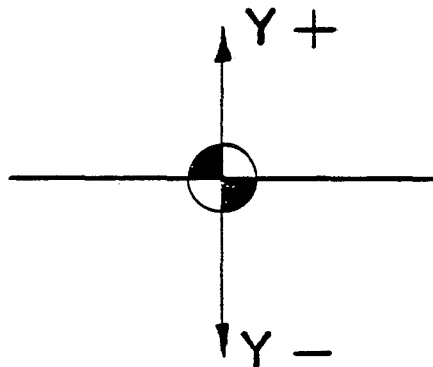
X AXIS DIRECTION FROM DATUM



X- LEFT FORWARD MOVEMENT OF TOOL

X+ RIGHT FORWARD MOVEMENT OF TOOL

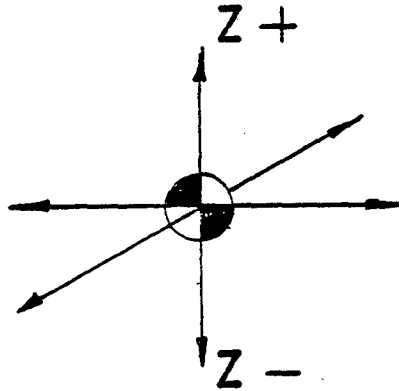
Y AXIS DIRECTION FROM DATUM



Y+ TOOL MOVES AWAY FROM THE OPERATOR

Y- TOOL MOVES TOWARDS THE OPERATOR

Z AXIS DIRECTION



Z- DOWNWARD MOVEMENT OF THE QUILL

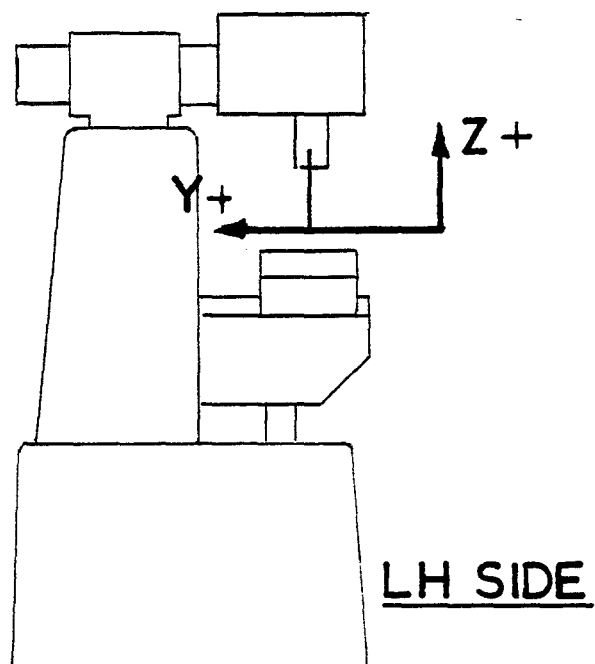
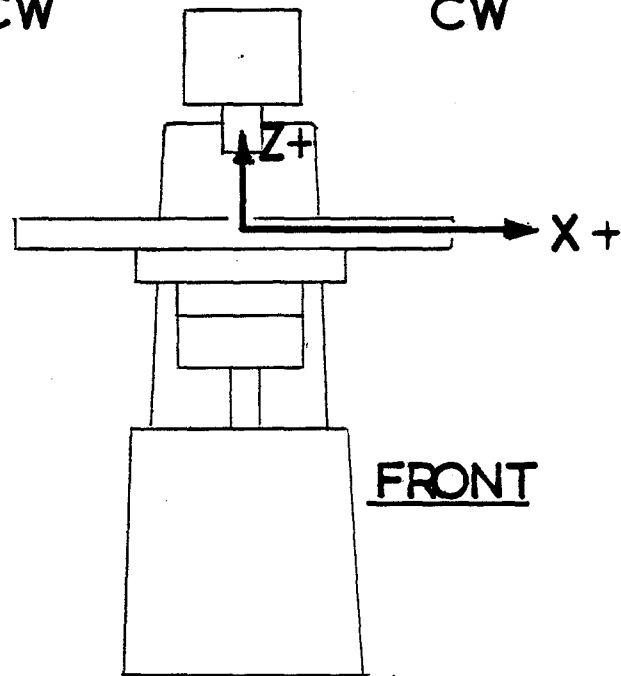
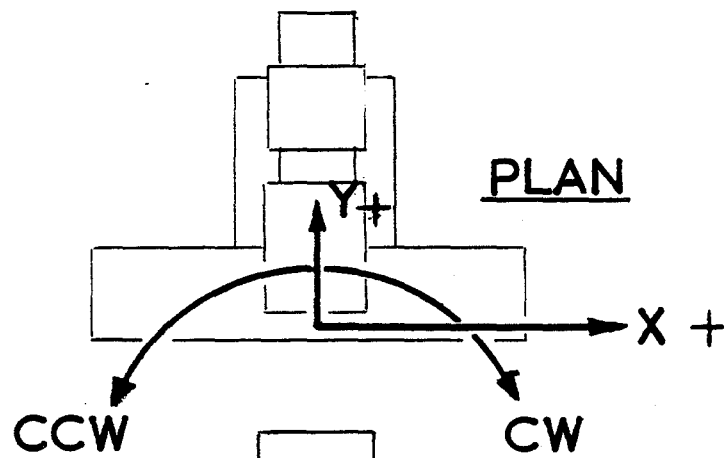
Z+ UPWARD MOVEMENT OF THE QUILL

MACHINE AXIS FIXED DATUM

Initially starting Easimill from a cold start, operation cannot commence until the machine has been datumed. Each axis is driven to a limit switch, Z, Y and X axis respectively. Each limit switch position is maximum positive motion for that axis. From this position the control can establish the parameter of movement for each axis.

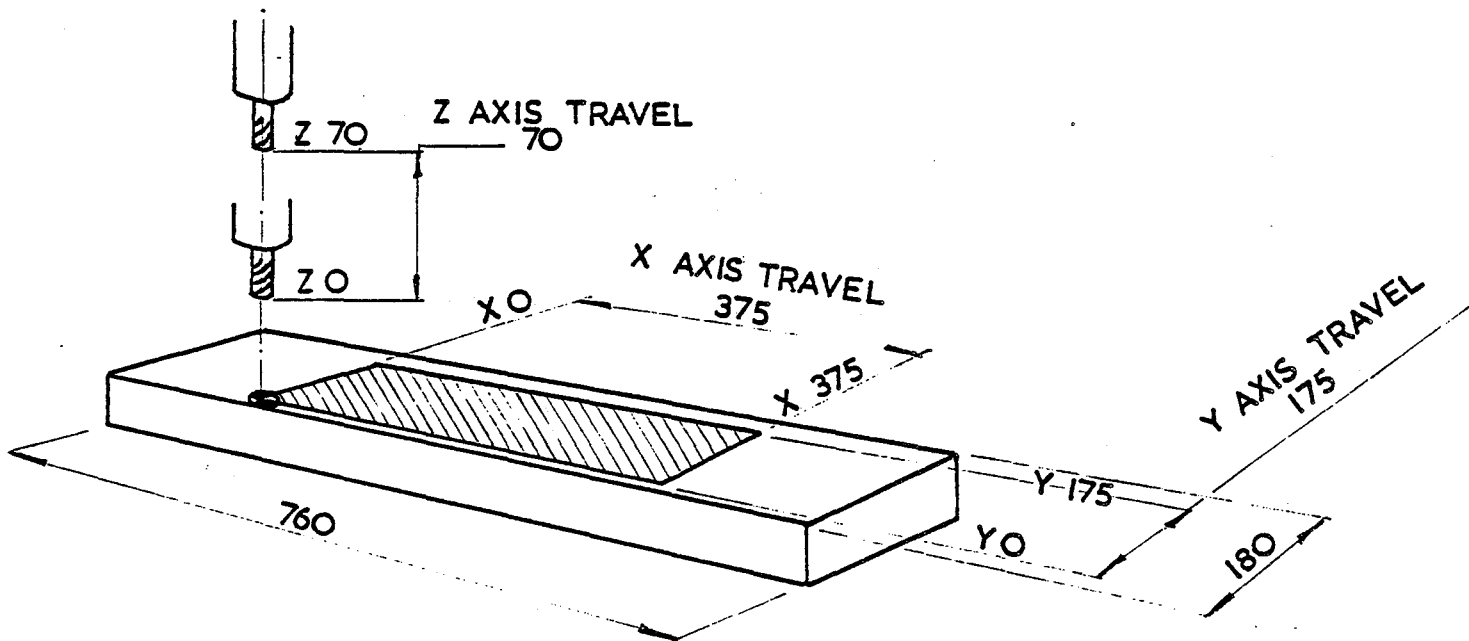
All movements programmed are positive. Any negative sign will respond. "MOVE EXCEEDS MACHINE LIMIT." The machine origin being the bottom left hand corner of the table for X and Y axis. Z axis origin being when the quill is at maximum distance from the headstock.

See sketch.



AXIS TRAVEL RANGES

MACHINE FIXED ZERO



X AXIS AND Y AXIS = 0 When the quill is at a point near the front left hand corner of the table.

Z AXIS = 0 When the quill is at maximum distance from the headstock.

X+ = Left forward movement of the table.

Y+ = Forward movement of the table towards the operator.

Z+ = Upward movement of the quill.

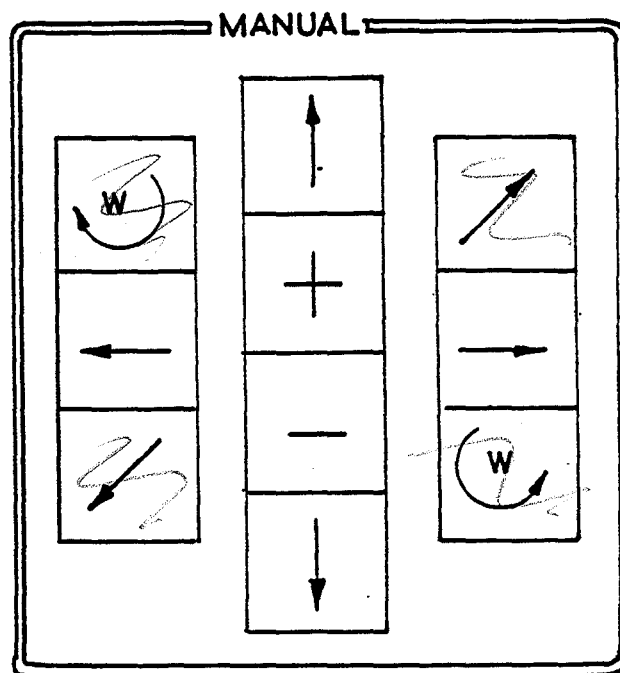
SECTION 6

MANUAL JOG MODES

For operating Easimill in the manual mode, that is without the use of programming, first set the key to "MAN" position. The machine can be jogged into position by depressing the directional arrow keys in the jog button cluster. One press represents 0.005 mm or 0.0002" jog. If a jog key is held depressed, continuous movement is maintained. Depress the appropriate arrowed key first which will select the slowest jog speed and then at the same time depress the + key which will accelerate the jog speed. Continually holding down the + key will accelerate the jog speed up to its maximum value 239 mm/min. the - key allows the jog speed to be decreased.

Movement will only occur as long as the appropriate arrowed key is depressed.

Caution should be taken when approaching an obstruction, such as a vice or the workpiece. Because the machine needs a short distance to decelerate.

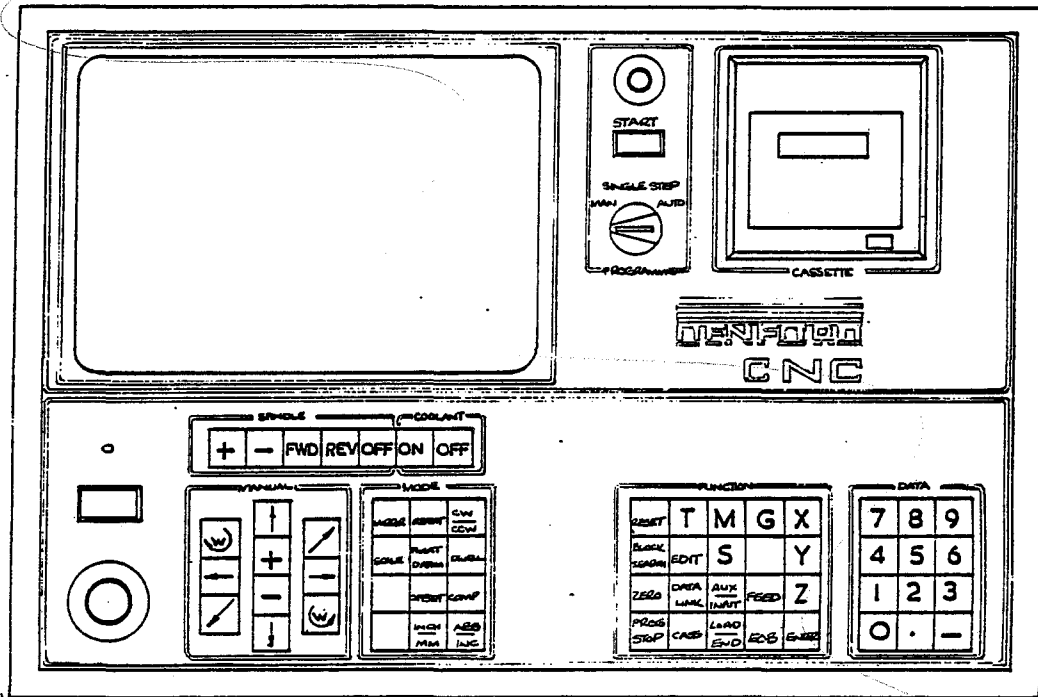


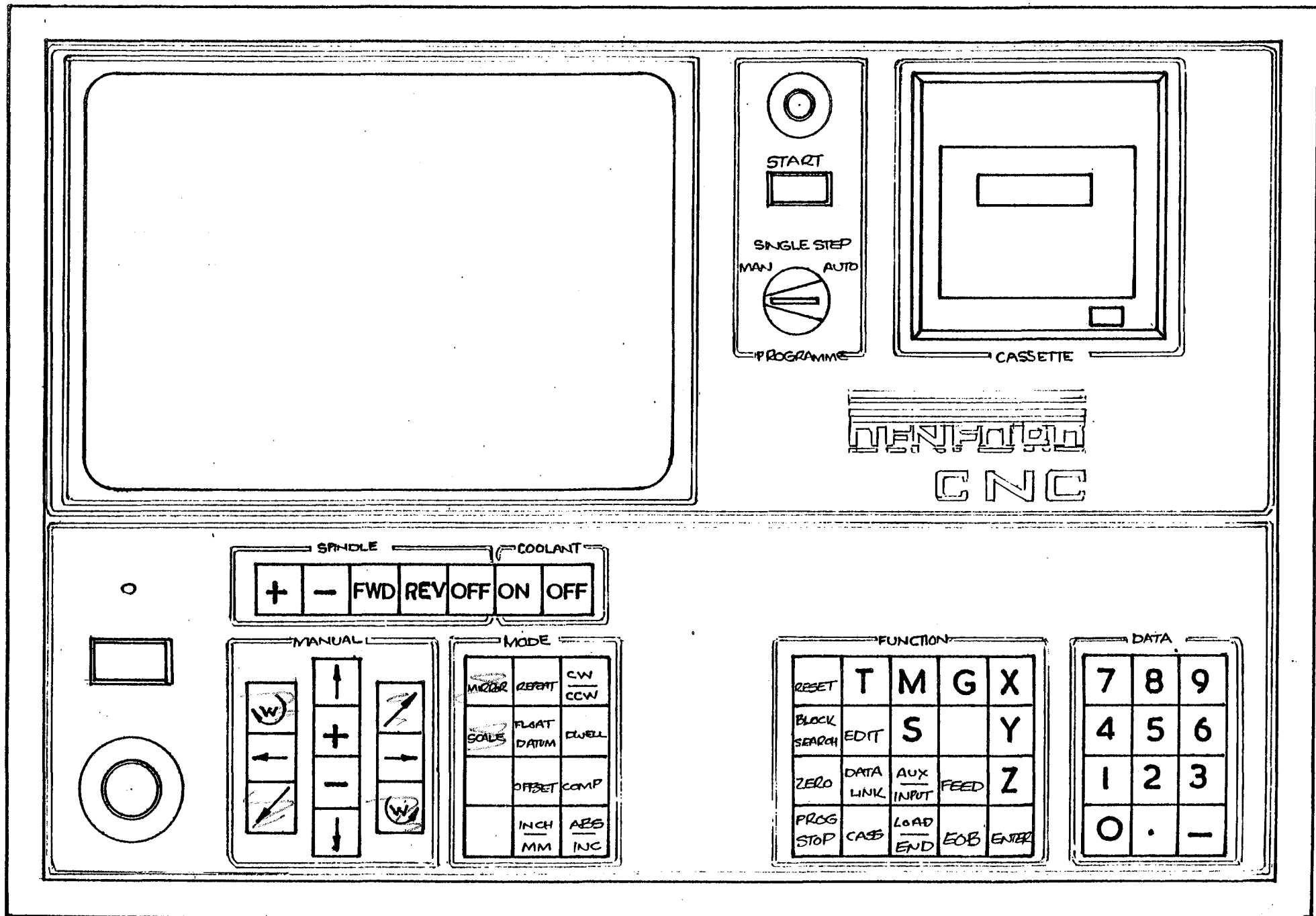
The directional arrow keys represents the movement of the spindle. If you want the table to move to the left parallel to the X axis (X plus), then depress the button with the right horizontal arrow. Similarly to move the Y axis then the appropriate diagonal arrow key is selected.

The up/down arrows are direction of the quill.

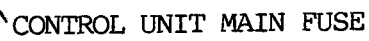
Any attempt to jog to a position outside the machine parameters the control will automatically reduce the jog speed into creep feed, before the overtravel limit is encountered. Then the screen will respond with "MOVE EXCEED MACHINE LIMIT". Depressing the reset key will clear the screen which responds with "PRESS ZERO TO DATUM THE MACHINE". Depress the zero key and datum procedure will commence. Normal operation is restored.

THE ALPHA-NUMERIC KEYBOARD



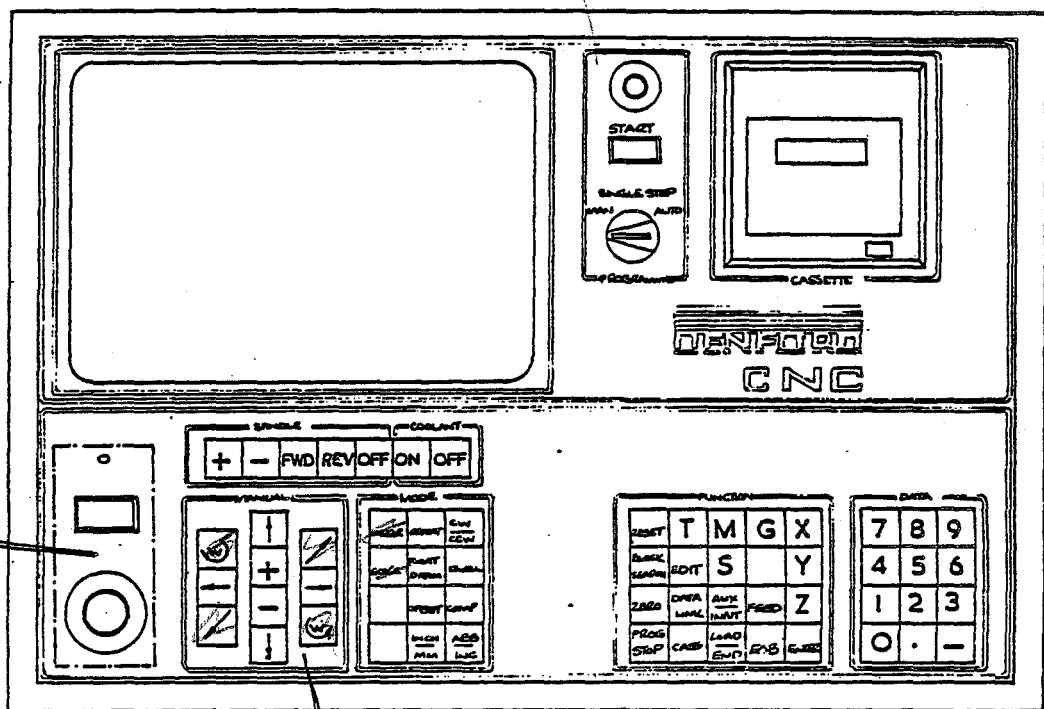


REAR VIEW CNC CONTROL UNIT



Program start/stop

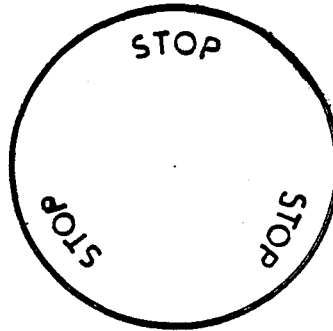
AXIS POWER &
EMERGENCY STOP



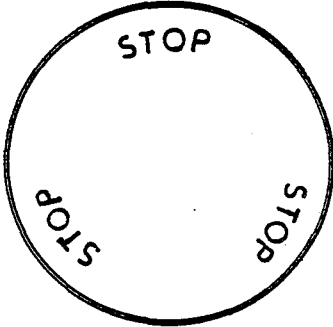


MANUAL JOG
BUTTONS

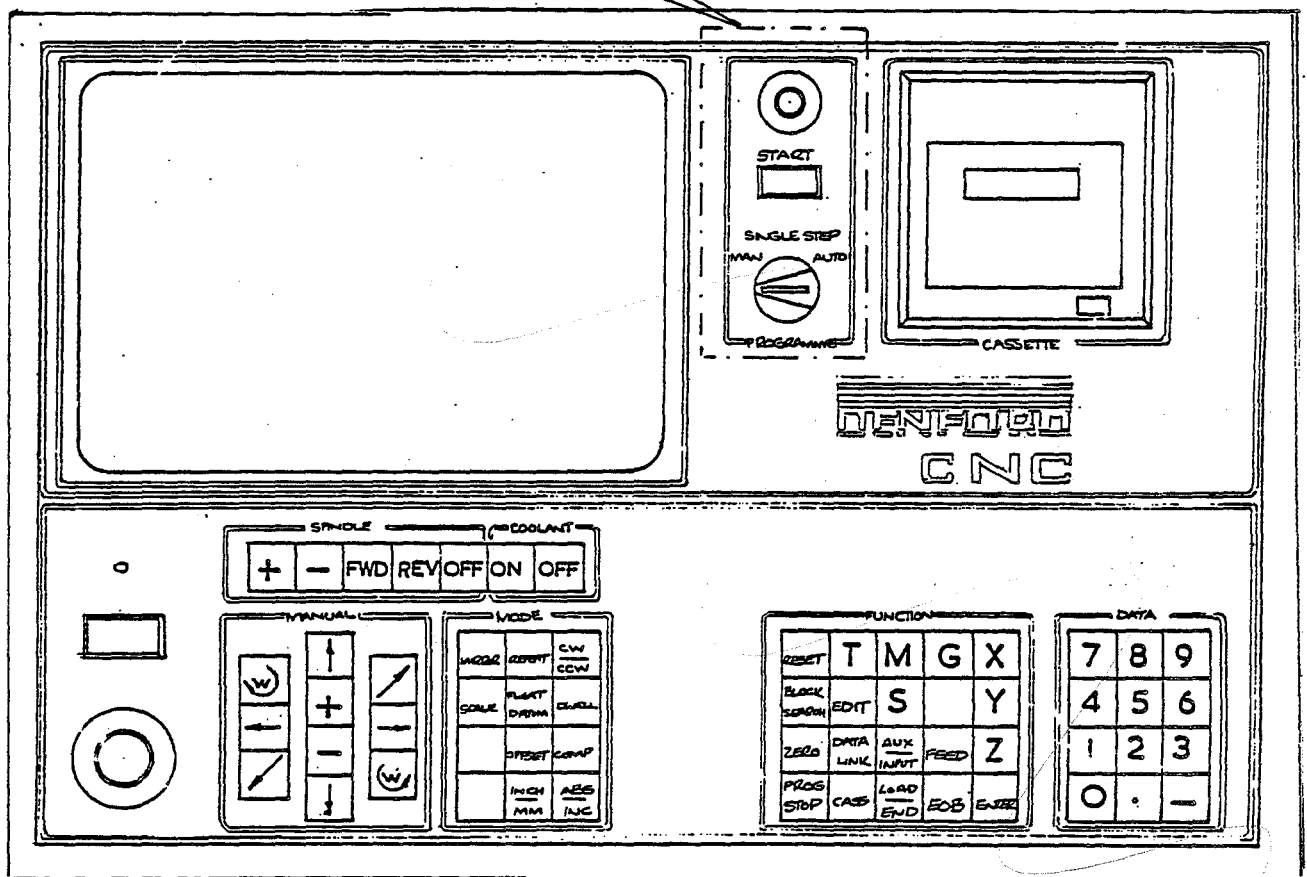
SECTION 7

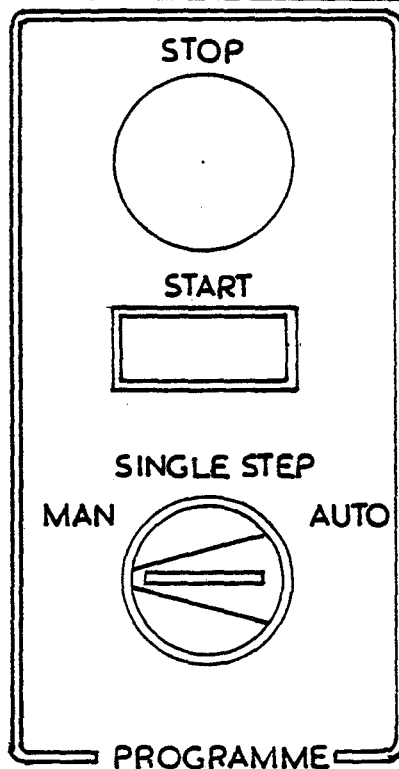
AXIS POWER AND EMERGENCY STOP

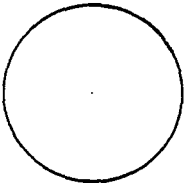

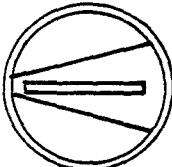


KEY	FUNCTION
	CONTROL POWER L E D.
	AXIS POWER BUTTON. THIS BUTTON ENERGIZES DRIVE MOTORS. SPINDLE DRIVE MOTOR.
	<u>EMERGENCY STOP</u> EACH TIME EMERGENCY STOP IS USED IT WILL KILL POWER TO ALL DRIVE MOTORS. THE MEMORY IS NOT AFFECTED. POWER IS TURNED BACK ON BY UNLOCKING EMERGENCY STOP AND DEPRESSING THE AXIS POWER BUTTON.

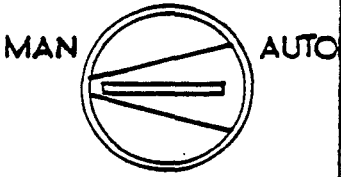
CYCLE MODE & OPERATION CONTROL





KEY	FUNCTION
<p>STOP</p> 	<p>DEPRESSING THE STOP BUTTON WILL HALT THE AXIS MOVEMENT IN THE CURRENT BLOCK. WHILST IN AUTO OR SINGLE STOP EXECUTION CAN BE RESTARTED USING THE START BUTTON. DEPRESSING THE STOP BUTTON WILL KILL ANY MDI MOVE AND CLEAR THE BLOCK.</p>
<p>START</p> 	<p>PROGRAM START BUTTON WILL EXECUTE THE CURRENT PROGRAM IN MEMORY OR EXCEUTE THE NEXT BLOCK IN SINGLE STEP MODE.</p>
<p>SINGLE STEP</p> <p>MAN  AUTO</p>	<p><u>THREE POSITION KEY SWITCH</u></p> <p>MAN POSITION - SELECT MANUAL ALLOWS THE USER TO ENTER NEW DATA, FOR MDI MOVES, ENTERING PROGRAMS, EDITING, CASSETTE ENTRIES AND DATA LINK. IN THIS MODE THE USER CAN MOVE ANY AXIS OR ALL THREE SIMULTANEOUSLY, DEPENDING ON REQUIREMENTS. STATING CO-ORDINATES FROM MACHINE FIXED DATUM, FLOATING DATUM AND MACHINE OFFSETS DO NOT AFFECT MDI MOVES. TOOL LENGTH OFFSETS ARE EFFECTIVE FOR Z.</p> <p>Continued.....</p>

SINGLE STEP

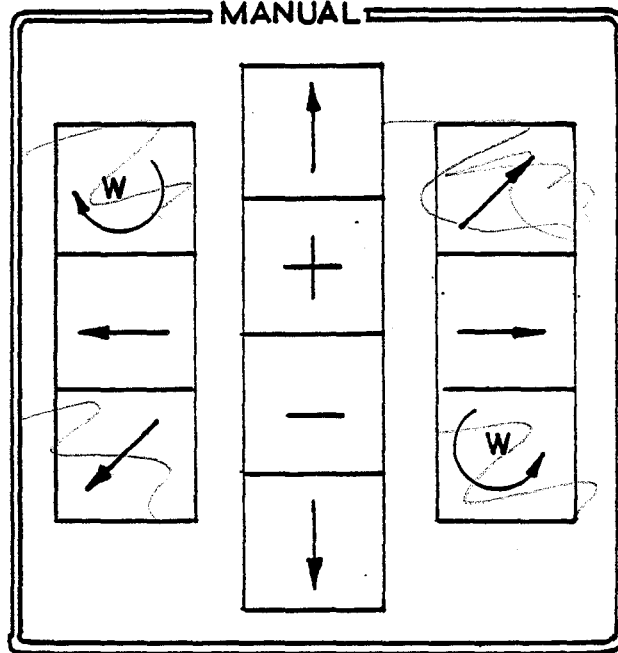








SINGLE STEP


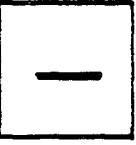
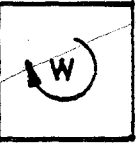

ALLOWS THE PROGRAM IN MEMORY TO BE EXECUTED ONE BLOCK AT A TIME. IN SINGLE STEP MODE, WITH THE DEPRESSION OF THE START BUTTON, THE MACHINE WILL EXECUTE ONE BLOCK OF INFORMATION ONLY AND STOP. IN ORDER TO EXECUTE THE NEXT BLOCK THE START BUTTON WILL ONCE AGAIN HAVE TO BE DEPRESSED.

AUTO

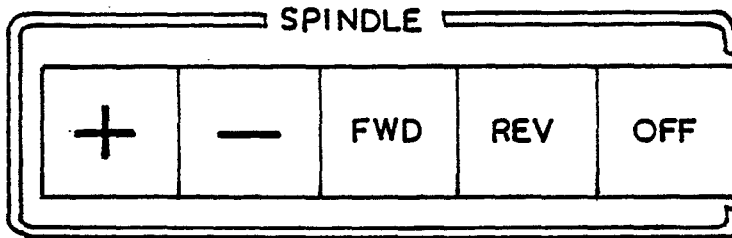
WHEN THE START BUTTON IS DEPRESSED, THE MACHINE WILL EXECUTE EACH BLOCK OF THE PROGRAM UNTIL EITHER THE END OF THE PROGRAM, PROGRAM STOP, OR A TOOLCHANGE IS REACHED. OPERATION CAN BE CONTINUED BY DEPRESSING THE START BUTTON.








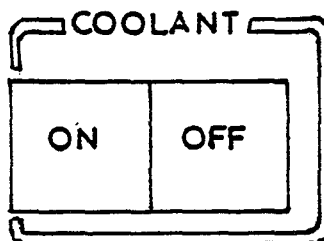
KEY	FUNCTION
	<p><u>AXIS JOG KEY</u></p> <p>PRESS THIS KEY TO MOVE THE SPINDLE UP (Z+ DIRECTION). MOVEMENT WILL ONLY OCCUR AS LONG AS THE KEY IS DEPRESSED.</p>
	<p>PRESS THIS KEY TO MOVE THE SPINDLE DOWN (Z- DIRECTION). MOVEMENT WILL ONLY OCCUR AS LONG AS THE KEY IS DEPRESSED.</p>
	<p>PRESS THIS KEY TO MOVE THE TABLE TO THE LEFT (SPINDLE TO THE RIGHT) (X+ DIRECTION).</p>
	<p>PRESS THIS KEY TO MOVE THE TABLE TO THE RIGHT (SPINDLE TO THE LEFT) (X- DIRECTION).</p>
	<p>PRESS THIS KEY TO MOVE THE TABLE AWAY FROM THE COLUMN (SPINDLE TOWARDS THE COLUMN) (Y+ DIRECTION).</p>
	<p>PRESS THIS KEY TO MOVE THE TABLE TOWARDS THE COLUMN (SPINDLE TOWARDS THE OPERATOR) (Y- DIRECTION).</p>



KEY	FUNCTION
	<p>THE + KEY WILL INCREASE MANUAL JOG FEED. HOLD DOWN THE APPROPRIATE ARROW KEY. THIS WILL SELECT THE SLOWEST JOG FEED. AT THE SAME TIME PRESS THE + KEY TO ACCELERATE THE JOG FEED. CONTINUALLY HOLDING DOWN THE + KEY WILL RAMP THE FEED UP TO ITS MAXIMUM.</p> <p>SUCCESSIVE PRESSES OF THE + KEY WILL GRADUALLY INCREASE AND STORE THE NEW FEED.</p> <p><u>ALTERNATIVELY</u> THE + KEY CAN ACT AS FEED RATE OVERRIDE WHEN RUN IN AUTO. PROGRAMMED FEED RATES WILL BE INCREASED AND REMAIN EFFECTIVE UNTIL A NEW FEED IS READ IN A SUCCESSIVE BLOCK.</p>
	<p>THE - KEY WILL DECREASE MANUAL JOG FEED. OPERATION PROCEDURE AS FOR THE + KEY.</p> <p><u>ALTERNATIVELY</u> THE - KEY CAN ACT AS PROGRAM FEED RATE OVERRIDE REDUCING PROGRAM FEED RATE. EFFECTIVE UNTIL NEW FEED IS READ.</p>
	OPTIONAL FOR FOURTH AXIS.
	OPTIONAL FOURTH AXIS.

MANUAL SPINDLE CONTROL KEYS



KEY	FUNCTION
	WILL MANUALLY INCREASE SPINDLE SPEED. <u>ONLY</u> EFFECTIVE ON MACHINE WITH PROGRAMMABLE VARIABLE SPEED.
	WILL MANUALLY DECREASE SPINDLE SPEED. <u>ONLY</u> EFFECTIVE ON MACHINE WITH PROGRAMMABLE VARIABLE SPEED.
	SELECTS FORWARD ROTATION OF SPINDLE. WHEN IN PROGRAM LOAD MODE, PRESSING FWD KEY WILL SELECT M03 COMMAND. VDU WILL PROMPT THE USER TO INPUT AN RPM VALUE.
	SELECTS REVERSE ROTATION OF SPINDLE. WHEN IN PROGRAM LOAD MODE, PRESSING REV KEY WILL SELECT AN M04 COMMAND. VDU WILL PROMPT THE USER TO INPUT AN RPM VALUE.
	SELECTS SPINDLE OFF. WHEN IN PROGRAM LOAD MODE, PRESSING OFF KEY WILL SELECT AN M05 COMMAND. SPINDLE OFF.



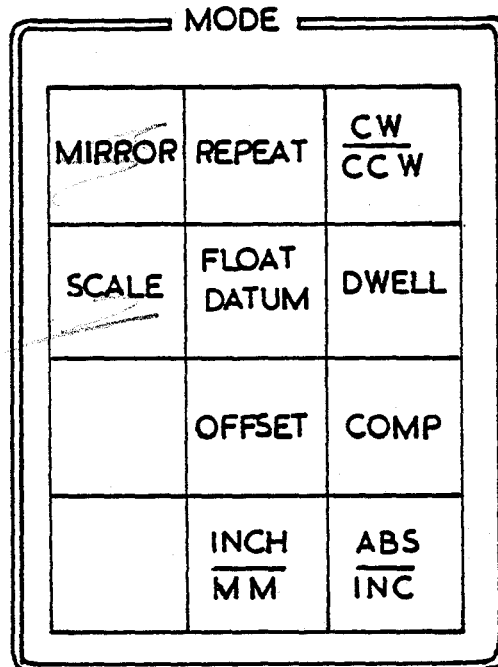
	SELECTS COOLANT ON. WHEN IN PROGRAM LOAD MODE, PRESSING THE COOLANT ON KEY WILL SELECT M08 COOLANT ON COMMAND.
	SELECTS COOLANT OFF. WHEN IN PROGRAM LOAD MODE, PRESSING THE COOLANT OFF KEY WILL SELECT M09 COOLANT OFF COMMAND.

FUNCTION				
RESET	T	M	G	X
BLOCK SEARCH	EDIT	S		Y
ZERO	DATA LINK	AUX INPUT	FEED	Z
PROG STOP	CASS	LOAD END	EOB	ENTER

KEY	FUNCTION
RESET	TO RESET FROM THE CURRENT MODE OR CANCEL THE MOST RECENT ENTRY.
T	IF THE CONTROL IS SET TO MDI AND THE T KEY IS PRESSED, A TOOL SELECTION MENU WILL APPEAR ON THE SCREEN FOR TOOL SETTING AND EDITING TOOL LENGTH OFFSETS, (SEE SECTION 20). IF THE CONTROL IS IN PROGRAM LOAD MODE THE T KEY CAN BE PROGRAMMED AS TOOLCHANGE M06 BLOCK. UPON SWITCHING THE CONTROL ON FROM COLD, PRESSING THE T KEY GIVES A SYSTEM TEST DISPLAY CHECKING INPUT SIGNALS AND MACHINE MOUNTED SWITCHES.
M	MISCELLANEOUS FUNCTION, REFERRED TO AS M FUNCTION. BY PRESSING M KEY THEN THE ENTER KEY, A LIST OF M CODES WILL BE DISPLAYED ON THE SCREEN.
G	PREPARATORY FUNCTIONS, REFERRED TO AS G FUNCTION, PRESS G KEY THEN THE ENTER KEY. A LIST OF G CODES WILL BE DISPLAYED ON THE SCREEN.
BLOCK SEARCH	THIS KEY WILL ALLOW THE PROGRAM TO BE EXECUTED FROM A SPECIFIC BLOCK IN THE PROGRAM.
EDIT	EDIT KEY PERMITS FULL EDITING FACILITIES OF THE PROGRAM IN MEMORY. (SEE SECTION 35.)

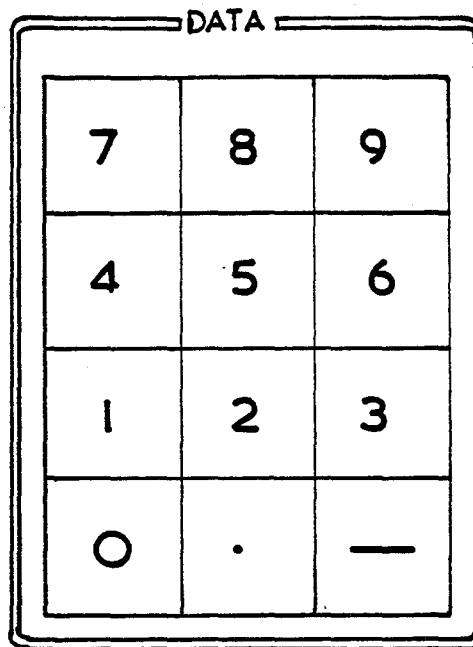
KEY	FUNCTION
S	S KEY GIVES A DIAGNOSTIC CHECK, UPON STARTING FROM COLD BEFORE AXIS POWER IS APPLIED, PRESSING THE S KEY GIVES A DIAGNOSTIC CHECK.
X	X AXIS KEY FOR X AXIS MOVES.
Y ^Z	Y AXIS KEY FOR Y AXIS MOVES.
Z	Z AXIS KEY FOR Z AXIS MOVES.
ZERO	ZERO KEY IS USED TO DATUM THE MACHINE. THIS DRIVES THE MACHINE TO ITS MAXIMUM TRAVEL FOR EACH AXIS, UPON STARTING FROM COLD OPERATION YOU CANNOT CONTINUE UNTIL MACHINE IS DATUMED.
DATA LINK	SELECTS A DATA LINK TO EXTERNAL EQUIPMENT, EG, AN EXTERNAL COMPUTER, TRANSMIT AND RECEIVE PROGRAMS OR PRINTER LINK.
AUX INPUT	<p>AUXILIARY OUTPUTS. THIS ALLOWS ANY OF THE FOUR AUXILIARY OUTPUT RELAYS TO BE OPERATED.</p> <p>AUXILIARY INPUTS. THIS INSTRUCTION ALLOWS THE PROGRAM TO BE HALTED BETWEEN MACHINING OPERATIONS. THE PROGRAM WILL ONLY PROCEED BEYOND THIS POINT IF ANY OF THE SEVEN AUXILIARY INPUTS ARE PROGRAMMED TO RECEIVE AN INPUT SIGNAL.</p>
FEED	<p>FEED KEY. USE TO SELECT A FEED RATE TO EXECUTE A MOVEMENT</p> <p>FEED RATES FROM 0 TO 1500 MM/MIN.</p> <p>FEED RATES FROM 0 TO 60 IN/MIN.</p>

KEY	FUNCTION
<div>PROG STOP</div>	<p>PROGRAM STOP BUTTON OR FEED HOLD.</p> <p>WHEN LOADING A NEW PROGRAM INTO MEMORY, THE PROGRAM STOP BUTTON IS USED AS A STOP IN THE PROGRAM M00. ADDITIONALLY, WHEN EXECUTING A PROGRAM IN AUTO OR SINGLE STEP, DEPRESSING THE PROGRAM STOP BUTTON WILL HALT THE CYCLE OF THE CURRENT WORK, OPERATION CAN BE CONTINUED BY PRESSING THE START KEY.</p>
<div>CASS</div>	<p><u>CASSETTE KEY</u></p> <p>PRESS CASS KEY AND A MENU OF CASSETTE FUNCTIONS WILL BE LISTED ON THE SCREEN. (SEE SECTION 36.)</p>
<div>LOAD END</div>	<p><u>PROGRAM LOAD KEY</u></p> <p>TO ENTER A PROGRAM INTO MEMORY OR END A PROGRAM LOADING SEQUENCE.</p>
<div>EOB</div>	<p><u>END OF BLOCK KEY</u></p> <p>END OF CURRENT LINE OF INFORMATION.</p>
<div>ENTER</div>	<p><u>ENTER KEY</u></p> <p>INFORMATION IS ACCEPTED INTO THE CONTROL AFTER THE ENTER HAS BEEN PRESSED.</p>



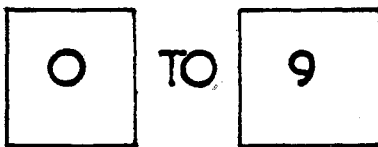
KEY	FUNCTION
MIRROR	MIRROR KEY SELECTS A MENU OF PERMITTED X AND Y MIRRORS. THESE FUNCTIONS CAN ONLY BE USED AS PART OF A PROGRAM LOADING SEQUENCE. MIRRORS CAUSE THE CONTROL TO ESTABLISH AN AXIS OF SYMMETRY AT ITS CURRENT POSITION. ALL SUCCEEDING BLOCKS ARE REFLECTED ABOUT THIS AXIS.
REPEAT	REPEAT FACILITY ENABLES SPECIFIED BLOCKS OF A PROGRAM TO BE REPEATED UP TO 99 TIMES WITH SPECIFIED OFFSETS. (SEE LATER TEXT.)
$\frac{CW}{CCW}$	CIRCULAR INTERPOLATION IS SELECTED USING THIS KEY. PRESS ONCE FOR G02 CLOCKWISE, PRESS AGAIN FOR G03. AFTER DEFINING THE END POINTS OF THE CIRCULAR MOVE, BY DEPRESSING CW/CCW KEY ALLOWS THE INPUT OF THE CIRCLE CENTRE CO-ORDINATES, XC AND YC (ARC CENTRE OFFSETS).
SCALE	PROGRAM SCALE OR MACHINE SCALE. SCALING RANGES 0.01% TO 650%. 100%=ACTUAL SIZE. PROGRAM SCALE G20 ALLOWS A SCALE INSIDE THE PROGRAM, MACHINE SCALE OR OVERALL SCALE IS SET OUTSIDE THE PROGRAM AND SCALES PROGRAM AND ANY MACHINE OFFSETS.
FLOAT DATUM	A FLOATING DATUM BLOCK G99 WILL CAUSE THE CONTROL TO ESTABLISH A DATUM POSITION, ALL AXES ARE SET TO ZERO ABOUT ITS CURRENT POSITION. FLOATING DATUM IS ONLY PERMITTED AS PART OF A PROGRAM LOAD SEQUENCE, THE DISPLAYED POSITION WILL BE RELATIVE TO THIS DATUM IF MACHINE OFFSETS (G55) ARE ZERO.

KEY	FUNCTION
DWELL	DWELL SELECTS A PROGRAMMABLE DWELL, G04 IN THE RANGE 0.1 TO 9999.9 SECONDS.
OFFSET	<p>SELECTS A MACHINE OFFSET G55. SET OUTSIDE THE PROGRAM AND ALLOWS THE DATUM TO BE ESTABLISHED ON THE COMPONENT. THE OFFSET VALUE EQUAL TO THE DISTANCE FROM THE MACHINE ORIGIN TO THE POSITION ON THE COMPONENT WHERE THE DATUM IS DESIRED.</p> <p>PROGRAM OFFSET G54 CAN BE USED TO OFFSET PARTS OF THE PROGRAM DURING PROGRAM LOAD SEQUENCE. THESE ARE INCREMENTAL IN OPERATION AND RESET TO ZERO EVERY TIME THE PROGRAM IS EXECUTED.</p>
COMP	<p>SELECTS TOOL RADIUS COMPENSATION.</p> <p>G41 CUTTER COMPENSATION LEFT IN DIRECTION OF TRAVEL.</p> <p>G42 CUTTER COMPENSATION RIGHT IN DIRECTION OF TRAVEL.</p> <p>CUTTER COMPENSATION SETS UP A NEW TOOL PATH AROUND THE PROGRAM TOOL PATH ON THE LEFT OR RIGHT SIDE OF THE LINE. (SEE SECTION 28.)</p>
INCH MM	SELECTS IMPERIAL OR METRIC UNITS.
ABS INC	<p>SELECTS ABSOLUTE CO/ORDINATES OR INCREMENTAL INPUT.</p> <p><u>NOTE</u>: ALLOWS INCREMENTAL INPUT, BUT DISPLAYS AS ABSOLUTE.</p>

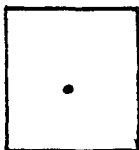


KEY

FUNCTION



NUMERICAL DATA KEYS.



DECIMAL POINT KEY.



MINUS SIGN KEY.

SECTION 13

MACHINE CODES (M FUNCTION AND G CODES)

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

M functions for use outside of Program

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

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 Input

G codes for use outside of Program

G00 Linear Rapid Traverse
G01 Linear
G02 Circular CLW
G03 Circular CCLW
G04 Dwell
G21 Machine Scale
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 (Machine Reference Point)

G codes for use inside the Program

G00 Linear Rapid Traverse
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)
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 (Machine Reference Point)
G99 Floating Datum

MISCELLANEOUS FUNCTIONS

One M function is permitted per block.

M and G codes cannot be entered on the same line.

M00 Programme Stop

When a program stop occurs then no further motion occurs until the cycle start key is depressed. Spindle speed and coolant remain unaffected by this function.

M02 or M2

This function will end the program. On reaching this point the spindle and coolant will stop and the tool will retract to its home position.

M03 or M3

This function starts the spindle rotation in clockwise direction. For controls fitted with programmable speed, then the desired rpm value can be entered. It is cancelled by M06, M05 or M02. Spindle direction cannot be changed whilst the spindle is rotating. For standard 10 speed machines this function will start the spindle in the high range.

M04 or M4

This function starts the spindle rotation in counter-clockwise direction. For controls fitted with programmable speed, then the desired rpm value can be entered. It is cancelled by M06, M05 or M02. For standard 10 speed machines this function will start the spindle in the low range.

M06 or M6 Tool Change

This function causes the spindle to stop and retract to its maximum position, before tool changing can be accomplished. Previous spindle speed will be stored in memory. Coolant control is unaffected. M06 and the tool number calls up the appropriate tool length offset from the tool library.

M08 or M8 Coolant On

This function selects flood or mist coolant which will then be on until cancelled by M09 or M02.

M09 or M9 Coolant Off

This function cancels coolant function.

M20

This function allows any of the four integral relays to be controlled either ON or OFF.

M21

This function allows the control to monitor seven user assigned input signals.

PREPARATORY FUNCTION G CODE

One G code is permitted per block.

M and G cannot be entered on the same line.

G00 or G0 Rapid Traverse

All motions rapid traverse in linear mode.

G01 or G1 Linear Interpolation

Is the mode of program to move the tool in a straight line that is parallel to an axis or at some angle to an axis. Depressing X, Y or Z key will default to G01 linear mode.

G02 or G2 Circular Clockwise

Is to be used when the tool is to follow the path of a circular arc while moving in a clockwise direction for X and Y axis.

G03 or G3 Circular Counterclockwise

Is to be used when the tool is to follow the path of a circular arc while moving in a counter-clockwise direction for X and Y axis.

G04 or G4 Dwell

No movement will occur while a timed dwell is performed.

G10 Mirror X

For changing over the positive and negative direction of the X axis, X mirror will cancel cutter compensation. M02 will cancel all mirrors.

G11 Cancel Mirror X

This function cancels X mirror function.

G12 Mirror Y

For changing over the positive and negative direction of the Y axis. Y mirror will cancel cutter compensation. M02 will cancel all mirrors.

G13 Cancel Mirror Y

This function cancels Y mirror function.

G20 Program Scale

This function allows a program scale 0.01% to 650% to be entered into the program. Program scales are cancelled with a new scale 100% or M02 end of programme for X and Y.

G21 Machine Scale

This function is set outside the program and scales the memory and any machine offsets, for X and Y axis, Z axis is unaffected. Scaling range 0.01 to 650%.

G40 Cancel Cutter Comp

This function is to cancel cutter compensation, program returns to centre line co-ordinates.

G41 Cutter Compensation Left

This function compensates the program for a new programmed path, on the left hand side of the line in the direction of travel. It is cancelled by G40, G42, G10, G11, G12, G13, G20, G54, G81, G98, G99, M06 and M02.

G42 Cutter Compensation Right

This function compensates the program for a new programmed path on the right hand side of the line in the direction of travel. It is cancelled by G40, G41, G10, G11, G12, G13, G20, G54, G81, G98, G99, M06 and M02.

G54 Programme Offset

This function allows an incremental offset within the program. It is cancelled by M02.

G70 Imperial Units

This function selects imperial units for the program.

G71 Metric Units

This function selects metric units for the program.

G81 Repeat Function

This function selects a repeat loop which will allow a programmed sequence to be repeated with specified offsets. G81 repeat loop will cancel cutter compensation.

G90 Absolute Input

This function selects absolute format.

G91 Incremental Input

This function selects incremental mode, and allows incremental input with absolute display.

G98 Absolute Datum (Machine Reference Point)

This function allows the control to establish its machine reference position for each axis in turn at a predetermined feed rate.

G99 Floating Datum

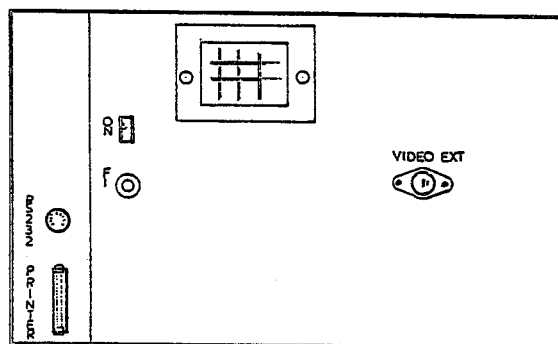
This function allows the control to establish a position where all axes are zero. Floating datum block will cancel cutter compensation.

STARTING THE MACHINE

Initially starting Easimill from a cold start.

1. Set the mains isolator at side of machine column for "ON" position.
2. Switch "ON" the power to CNC control (figure 1). This switch is located at the rear of control unit, a red LED on the front panel will indicate power is on.

FIGURE 1

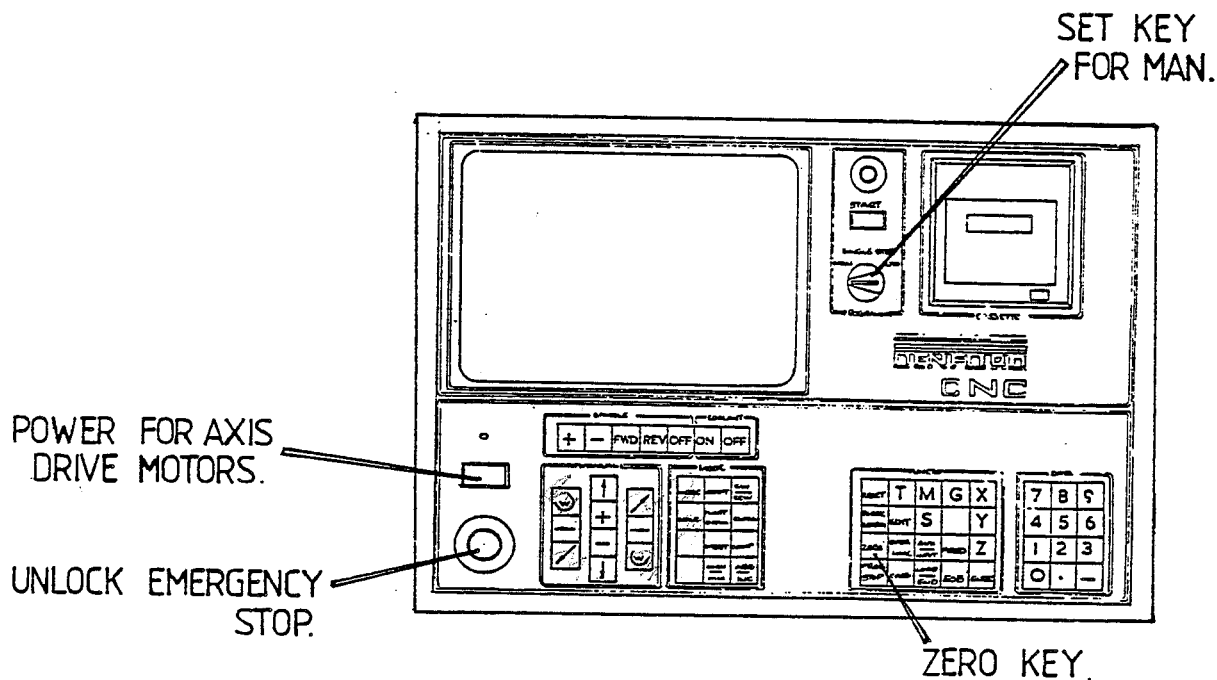


REAR VIEW

CNC CONTROL UNIT

The VDU will prompt the user to press <ZERO> to datum the machine.

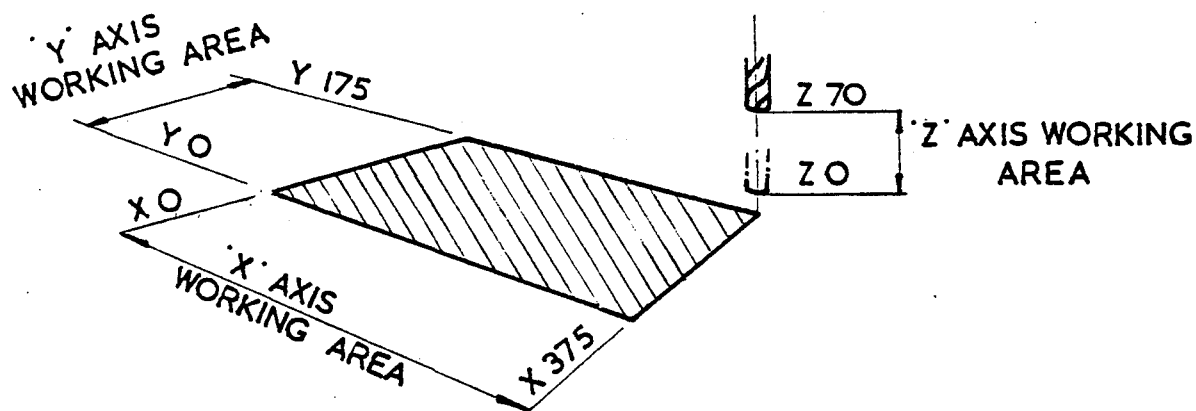
3. Unlock the emergency stop button on front panel (figure 2).
4. Set the key for MAN position. This allows manual data input (figure 2).



5. Depress the square green button for power to axis drive motors and spindle motor.

6. Depress ZERO key.

This drives Z, Y and X respectively to machine limit, the maximum movement on each axis. The control has built in machine limits from this position. The parameters of working area are defined X375, Y175 and Z70.



Working Area for Each Axis

375 mm for X,

175 mm for Y,

70 mm for Z.

For a manual move to drive X and Y to zero dimension depress keys

G01 ENTER X0 ENTER Y0 ENTER F1500 ENTER EOB.

G01 = Linear interpolation

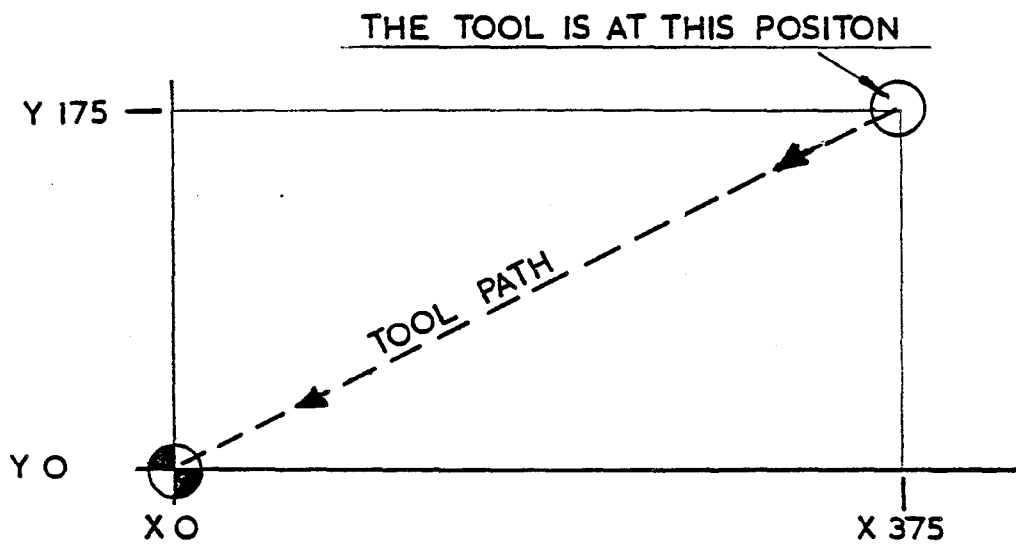
X0 = Machine fixed datum position for X

Y0 = Machine fixed datum position for Y

ENTER = Enter data into control

F1500 = Feed rate 1500 mm/min

EOB = End of block. When in MAN mode this key executes the command



SECTION 15

MOVEMENTS AND ASSOCIATED FEED INPUT

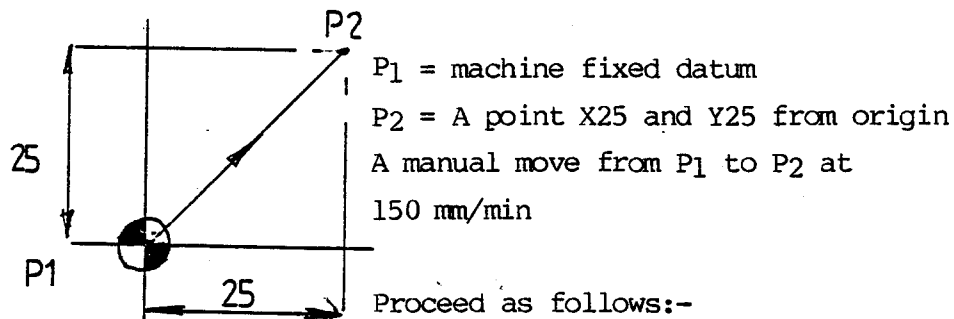
A movement in one or more axes can be input by pressing the desired axis key followed by the required dimensions. These co-ordinate dimensions and associated FEEDS may be input either as a single block of data which is to be executed immediately or as blocks of data which forms part of a programmed sequence.

Before co-ordinate dimensions keyed into the PNC 3 are executed a check is made to ensure that the machine parameters are not exceeded, should this be the case a warning message is displayed. RESET restores normal operation.

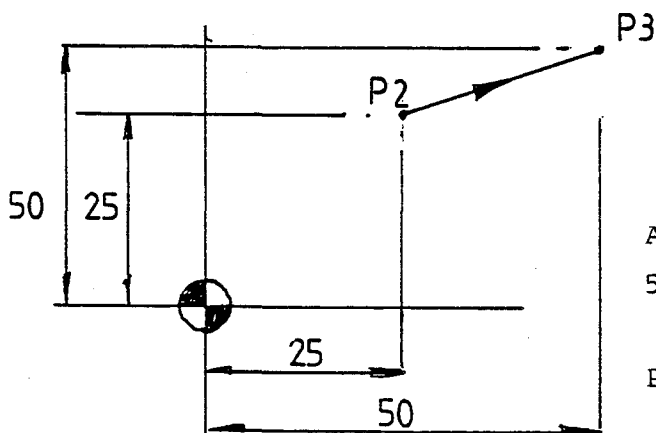
The required FEED is keyed in as millimetres per minute or inches per minute.

If no feed is programmed the default feed of 234 mm/min is assumed.

MANUAL MOVE IN ABSOLUTE MODE FROM MACHINE FIXED DATUM

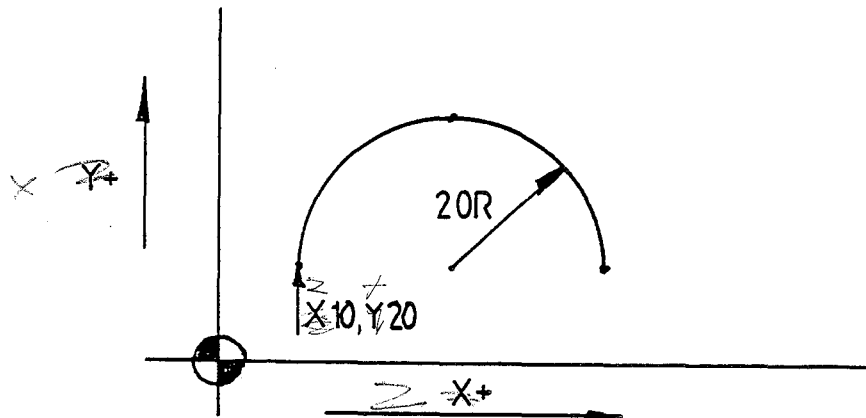


1. Set the MAN, S step, auto key TO MAN position.
This allows manual data input.
2. Press keys X25 ENTER Y25 ENTER F150 ENTER EOB



* NOTE: X and Y manual data entries are always co-ordinates from the machine fixed datum, therefore X and Y data entries are always in the positive quadrant and any negative dimension would be outside machine limits.

CIRCULAR INTERPOLATION



Absolute

1. G01 ENTER X10 ENTER Y20 ENTER F250 ENTER EOB
2. G02 ENTER X30 ENTER Y40 ENTER F150 ENTER CW/CCW ENTER 30 ENTER 20 ENTER EOB
3. G02 ENTER X50 ENTER Y20 ENTER F150 ENTER CW/CCW ENTER 30 ENTER 20 ENTER EOB

Incremental

1. G01 ENTER X10 ENTER Y20 ENTER F250 ENTER EOB
2. G02 ENTER X20 ENTER Y20 ENTER F150 ENTER CW/CCW ENTER 20 ENTER 0 ENTER EOB
3. G02 ENTER X20 ENTER Y-20 ENTER F150 ENTER CW/CCW ENTER 0 ENTER - 20 ENTER EOB

SECTION 16

CIRCULAR MOVEMENTS

Circular movements for X and Y axes are defined by using G02 for clockwise or G03 for counter-clockwise and are limited to quadrant boundaries OR, by using CW/CCW key, alternative depression of the key changes the code from clockwise to counter-clockwise. Define the end points of the circular movement. Press CW/CCW key and input the circle centre origin co-ordinates (XC and YC).

If you are programming in absolute mode the circle centre origin is the measured distance from the program datum to the circle centre. Alternatively, if you are programming in incremental the + OR - of the circle centre dimension is determined by the incremental distance from start point to arc centre.

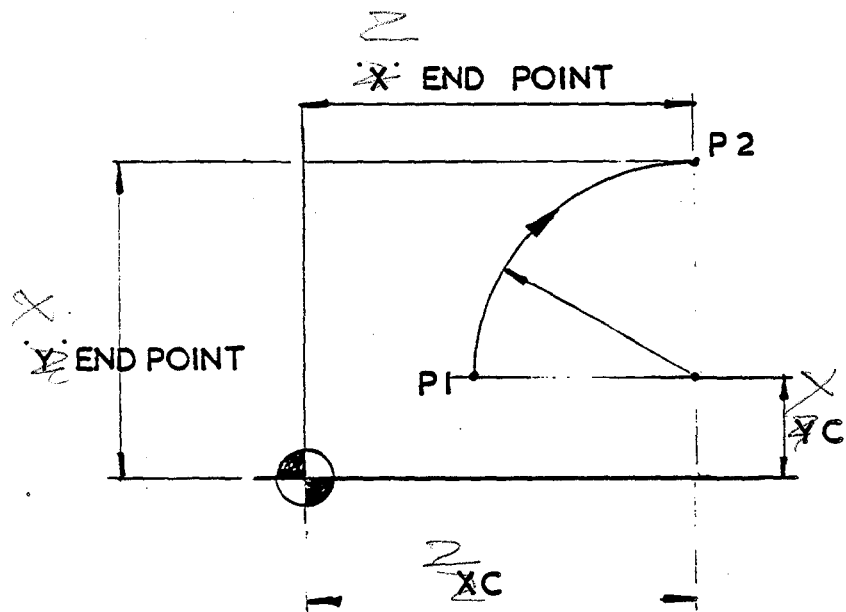
When a circular block is to be entered, G02 for clockwise or G03 counter-clockwise. Enter G02 OR G03 and the code is given on the screen, depress the enter key to continue. The program block grid will then appear on the screen and input can commence.

Enter first the end points, X and Y and a feedrate. To input the circle centre origin, about which the circular movement is to act, press CW/CCW key, the prompt will then request the input of circular centre XC and YC. When complete, the EOB key is used to signify the end of input for that block.

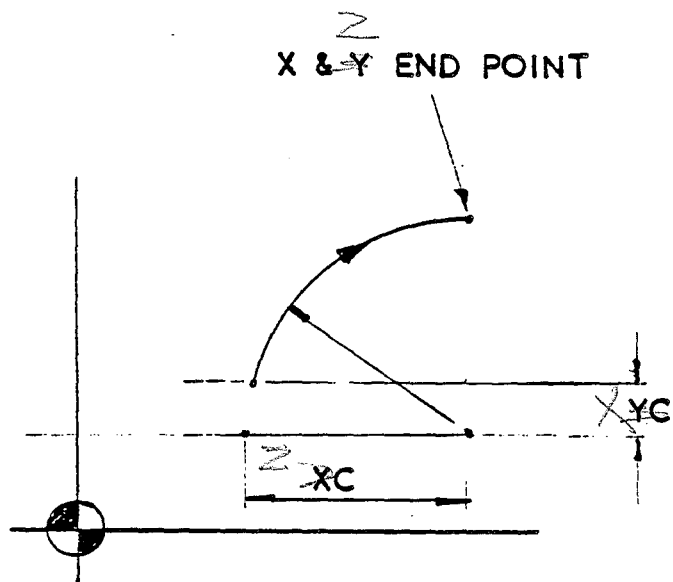
If incorrect or impossible end points are programmed, the control will respond with error in circle centre OR, during operation, the table will move continuously in a circle attempting to find the programmed end point. The movement may be stopped by depressing PROG STOP key or red stop button.

NOTE: When calculating end points and circle centres these must be accurate to ± 0.003 mm. Make sure that the radius at the start is equal to the radius at the end point. Circle centres can be established outside machine limits, although the start point and end points must be within the machine limits.

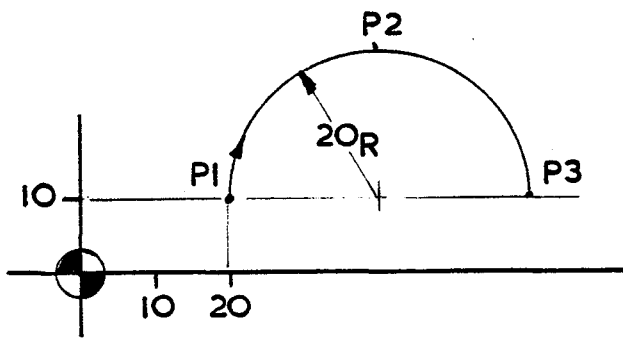
ABSOLUTE



INCREMENTAL



G02 CLOCKWISE



Absolute

P₁ = G01 X20 Y10 F500

P₂ = G02 X40 Y30 F150 CW/CCW XC40 YC10

P₃ = G02 X60 Y10 F150 CW/CCW XC40 YC10

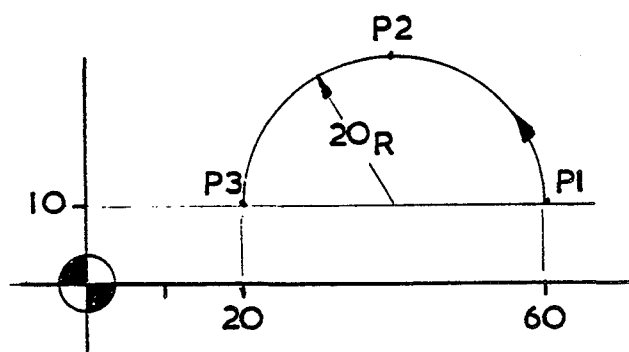
Incremental (Control will allow incremental input, executes and displays in absolute format.)

P₁ = G01 X20 Y10 F500

P₂ = G02 X20 Y20 F150 CW/CCW XC20 YC0

P₂ = G02 X20 Y-20 F150 CW/CCW XC0 YC-20

G03 COUNTER-CLOCKWISE



Absolute

P₁ = G01 X60 Y10 F500

P₂ = G03 X40 Y30 F150 CW/CCW XC40 YC10

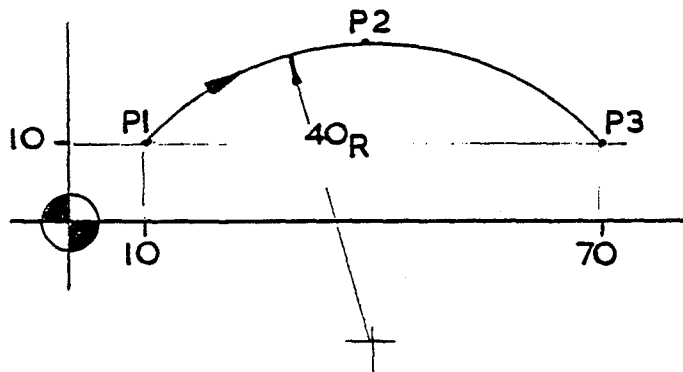
P₃ = G03 X20 Y10 F150 CW/CCW XC40 YC10

Incremental

P₁ = G01 X60 Y10 F500

P₂ = G03 X-20 Y20 F150 CW/CCW XC-20 YC0

P₃ = G03 X-20 Y-20 F150 CW/CCW XC0 YC-20



Absolute

P₁ = G01 X10 Y10 F500

P₂ = G02 X40 Y23.542 F150 CW/CCW XC40 YC-16.457

P₃ = G02 X70 Y10 F150 CW/CCW XC40 YC-16.457

Incremental

P₁ = G01 X10 Y10 F500

P₂ = G02 X30 Y13.542 F150 CW/CCW XC30 YC-26.457

P₃ = G02 X30 Y-13.542 F150 CW/CCW XC0 YC-40

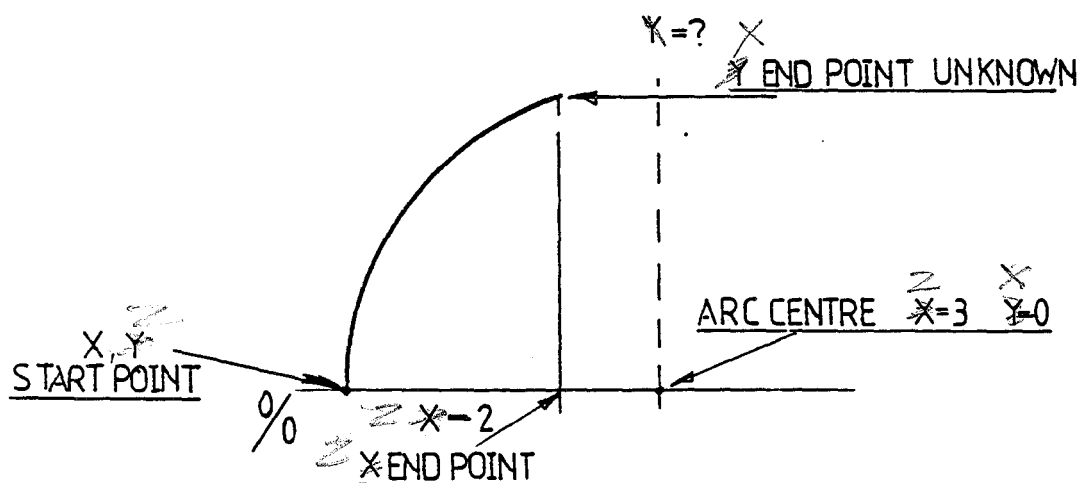
SECTION 17

AUTO/SINGLE STEP POSITIONAL ERROR CORRECTION FOR CIRCULAR MOVEMENTS

For linear moves, the control will always get to the desired X, Y position. However, in the case of circular moves, unless the circle centre is very accurately defined, it may not be possible for the control to achieve the programmed 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 control from a computer, the control will do a certain amount of positional error correction. The amount of correction (or error tolerance) available to the user is selectable from + or - 1 motor step in MDI or Single Step mode to + or - 0.1 mm in AUTO. (1 step = 0.00625 mm.) The error correction works in the following way, when the control 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 0.11 mm in both axes the control will set up and execute a linear move from its current position to the desired end point. If the error is greater than 0.1 mm then the PNC 3 will display the error message "Machine did not get to correct position".

This can be demonstrated with the following example.



BLOCK

1. G01 X 0 Y 0
2. G02 X 2 Y 2
CENTRE X 3 Y 0
3. M02 End of program

When this program is executed (in Single Step or AUTO) the control 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 -0.02 mm by altering the end point of block 2 and be X=2, Y=2.81. Now run the program in Single Step mode and note that the control 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 control 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 0.02 mm but in the opposite direction. If the program is run in Single Step mode it will be seen that the actual position is X=2.055, Y=2.85. If the program is run in AUTO, the control will self correct from this position to the desired position.

Now alter block 2 to read:

```
G02 X 2.625 Y 2.98  
CENTRE X 3 Y 0
```

Run the program in Single Step to prove that the end point is valid and that no error message will occur. If we now introduce an error of +0.02 mm 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 0.23 mm which is outside the tolerance band even when switched to AUTO. And this total error was introduced by an error of only 0.02 mm 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 (ie, ± 0.003125 mm). Programmers who are involved in writing post-processors for the PNC3 are particularly advised to be aware of this fact.

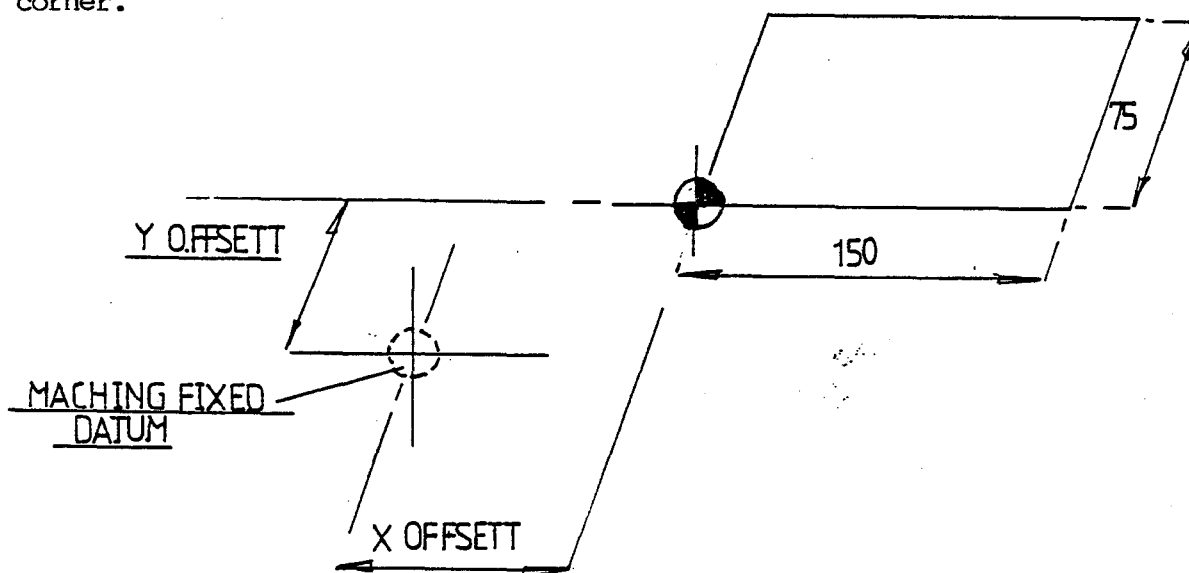
SECTION 18

SETTING AN ABSOLUTE ZERO

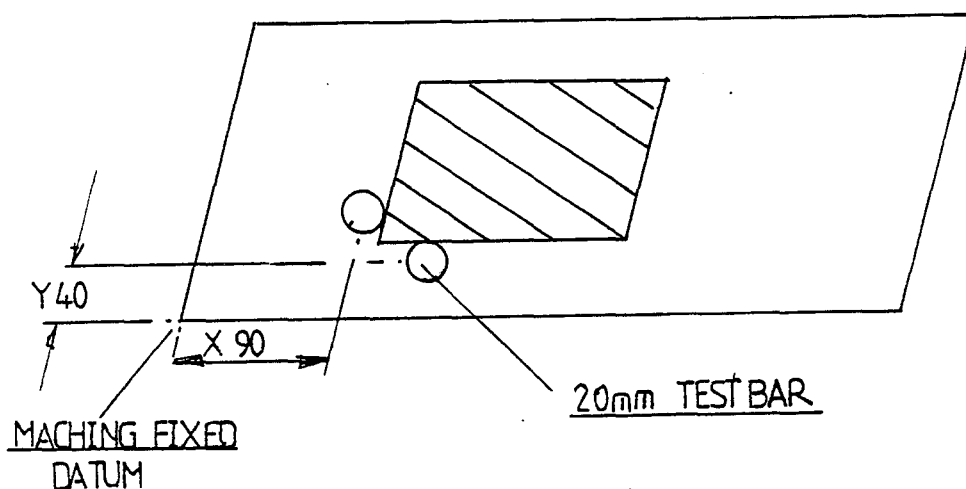
G55 machine offset is only permitted outside a program load mode. A machine offset offsets the program in memory and does not affect manual data moves.

To set a zero or datum from the component the user must enter the machine offset G55 required for the zero position. These values can be taken from the digital readout on the screen.

Assuming a component 150 long x 75 wide requires a datum from the bottom left hand corner.



With a test bar, wobble bar or setting probe in the spindle, use the axis jog keys to align the test bar against one edge of the component. Assuming the left hand edge on the Y axis after completion of this procedure. Note the reading on the digital readout for X axis. Add the radius of the test bar to the dimension. Record this new dimension as the offset value required to set X zero. Enter this dimension into the control as G55 X offset.



Example using 20 mm diameter test bar.

Reading taken from digital readout:-

X axis readout value = X90 +

Radius test bar = 10

Offset for X = X100

Enter offset into the control for X.

Press G55 ENTER X100 ENTER EOB

Repeat this procedure for Y offset.

Align the test bar against the front edge on the component, take the reading from digital readout for Y axis, add the radius of the test bar to this dimension as the offset value requires for Y zero.

Example

Y readout value = Y40

Radius test bar = 10

Y50

Enter the offset into the control for Y.

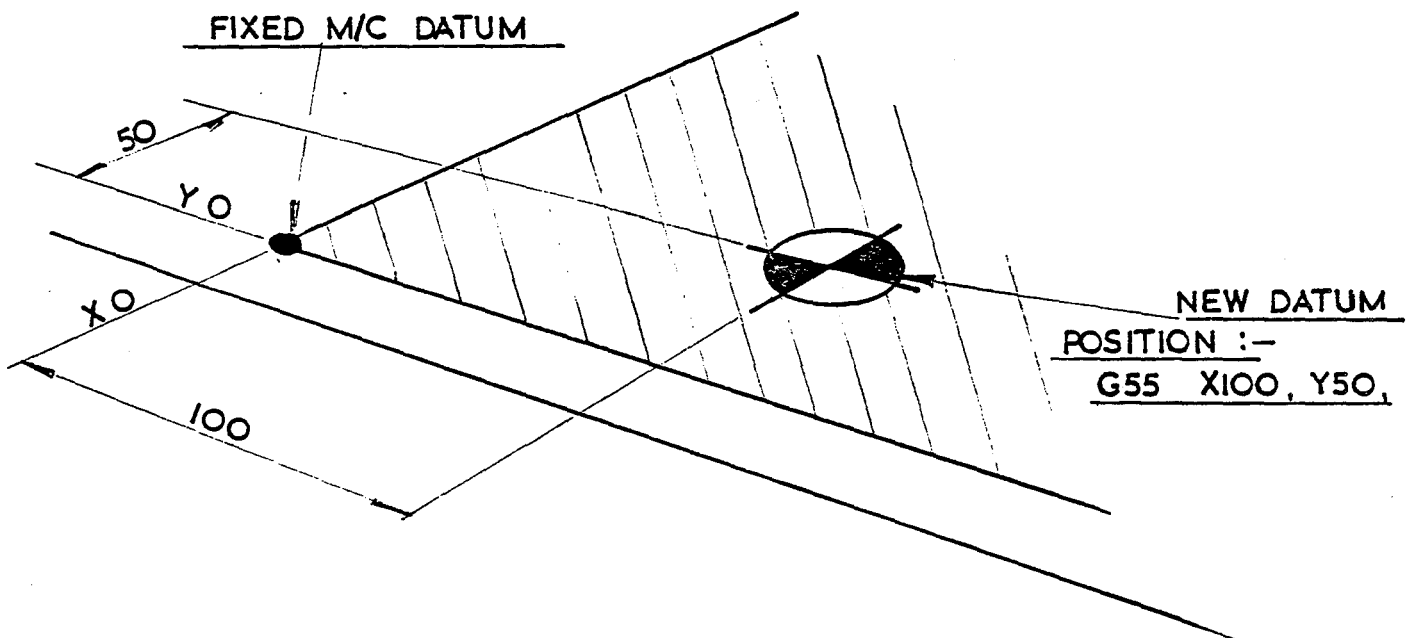
Press G55 ENTER Y40 ENTER EOB

.The offsets G55, X100, Y50 are the values required to set a zero or absolute datum relative to the component.

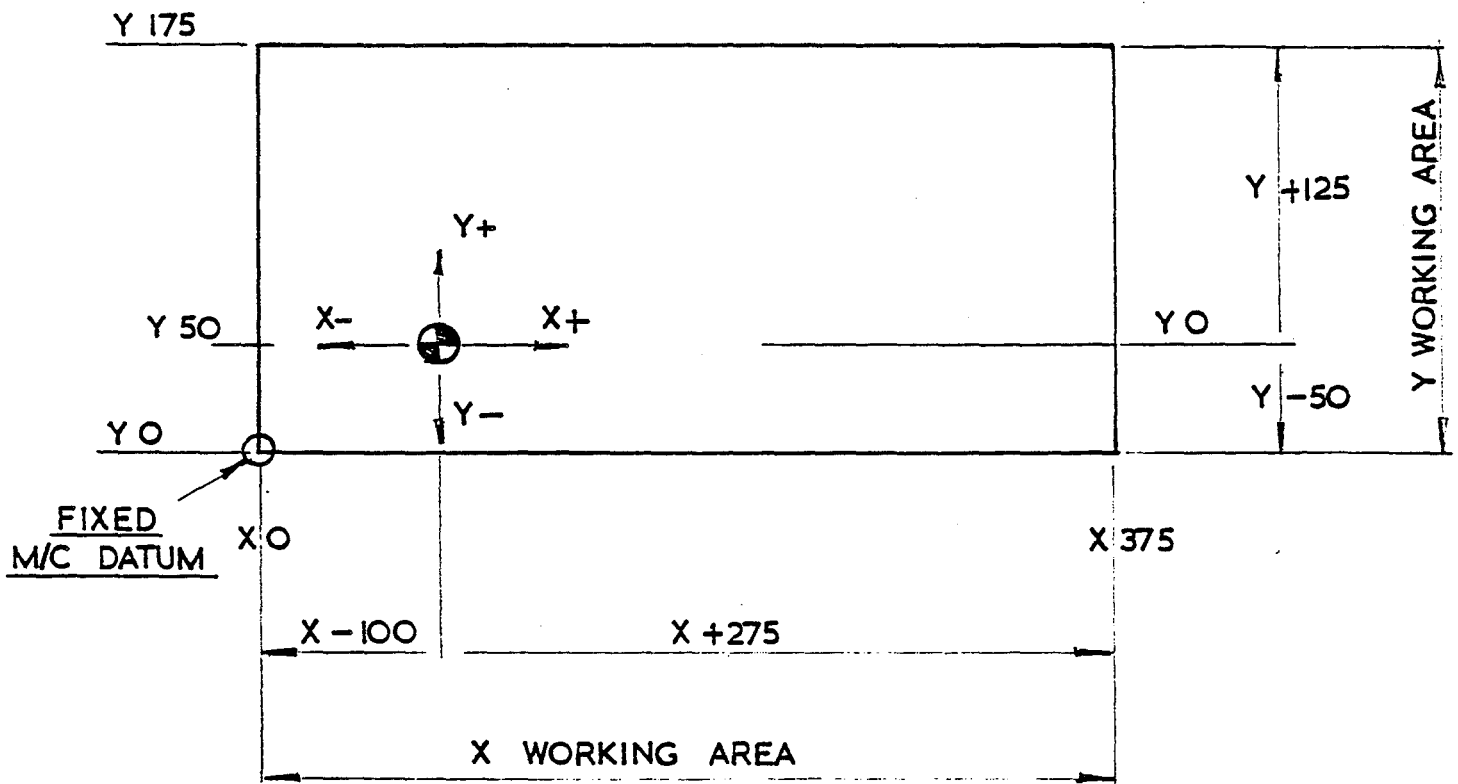
NOTE: G55 machine offsets must be entered into the control after the program is loaded into memory. When entering a new program all previous G55 machine offsets are reset to zero.

G55 machine offsets only affect the program in memory. All manual data entries are calculated and executed from machine fixed zero.

A G55 value will not be saved on cassette with the program in memory.



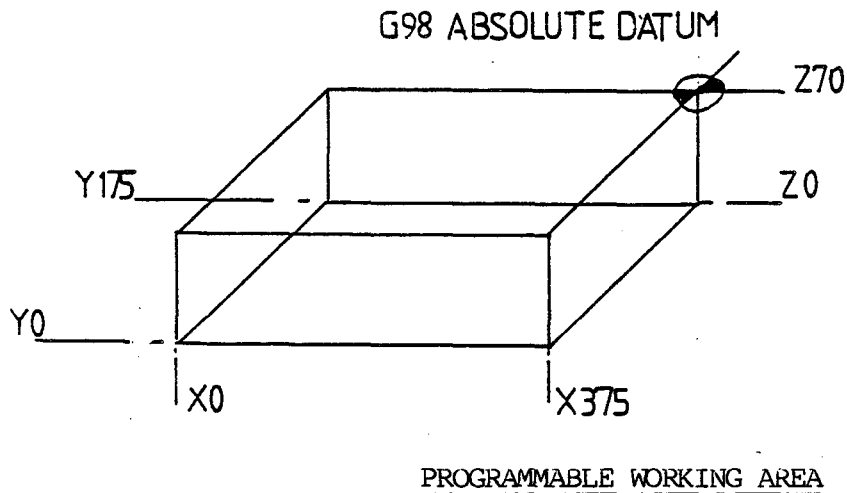
NEW PROGRAM PARAMETERS
DEFINE WORKING AREA



G98 ABSOLUTE DATUM/MACHINE REFERENCE POINT

An absolute datum block G98 can be programmed to cause the control to seek its machine reference point for each axis in turn, (ie, the maximum parameter for each axis.) This is executed at a predetermined feedrate and cannot be increased via feedrate override.

G98 ABSOLUTE DATUM



Pressing the zero key during a program load sequence will select a G98 absolute datum block to be entered. At this point in the program the machine will return to its absolute Datum X375 Y175 Z70.

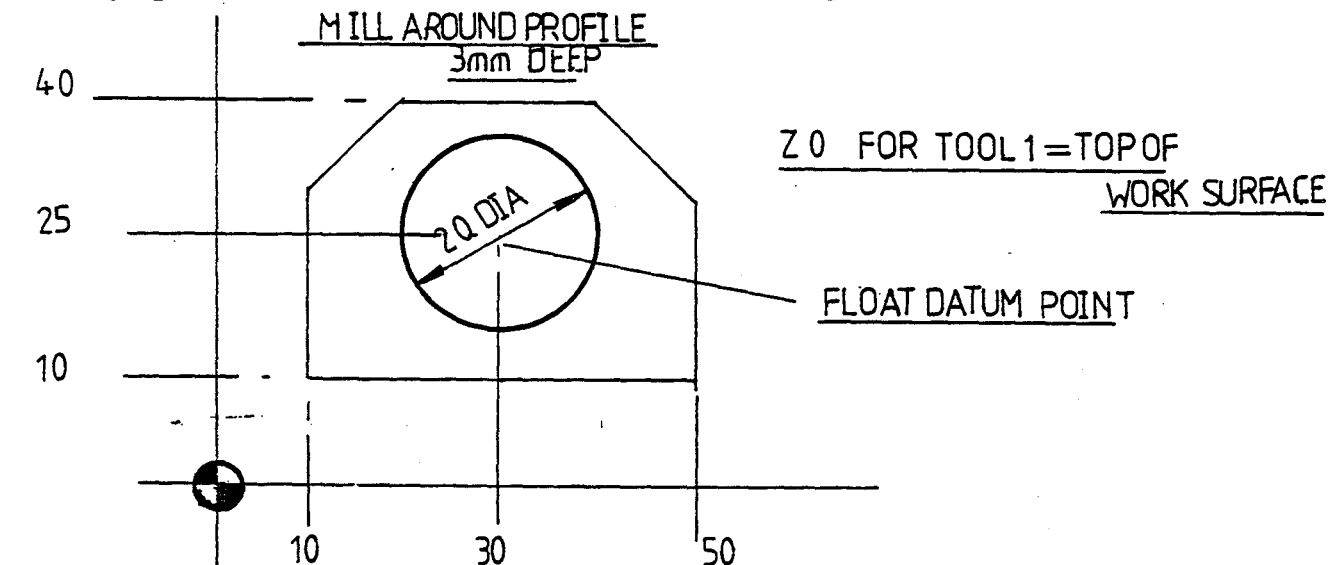
SECTION 19

FLOAT DATUM G99

Float datum is only permitted as part of program load sequence.

Float datum allows the programmer to set up a secondary datum within the program. A float datum block will establish a datum position with all axes set to zero about its current position. All subsequent blocks will be relative to this datum position. The digital readout will be relative to this datum position. If G55 machine offsets are zero Z0 tool length offsets will also be relative to this datum position.

Assuming the previous block X50, Y50, Z3, position is followed by G99 float datum block. Then the X, Y and Z axes are set to zero about that position. Calling up a float datum will deactivate cutter compensation.



1. M06 1	16. G00 X30 Y25
2. M03 1000	17. G99
3. G00 X30 Y10 Z3	18. G00 X0 Y10
4. G41	19. G42
5. G00 X30 Y10	20. G00 X10 Y10
6. G01 Z-3 F60	21. G01 Z-6 F60
7. G01 X10 F100	22. G2 X10 Y0 F100 XC0 YC0
8. G01 Y30	23. G2 X10 Y-10 XC0 YC0
9. G01 X20 Y40	24. G2 X-10 Y0 XC0 YC0
10. G01 X40	25. G2 X0 Y10 XC0 YC0
11. G01 X50 Y30	26. G00 Z0
12. G01 Y10	27. G40
13. G01 X25	28. G00 X-30 Y-25
14. G00 Z3	29. M05
15. G40	M02

SECTION 20

TOOL LENGTH OFFSETS

This feature can be used with a quick change tooling system.

Tool length offsets are described as a measured distance from the machine fixed zero to a plane at which the part is programmed, usually the top of the workpiece.

The ability exists to use several tools with independant lengths in one program. To program the Z axis we must know where the tip of the tool is at all times. This is done by the program loading up tool number and using the tool length offsets stored in the tool library. The control automatically adds or subtracts the tool length and places the tool point at the desired location.

If a move Z-3 is programmed using tool 1 the tool moves -3 from Z0.

Similarly if a move Z6 is programmed the tool moves +6 from Z0.

This feature eliminates the need for preset tooling, each tool offset may be measured on the machine.

SETTING AND RECORDING THE TOOL LENGTHS

Put the longest tool in the spindle and move the quill to a retract plane above the work surface, usually a position of maximum dimension.

Next raise or lower the knee to a position which the tool will clear the top of the worksurface.

To set a Z0 for tool 1 whereas Z0 = top of the worksurface:

1. set the key for MAN mode,
2. depress T key.

Displays a tool setting menu:-

1. display and edit tool offsets,
2. set tool offset,
3. change current tool.

Choose mode 2 from the menu (set tool offset).

The VDU will prompt the user to enter the number of the tool to be set.

When the number is entered the current offsets for that tool will be displayed and the prompt will change to <PRESS ENTER> to fix offset. Using the axis JOG keys advance the tool until it just touches the surface of the workpiece. Pressing the <ENTER> key causes the actual Z position of the tool to be loaded as the Z offset for that tool, and consequently the Z axis digital readout, changes to ZERO for Z axis.

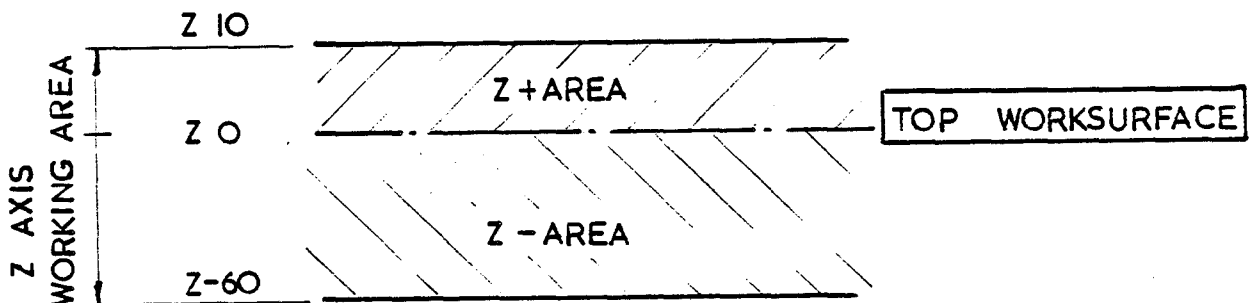
The prompt will now change to "ENTER tool offset changes D" and the user can enter the current tool diameter. The tool diameter cannot be less than ZERO, and if the user tries to enter a negative diameter then the control automatically sets the diameter offset to ZERO. When the diameter has been entered, the prompt will change to "Press <EOB> " to accept data. When <EOB> is pressed the control exits from the toolsetting mode.

Follow this sequence for each tool.

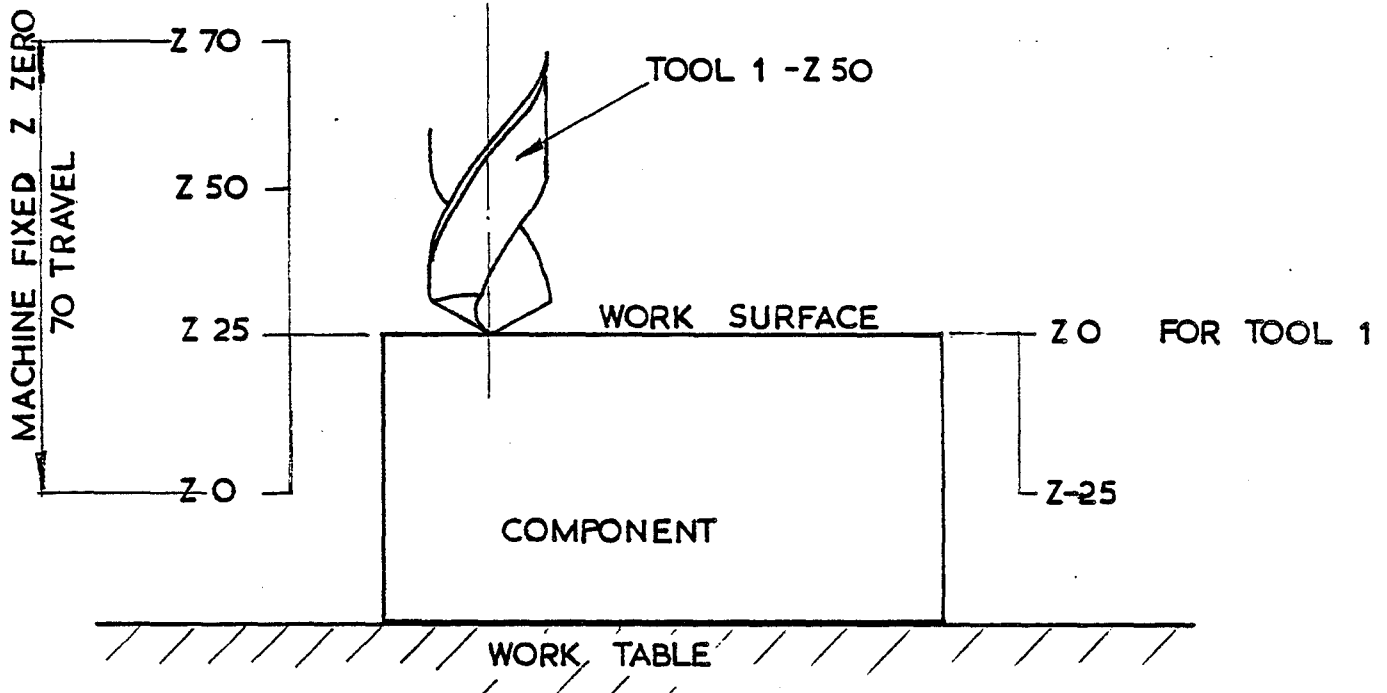
Reset returns to the previous menu.

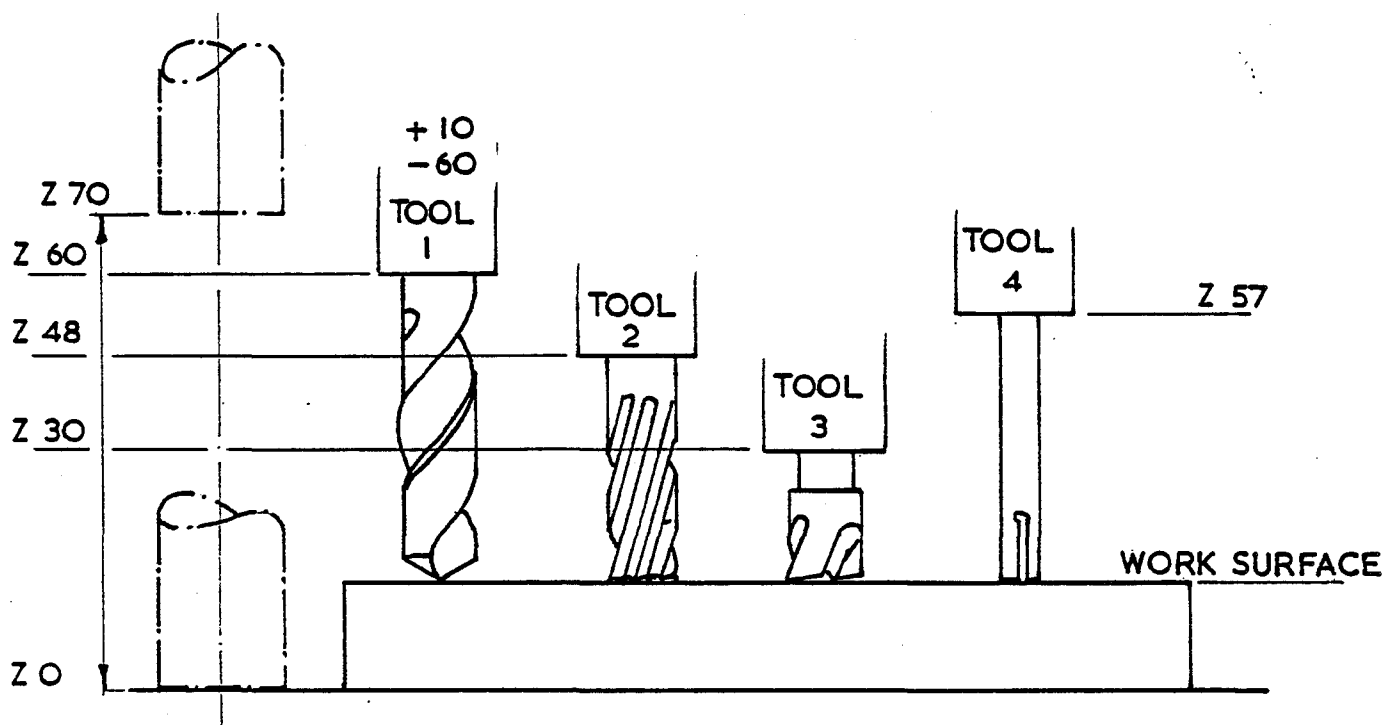
For example, if Z offset for tool 1 = 60 then the parameters of movement for that tool are:-

$Z+10$ to $Z-60 = 70$ mm movement on the quill.



TOOL 1 10 MM DRILL





PARAMETERS MOVEMENT FOR TOOLS SET FOR WORKSURFACE

$T_1 = Z+10$ POSITIVE	$Z-60$ NEGATIVE
$T_2 = Z+22$ POSITIVE	$Z-48$ NEGATIVE
$T_3 = Z+40$ POSITIVE	$Z-30$ NEGATIVE
$T_4 = Z+13$ POSITIVE	$Z-57$ NEGATIVE

SECTION 21

DISPLAY AND EDIT TOOL OFFSETS

The current tool offsets can be displays by pressing the T key followed by selection 1 from the menu. Each tool has its own length (Z) and diameter offset.

All 16 pairs of offsets will be displayed and the control will 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 to the previous menu.

Tool offset editing can be accomplished by entering the tool number to be edited and pressing enter.

The selected tools offset will then appear on the screen and the prompt will change to "ENTER TOOL OFFSET changes" followed by "Z" indicating the axis to be changed. Values entered at this point are incremental and will be added OR subtracted to the current offset.

If it is required to reduce the offset then a negative value must be entered. To increase the offset then a positive value must be entered.

When enter is pressed the prompt will change to "DIA" and the offset value can be changed in the same way.

Tool diameter offsets cannot be negative and if the user tries to enter an offset change value which would result in a negative diameter. The control will automatically enter a total diameter offset of ZERO.

NOTE: When a tool is edited the control also assumes that the tool is to become the current tool and the digital readout will change and display position will change accordingly.

SECTION 22

SPINDLE SPEED CONTROL

Easimill 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 screen.

When FWD or REV or M03 or M04 is selected the control 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 changed 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 control is first powered up).

Some PNC 3's have a 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 screen) or using the following codes.

M08 Coolant On

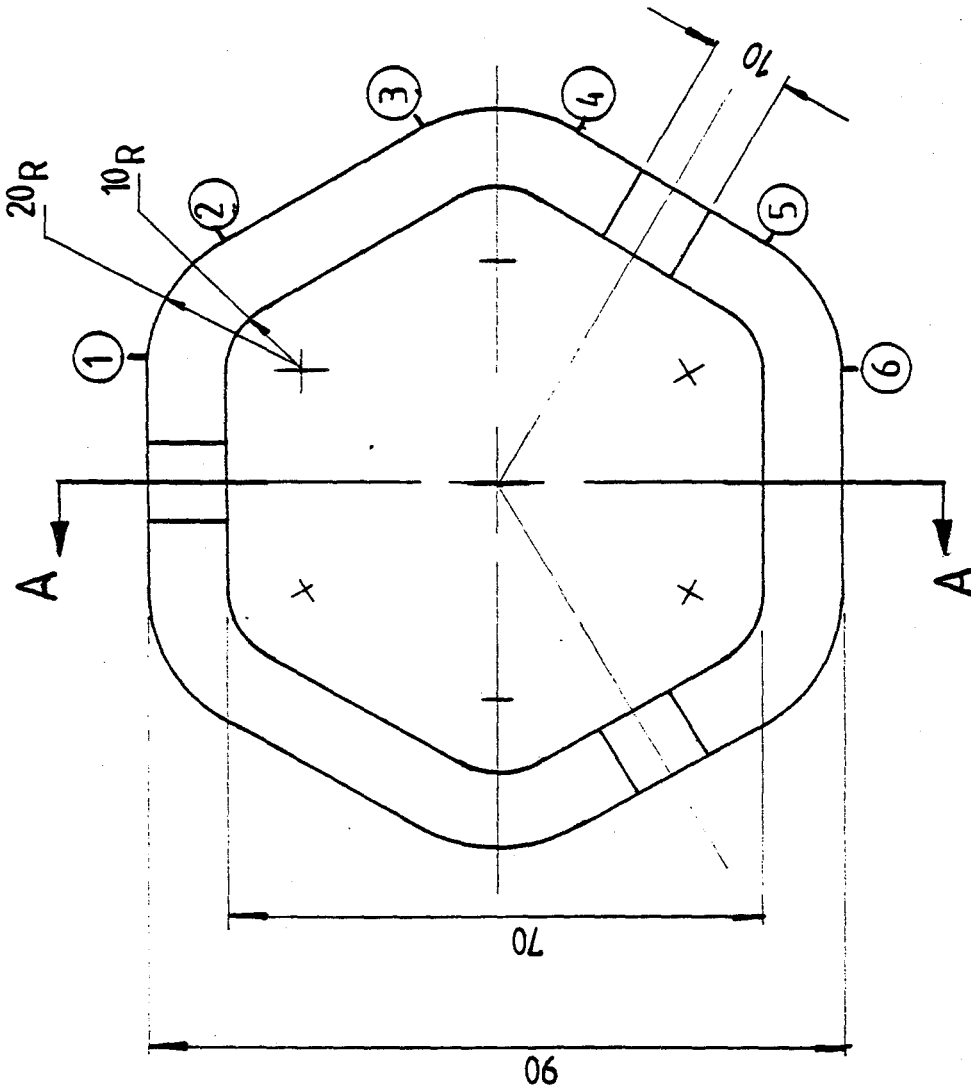
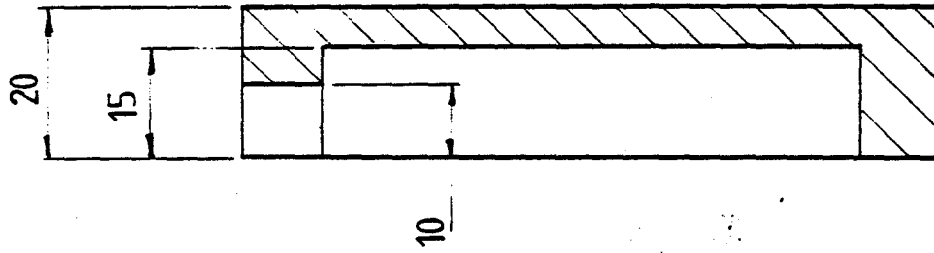
M09 Coolant Off

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 PNC 3 when that block in the program is executed.

FIG 4

EASIMILL 3 CNC

PROGRAM NO 4



PROGRAMMING

SECTION 23

ABSOLUTE/INCREMENTAL G90/G91

The mode of the machine can be altered to and from absolute and incremental by pressing the abs/inc key or stating the appropriate G code. The default condition is always absolute format. The current mode of the machine is displayed in the top right hand area of the screen, along with the current units inch or mm. In absolute mode all co-ordinates refer to absolute positions from the origin.

In incremental mode any keyed in co-ordinates are added to the previous co-ordinates or the current position, therefore it is incremental input with absolute display and execution.

During EDIT mode, the new data keyed in will be related to the data in the previous block.

SECTION 24

ENTERING A NEW PROGRAM

Easimill's memory will store 500 blocks of information. To enter a new program into memory set the key to Man position and depress the LOAD key.
END

The VDU will prompt the user for a program number which will be assigned to the new program. This can be any number with a maximum of six digits. Key in the new number and press enter to accept the data. The control is now set to load mode, this enables the control memory to be loaded with a series of blocks which will be executed consecutively when the program is run. To end the program load sequence, depress the LOAD key, which will add M02 onto the end
END

of the program and end loading. Alternatively, keying in M02 will end program loading.

Two different load operations are possible when the memory is loaded. Depress the LOAD key and depress enter key, the VDU will prompt the user:-
END

1. load memory from keyboard,
 2. continue memory load from keyboard.
-
1. Load is used to enter a new program into memory. Any previously loaded program is overwritten, ie destroyed.
 2. Continue memory load from keyboard, enables an existing program to be continued, ie extended, and will cancel the end program code, loading will commence from the last block in memory.

Upon completion of a load or a continue memory load, the control displays the program number, how many blocks there are in memory and how much memory remains for a period of six seconds after which the control displays normal data. Any key pressed clears the memory status display.

Should too much program data be keyed into the control, such that the memory becomes full, "memory is full" is displayed and no more data can be entered. The reset key will restore normal operation.

Once EOB has been entered it is not possible to step back and update erroneous data without ending the loading sequence and calling up edit mode. (See Edit Section Page 91.)

See edit text.

REPEAT G81

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

1. The start block number to be repeated, this must be linear block with all axes defined. X, Y and Z dimension within the start block.
2. The end block number to be repeated.
3. The number of repeats required.
4. The required offset dimension, this being incremental offset for each repeat loop.
5. Feed. Entering a feed into the repeat loop will change all feeds programmed within the loop to the new feedrate.
Omitting a feedrate value will leave all feeds as initially programmed.

Repeats may be programmed up to a nested level of three with a maximum repeat loop 99 times.

Should this level be exceeded "Nest error in repeat levels" is displayed. The reset key restores normal operation.

When each repeat is programmed the control checks all the dimensions being repeated, adding the programmed offset to the number of repeats to ensure that the machine limits are not exceeded. This process may take a few seconds. Should the limits be exceeded, "Move exceed machine limit" is displayed, the reset key restores normal operation and corrected data keyed in before program can continue.

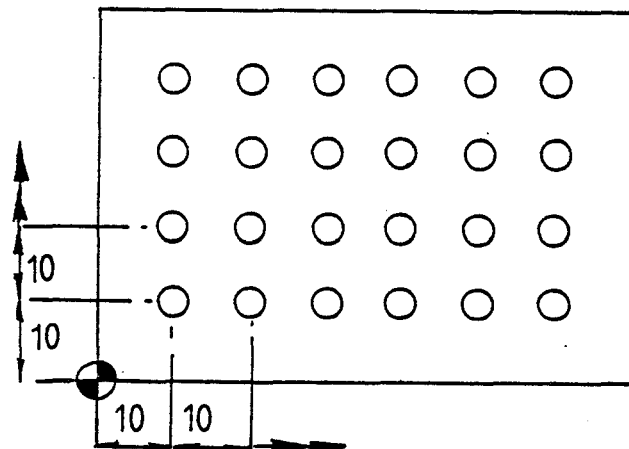
Calling up a G81 repeat block will deactivate cutter compensation.

Below is the information required for a repeat block.

G81 REPEAT FROM.....TO.....REP.....OFFSET.....FEED.....

G81 - REPEAT CODE
FROM - START BLOCK FROM WHERE REPEATING WILL COMMENCE
TO - END BLOCK OF REPEAT LOOP
REP - ENTER THE NUMBER OF REPEATS, MAXIMUM 99
OFFSET - ENTER X, Y OR Z INCREMENTAL OFFSET FOR EACH LOOP
FEED - ENTERING A FEEDRATE VALUE WILL REPLACE ALL FEEDRATES WITHIN THE LOOP

NOTE: G20 Program Scale, does not effect the offset within the repeat loop.
The offset values remain unaffected by any program scale factor active in the program.



DRILL 24 HOLES
EQUI SPACED 6mm DEEP

1. M03
2. G00 Z3
3. G00 X10 Y10 Z3
4. G01 Z-6 F100
5. G00 Z3
6. G81 FROM 3 TO 5 REP 5 X+10
7. G81 FROM 3 TO 6 REP 3 Y+10
8. G00 X0 Y0
9. M05
10. M02

SECTION 26

MIRROR IMAGE FACILITY

Mirror image facility is only permitted as part of a program load sequence acting upon memory.

Mirror images are defined by using the mirror key or the appropriate G code.

Depress either the respective G codes or the mirror key and the mirror menu appears on the screen:-

1. G10 mirror X,
2. G11 cancel mirror X,
3. G12 mirror Y,
4. G13 cancel mirror Y.

Mirror is only available for X and Y axes.

Calling up a mirror in the program causes the control to establish an axis of symmetry about its current position.

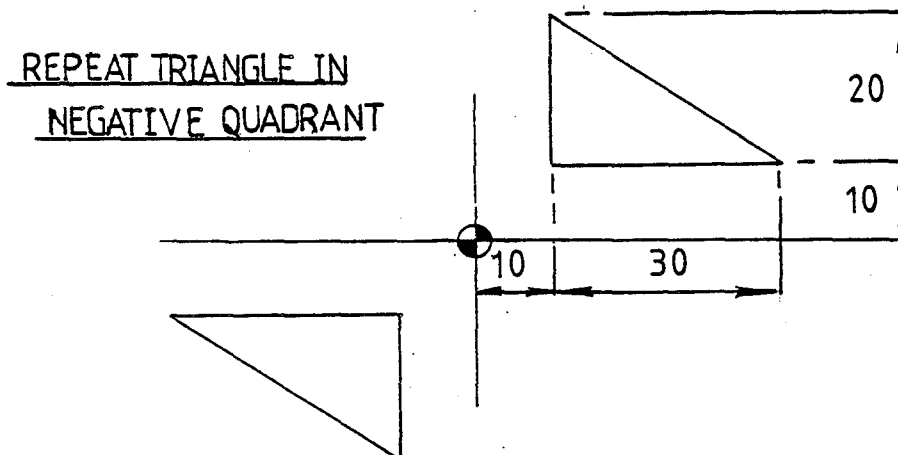
All succeeding blocks are reflected about this axis.

This function uses the actual position from the previous blocks as its axis of symmetry, taking into account all scaling factors and offsets including repeats.

Calling up a mirror or cancel mirror mode will deactivate cutter compensation.

EXAMPLE SHOWING MIRRORS

REPEAT TRIANGLE IN NEGATIVE QUADRANT.



1. G00 X0 Y0 Z3	13. G10
2. M03	14. G12
3. G00 X25 Y10 Z3	15. G81 FROM 3 TO 12 REP 1
4. G42	16. M05
5. G01 Z-3 F75	17. G11
6. G01 X10 F75	18. G13
7. G01 Y30	19. G00 X-50 Y0
8. G01 X40 Y10	M02
9. G01 X23	
10. G00 Z3	
11. G40	
12. G00 X0 Y0	

SECTION 27

SCALE FACILITY

Program scale G20
Machine scale G21
Scale range 0.01%-650% 100% = Actual size

Program scaling can be accomplished outside of a program. This is called machine scaling G21. Or inside a program loading sequence. This is called program scale G20. Program scale and machine scale allows the user to key in a value in the range of 0.01% to 650% as a scale factor for all subsequent X and Y dimensions. Z axis dimension remain unchanged.

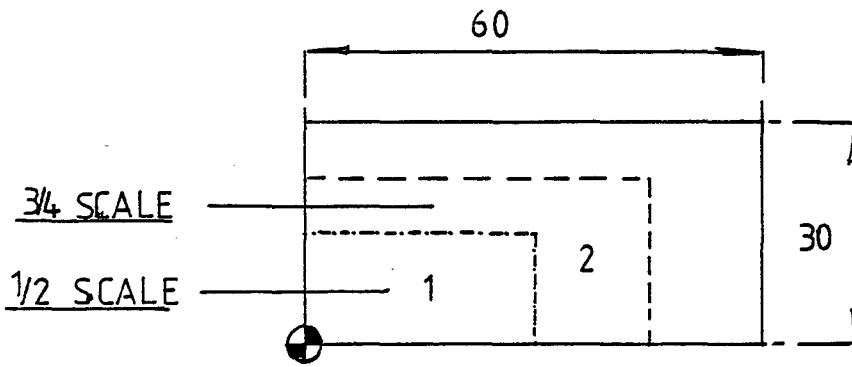
Its effect can be cancelled by inserting a program scale of 100%, where 100% = full size.

Machine scale G21 permitted outside program loading sequence allows the user to key in a value in the range 0.01% to 650% as a scale factor for the complete program, data is automatically scaled. Entering a new program into memory will cancel any machine scale.

NOTE: Any program scale active in the program will be further scaled by the machine scale function so dimensions can be reduced by up to 0.01%x0.01%. Example 10,000 times or increased by up to 6.5x6.5 example 42.25 times.

Program Scale does not effect the offsets within a repeat block, any offsets remain unaffected by the scale factors within the program.

PROGRAMME SCALE



RECTANGLE 60x30 mm

AREA 1 = 50%

AREA 2 = 75%

1. M03
2. G20 50 PROG SCALE 50%
3. G00 X0 Y0 Z3
4. G01 Z-3 F150
6. G01 X60 F150
7. G01 Y30
8. G01 X0
9. G01 Y0
10. G20 75 PROG SCALE 75%
11. G81 FROM 3 TO 9 REP 1
12. G20 100 PROG SCALE 100%
13. G81 FROM 3 TO 9 REP 1

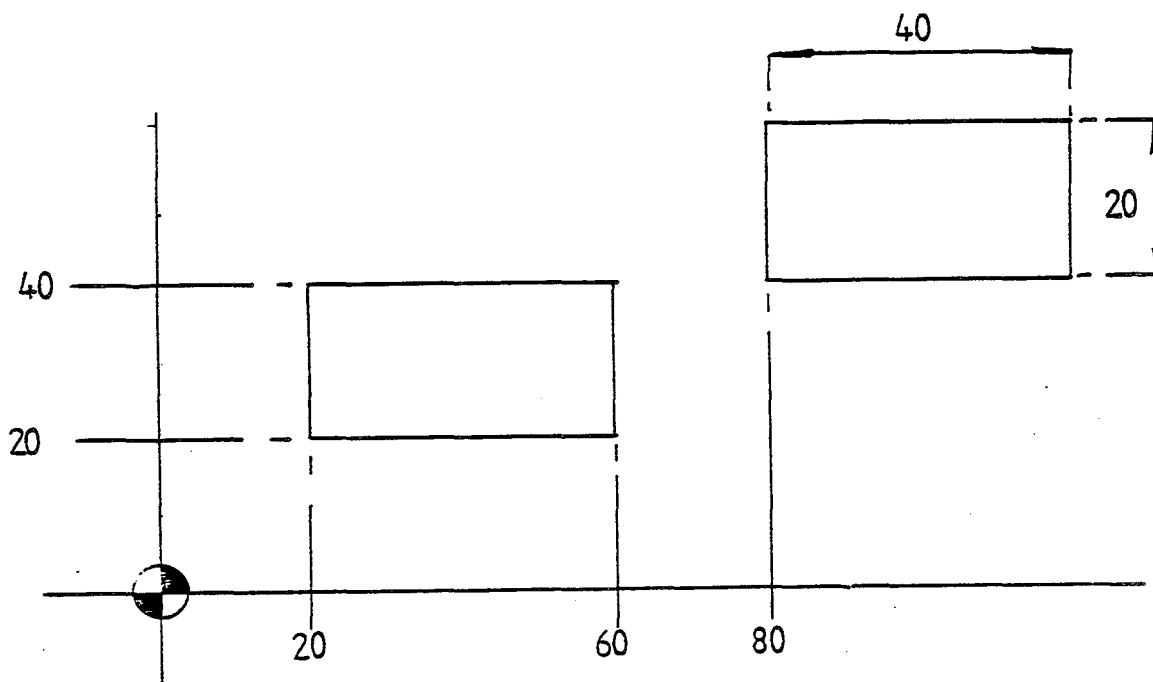
OFFSET FUNCTIONOFFSET FACILITY

Two offset functions are permitted, G55 is a machine offset, set outside the program and will act upon the memory allowing the start position to be established at any point within the machine limits and used to offset the entire program. This facility can be used to establish a datum on the component or used as a dry run facility. Entering a new program into memory will automatically reset any previous machine offsets, G55 to zero.

Pro ram offset G54 can be used inside a program loading sequence to offset parts of the program and is incremental in operation.

For example, if block 1 is G54 X10 mm and block 2 is G1 X5 mm, programmed moves from block 3 onwards will be offset by a total of 15 mm. If a program offset appears as a block within a repeat loop then the offset will be added onto the total offset everytime the repeat loop is executed. Should you wish to cancel the offset, then a new offset could be inserted into the program.

If an equal value but the opposite sign, example G54 X-10 will cancel out the positive offset. Program offsets are reset to zero everytime the program is executed from the start.



1. M03
2. G00 X0 Y0 Z20
3. G00 X20 Y20 Z3
4. G01 Z-3 F150
5. G01 X60 F150
6. G01 Y40
7. G01 X20
8. G01 Y20
9. G00 Z3
10. G54 X60 Y20
11. G81 FROM 3 TO 9 REP 1
12. G00 X-60 Y-20 Z20
M02

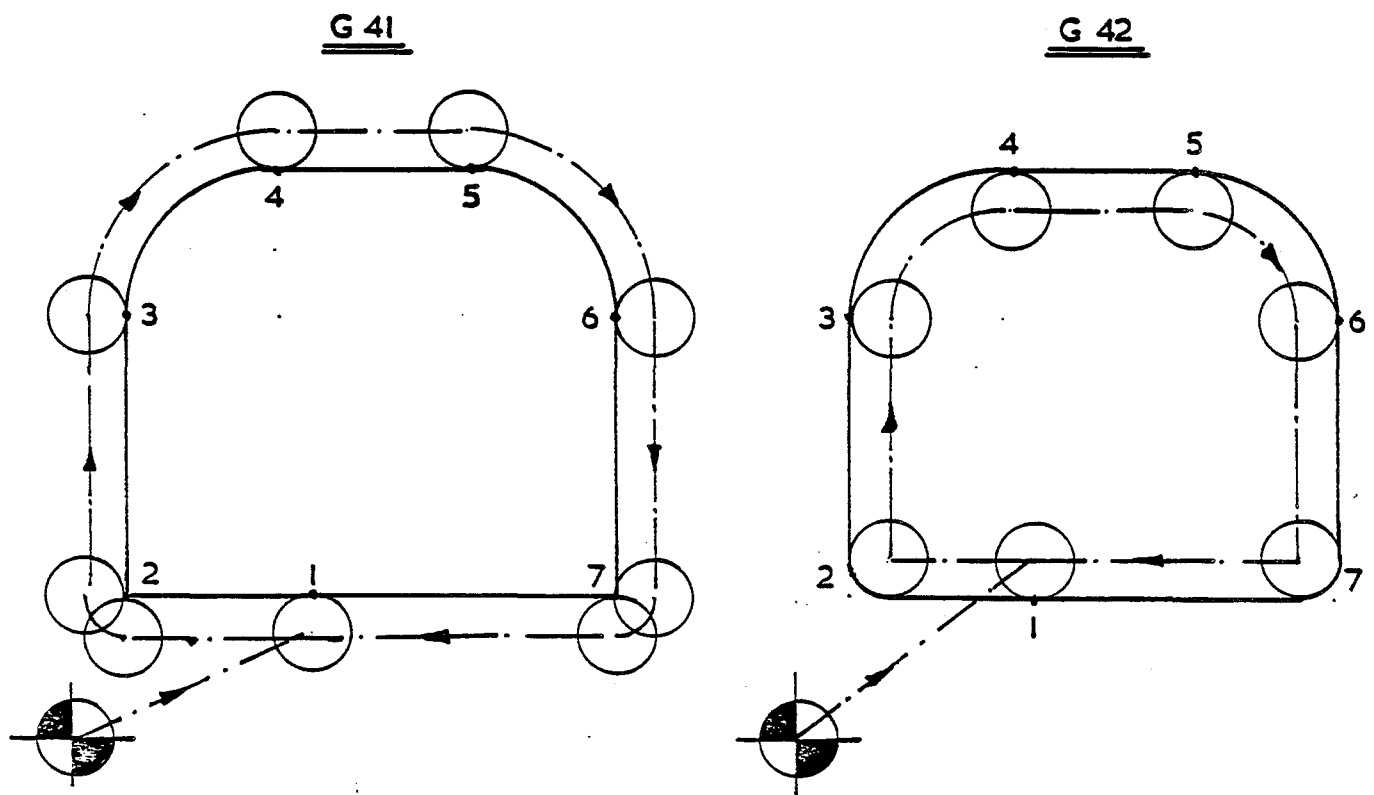
CUTTER DIAMETER COMPENSATION

Tool diameter compensation (or automatic cutter offset calculation) allows the control to compensate for the diameter of the current tool. This means the program need only specify the true dimension of the desired finished product. This facility is obtained by pressing the COMP key, either inside or outside a program load sequence. Alternatively the appropriate G code can be used. The control will generate a new cutter path at all times equidistant from the programmed tool path. The distance the new path is separate from the programmed path is equal to half the tool diameter. For example, if Tool 1 was set as 10 mm diameter during toolsetting. The control will generate a cutter path 5 mm equidistant from the original programmed path.

G40 - cancel cutter compensation.

G41 - Turns cutter compensation on and compensates to the left of the original cutter path in the direction the tool is moving.

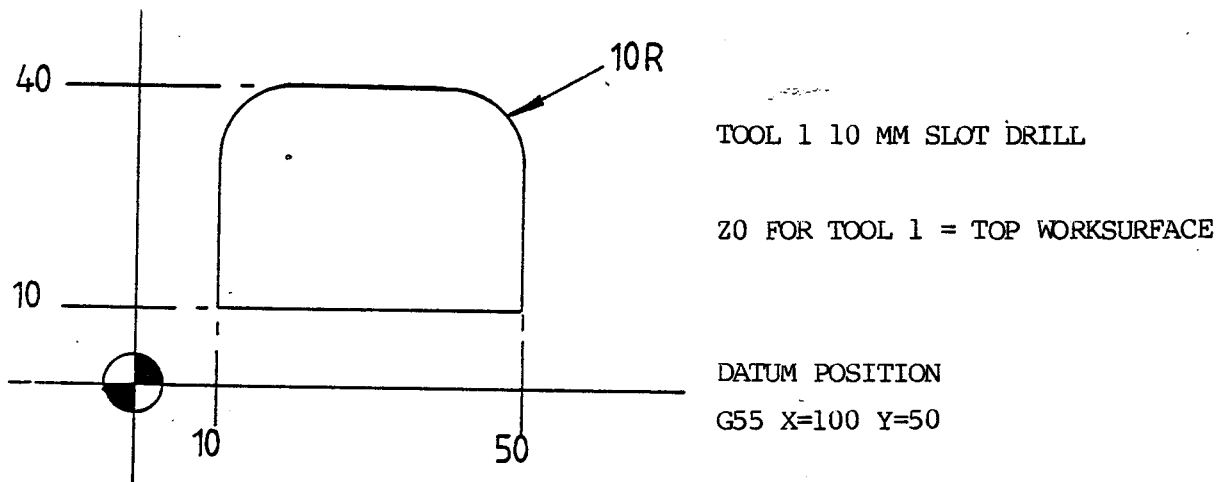
G42 - Turns cutter compensation on and compensates to the right of the original tool path in the direction the tool is moving.

Examples

When cutter compensation is activated the control checks through the program and determines the new tool path and compensates the offset taking up a new position perpendicular to its present position. Therefore, before calling up cutter compensation in the program, the previous block must position the tool at a clearance plane above the workpiece to avoid damage to the workpiece or tool.

Consequently, before cancelling tool compensation, retract the tool clear of the workpiece to avoid damage to the workpiece when the tool offsets are deactivated.

NOTE: Do not call cutter compensation at a corner, but part way along a straight line. The action of the control on an external corner is to roll around it in an arc, creating a burr-free corner.



1.	M06 1		- CALL UP TOOL 1
2.	G00	Z3	- SETS T1 CLEAR OF WORKSURFACE
3.	M03 800		- SPINDLE STARTS AT 800 RPM
4.	G00 X30	Y10	- DRIVE TO POSITION ON PROFILE
5.	G41		- ACTIVATE TOOL COMPENSATION LEFT
6.	G00 X30	Y10	- STATE SAME X AND Y CO-ORDINATES SO TOOL TAKES UP COMPENSATED POSITION
7.	G01	Z-3 F50	- SINK IN THREE DEEP
8.	G01 X0	F100	- CUT TO CORNER
8.	G01	Y30	- POSITION TO START RADIUS USING PREVIOUS FEED
9.	G02 X10	Y40 XC20 YC30	- CIRCULAR INTERPOLATION
10.	G01 X40		- POSITION TO START RADIUS
11.	G02 X50	Y30 XC40 YC30	- CIRCULAR INTERPOLATION
12.	G01	Y10	- POSITION TO BOTTOM CORNER
13.	G01 X28		- POSITION WHERE MILL STARTED
14.	G00	Z3	- RETRACT TOOL ABOVE WORKSURFACE
15.	G40		- CANCEL CUTTER COMPENSATION
16.	M05		- STOP SPINDLE
17.	G00 X-50	Y0	- TOOLCHANGE POSITION
	M02		
	G55 X150	Y60 Z0	- DATUM POSITION

SECTION 30

EFFECTS OF CUTTER OFFSETS

In the following section all possible combinations of two adjacent blocks will be considered. For the purpose of the examples the current block will be known as block 1 moving from point P1 to point P2 and the next block will be known as block 2 moving from point P2 to point P3. 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 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 deactivate cutter offsets before including the mirror imaging block. The cutter offsets can be reactivated in the normal way.

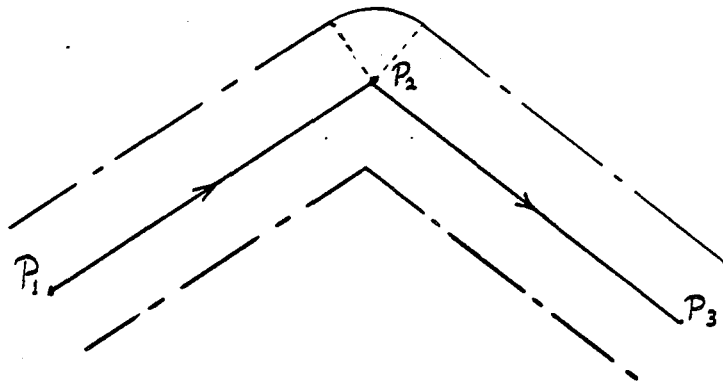
In the following examples the terms internal and external corner will be used. They are defined as follows:

External Corner - When the angle between the two blocks is greater than 180° on the side of the material being worked, ie 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° , ie 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.

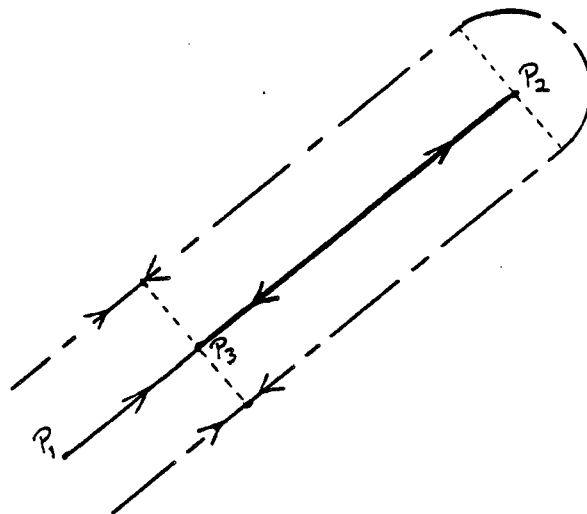
As will be seen from the following examples, the PNC's action on encountering an external corner is to move around it in an arc of radius equal to the cutter radius about a centre located at point P2. During this circular motion the tool will always be in contact with point P2 thus creating a burr-free corner.

STRAIGHT LINE TO STRAIGHT LINE INTERSECTION

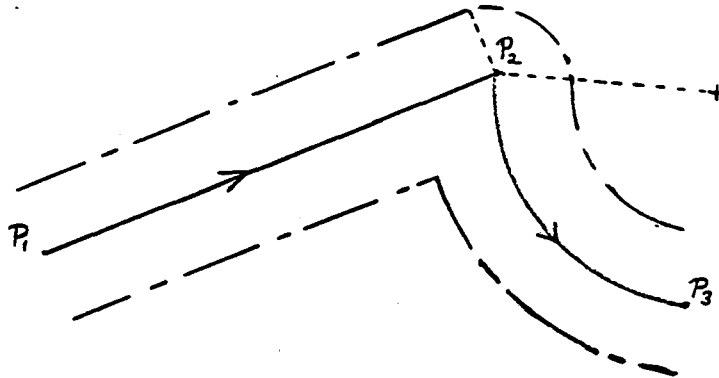


In the extreme case where block 2 actually doubles back onto itself this will always be taken as an external corner independent of whether a right hand or a left hand offset is being used.

Example

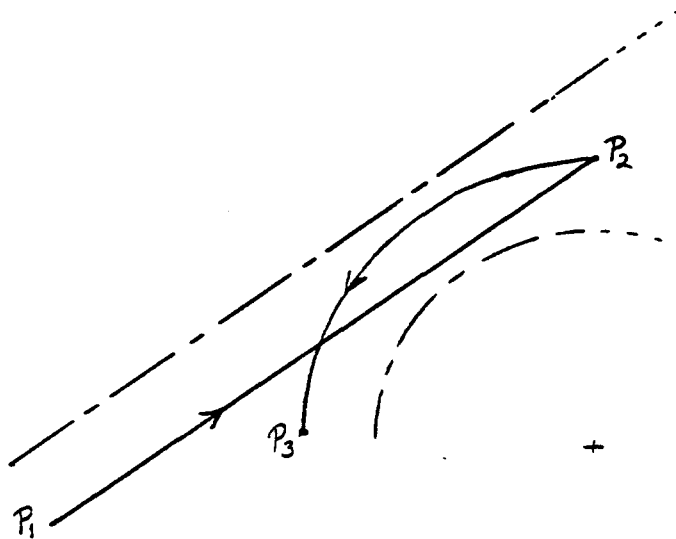


STRAIGHT LINE TO ARC INTERSECTION



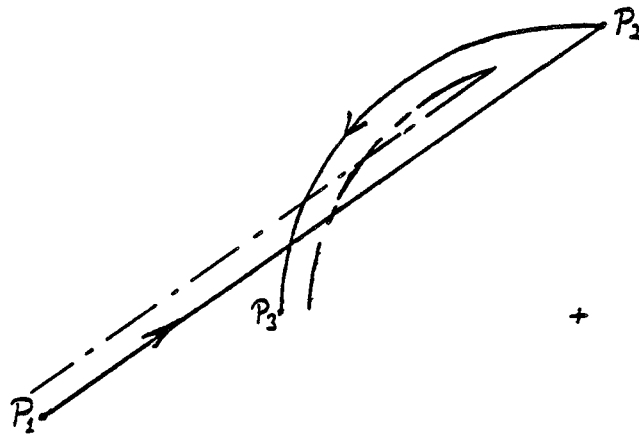
The PNC will cope with any kind of Line to Arc intersection except where no intersection is possible.

Example

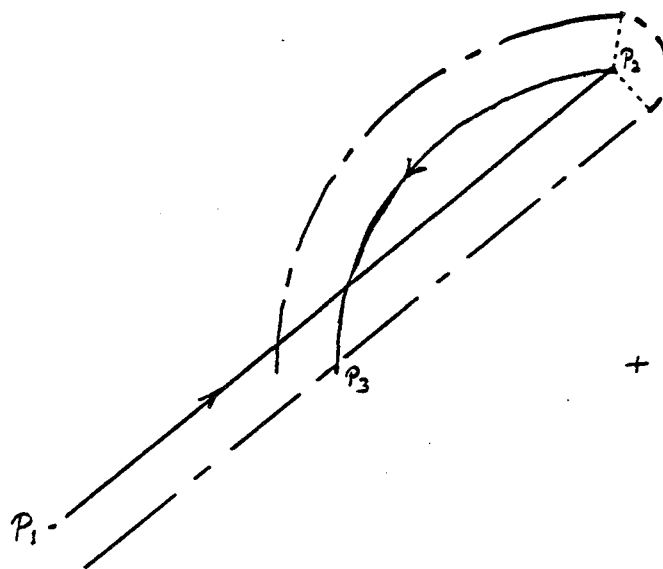


In which case there will be a pause during the calculation period, followed by an error message on the display indicating that perhaps the wrong tool diameter has been input. Note that in this case if the cutter diameter is small enough an intersection may be possible.

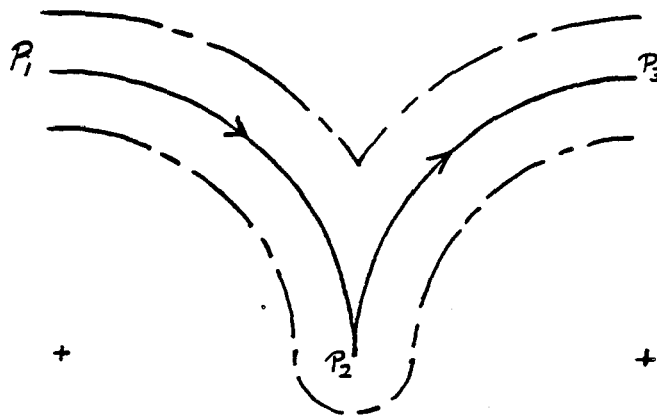
Example



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:

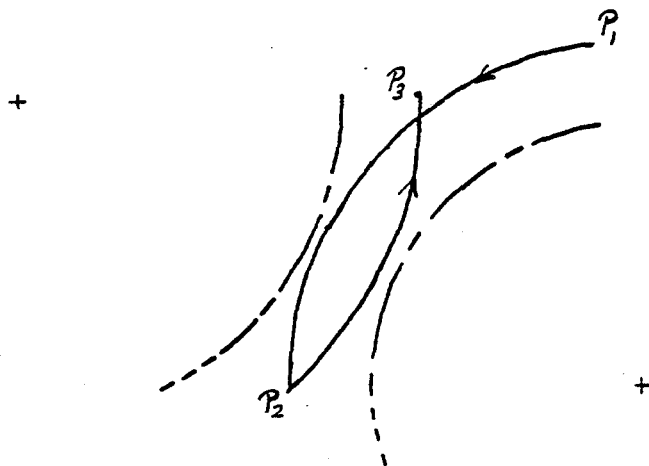


ARC TO ARC INTERSECTION



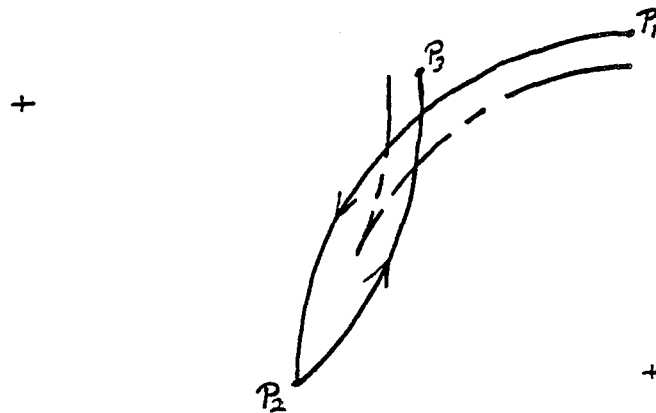
The PNC will cope with any kind of Arc to Arc intersection except where no intersection is possible.

Examples

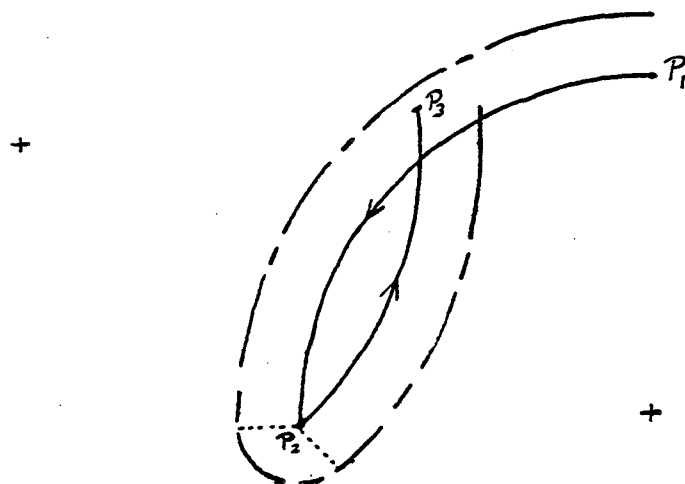


In which case there will be a pause while the PNC tries to find an intersect following by an error message on the display indicating that perhaps the wrong tool diameter has been input. Note that in this case if the cutter diameter is small enough an intersection may be possible.

Example



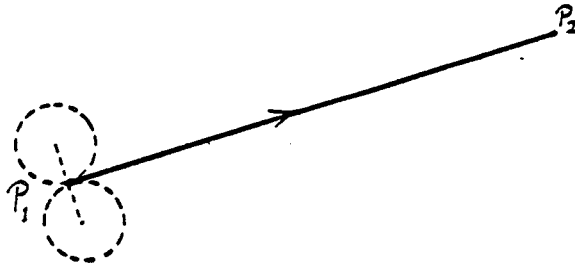
Note that in both the previous examples if a right hand (or positive) offset is applied the P_2 will be taken as an external corner and the following will result.



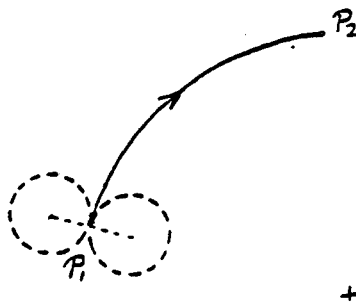
FIRST OFFSET AFTER ACTIVATION

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.

Example



or if the line P_1 to P_2 is circular:-



LAST OFFSET WHEN DEACTIVATED

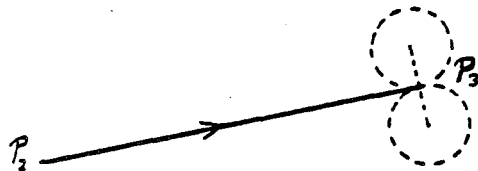
Apart from a G40 Cancel Tool Comp block the following types of block will all deactivate the cutter offset.

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 reactivate)
 G98 Absolute Datum/Machine Reference Point
 G99 Floating Datum

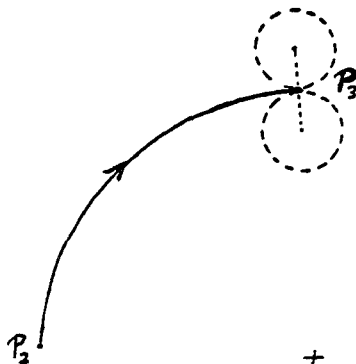
Also the last block in any program will automatically deactivate cutter offsets.

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

Example



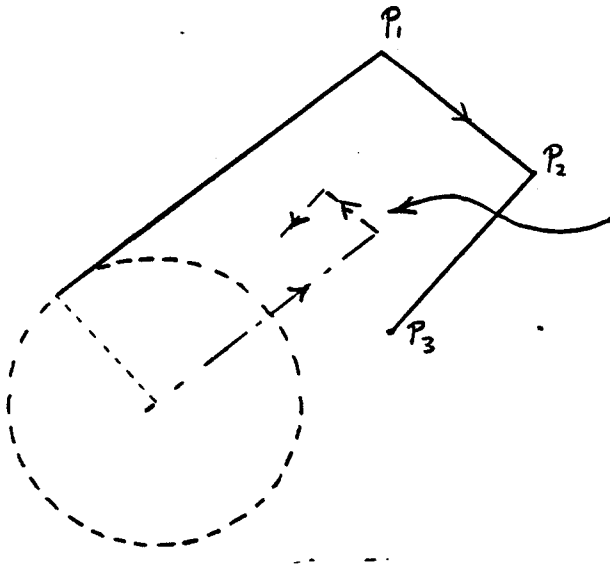
or if the line P_2 to P_3 is circular.



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 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 tool small for the cutter diameter.

Example



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.

SECTION 31

AUXILIARY FUNCTIONS

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

The three types are:

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

- a) ON/OFF auxiliaries are set when programmed. If the auxiliary is programmed ON it will remain ON until programmed OFF. Such auxiliaries could be used to, for example, control lubricant.
- b) MOMENTARY auxiliaries are switched ON (if programmed to be on) only when the machine is at a programmed position. When the axes are moving momentary auxiliaries are always OFF. This type of auxiliary can be used to, for example, provide a table locking signal or to activate a rotary table to index for a milling operation.
- c) PULSED auxiliaries provide a pulse output of 50 milliseconds (if programmed 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 programmed 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 programmed OFF. More than one auxiliary can be programmed off by repeatedly entering numbers. When the auxiliaries have been set up ON or OFF, as desired, pressing `<EOB>` will end that block of information.

SECTION 32

INPUT FACILITIES

The PNC 3 is equipped to be able to monitor seven 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 programmed state, sequence execution waits until the switch signals are as programmed before proceeding. Switch levels may be programmed 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 table, 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.

SECTION 33

DWELL FUNCTION

When a G04 dwell preparatory function is called up, a variable time delay from 0.1 to 9999.9 seconds can be programmed and signifies that no movement of the machine is to occur while this block is being performed.

It may be used for spot face cycle or boring cycle, example:- M03

G01 Z-20 F100

G04 2

G01 Z3 F150

In the example the tool travels 20 mm and dwells for two seconds before retracting.

G04 dwells can be programmed inside a program load sequence or outside the programmed sequence.

SECTION 34

EDIT FACILITY

The edit facility enables a program in memory to be edited. To select edit mode set the key for Man position and depress the Edit key. When edit mode is selected the user may choose to display any block of data. Key in the desired block number to be edited and depress enter key. Seven edit functions are permitted in edit mode.

They are:-

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

During edit mode three blocks will be displayed, the current block and the two previous blocks plus the system editor, along the bottom of the screen. The selected function is performed on the bottom block.

FUNCTIONS

1. Previous Depression of key 1 - Decrease the block numbers displayed by 1. Therefore each time key 1 is depressed the previous block will be displayed.
2. Next Depress key 2 - Increments the block numbers displayed by 1. Therefore each time key 2 is depressed the next block will be displayed.
3. Replace Depress key 3 - Replace function. The current block displayed can be replaced by a new block on the same line number. Keying in the new block and accepting the data with EOB key.
4. Delete Depress key 4 - The current block displayed will be deleted from the program and all block numbers will decrease by one.
Action All blocks will automatically be renumbered but block numbers, within a repeat block G81, are not updated.

5. Add Depress key 5 - Add a new block into the program. A new block will be inserted into the program on the current line number. The current block will move down one and all block numbers after the current block will automatically renumber.

If it is desired to add a block or a number of blocks to the end of a programmed sequence, the LOAD continue facility should be used.

Note Block numbers are automatically renumbered except those within a repeat G81 block.

6. Alter Depress key 6 - To alter the current block. This allows the user to alter existing data. Alter mode cannot change the status of a block, ie G01 cannot be altered into G02. If any attempt is made to alter data that is not permitted, the system will display "use replace function", the reset key will restore normal edit mode.

Feed notes cannot be altered directly as any one of the accompanying X, Y or Z data must be keyed in first before the feed can be accepted.

7. Search Depress key 7 - Block search. The system will display the number of blocks in the program. Key in the appropriate block number and depress enter and that block will be displayed.

8. RESET The reset key will cancel edit mode and return to normal operation.

SECTION 35

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, 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 five seconds, 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 five seconds to pass the clear leader. If the end of tape clear leader is detected during a cassette operation, eg 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, ie 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 to be 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 effort is made to record a program, the message "CASSETTE IS WRITE PROTECTED" will be displayed.

2. ERASE CASSETTE enables a cassette to be erased, ie cleared of programs. The cassette should first be rewound using option 1, ie rewind cassette. When a cassette is erased "PROGRAM END" is recorded at the start of the cassette to indicate that this is the end of the cassette. The cassette erase operation takes approximately three 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, ie to the message "Program end found", the cassette is then ready for other programs to be stored. Depress the RESET key, returns to cassette menu.
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, displayed in turn the numbers of the programs on tape which are found, until the program required is found or until the tape end is found. Depressing ENTER key will load the program into memory.

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 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 pre-recorded 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, ie 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 one to six numerals.

Each program is stored as four elements separated by blank tape.

- i) The cassette program identifier.
- ii) The program.
- iii) The tool offsets associated with the program.
- iv) 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 three 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.

SECTION 36

DATA LINK FACILITY

Four operation 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. Store program to data link. (Enhanced RS232C option only).
4. Print program, ie 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, ie destroyed. The format of the program data is shown in the RS232C interface specification.
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.
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".

When function 3 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 than the whole of the program in memory is transmitted via the RS232C link.

If 2. is selected than 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.

(NB: 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.

IE

< 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.

PNC 3

Standard PNC 3 RS232 Interface Specification

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 7, 6 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	54H	
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, ie 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 unrecognized character is received by the PNC 3 (see 2, above) 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 (to a maximum of 60 (for 2 axis), 80 (for 3 axis) characters):-

<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> <p> .

If the program block contains L as the last character (ie the character before <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.

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 25.7 mm/min and the Z axis would move at a feedrate of 100 mm/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 replace Y.)

X<d> Y<d> CCX<XCENTRE> Y<YCENTRE> for Clockwise Moves

or X<d> Y<d> CAX<XCENTRE> Y<YCENTRE> for Anti-Clockwise 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>

PNC 3 RS232 Interface Specification

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, ie, 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> rpm
FSF<n>	Spindle forward direction, <n> rpm
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 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)

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

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

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.

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 Ycccc

Counter-Clockwise

G03 Xnnnn Znnnn FXffff CXcccc Ycccc

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

ffff = Feedrate in mm/min or ins/min

cccc = XY 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

Machine Scale

G21 Sssss

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

Program Offsets

Mill

G54 Xnnnn Ynnnn Znnnn L

Lathe

G54 Xnnnn ZnnnnL

Machine Offsets

Mill

G55 Xnnnn Ynnnn Znnnn

Lathe

G55 Xnnnn Znnnn

9. Imperial Units

G70

10. Metric Units

G71

11. Repeat Loops

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

G99L

16. Program Stop

M00L

17. Spindle Speed Control

FOR UNITS WITH PROGRAMMABLE SPINDLE SPEEDS

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
eg

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

eg

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 (ie to clear PNC 3 memory)

E will erase the program

Section B Output from PNC 3

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.

PNC 3 RS232 ERROR CODES (Enhanced specification only)

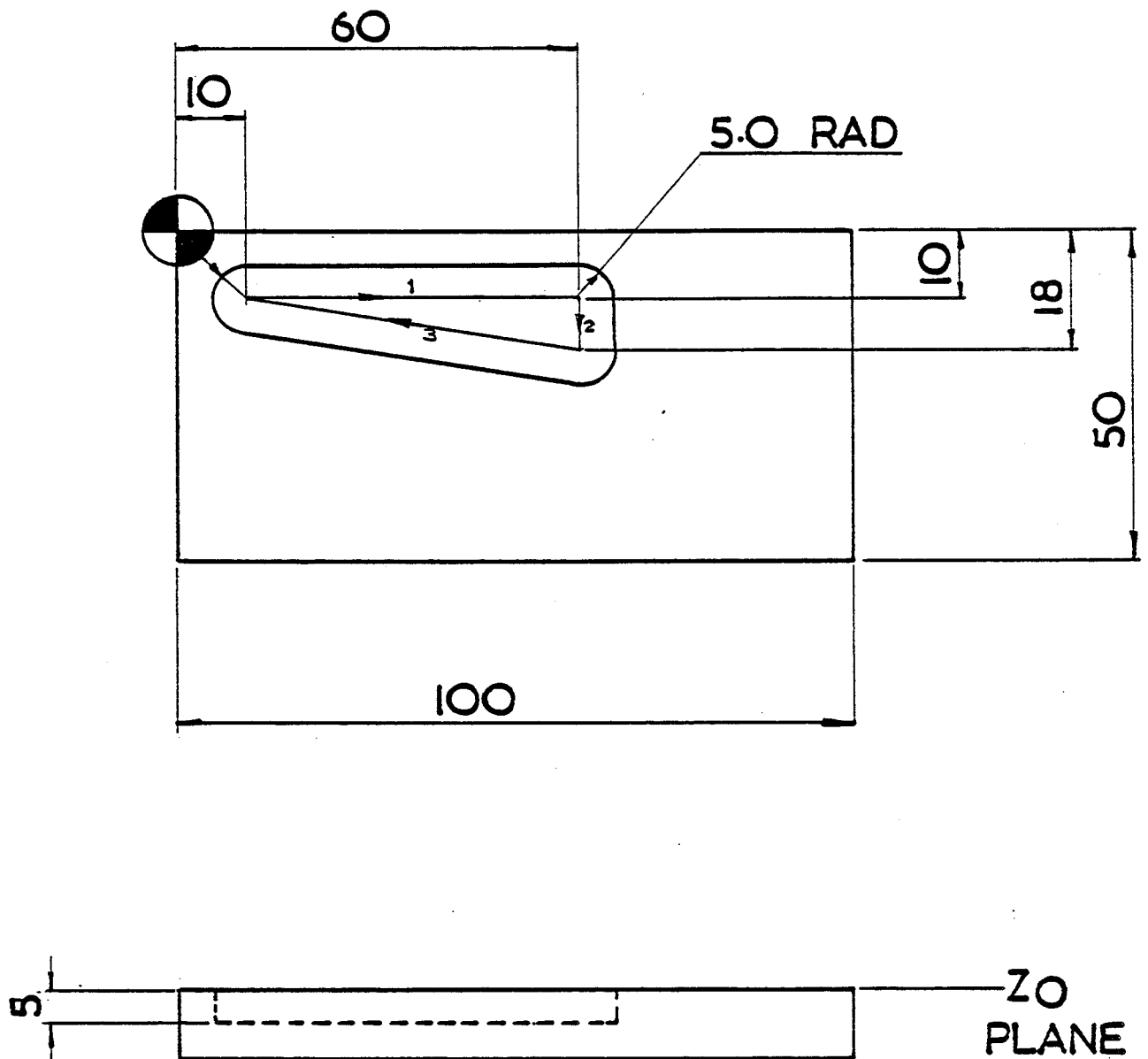
<u>Error No.</u>	<u>Meaning</u>
01	Not assigned
02	Illegal G code received
03	Illegal M code received
04	Illegal character for this block
05	Move exceeds machine limits
06	Block not completed successfully
07	This block not allowed to execute immediately (Must be ended with L)
08	This block not allowed 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 co-ordinate
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

<u>Error No.</u>	<u>Meaning</u>
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

SECTION 37

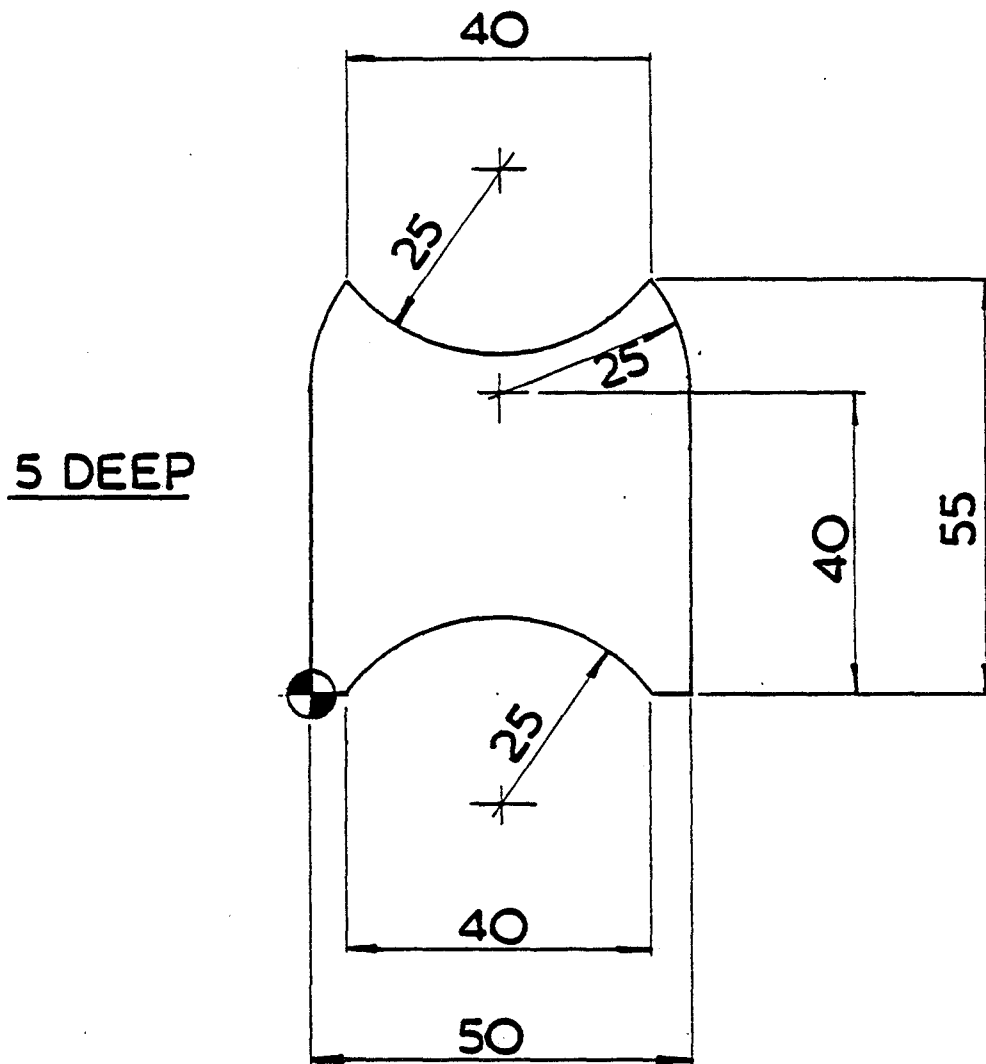
EXAMPLES

FIG 1



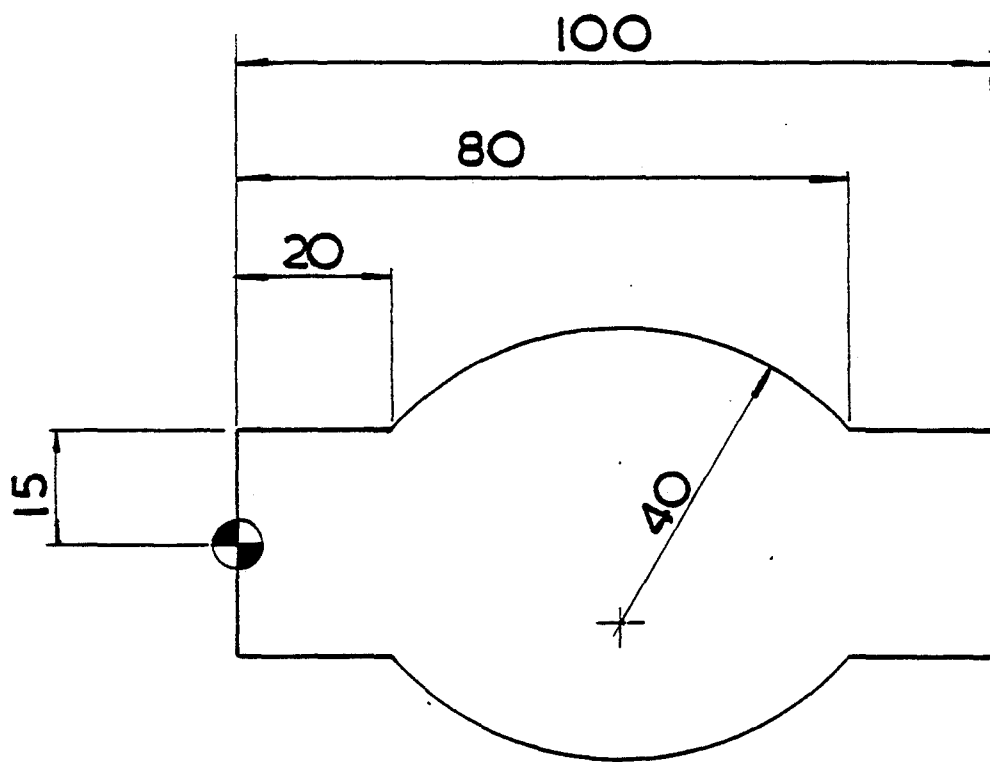
TOOL N° 1
10 DIA SLOT DRILL

FIG 2



⊕ DATUM POSITION G55 X 120 Y 50
TOOL N°1 6 DIA SLOT DRILL
Z0 DIMENSION = TOP OF WORKSURFACE

FIG 3



DRAWING No.

4

DESCRIPTION

PROGRAMME No.

4

MATERIAL

PROGRAMME BY

DATE



TOOLING & SETTING DATA

TOOL 1 = 14mm DIA.

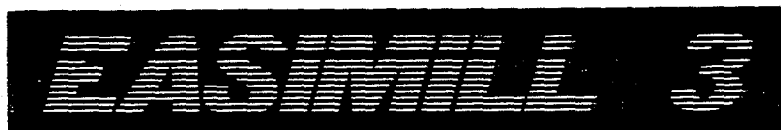
TOOL 2 = 12mm DIA. (USE SAME TOOL - BUT
INPUT DIFFERENT DIAMETERS INTO

MM & ABSOLUTE

CONTROL)

N	G	M	X	Y	XY FEED	CIRCULAR MOVEMENT		Z	Z FEED	REMARKS
						XC	YC			
01		06								TOOL 1
02	00		X 177	Y 110						RAPID TRAVERSE
03		03								SPINDLE FWD
04		08								COOLANT ON
05	00							Z 3		RAPID TRAVERSE
06	99									FLOATING DATUM
07	00		X -40	Y 45				Z 0		RAPID TRAVERSE
08	41									TOOL COMP LEFT
09	00		X -40	Y 45						RAPID TRAVERSE
10	01							Z -8	F 498	
11	01		X 0	Y 45	F 348			Z -8		
12	01		X 14.431							
13	02		X 31.750	Y 35		14.431	25			
14	01		X 46.187	Y 10						
15	02		X 48.869	Y 0		28.869	0			
16	02		X 46.187	Y -10		28.869	0			
17	01		X 31.750	Y -35						
18	02		X 14.431	Y -45		14.431	-25			
19	01		X -14.431							
20	02		X -31.750	Y -35		-14.431	-25			
21	01		X -46.187	Y -10						
22	02		X -48.869	Y 0		-28.869	0			
23	02		X -46.187	Y 10		-28.869	0			
24	01		X -31.750	Y 35						
25	02		X -14.431	Y 45		-14.431	25			
26	01		X 0							
27	01			Y 55	F 498					
28	40									CANCEL TOOL CO
29	00		X -40							RAPID TRAVERS.
30	81		FROM 7 TO 29 REPEATS 3			OFFSET Z	-5			
31		06								TOOL 2
32	81		FROM 7 TO 28 REPEATS 1			OFFSET Z	-15			
33	00							Z 0		RAPID TRAVERS
34	00		X 0	Y 0						RAPID TRAVERS
35	01			Y 26	F 79.9			Z -18	F 55.4	
36	01			Y 0						

RAWING No.	4
DESCRIPTION	
PROGRAMME No.	4
MATERIAL	
PROGRAMME BY	
DATE	



TOOLING & SETTING DATA
PAGE 2

MM & ABSOLUTE

N	G	M	X	Y	XY FEED	CIRCULAR MOVEMENT		Z	Z FEED	REMARKS
						XC	YC			
37	01		X 0	Y 4	F 252			Z -18		
38	02		X 4	Y 0		0	0			
39	02		X 0	Y -4		0	0			
40	02		X -4	Y 0		0	0			
41	02		X 0	Y 4		0	0			
42	20		200%							PROGRAM SCALE
43	81		FROM 37 TO 41 REPEATS 1							
44	20		300%							PROGRAM SCALE
45	81		FROM 37 TO 41 REPEATS 1							
46	20		400%							PROGRAM SCALE
47	81		FROM 37 TO 41 REPEATS 1							
48	20		500%							PROGRAM SCALE
49	81		FROM 37 TO 41 REPEATS 1							
50	20		600%							PROGRAM SCALE
51	81		FROM 37 TO 41 REPEATS 1							
52	20		650%							PROGRAM SCALE
53	81		FROM 37 TO 41 REPEATS 1							
54	20		100%							PROGRAM SCALE
55	00		X 0	Y 20						RAPID TRAVERSE
56	01		X 0	Y 28.8	F 102					
57	20		64%							PROGRAM SCALE
58	81		FROM 11 TO 26 REPEATS 1			OFFSET Z -2				
59	20		100%							PROGRAM SCALE
60	00							Z -3		RAPID TRAVERSE
61	01		X 0	Y 40	F 150					
62	20		88%							PROGRAM SCALE
63	81		FROM 11 TO 26 REPEATS 1			OFFSET Z 5				
64	20		100%							PROGRAM SCALE
65	00			Y 26						RAPID TRAVERSE
66	00							Z -6		RAPID TRAVERSE
67	01			Y 54	F 252					
68	00							Z 0		RAPID TRAVERSE
69	00		X 22.519	Y -13						RAPID TRAVERSE
70	00		X 22.519	Y -13				Z -6		RAPID TRAVERSE
71	01		X 46.762	Y -27	F 252					
72	00							Z 0		RAPID TRAVERSE

DRAWING No.	5
DESCRIPTION	MIRROR/REPEAT/TAPER 'Z'
PROGRAMME No.	5
MATERIAL	TOY STEEL
PROGRAMME BY	
DATE	



TOOLING & SETTING DATA

PAGE 2

TOOL 1 06 DRILL

TOOL 2 10 SLOT/DRILL

TOOL 3 08 SLOT/DRILL

G55 X150 Y00

N	G	M	X	Y	XY FEED	CIRCULAR MOVEMENT		Z	Z FEED	REMARKS
						XC	YC			
32	81	S18	E27	REP 1						(REPEAT) 3 HOLES
33	12		(MIRROR 'Y')							MIRROR Y (4)
34	81	S18	E27	REP 1						DRILL (REPEAT)
35	11		(CANCEL MIRROR 'X')							CANCEL MIRROR 'X'
36	13		(CANCEL MIRROR 'Y')							CANCEL MIRROR 'Y'
37	01							Z 10	F 1500	TO CLEAR WORK
38		05								SPINDLE OFF
39	01		X -60	Y -60	F 1500					T/CHANGE
40		06	T2							Ø10 SLOT/DRILL
41		03								SPINDLE ON
42	01					X*	X	Z 10	F 1000	TO GIVE 'Z'
										CLEARANCE
43	01		X -45	Y 65	F 1500	(G81.SbL)		Z 3	F 1500	COUNTER/BORE
44	01							Z -5	F 200	HOLES
45	01							Z 3	F 1000	
46	01		X -65							
47	01							Z -5	F 200	(1)
48	01							Z 3	F 1000	
49	01			Y 45						
50	01							Z -5	F 200	
51	01							Z 3	F 1000	
52	01		X 0	Y 0	F 1500					
53	12		(MIRROR 'Y')							(2)
54	81	S43	E52	REP 1						
55	13		(CANCEL MIRROR 'Y')							
56	10		(MIRROR 'X')							(3)
57	81	S43	E52	REP 1						
58	12		(MIRROR 'Y')							
59	81	S43	E52	REP 1						(4)
60	11		(CANCEL MIRROR 'X')							
61	13		(CANCEL MIRROR 'Y')							
62	01							Z 10	F 1000	(TO CLEAR WORK)
63		05								SPINDLE OFF
64	01		X -60	Y -60	F 1500					T/C
65		06	T2			(SAME TOOL 'Z' OK AT Z 70 WHERE FINISHED)				Ø10 SLOT DRILL

DRAWING No.	5
DESCRIPTION	MIRROR/REPEAT/TAPER 'Z'
PROGRAMME No.	5
MATERIAL	TOY STEEL
PROGRAMME BY	
DATE	



TOOLING & SETTING DATA

PAGE 3

TOOL 1 06 DRILL

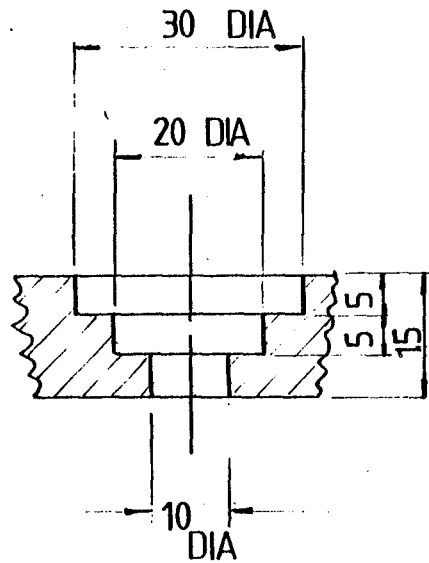
TOOL 2 10 SLOT/DRILL

TOOL 3 08 SLOT/DRILL

G55 X150 Y00

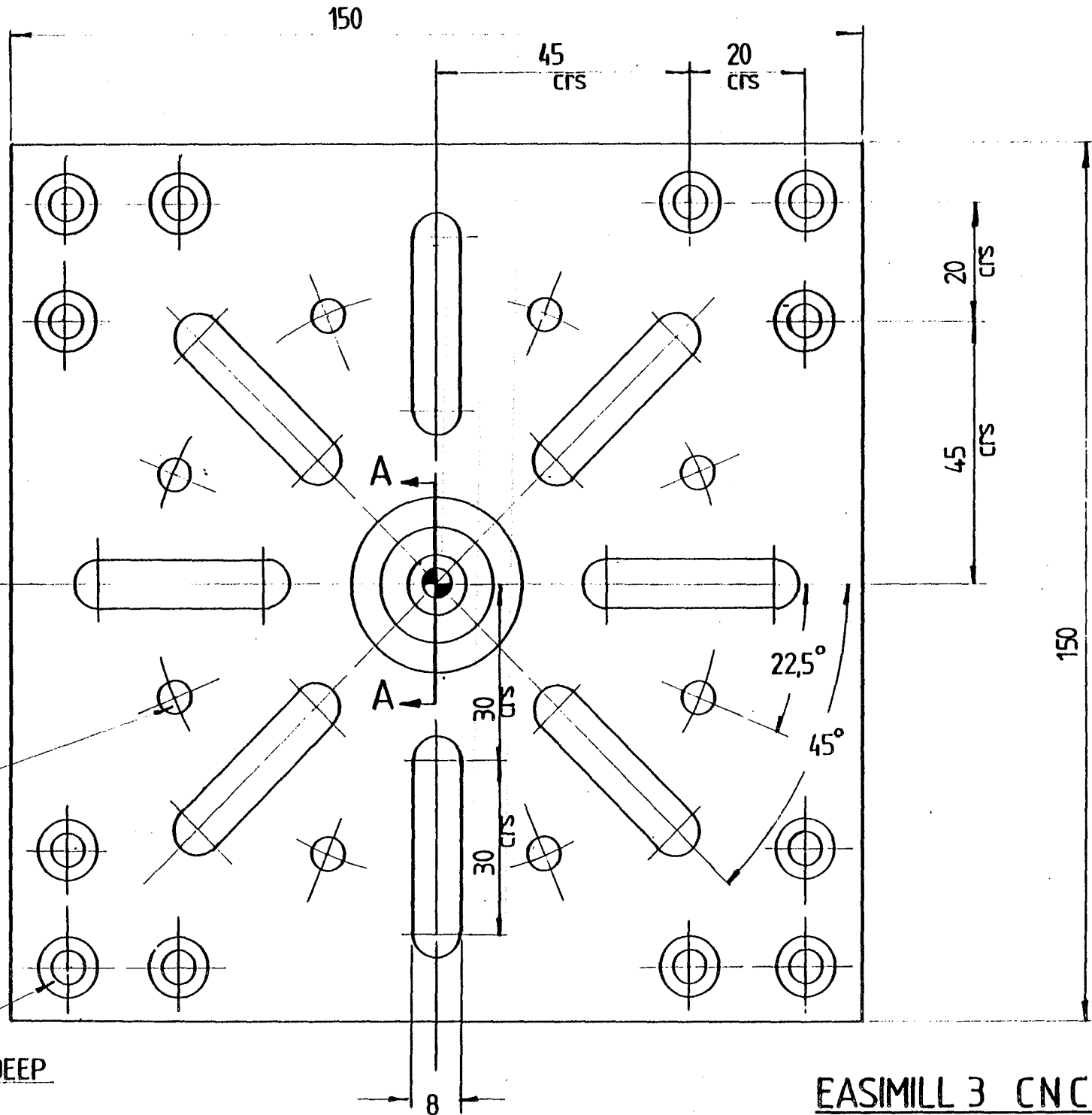
N	G	M	X	Y	XY FEED	CIRCULAR MOVEMENT		Z	Z FEED	REMARKS
						XC	YC			
66		03								
67	01		X 0	Y 0	F 1500					CENTRE HOLE Ø1
68	01							Z -15	F 200	
69	01							Z 3	F 1000	
70	41		(CUTTER COMP LEFT)							
71	01		X -10	Y 0	F 500					
72	01							Z -10	F 200	
73	03		X 0	Y -10	F 200	0	0			Ø20 BORE
74	03		X 10	Y 0		0	0			
75	03		X 0	Y 10		0	0			
76	03		X -10	Y 0		0	0			
77	40		(CANCEL CUTTER COMPENSATION TO DO NEXT BORE)							
78	01							Z 3	F 1000	TOOL UP
79	41		(CUTTER COMPENSATION FOR NEXT BORE)							(COMP LEFT)
80	01		X -15	Y 0	F 500					
81	01							Z -5	F 200	
82	03		X 0	Y -15	F 200	0	0			BORE
83	03		X 15	Y 0		0	0			Ø30X
84	03		X 0	Y 15		0	0			50P
85	03		X -15	Y 0		0	0			
86	01							Z 10	F 1000	
87	40		(CANCEL CUTTER COMPENSATION)							
88		05								
89	01		X -60	Y -60	F 1500					T/CHANGE
90		06	T3							TOOL 3 Ø8 SLO.
										DRILL
91								Z 10	F 1000	TO CLEAR WORK
92		03				X	X			SPINDLE ON
93	01		X 0	Y 60	F 1500	(G8 SbL XYZ)		Z 3	F 1500)	
94	01							Z 0	F 200)	SLOT 1
95	01		X 0	Y 30	F 200			Z -8	F 55)	
96	01							Z 3	F 1000)	
97	01		X 0	Y 0	F 500					
98	12		(MIRROR 'Y')							
99	01	S93	E97	REP 1						(SLOT 3)
100	13		(CANCEL MIRROR 'Y')			X	X			

SECTION A-A



8 HOLES DRILL 6 DIA
10. DEEP ON A 100 P.C.D

12 HOLES DRILL 6 DIA - 10 DEEP
C/BORE 10 DIA - 5 DEEP



EASIMILL 3 CNC

PROGRAM NO 5