5. OPERATION OF STARTURN PC

This operation section of the manual is intended to enable a competent turner to operate the machine effectively.

In the control description section(Page 5.5 to 5.8), each button is identified in turn together with a brief description of its function.

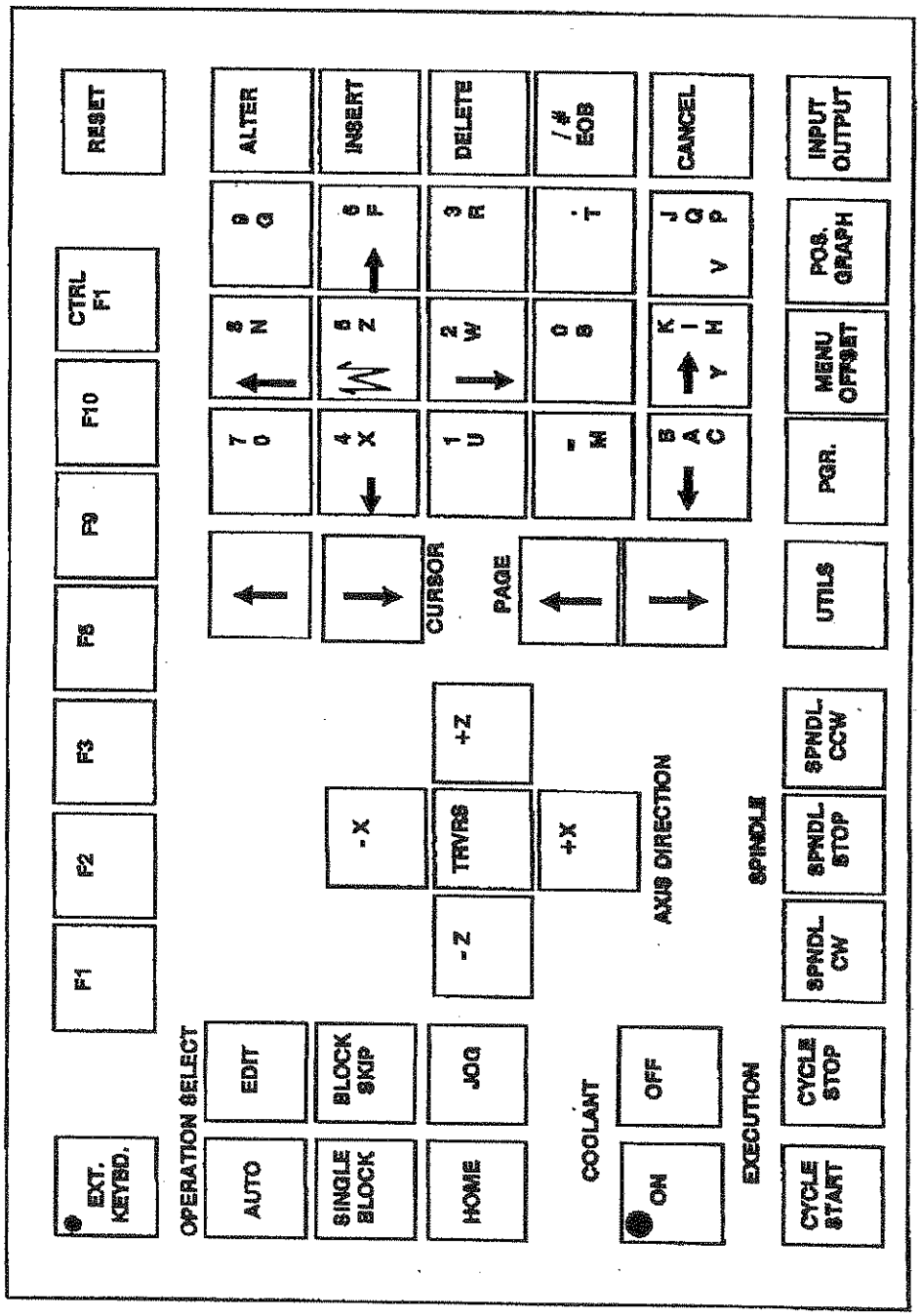
This section also covers the type of tooling and chucking used on STARTURN PC and describes how to mount both tools and chucks.

The tailstock is also described for those STARTURN PC'S which have one fitted.

The final part of the section details the correct procedure for starting up STARTURN PC. The operator should ensure that he/she reads the start up procedures thoroughly and adheres to them whenever turning on the machine.

- When indexing the turret head, ensure it is clear of the chuck,

STARTURN PC OPERATION PANEL



STARTURN PC M.D.I.PANEL(DRAWING ON PREVIOUS PAGE)

KEY TO MANUAL DATA INPUT PANEL ON PREVIOUS PAGE

- RESET - Resets any alarm messages. Resets program to start in edit mode.
- ALPHA/NUMERIC PAD - Inputs characters expected by controller when inputting program. Multi- character keys toggle between characters shown.
- CURSOR - Moves cursor through program, element by element, in defined direction.
- PAGE - Moves cursor through program, page by page, in defined direction.
- SELECTION KEYS EACH HAVING SEVERAL PAGES:-
- UTILS - Toggles between directives.
- PRG - Selects mode, simulate only, edit only, or edit and simulate.
- MENU OFFSET- Toggles between M.D.I. and Tool setting.
- POS. GRAPH - Selects simulate, edit, and M.D.I.
- INPUT OUTPUT - Automatically loads remote device link menu. This menu allows the user to send or receive from external peripherals.

EDITING KEYS

- ALTER - Alters addresses.
- INSERT - Inserts addresses,also used to initialise new programs.
- DELETE - Deletes addresses.
- / ; # E.O.B. - characters shown are toggle EOB operated when editing.
- CANCEL- Cancels an address, before insert is executed.

OPERATION SELECT

- AUTO - Select to run program.
- EDIT - Select to edit program.
- SINGLE BLOCK - Allows single step execution of program.
- BLOCK SKIP- Select in edit mode to ignore block when running program (Activates \ in front of block).
- HOME - Zeros machine around its own reference points.
- JOG - Moves axes around at feeds as set on override. When in manual mode, moves axes at 0.01, 0.1, and 1 increments and continuous feed.

EXECUTION

- CYCLE START - Starts program.
- CYCLE STOP - Stops program.

AXIS/DIRECTION - PRESS EITHER JOG OR MPG TO OPERATE.

- -X - Movement in -X direction.
- +X - Movement in +X direction.
- -Z - Movement in -Z direction.
- +Z - Movement in +Z direction.
- TRVRS - Rapid Traverse (toggle switch).

SPINDLE

- CW - Spindle movement clockwise.
- STOP - Spindle Stop.
- CCW - Spindle movement counter clockwise

DATUM OF AXES

DATUM AXIS AS FOLLOWS:-

- Press HOME button on the Operations Panel.
- Press "+ X" button on Operations Panel and wait for Cross slide to come to a stand still.
- Press "+ Z" button on Operations Panel and wait for saddle to come to rest
- Press the JOG button on the Operations Panel.
- Ensure the X,Z values read ZERO.
- Pressing the +X , -X, +Z, or -Z Keys and the TRVRS Key simultaneously will move the slides at rapid traverse speed.
- The machine is now ready for use.

NOTE:-

If X and Z values on machine position are not ZERO,ring DENFORD SERVICE DEPARTMENT for advice.

RUN IN PROCEDURES

Before operating STARTURN PC for the first time, it is important that the spindle is "RUN-IN"; for a total period of 45 minutes, broken down as follows :-

This is carried out by following the procedure below :-

1. Datum the machine as described above.
2. Press EOB.
3. Press S500 EOB (A speed of 500 rpm has now been entered).
4. Press SPNDL CW on MDI Panel (the spindle will run at 500 rpm clockwise).
5. After 15 min, change the spindle speed to 1000 rpm,by pressing S1000 EOB.
6. After 10 min,change the spindle speed to 2000rpm, by pressing S2000 EOB.

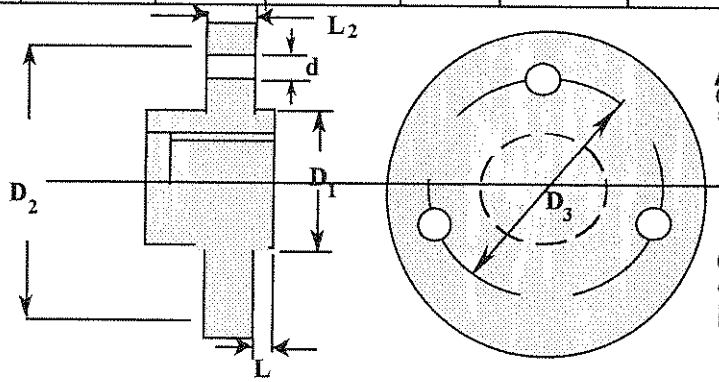
7. After 10 mins,change the spindle speed to 3000 rpm, by pressing S3000 EOB.

8. After a further 10 min, stop the spindle by pressing SPINDLE STOP on the MDI panel

MANUAL CHUCK INFORMATION

Units mm

TYPE		INTERNAL JAW RUNOUT		EXTERNAL JAW RUNOUT		FACE AND PERIPHERAL RUNOUT	GRIPPING FORCE		MAXIMUM UNBALANCE AT PERIPHERAL DIAMETER
SC (OD)	SK (OD)	DIA OF TEST ROUND BAR	ACCURACY	TEST DISC	ACCURACY		PINION TORQUE	G.F. OF 3 JAWS	
inch mm	inch mm			OD Width			Kg .m	Kg	g
3 (85)		8, 10, 15.	A	60 20	B	D	3	900	
4 (110)		8, 10, 20.		80 20			4.5	1200	11
5 (130)		10, 15, 25.		100 20			6.5	1500	
6 (165)	6 (165)	10, 20, 30.		125 30			9.0	2100	
7 (190)	7 (190)	10, 20, 30.		145 30			11.0	2400	21
9 (232)	9 (232)	20, 30, 50.		170 30			15.0	3000	
10 (273)	10 (273)	20, 40, 50.		205 40			18.0	3600	
12 (310)	12 (310)	30, 50, 70.		235 40			21.0	3900	
14 (355)		30, 50, 70.		275	C		23.0	4000	42
16 (405)		50, 70, 90.		315			25.0	4500	



KEY

A When the round test bar is clamped by the master pinion (arrow marked), the accuracy should be within 0.030 mm T.I.R. at each position of 100mm from the root (50mm in the case of the test bar being 8 dia. or 10 dia.). In the case that it is clamped by the other pinions, the accuracy should be 0.050mm T.I.R.

B Peripheral run out of test disc - within 0.050mm Face run out of test disc within 0.020

C Peripheral run out of test disc - within 0.05 mm. Face run out of test disc within 0.03mm

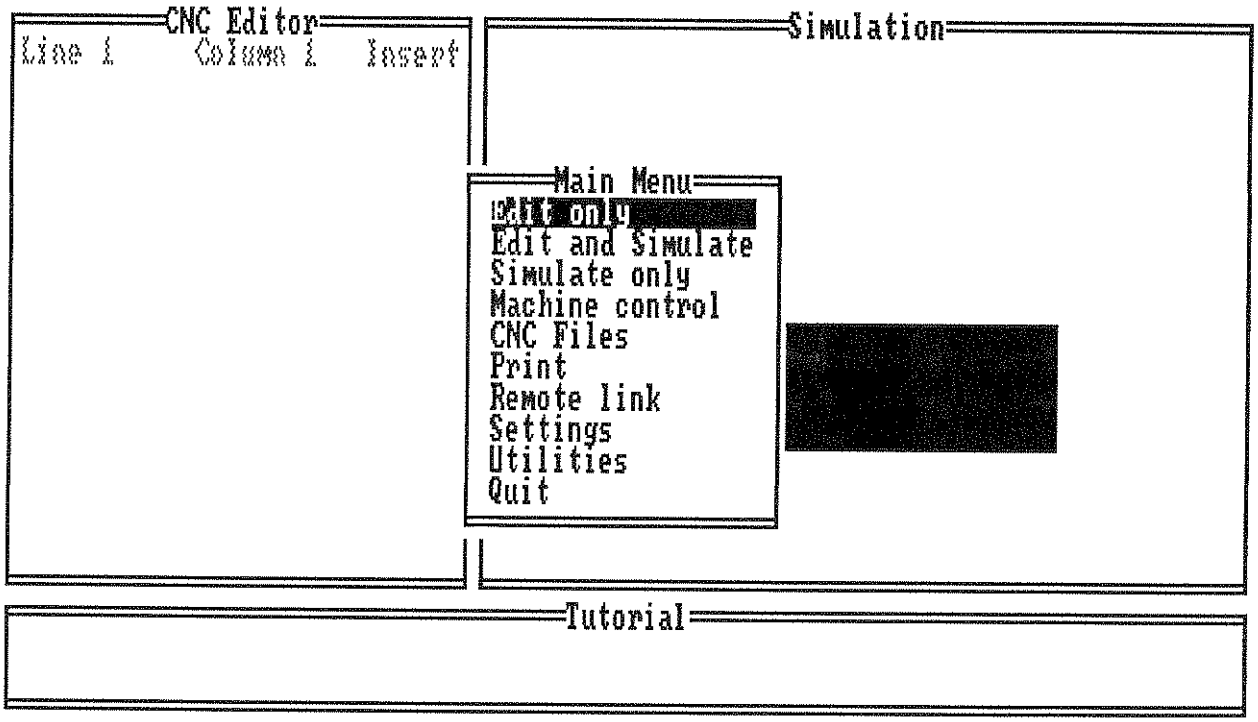
D Peripheral run out of test disc- within 0.020mm. Face run out of test disc within 0.020.

Type		D1		D2		D3	L1	L2	Mounting Bolt		Max. dia gripped by external jaw		Allowable Max. R.P.M.
SC	SK	Pilot	Tolerance	P.C.D.	Tolerance				d	Thread	SC	SK	
3		60	+0.021 +0.002	73	±0.2	88	3	10	6.6	3 - M6	inch mm 2.75 70	inch mm	3000
4		80	"	95	"	115	4	12	9.0	3 -M8	3.75 95		2500
5		100	+0.025 +0.003	115	"	135	4.5	12	9.0	3 -M8	4.33 110		2500
6	6	130	+0.028 +0.003	147	"	170		15	11.0	3 - M10	5.70 145	5.90 150	2000
7	7	155	"	172	"	195	4.5	18	11.0	3 - M10	6.5 165	6.89 175	2000
9	9	190	+0.033 +0.004	210	"	235	5.5	20	13.0	3 - M12	7.87 200	8.67 220	2000
10	10	230	"	250	"	275	5.5	20	13.0	3 - M12	9.45 240	9.66 245	1800
12	12	260	+0.036 +0.004	285	±0.3	310	6.5	22	13.0	3 - M12	10.83 275	11.22 285	1800
14		300	"	328	"	355	6.5	26	13.0	6 - M12	12.41 315		1500
16		345	+0.040 +0.005	375	"	405	7.5	26	13.0	6 - M12	14.17 360		1500

SC = type with solid hard jaw, with plain back. SK = type with 2-piece hard jaw, plain back.

6. MAIN MENU

PRESSING "F10" AT ANY TIME WILL ACTIVATE THE MAIN MENU.



EDIT ONLY

Displays full screen editor,with 241 character sideways scrolling facility.

Simulation is not available within this function but pressing the F9 key will run a syntax check on the CNC code.

EDIT AND SIMULATE

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

SIMULATE ONLY

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

MACHINE CONTROL

Switches to theStarturn 5 control, allowing tool set-up and machining of your CNC program.

CNC FILES

A very powerful Filing system is integrated within the software. The following points should be noted when using an edit window for Loading, Saving or Listing Files.

FILE NAME WINDOW FEATURES.

a)To edit the window, Press the ALTER Key or Type any number with the cursor positioned at the far left.

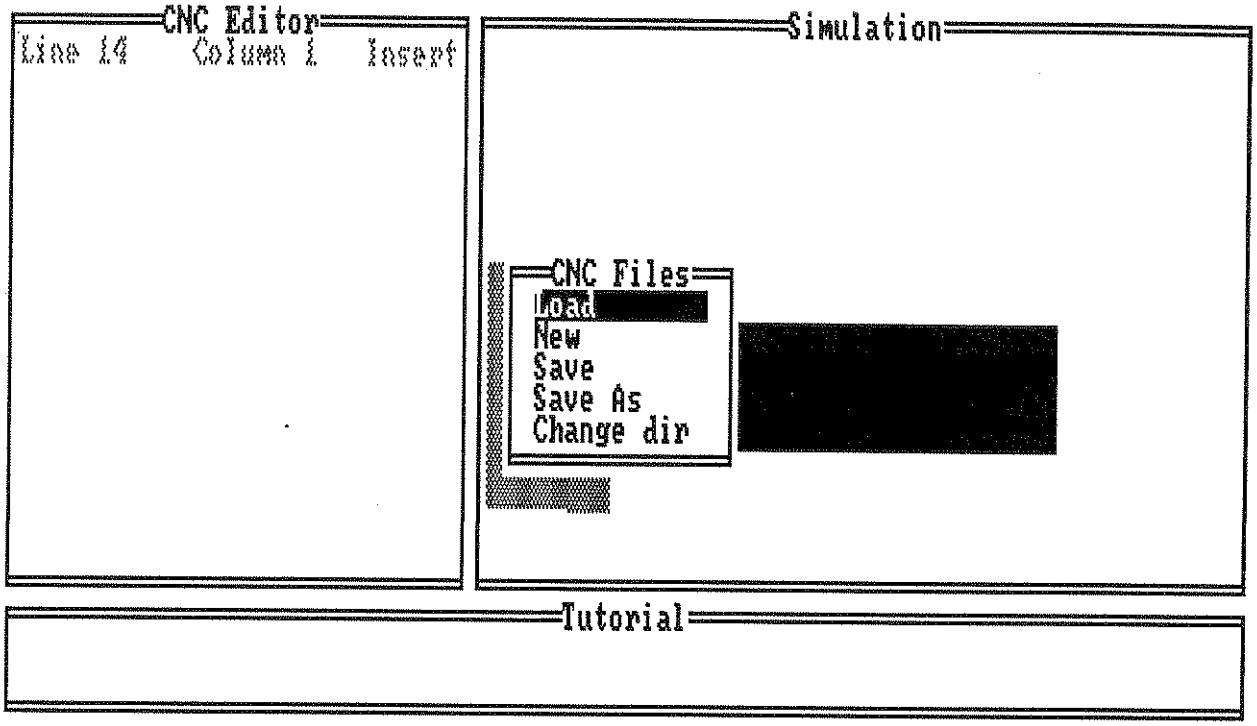
b) Use the cursor keys to position the cursor.

c) To list all files, press ALTER followed by EOB.

d) Press EOB to confirm or RESET to Exit.

e) All Files are listed alphabetically in a window.

f) Use the Cursor UP/DOWN and Page UP/DOWN keys to select a file.



LOAD

If there is a program currently in the Editor, you will be asked whether you wish to merge the program from Disk. Press the "Y"key to merge and the "N" key to clear the current program from the Editor and to load the selected program from Disk.

Note: F3 can be used as a quick- load fast key. No merge available with F3 key.

NEW

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

SAVE

Saves the current program to Disk in its default directory and with its default

filename. If the program has no name, then you will be prompted to enter a name. The suffix assigned to the filenames will be ".FNC".

If you want to save to Disk and Input a filename, do not select this option.

Note: F2 can be used as a quick save fast key.

SAVE AS

Prompts for a numerical filename which can be entered in the Input Window. Press the EOB key to confirm.

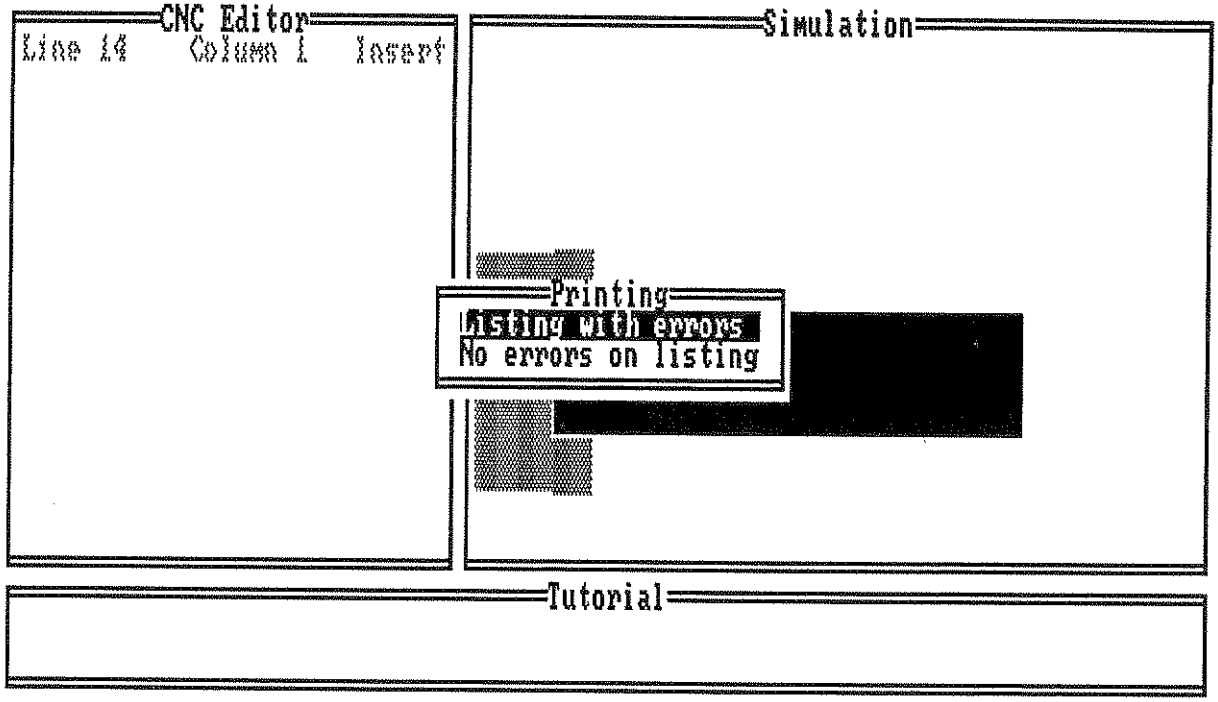
CHANGE DIR

Sets the current Directory for saving CNC programs.

First select the drive you wish to work with, then select the Directory you want to work in.

NB. Be shure to save all your programs in a personal directory.

7. LINKS


CNC Editor
Line 14 Colum 1 Insert
Simulation
Printing
Listing With errors
No errors on listing
Tutorial

PRINT

Allows you to get a paper copy of your program in paginated form.

The layout for the copy can be set in the Settings Menu under "Print Page Layout".

a)Listing with errors

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

b)Listing without errors

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

Note: Errors can be checked with the "Dry Run" option, but a printout with errors can be useful for certain programs.

PRINTING ERRORS.

1 - Printer does not respond

Remedy:-

Check the cabling between the computer and the printer.

Is the printer set for Parallel or Serial communications?

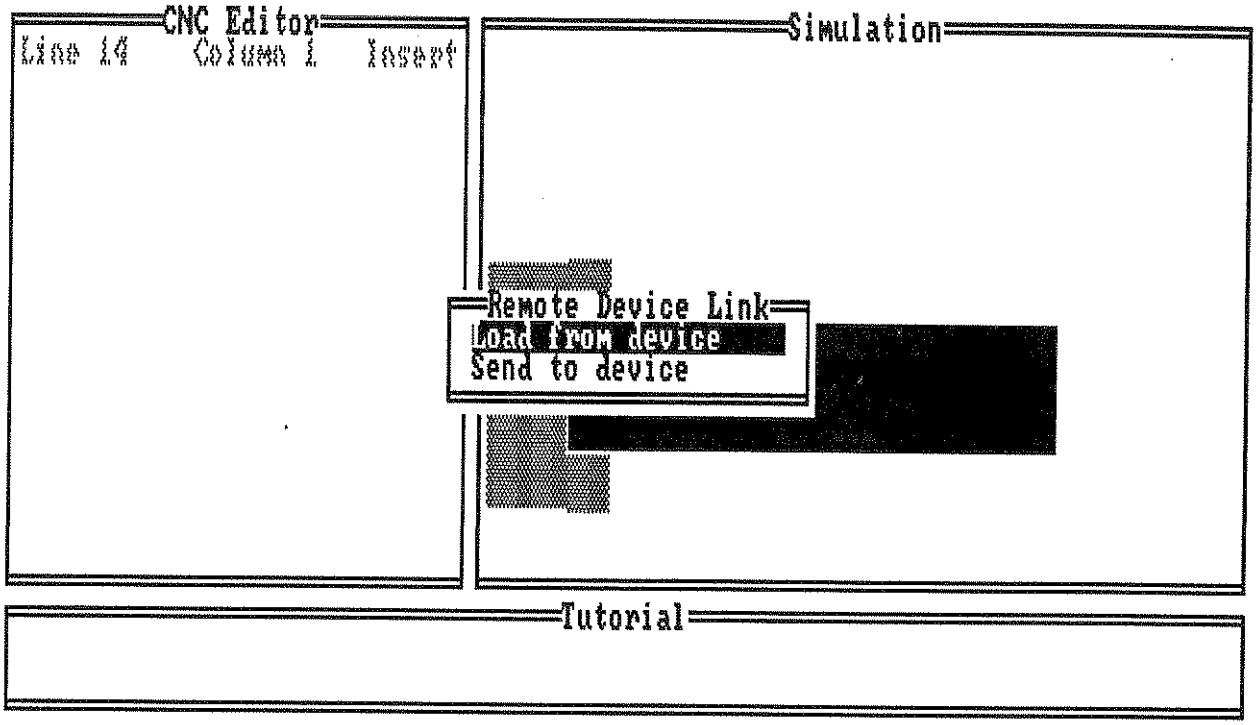
Have you set the correct parameters in the Settings Menu for"Print Device".?

Check that the printer is switched on and there is enough paper available for the printout.

2 - Page Layout Incorrect

Remedy:-

Any problems with page layout, linefeeds, or paper widths can be altered by selecting the Page Layout option from the Settings Menu.



REMOTE DEVICE LINK

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

LOAD FROM DEVICE

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

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

SEND TO DEVICE

You will be prompted with "Ready To Send?". Press the "Y" key to send or the "N" key to abort.

During transmission a "Transmitting to Device" message with the number of bytes and lines sent will be displayed in a window. A "Transmission Completed" message confirms the whole program has been sent. Press the RESET key to clear the transmit window.

COMMUNICATIONS ERRORS

Check that the Settings options for Remote Link are set correctly, if so then check the following points:

a) The Cable is located in the Port at the side of the control box and at the back of the Remote Device. A 25pin RS232 cable is required to link the machine.

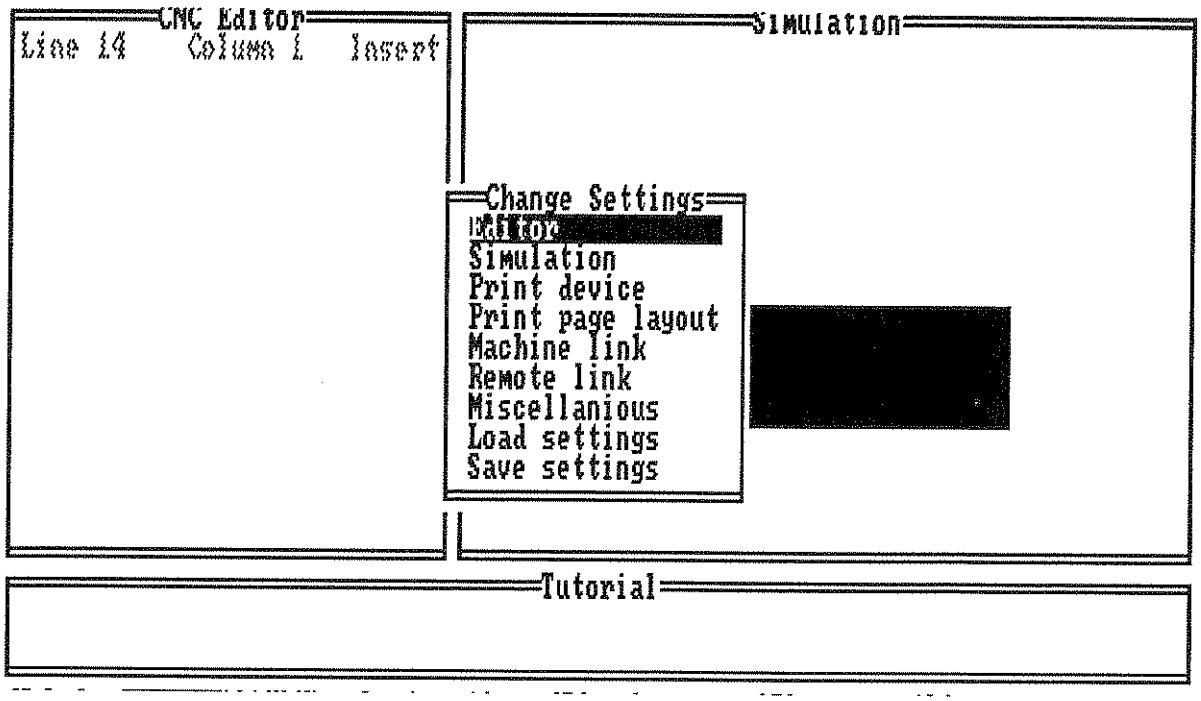
b) Whether the cable connections are faulty.

c) That the Remote device is set "Ready To Receive" or "Send", before sendingor receiving Part Programs. Check the Device Manual for setting"Ready to Receive" if you are not sure how to do it.

d) Ensure that the cable is plugged into the correct Port of the Computer.

NOTE:- The RS232 connector on the side of the CONTROL BOX is named RS232

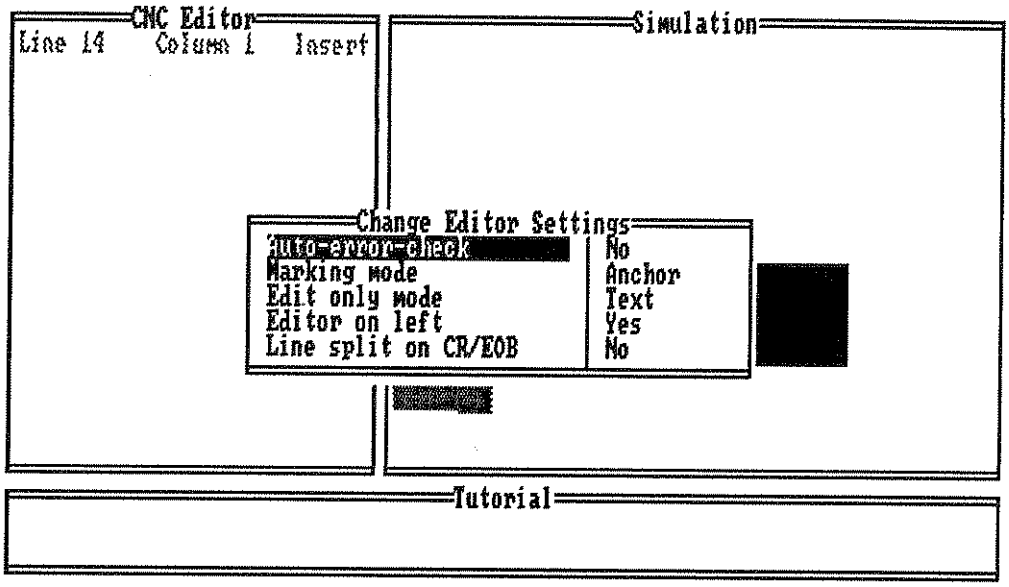
8. SETTINGS



There are many Settings within the software which allow customisation to suit the end user. Once you have set all the options, remember to save the Settings to Disk.

By selecting EDITOR from the SETTINGS MENU the CHANGE EDITOR SETTINGS

MENU comes up on screen.



AUTO-ERROR-CHECK

Toggles automatic error checking of each program line, as it is entered on or off. With error checking on, moving from a line will display a description of any error. Press the RESET key to clear and the cursor will highlight where abouts on the line the error occurred.

MARKING MODE

Toggles the marking mode between Anchor or Drag.

NOTE:- This option has no effect on the STARTURN PC machine and is only displayed for compatability with other DENFORD software products.

EDIT ONLY MODE

Toggles between Text or Graphics Mode when the Edit Only option is selected from

the Main Menu. The only advantage of selecting Text Mode over the Graphics Mode is a slight speed difference when scrolling. If Edit and Simulate is selected, then Graphics Mode is set automatically.

EDITOR ON THE LEFT

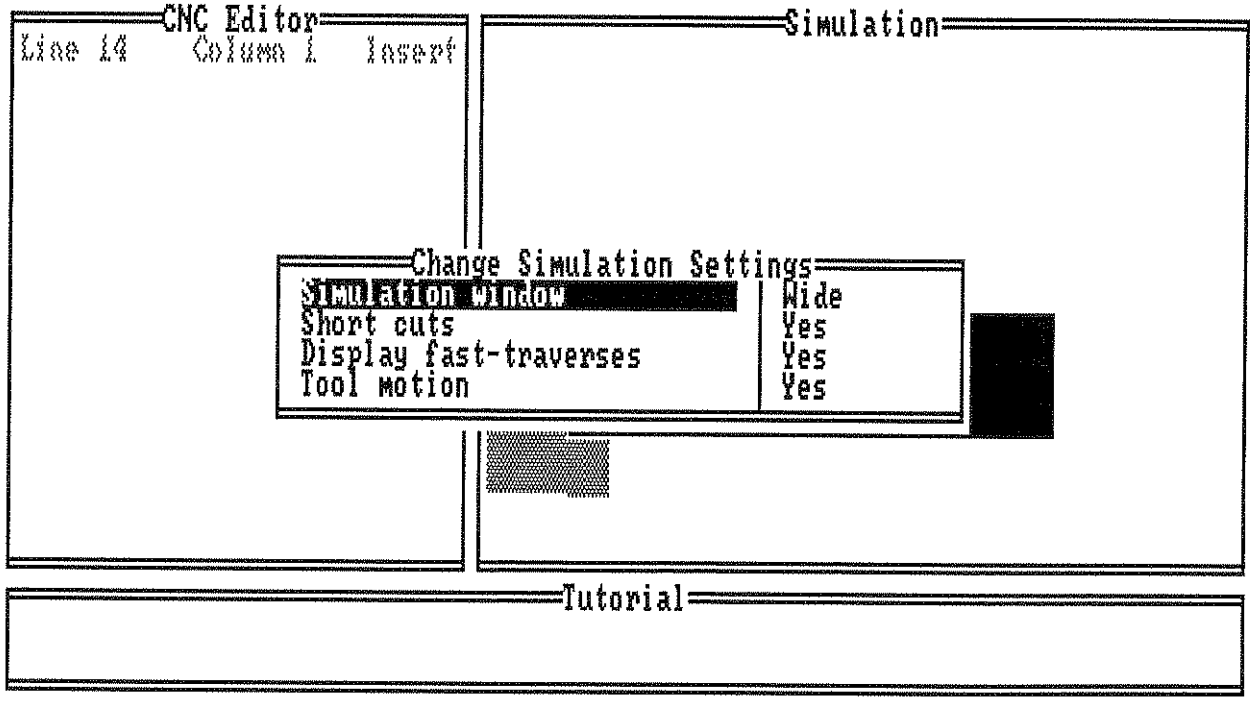
If you prefer the Editor on left and the Simulation on the right, (it can be toggled to either position), these can be toggled to the preferred position.

LINE SPLIT ON CR/EOB

Splits the line at the cursor position after the Return key if set to ON.

NOTE:- Inefective on STARTURN PC compatability.

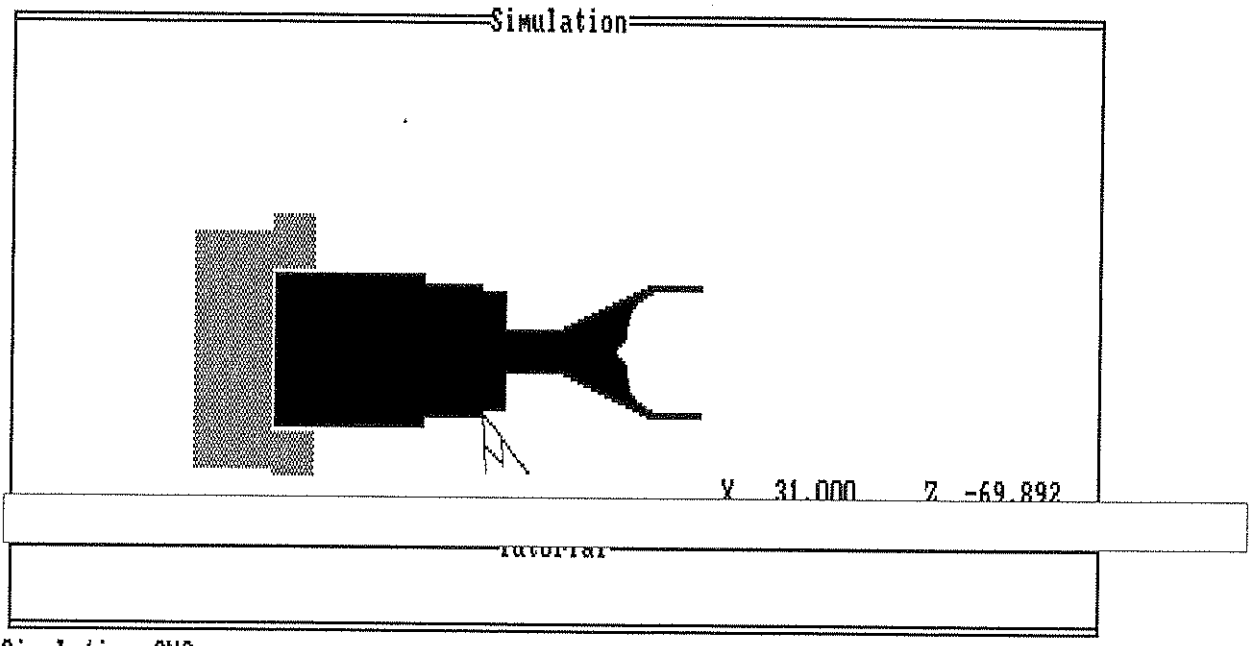
CHANGE SIMULATION SETTINGS



SIMULATION WINDOW

Switches between a normal sized simulation window to a wider one. Displays a larger view of the Simulation, if the Wide option is selected - see screen on following page(8.5).

EXAMPLE SIMULATION ON WIDER WINDOW SETTING



SHORT CUTS

This option is active when the tool motion is set to ON. This then speeds up the simulation by not displaying all Tool Movements, when SHORT CUTS is selected.

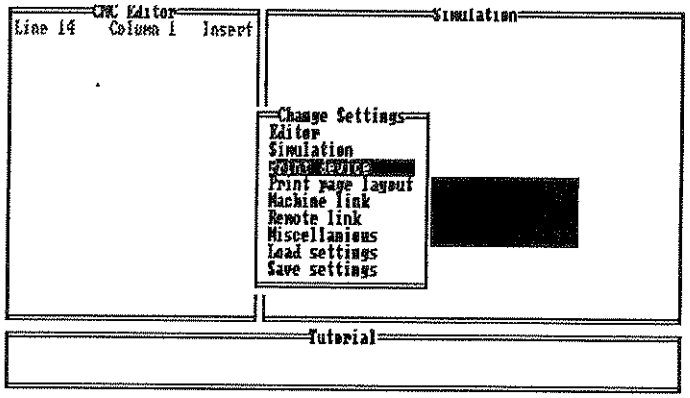
DISPLAY FAST TRAVERSE

Toggles Fast Traverse display ON and OFF. If Fast Traverse is set to ON, the software displays dotted lines.

TOOL MOTION

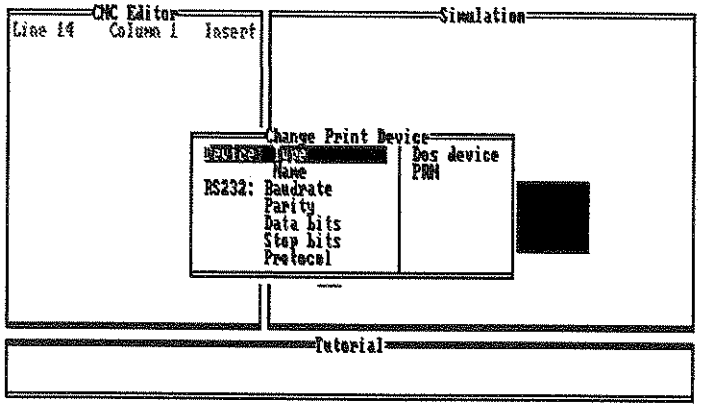
Toggles Tool Display ON and OFF. If it is set to ON, the Tool Shape is displayed. If set to OFF, a Toolpath Plot only is displayed.

9. PRINT DEVICE



CHANGE PRINT DEVICE

DEVICE : TYPE This can be set to :-



DOS DEVICE

A DOS device is normally the Parallel Port ,which can be set to LPT1, LPT2 or PRN. Select this option, if your Printer has a Parallel Port.

SERIAL

There are several settings required with the serial Port.

Baudrate

Parity

Data bits

Stop bits

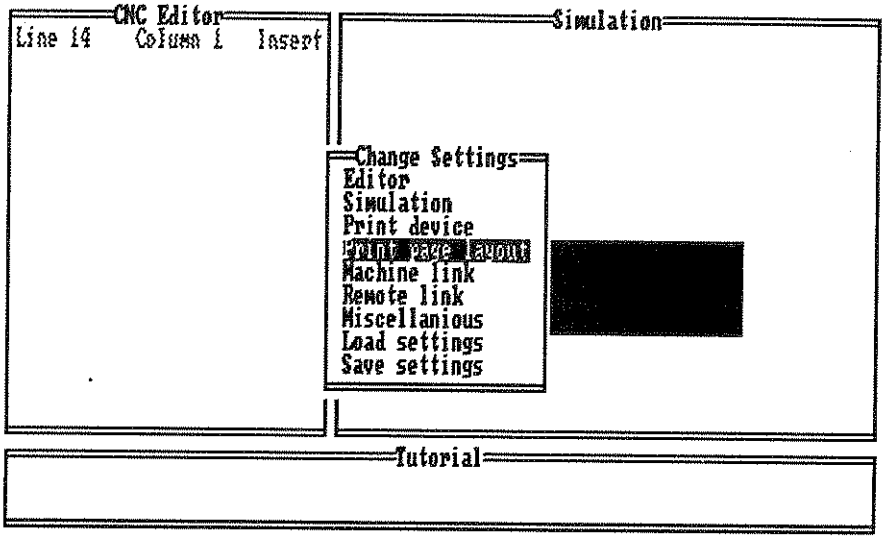
Protocol

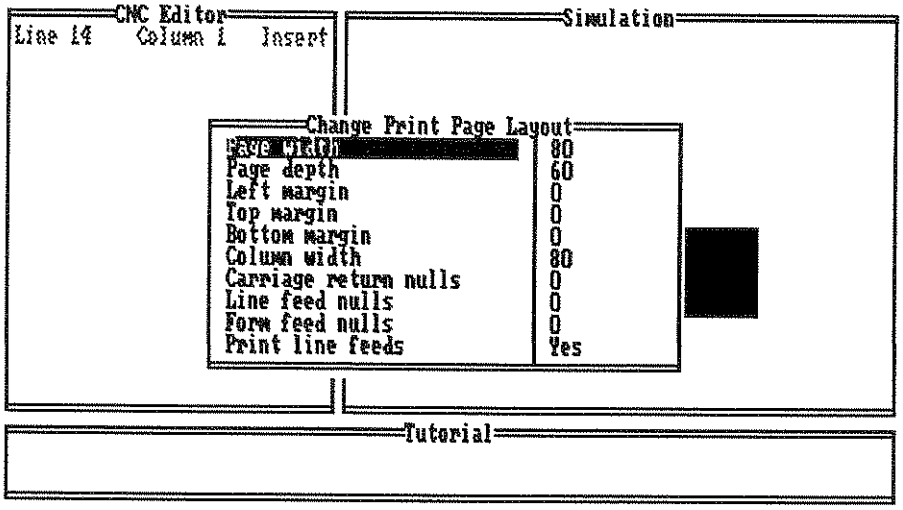
To change any of the above parameters, make sure that the Device is set to Serial. Highlight the option with the cursor keys, and press the RETURN key to scroll between the different options.

FILE

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

CHANGE PRINT PAGE LAYOUT





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

Page width

Page length

Left margin

Top margin

Bottom margin

Column width

Carriage return nulls

Line feed nulls

Form feed nulls

Print line feeds

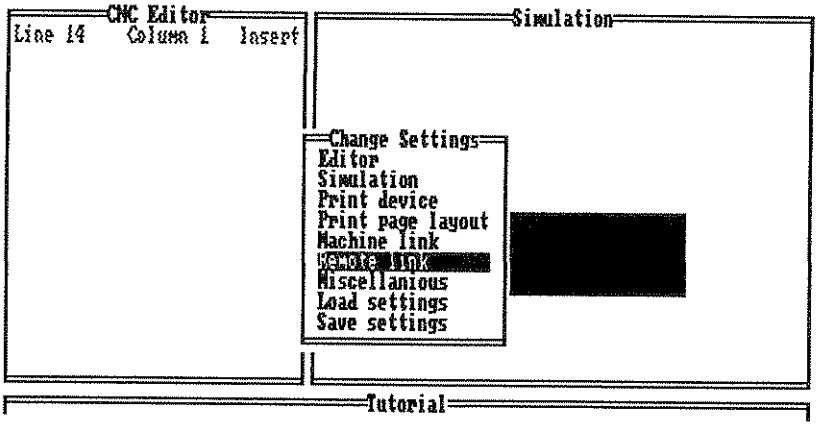
Line feeds and form feeds depend on the Printer's settings. If you are not sure, try different options until a correct printout is obtained.

10. CHANGE SETTINGS

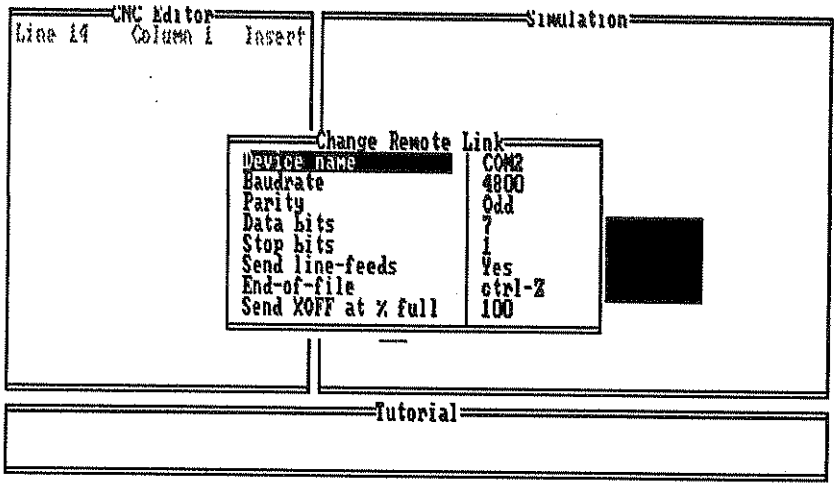
CHANGE MACHINE LINK (The machine link is always through the SERIAL PORT)

This option is ineffective on the STARTURN PC and should therefore never be altered.

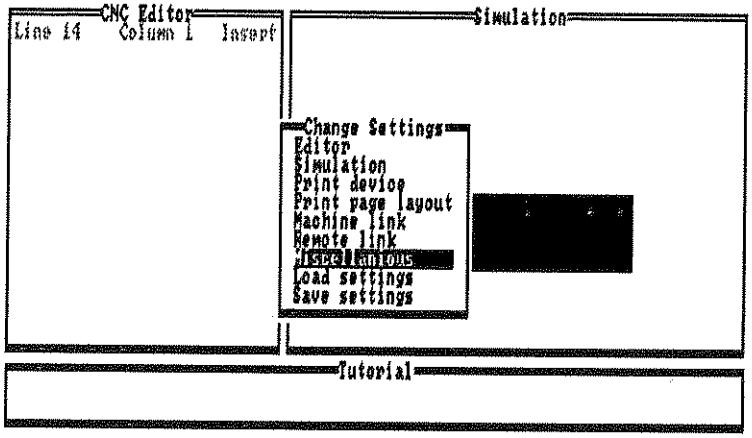
REMOTE LINK

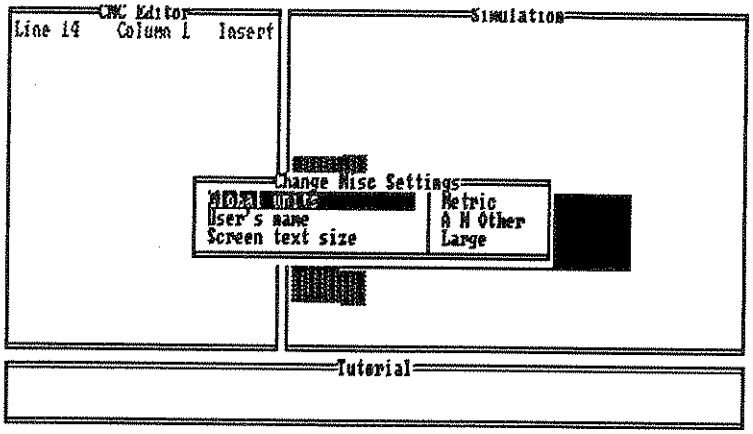


The settings for the remote link are for the Serial Port. This sets the protocol when communicating to an external device, such as a remote computer or paper tape punch.



CHANGE MISCELLANEOUS SETTINGS





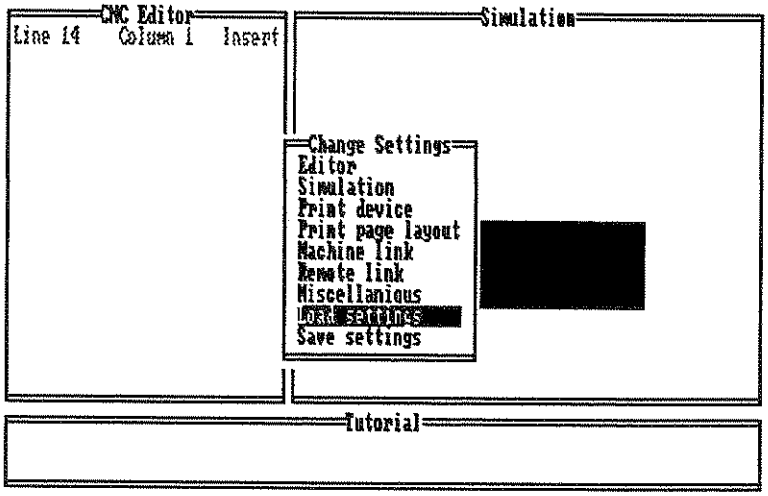
GLOBAL UNITS - USERS NAME - SCREEN TEXT SIZE

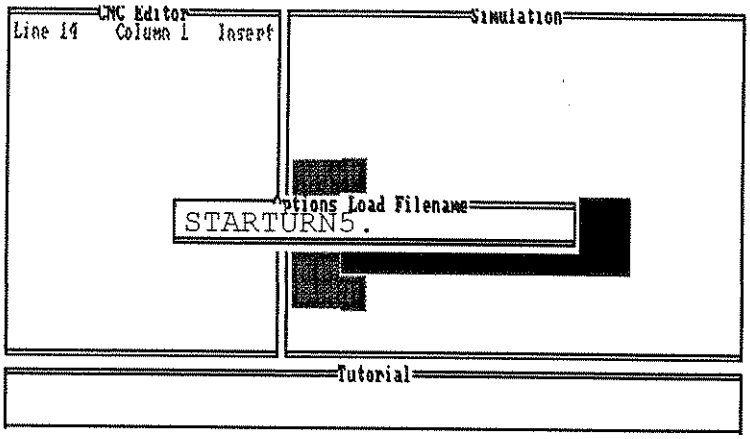
Global Units toggles between Imperial or Metric programming as the Default setting.

If the User's name is used, it will be printed off on all CNC program printouts.

The Screen Text Size can be toggled between 25 lines or 43 lines on screen.

LOAD SETTINGS

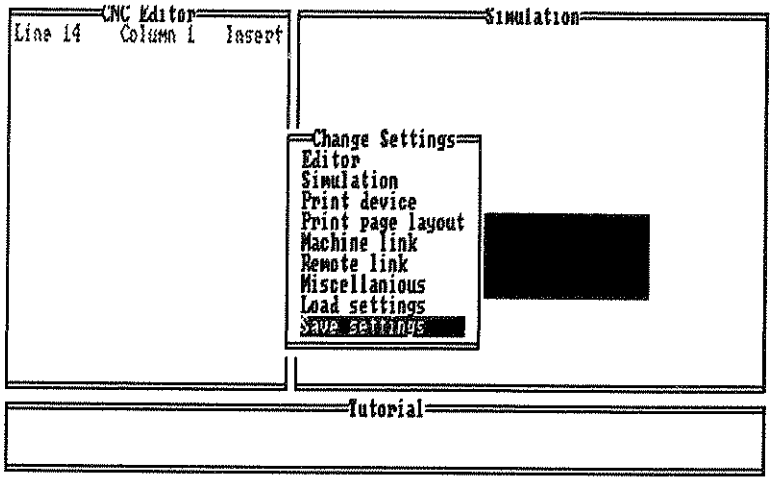


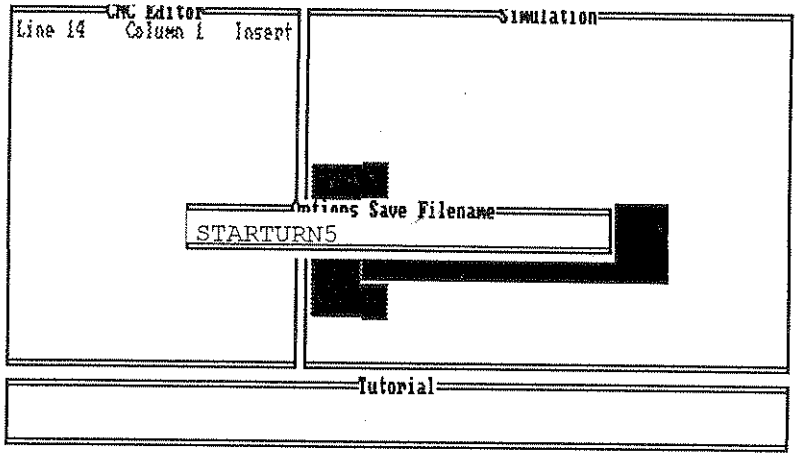


Any number of settings files can stored to Disk with the Extension ".OPT".

Select the Load Settings option, and the Default Settings File appears in the Edit Window. The default filename will be STARTURNPC.OPT. If you want a listing of all the available Settings files, clear the Edit Window with the ALTER Key, then press the EOB key.

SAVE SETTINGS





Select Save Settings option:

Type in a filename, or accept the default and press the RETURN key to confirm.

11. DIRECTIVES

BILLET DEFINITION.

This directive allows the billet in the simulation window to be given a size.

The billet definition should be placed at the start of the program, after the units of measure have been set.

TURNING SIMULATION.

[BILLET X30.0 Z50.0 defines the billet as 50mm long with a diameter of 30mm (if diameter programming is active).

CLEAR DIRECTIVE

Example :- [CLEAR

STEP DIRECTIVE

Switches over to single step execution.

Example:- [STEP

SINGLE STEP OFF DIRECTIVE

This directive switches off single step execution both on screen and when linked to the machine.

Example :- [NOSTEP

12. TUTORIALS AND COMMENTS

This option is intended as a teaching aid.Tutorials and comments can only be entered on a TEXT EDITOR via a QWERTY Keyboard.

TUTORIALS

Interactive lessons can be developed through the Tutorials facility. Messages and Questions can be placed within the CNC program.

! Displays message without stopping.

? Displays message but stops for keypress.

TUTORIAL MESSAGE

Tutorial message instructions begin with the "!" exclamation mark, which is followed by some text.

When the CNC program is executed, your text will appear in the Tutorial Window at the bottom of the screen.

Example: !Using tool 2 ....

TUTORIAL PAUSE

Tutorial pause instructions begin with the "?" question mark, which is followed by some text.

When the CNC program is executed, your text will appear in the Tutorial Window at the bottom of the screen. You will then be prompted to press RETURN to continue.

Example: ?Check the position

COMMENTS

Comments begin with the "(" open bracket character. They can be used to annotate

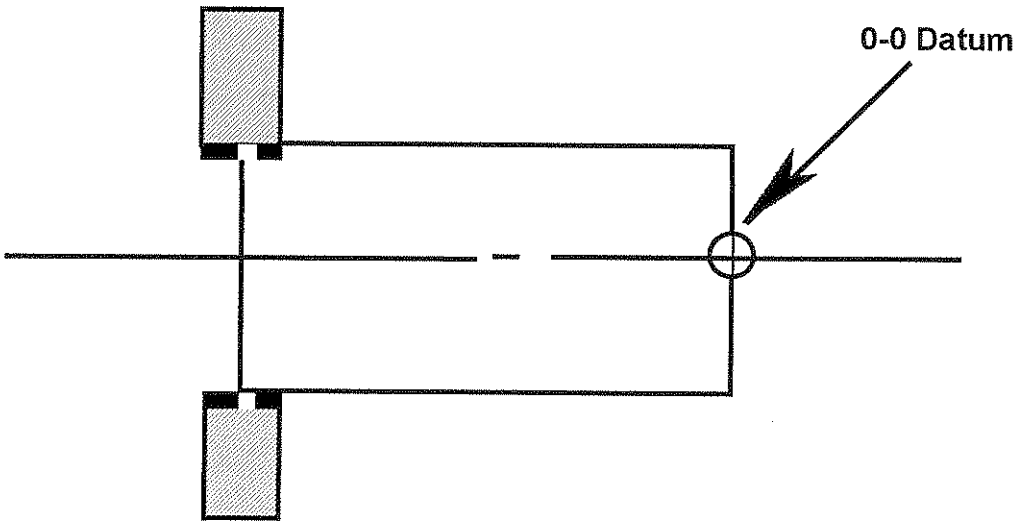
a program, and are ignored when it is executed.

Example: (Entering circular cycle

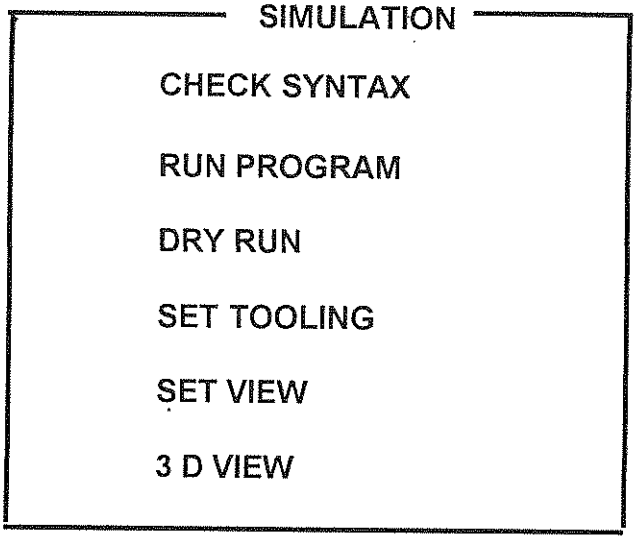
13. SIMULATION MENU

THE DATUM

The turning simulation always takes the centre-line and the end of the bar as the 0 - 0 datum position.



By pressing F9 the SIMULATION MENU is brought up on screen.



CHECK SYNTAX

This facility checks through the whole program for format errors.

RUN PROGRAM

This starts the on screen simulation of the program.

DRY RUN

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

SET TOOLING

Allows a tool shape to be allocated to a tool number. After highlighting the SET TOOLING Menu option, press the EOB KEY. Use the CURSOR UP and DOWN to highlight a tool number and select by pressing the EOB key. The following keys may be used to select the tools:

JOG KEY - to flip from left to right, and from front to back toolpost.

UP CUSOR - displays the next toolshape.

DOWN CURSOR - displays the previous toolshape.

EOB - to confirm.

RESET - to quit.

We supply over thirty predefined toolshapes with the software. You can add more shapes of your own design (see page 19.7 for details).

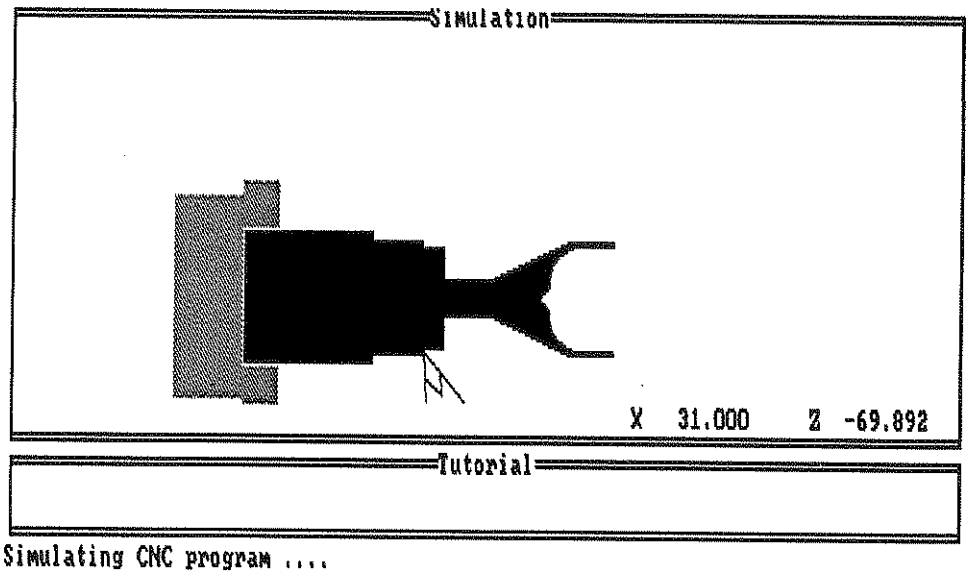
SET VIEW

Use this facility to indicate the view you require for on screen simulation. Press the JOG KEY to change the view window, which is highlighted by a box. Press the EOB key to confirm.

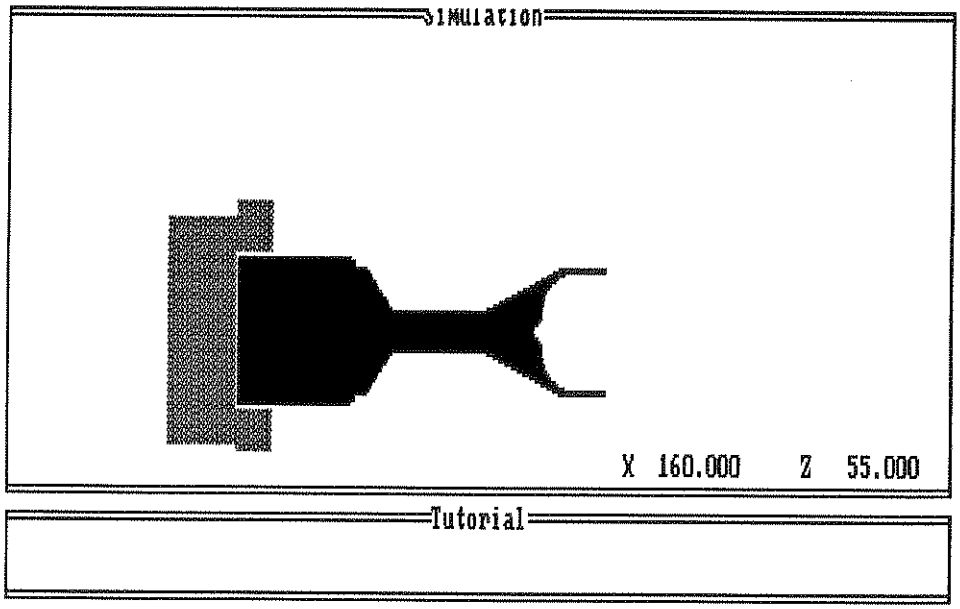
3D VIEW

Shows the simulation in a THREE DIMENTIONAL view.

EXAMPLE SIMULATION OF TURNED PART.



Simulating CNC program ....



14. EDITING

EDITING CNC PROGRAM

WHEN EDITING A CNC PROGRAM THE FOLLOWING KEYS CAN BE USED:

- ALTER - alters the address.
- INSERT - inserts the address. This is also used to initialise a new program.
- DELETE - deletes addresses.
- / ; # EOB characters shown are toggle EOB operated when editing.
- CANCEL - cancels an address(before insert is executed).

EDITING A PROGRAM

1. PRESS [EDIT] THIS BRINGS UP A SPLIT SCREEN-THE PROGRAM ON THE LEFT, AND THE SIMULATION SCREEN ON THE RIGHT.

2. PRESS [PGR] THIS BRINGS UP THE PROGRAM ONLY ON SCREEN WITH THE CURSOR AT THE TOP OF THE PROGRAM.

3. THERE ARE SEVERAL WAYS TO EDIT THE PROGRAM DEPENDING ON WHAT ALTERATIONS TO THE PROGRAM ARE REQUIRED, ALL OF WHICH, HOWEVER, REQUIRE THE MOVING OF THE CURSOR USING THE [↑] [↓] KEYS.

4. EXAMPLE: IF THE BLOCK TO EDIT BEGINS WITH G41, PRESS G41 THEN THE APPROPRIATE CURSOR KEY. THIS WILL POSITION THE CURSOR UNDER G41.TO MOVE THE CURSOR TO THE RIGHT AND DOWN PRESS [↓] TO MOVE THECURSOR TO THE LEFT AND UP PRESS [↑]

5. TO ALTER G41 TO SAY G42, KEY IN G42 AND PRESS [ALTER]

6. TO INSERT INFORMATION WITHIN A BLOCK , POSITION THE CURSOR IMMEDIATELY BEFORE THE POSITION YOU WISH TO PLACE IT ,THEN KEY IN THE INFORMATION AND PRESS [INSERT]

7. TO INSERT A FULL LINE OF INFORMATION, POSITION THE CURSOR BENEATH THE" ; "ON THE PREVIOUS LINE THEN KEY IN THE LINE REQUIRED AND PRESS [NSERT] [EOB]

8. TO DELETE A SINGLE CHARACTER, POSITION THE CURSOR BENEATH THAT CHARACTER AND PRESS [DELETE]

9. TO DELETE A FULL LINE OR BLOCK OF INFORMATION POSITION THE CURSOR BEFORE THE FIRST CHARACTER IN THE BLOCK AND CONTINUE TO PRESS [DELETE] UNTIL THE FULL BLOCK IS DELETED.

10. IF THE PROGRAM THAT HAS BEEN EDITED IS SAVED, PRESS [F2] FOR QUICK SAVE OF THE AMENDED PROGRAM.

15. WORD DETAILS

WORD DETAILS

Although the Control will, in general, accept part programming words in any sequence, it is recommended that the following word order for each block is used :-

N; G; X or U; Z or W; I; K; F; S; T; M.

O: PROGRAM NUMBER

The O followed by a 4 digit numerical value is used to assign a program number-

e.g. O1002

N : Sequence Number

The N word may be omitted. When programmed, the sequence number following the N address is a four digit numerical value and is used to identify a complete block of information. Although ascending, decending, or duplicate numbering is allowed, it is best to program in ascending order in increments of 10. This allows for future editing and simplified sequence number search.

G : Preparatory Command

The two digit G command is programmed to set up the control to perform an automatic machine operation. A full list of G codes are given, one G word from each modal group and one non modal G word can be programmed on the same block.

EXAMPLE :

Valid N100 G00 G40 G90 G95

Non valid N100 G00 G40 G41 G90 G95

*G40 & G41 are from the same group.

A retained G word (Modal) from one group remains active until another G word from the same group is programmed.

One-shot G words (Non-Modal) must be programmed in every block when required.

G CODES LISTING FOR DENFORD FANUC LATHES

NOTE:- NOT ALL G CODES APPLY TO EACH MACHINE.

Group			
	1	G00	Positioning (Rapid Traverse)
	1	G01	Linear Interpolation (Feed)
	1	G02	Circular Interpolation CW
	1	G03	Circular Interpolation CCW
	0	G04	Dwell
	0	G10	Offset Value Setting By Program
	6	G20	Inch Data Input
	6	G21	Metric Data Input
	9	G22	Stored Stroke Check On
	9	G23	Stored Stroke Check Off
	0	G27	Reference Point Return Check
	0	G28	Reference Point Return
	0	G29	Return From Reference Point
	0	G30	Return To 2nd Reference Point
	0	G31	Skip Function
	1	G32	Thread Cutting
	1	G34	Variable Lead Thread Cutting

0	G36	Automatic Tool Compensation X
0	G37	Automatic Tool Compensation Z
7	G40	Tool Nose Radius Compensation Cancel
7	G41	Tool Nose Radius Compensation Left
7	G42	Tool Nose Radius Compensation Right
0	G50	Work Co-ord. Change/Max. Spindle Speed setting
0	G65	Macro Call
12	G66	Macro Modal Call
12	G67	Macro Modal Call Cancel
4	G70	Finishing Cycle
4	G71	Stock Removal in Turning
0	G72	Stock Removal in Facing
0	G73	Pattern Repeating
0	G74	Peck Drilling in Z Axis
0	G75	Grooving in X Axis
0	G76	Thread Cutting Cycle
1	G90	Cutting Cycle A
1	G92	Thread Cutting Cycle
1	G94	Cutting Cycle B
2	G96	Constant Surface Speed Control
2	G97	Constant Surface Speed Control Cancel
11	G98	Feed Per Minute
11	G99	Feed Per Revolution

NOTES FOR G CODE LISTING

Note 1:

G Codes of 0 group represent those non modal and are effective to the designated block.

Note 2:

G Codes of different groups can be commanded to the same block. If more than one G code from the same group are commanded, the latter becomes effective.

AXIS DEFINITIONS

Z AXIS

The Z axis is along a line between the spindle and the tailstock, or the centre line of rotation of the spindle. Minus (-) movements of the turret are left toward the headstock; positive (+) movements are right toward the tailstock.

X AXIS

The X axis is 90 degrees from the Z axis (perpendicular to the Z axis). Minus (-) movements of the turret are toward the centre-line of rotation,and positive (+) movements are away from the centre-line of rotation.

X : X AXIS COMMAND

The X word is programmed as a diameter which is used to command a change in position perpendicular to the spindle centre-line.

U : X AXIS COMMAND

The U word is an incremental distance (diameter value) which is used to command a change in position perpendicular to the spindle centre-line. The movement is the programmed value.

Z : Z AXIS COMMAND

The Z word is an absolute dimension which is used to command a change in position parallel to the spindle centre-line.

W : Z AXIS COMMAND

The W word is an incremental distance which is used to command a change of position parallel to the spindle centre-line.

Do not program X & U or Z & W in the same block. If an X axis command calls for no movement it may be omitted.

X ,U or P : DWELL

The X word is used with G04 to command a dwell in seconds.

The P word is used with G04 to command a dwell in milliseconds.

I WORD

For arc programming (G02 or G03) the I value (with sign) is programmed to define the incremental distance parallel to the X axis, between the start of the arc and the arc centre.

K WORD

For arc programming (G02 or G03), the K value (with sign) is programmed to define the incremental distance parallel to the Z axis, between the start of the arc and the arc centre.

The maximum arc for I & K programming is limited to the quadrant. If I or K is zero, it must be omitted.

F WORD

a) In G99 mode the F word is used to command feed/rev.

b) In G98 mode the F word is used to command feed/min.

c) In G32 mode the F word specifies the lead (pitch) of the thread.

P WORD

a) Used in automatic cycles to define the first block of a contour.

b) Used with M98 to define a subroutine number.

Q WORD

Used in automatic cycles to define the last block of a contour.

R WORD

For circular interpolation (G02 or G03)the R word defines the arc radius from the centre of the tool nose radius (G40 active) - or the actual radius required (G41/G42 active).

S WORD

a) In the constant surface speed mode (G96) the four digit S word is used to command the required surface speed in either feet or metres per minute.

b) In the direct r.p.m. mode (G97), the four digit S word is used to command the spindle speeds incrementally, in r.p.m., between the ranges available for the machine (see Machine Specification).

c) Prior to entering constant surface speed mode (G96) the S word is used to specify a speed constraint, the maximum speed you wish the spindle to run at. To set this restraint the S word is programmed in conjunction with the G50 word.

T WORD

The T word used in conjunction with "M06", is used to call up the required tool on an automatic indexing turret machine, and to activate its tool offsets.

M WORD

An M Word is used to initiate auxiliary functions particular to the machine. One M code can be programmed within one program block together with other part program information.

M CODE LIST FOR DENFORD FANUC LATHES

All M codes marked with an asterisk will be executed at the end of a block (i. e. after the axis movement). NOTE:- not all M codes are available on each machine.

* M00 PROGRAM STOP

* M01 OPTIONAL STOP

* M02 PROGRAM RESET

M03 SPINDLE FORWARD

M04 SPINDLE REVERSE

* M05 SPINDLE STOP

M06 AUTO TOOL CHANGE

M07 COOLANT "B" ON

M08 COOLANT "A" ON

* M09 COOLANT OFF

M10 CHUCK OPEN

M11 CHUCK CLOSE

M12

M13 SPINDLE FORWARD & COOLANT ON

M14 SPINDLE REVERSE & COOLANT ON

M15 PROGRAM INPUT USING, "MIN P" (SPECIAL FUNCTION)

M16 SPECIAL TOOL CALL (TOOL CALL IGNORES TURRET)

M17

M18

M19 SPINDLE ORIENTATE

M20 SPINDLE INDEX A

M21 SPINDLE INDEX 2A

M22 SPINDLE INDEX 3A

M23 SPINDLE INDEX 4A

M24

M25 QUILL EXTEND

M26 QUILL RETRACT

M27

M28

M29 SELECT "DNC" MODE

M30 PROGRAM RESET & REWIND

M31 INCREMENT PARTS COUNTER

M32

M33

M34

* M35

M36

M37 DOOR OPEN TO STOP

M38 DOOR OPEN

M39 DOOR CLOSE

M40 PARTS CATCHER EXTEND

M41 PARTS CATCHER RETRACT

M42

M43 SWARF CONVEYOR FORWARD

M44 SWARF CONVEYOR REVERSE

* M45 SWARF CONVEYOR STOP

M46

* M47

M48 LOCK % FEED AND % SPEED AT I00%

M 49 CANCEL M48 (DEFAULT)

M50 WAIT FOR AXIS IN POSITION SIGNAL (CANCELS CONTINUOUS PATH)

M51 CANCEL M5O (DEFAULT)

M52 PULL-OUT IN THREADING=90 DEGREES (DEFAULT)

M53 CANCEL M52

M54 DISABLE SPINDLE FLUCTUATION TESTING (DEFAULT)

M56 SELECT INTERNAL CHUCKING (FROM PLC EDITION "F")

M57 SELECT EXTERNAL CHUCKING (FROM PLC EDITION"F")

M58

M59

M60

M61

M62 AUX. 1 ON

M63 AUX. 2 ON

M64 AUX. 1 OFF

M65 AUX. 2 OFF

* M66 WAIT FOR INPUT 1

* M67 WAIT FOR INPUT 2

M68 ONLY INDEX WITH ALL AXIS AT HOME POSITION

M69 INDEX TURRET ANYWHERE

M70 MIRROR IN X ON

M71

*M72

M73

M74

M75

M76 WAIT FOR INPUT 1 TO GO LOW (from revision C)

M77 WAIT FOR INPUT 2 TO GO LOW (from revision C)

M78

M79

M80 MIRROR IN X OFF

M81

M82

M83

* M84

M85

M86

M87

M88

M89

M90

M91

M92

M93

M94

M95

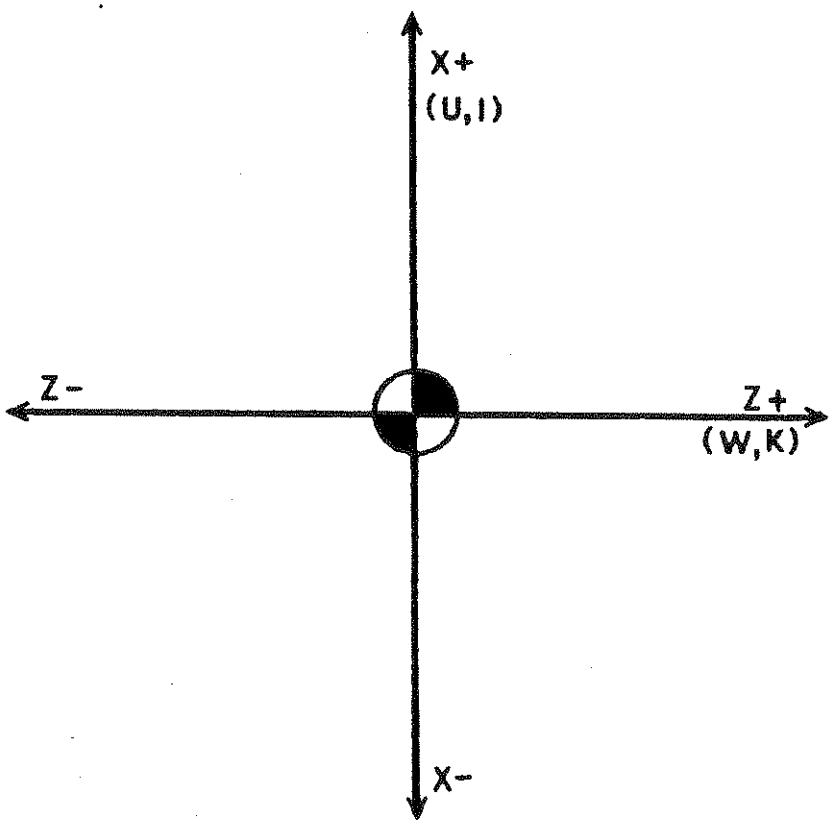
M96

M97

M98 SUB PROGRAM CALL

M99 SUB PROGRAM END

16. BASIC MOVEMENTS



BASIC MOVEMENTS

STAIGHT LINE MOTION (G00, G01)

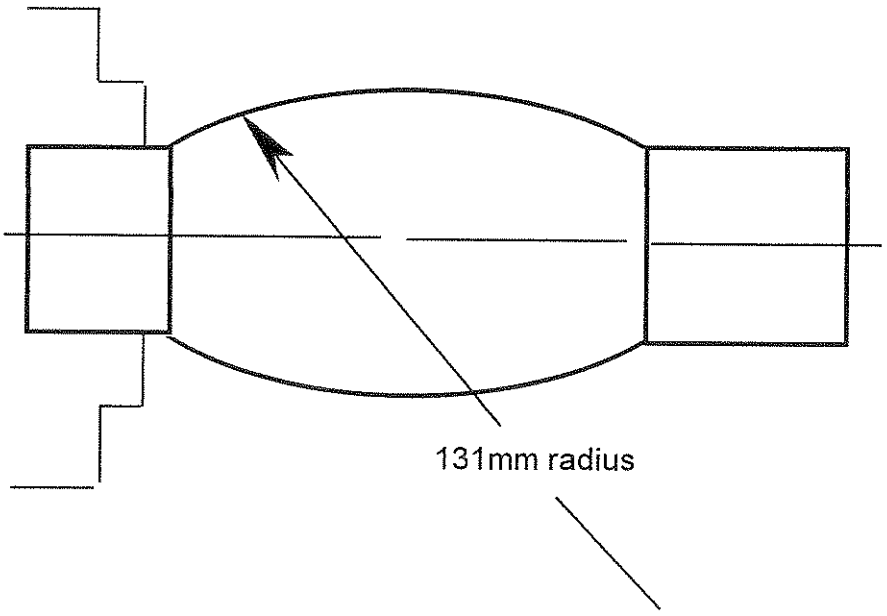
Almost all motion commands will be programmed as straight line motion for moving to, or cutting along diameters, faces and angles. During cutting statements, it is only necessary to change this mode of operation when an arc is to be generated.

Any point on a component can be defined by X & Z dimensions.

In absolute programming, the programmer commands the tool to move to a position which is relevant to a component zero. The direction of motion is determined by the system and is dependent on the previous position of the tool. If the new position in"X" is larger that the last position, then the tool will move away from the spindle centre-line.

In incremental programming, the programmer commands the tool point to move to a new position which is a specific distance and direction from its previous position. The direction of motion is determined by the sign of the value for the command. If the command in "X" is U-1.0 then the tool will move towards the spindle centre-line.

NB. The maximum arc that can be generated is 131 mm.

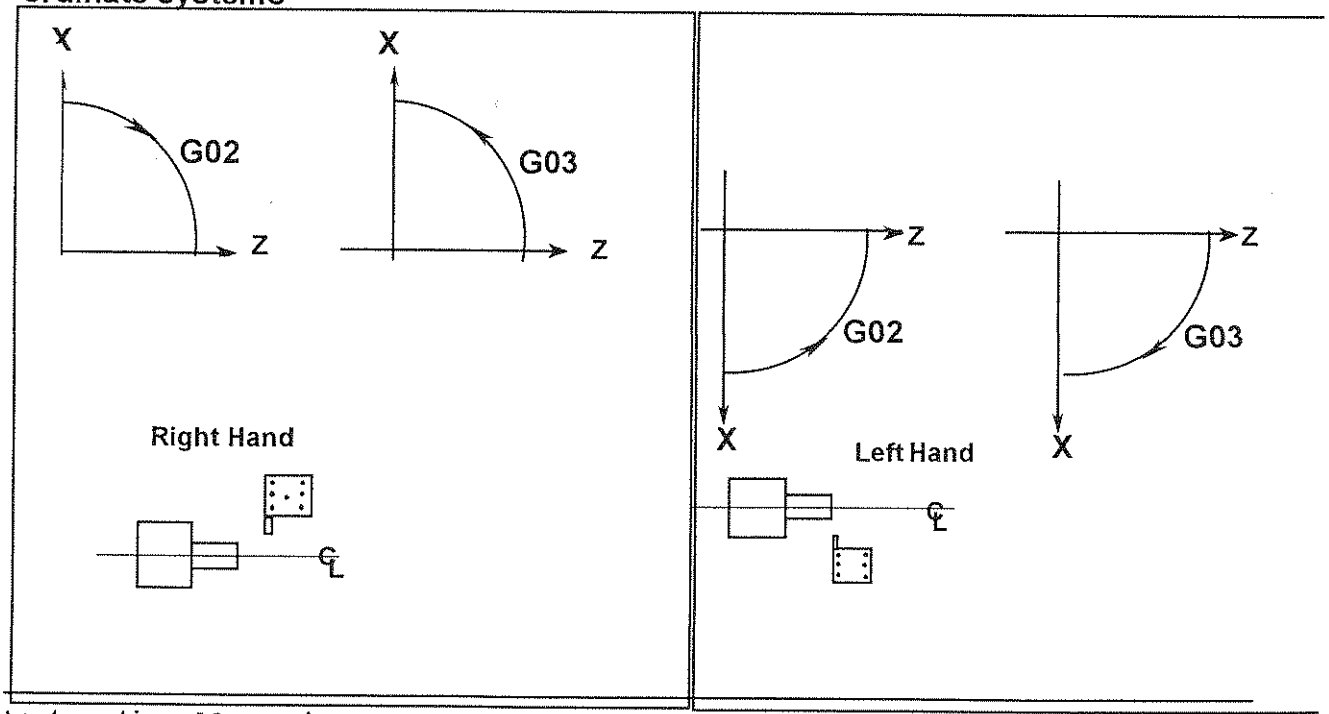


CIRCULAR INTERPOLATION (G02 AND G03)

The following commands will move a tool along a circular arc.

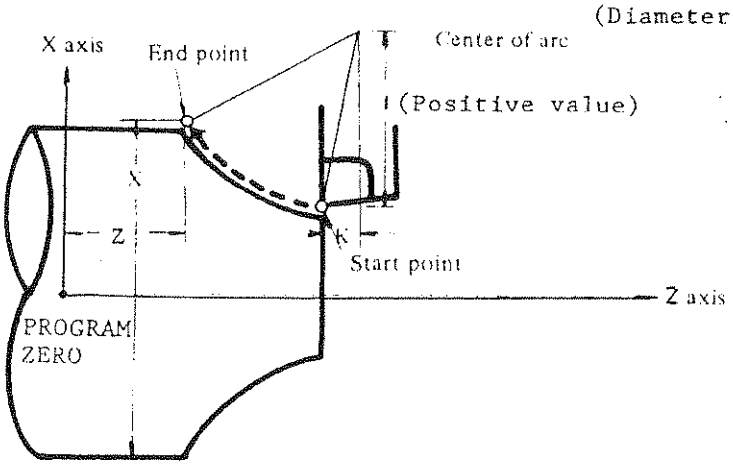
	Data to be given		Command	Meaning
1	Rotation Directiuon		G02	Clockwise Direction (CW)
			G03	Counterclockwise Direction(CCW)
2	End Point Position	Absolute Command	X, Z	Endpoint Position in the Work Co-Ordinate Position
		Incremental Command	U,W	Distance from Start Point to End Point
3	Distance from Start point to Centre		I, K	Distance with Direction from Start Point to Arc Centre. (Always radius value and Incremental Value)
	Radius of Arc		R	Radius of Arc. (Always a Radius Value)
4	Feedrate		F	Feedrate along the Arc

The clockwise or counterclockwise direction varies in Right or Left Hand co-ordinate systems



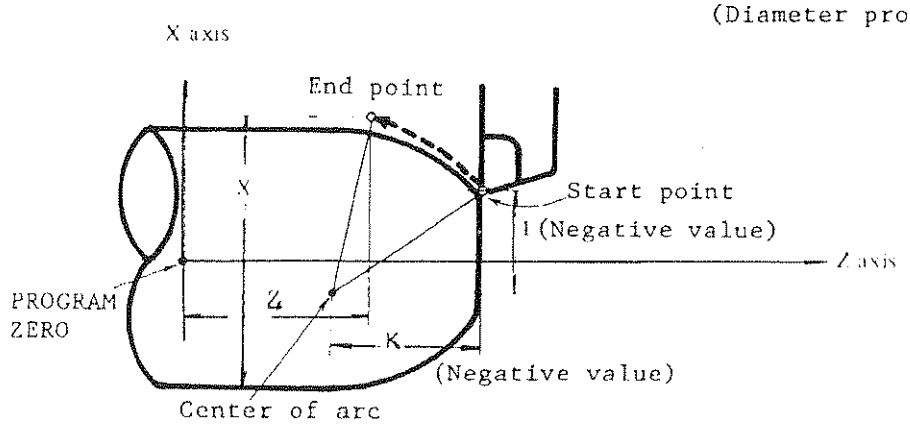
G02X ---- Z ---- I ---- K ---- F ----;
(absolute programming)

(Diameter programming)



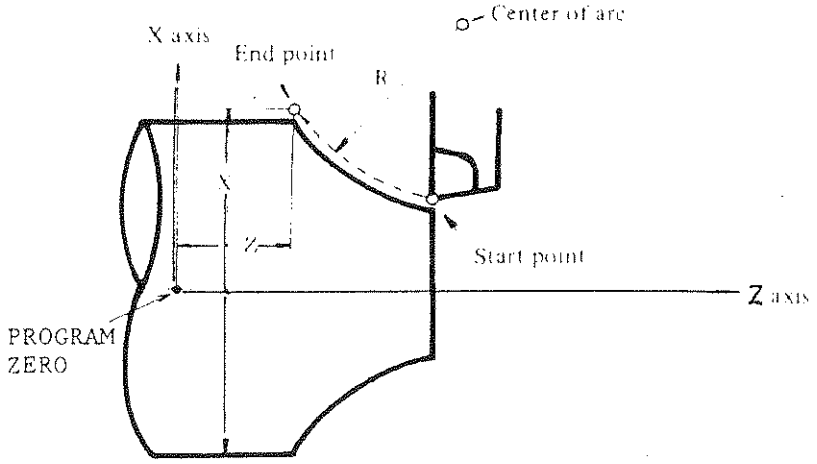
G03X ---- Z ---- I ---- K ---- F ----;
(absolute programming)

(Diameter programming)



G02X ---- Z ---- R ---- F ----;
(absolute programming)

(Diameter programming)



Program Example (Absolute Command)

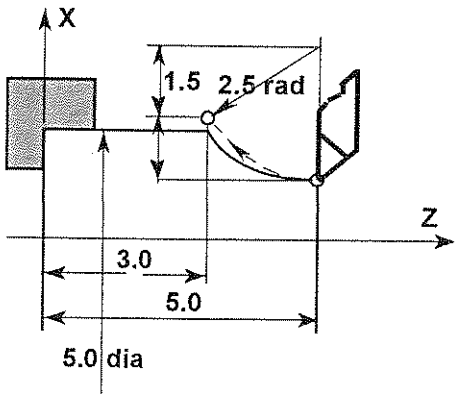
(Diameter Programming)

G02 X5.0 Z3.0 I2.5 F0.03;

or G02 U2.0 W-2.0 I2.5 F0.03;

or G02 X5.0 Z3.0 R2.5 F0.03;

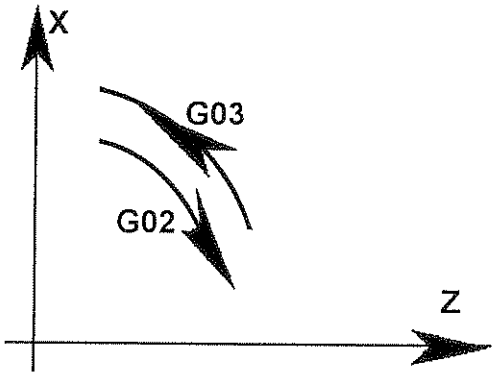
or G02 U2.0 W-2.0R2.5 F0.03;



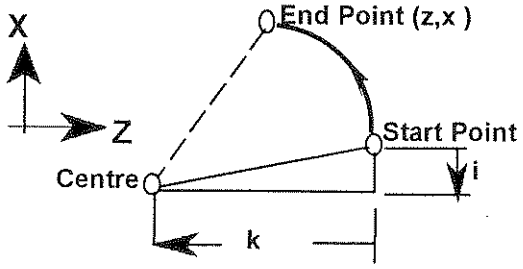
The feedrate for circular interpolation is specified by the F Code. The feedrate along an arc (tangent to an arc) is controlled to maintain the specified feedrate.

The clockwise and counterclockwise directions are as shown below.

The view is from the positive direction of the Z Axis (X Axis) to the negative direction in the ZX plane in the right hand cartesian co-ordinate system.



The end point of an arc is specified by address X, Z, or U, W, and is expressed as an absolute or incremental value. For the incremental value, the co-ordinate of the end point which is viewed from the start point of the arc is specified. The Arc Centre is specified by the address I, and K for the X and Z Axes respectively. The numerical value following I, J, or K, however, is a vector component in which the arc centre is seen from the start point, and is always specified as an incremental value as shown in the figure below.



I and K must be aas signed according to the direction.

The Radius can be specified with the address R instead of specifying the centre by I or K.

The command format is as follows:-

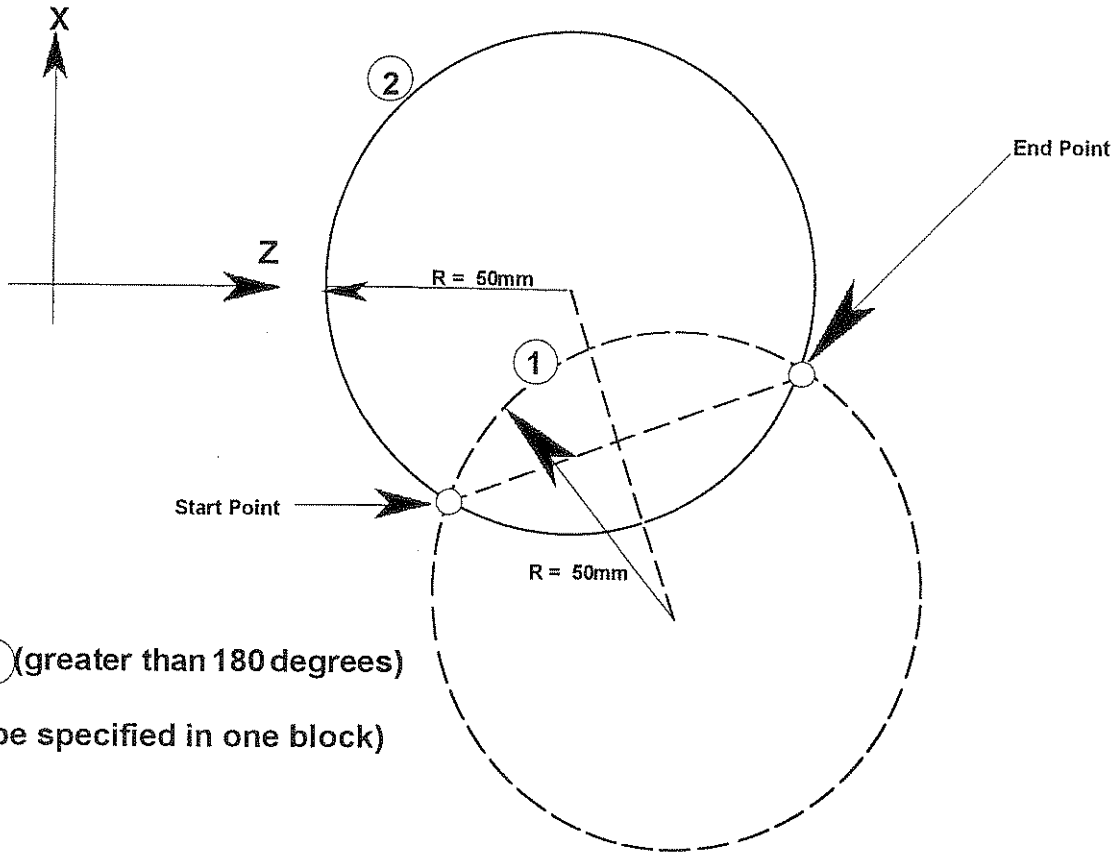
G02 }
    } X .....Z..... R..... ;
G03 }

In this case, two types of arcs (one arc is less than 180degrees; the other is more than 180 degrees) are considered, as shown in the figure below. An arc exceeding 180 degrees can not be commanded.

EXAMPLES

For arc (1) (less than 180degrees)

G91 G02 Z60.0 X20.0 R50.0 F300.0;



For arc (2) (greater than 180 degrees)

(cannot be specified in one block)

NOTES

1. I0 and K0 can be omitted.

2. If X(U) and Z(W) are both omitted or if the end point is located at the same position as the start point, and when the centre is commanded by I and K, an arc of 360 degrees (a full circle) is assumed.

3. If I, K and R addresses are specified simultaneously, the arc specified by address R takes precedence and the others are ignored.

17. PROGRAMMING INSTRUCTIONS

PLANNING AND PROGRAMMING PROCEDURE

The following procedure may be used as a guide to assist the programmer, by describing each step required in preparing the method of production.

Before writing the part program, a detailed planning procedure is required:

PLANNING PROCEDURE

- 1. Receive part drawing. From part drawing information, check suitability of part to be machined against the machine capacity (clearances).

- 2. Determine a method of driving the component (chuck type, chuck size, type of jaw, collet, collet size, face driver etc) and the method of machining.

- 3. Determine the tooling required to suit the method of machining and utilise the tools which are permanently set up in the turret wherever possible

- 4. Determine the order of machining and the tooling stations.

- 5. Determine planned stops (cycle interrupt procedure, incorporating block delete codes) for checking dimensional sizes,where required by the operator.

- 6. Determine cutting speeds, bearing in mind the following:

  a) Component material, method of driving, rigidity of component.

  b) The tooling selected for roughing and finishing: toolholders with carbide inserts, the grade of the carbide insert selected, carbide drills, high speed steel drills, and ceramics.

- 7. Determine depths of cut and feeds for roughing operations, bearing in mind the horsepower/kilowatts available for cutting and the rigidity of the part.

- 8. Determine, from surface finish requirements, the tool nose radius most suited for the finishing operations and determine feedrates.

- 9. Allocate tool offsets as required.

- 10. Complete planning sheet.

PROGRAMMING PROCEDURE

- 1. After completing the planning sheet draw the component to scale showing the tool paths. A scale drawings is not mandatory where the component shape and the cutting paths are simple. A sketch may prove sufficient.

- 2. Select a component datum and dimension in scale drawing (or sketch) with the length and diameter of cuts relative to the component datum. Carry out necessary calculations at slopes and arcs.

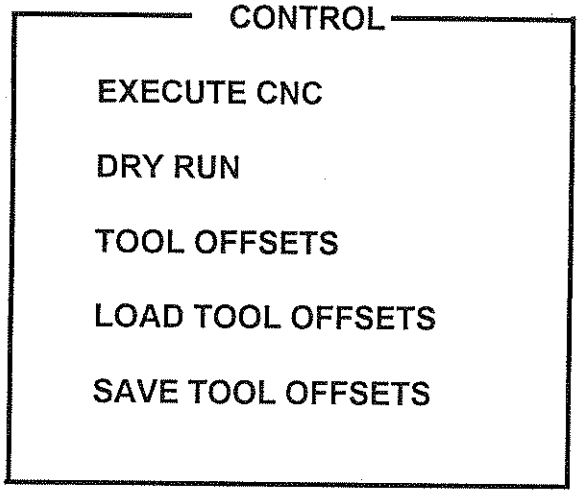
- 3. Draw tooling layout sheet showing tools to be used in the program and indicate the station numbers for each tool.

- 4. Complete the tooling layout sheet by indicating the ordering code for each tool and the grade and type of inserts to be used.

- 5. Commence writing starting procedure on to program sheets.

CONTROL MENU

IN THE MACHINE CONTROL MODE, BY PRESSING F9 THE FOLLOWING MENU APPEARS: USE CURSOR UP/DOWN TO SELECT.



EXECUTE CNC.

Starts execution of CNC PROGRAM

DRY RUN

Runs the machine program without actual machine movement. This mode provides fast overtravel checking so that a "run in" can be avoided.

TOOL OFFSETS

Lists the current tool offsets and allows you to edit them.

LOAD TOOL OFFSETS

Loads a new set of offsets from disk.

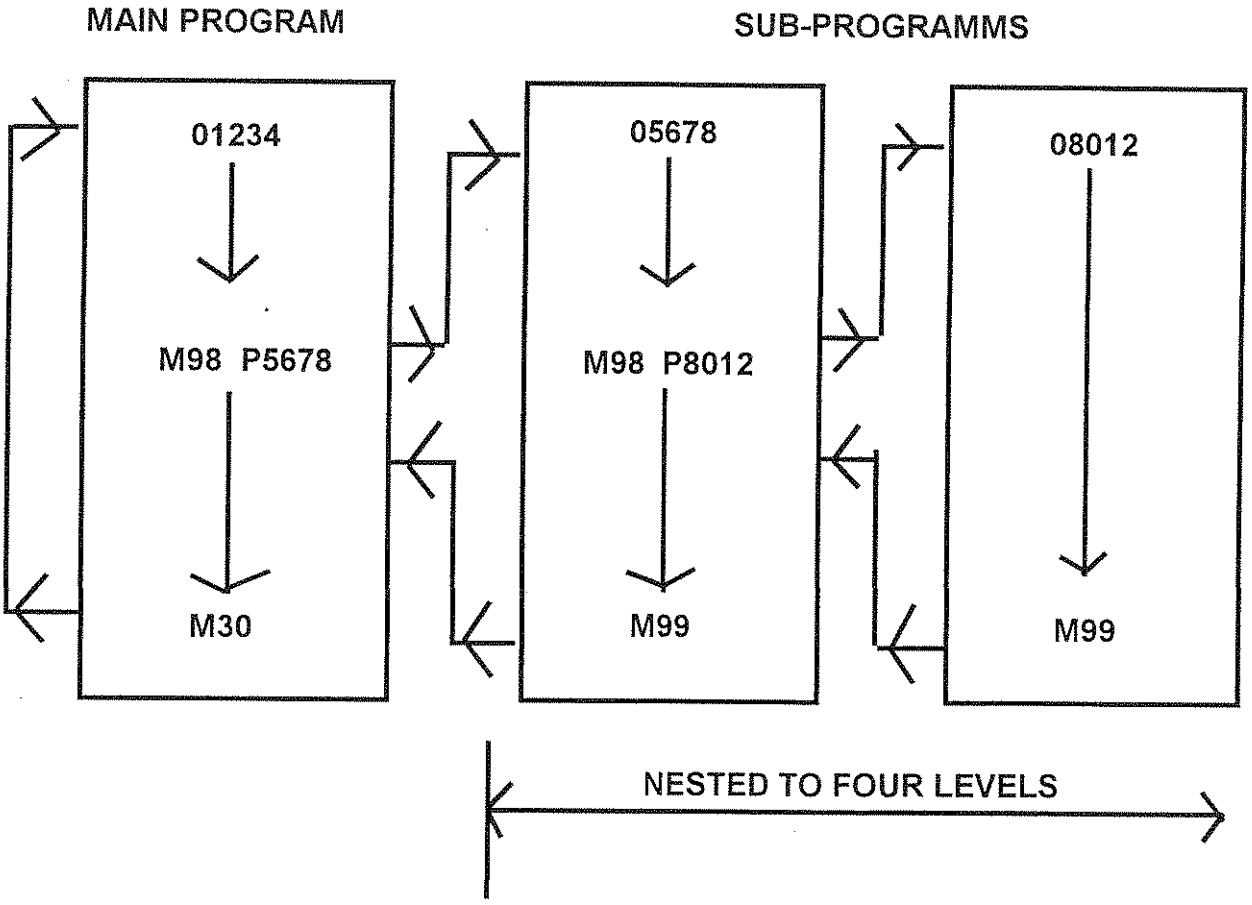
SAVE TOOL OFFSETS

Saves current offsets to disk.

PROGRAM AND SUBROUTINE IDENTIFICATION

The first block of a program/subroutine must contain a program number "O".

The program would be as follows:-



M98

SUB PPROGRAM CALL

P9999 ——— 5678

No

No. OF REPEATS

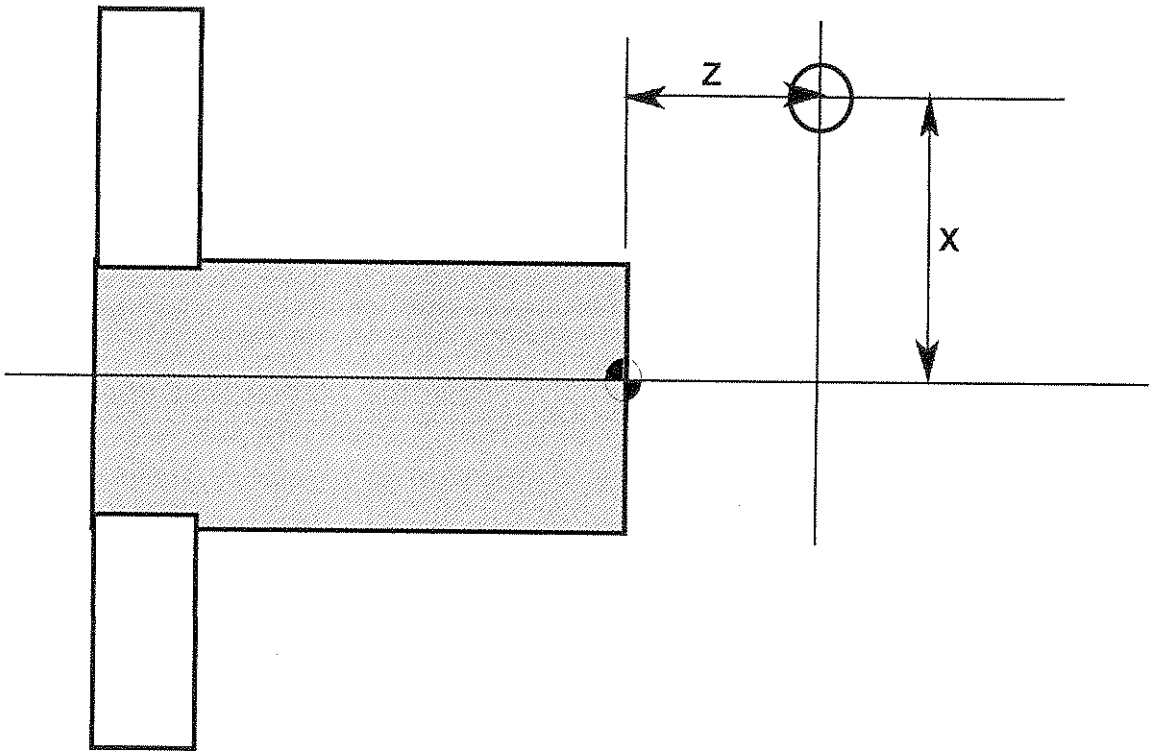
See page18.1 for further details.

COMPONENT ZERO

During the Programming Procedure step 2 (Page 17.3) indicates that a component datum be selected in order to dimension a scale drawing. The programmer chooses zero to lie on some position in the Z plane (the zero position for X will always lie on the spindle centre-line), at a position which is convenient to describe component lengths.

For the current operation, a zero position which has been selected for a first operation may not be the required position for the second operation.

The zero position is the datum from which all component co-ordinate values are referred.



TOOL SETTING PROCEDURE

The following procedure should be adhered to :-

- Datum the machine
- Any offsets already in the file can be overwritten.

· Select tool in MDI,(JOG mode).

· Press T, then tool number required, then EOB. The correct tool will now be selected.

· Start the Spindle - Press S, then a spindle speed, then press EOB.

· Press SPINDLE CW.

· Using AXIS JOG keys, touch on the end of the bar.

· Press the MENU\OFFSET Key.

· Press MZ 0 EOB Z now reads zero.

· Press RESET or CANCEL Key.

· Turn an outside diameter and move away in the Z Axis only.

· Stop the Spindle and measure the outside diameter just turned.

· Press MENU OFFSET Key.

· Press MX and input the measured diameter.

· Press RESET or CANCEL Key.

· Return machine to " home " position.

· Repeat the above procedure for the remaining tools.

·To save the OFFSETS press F9,which brings up the offset menu,then select SAVE OFFSETS.

FORMATTING (INITIALISATION)

Formatting is utilised to provide the control with a standardised pattern of input data. This is also referred to as Initialisation or Safe Start.

The control system will only perform the operations that it has been instructed to perform and no more.

Typical formatting conditions are:-

a) G20, G21 Inch, Metric

b) G00, G01 Rapid, Feed Movement.

c) G40, G41, G42 Tool Nose Radius Compensation

d) G98, G99 Feed /min, /rev.

e) G96, G97 CSS, Direct r.p.m.

f) S .... M03, M04 Spindle Speed and Direction.

g) M06 T .... Tool No.

h) G50 S .... CSS r.p.m. limit

i) G96 S .... CSS and Cutting Speed

Example Program Start:-

: O0001

N10 G21 G97 G99 G40

N15 G96 S350

N20 M06 T0101

N30 M03 G0 X100 Z100

N40 G50 S3000

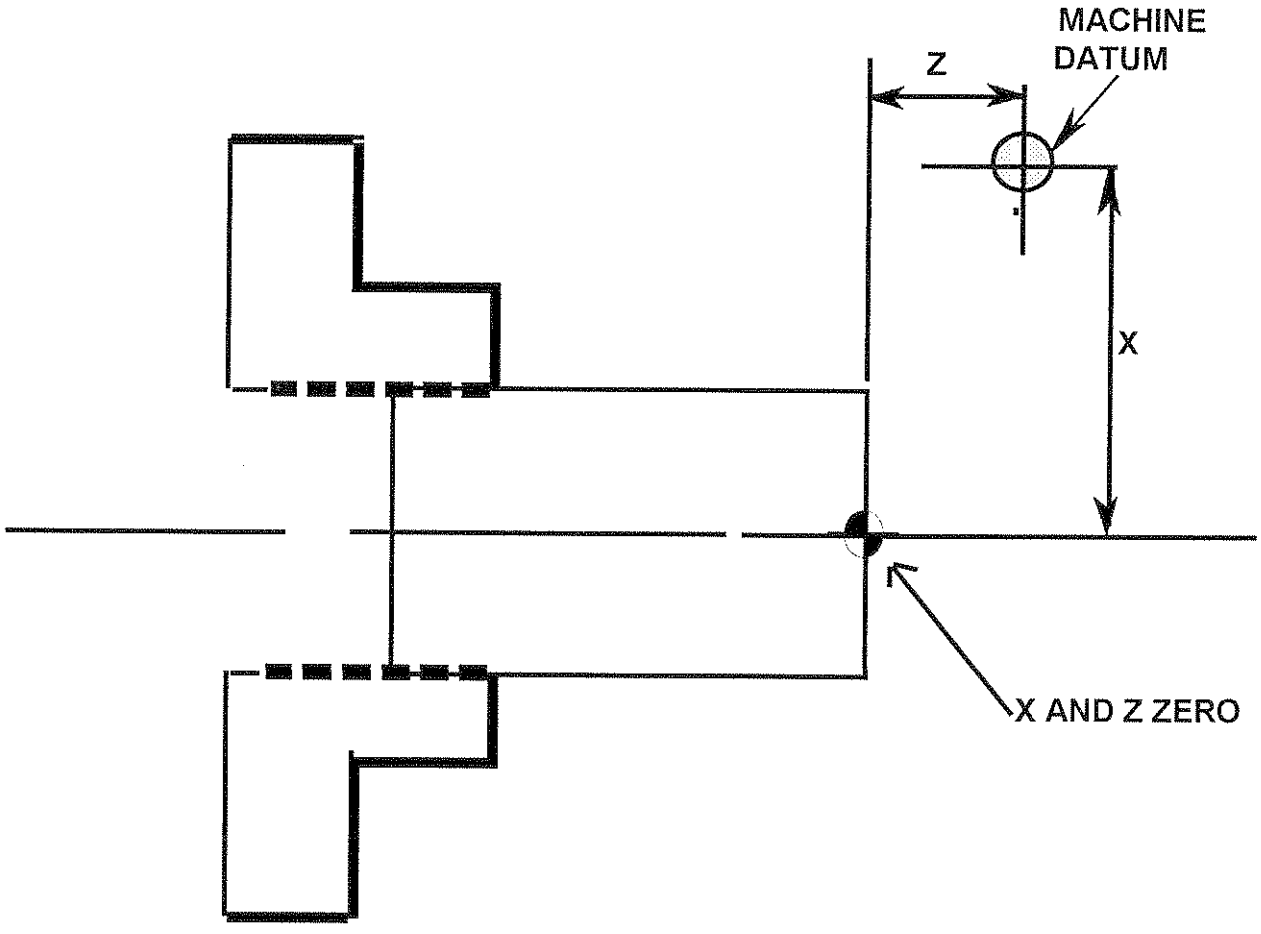
N60 G01 X.... Z.... F....

TOOL DATA

By using an OFFSET FILE, the program can be prepared using a common reference point.

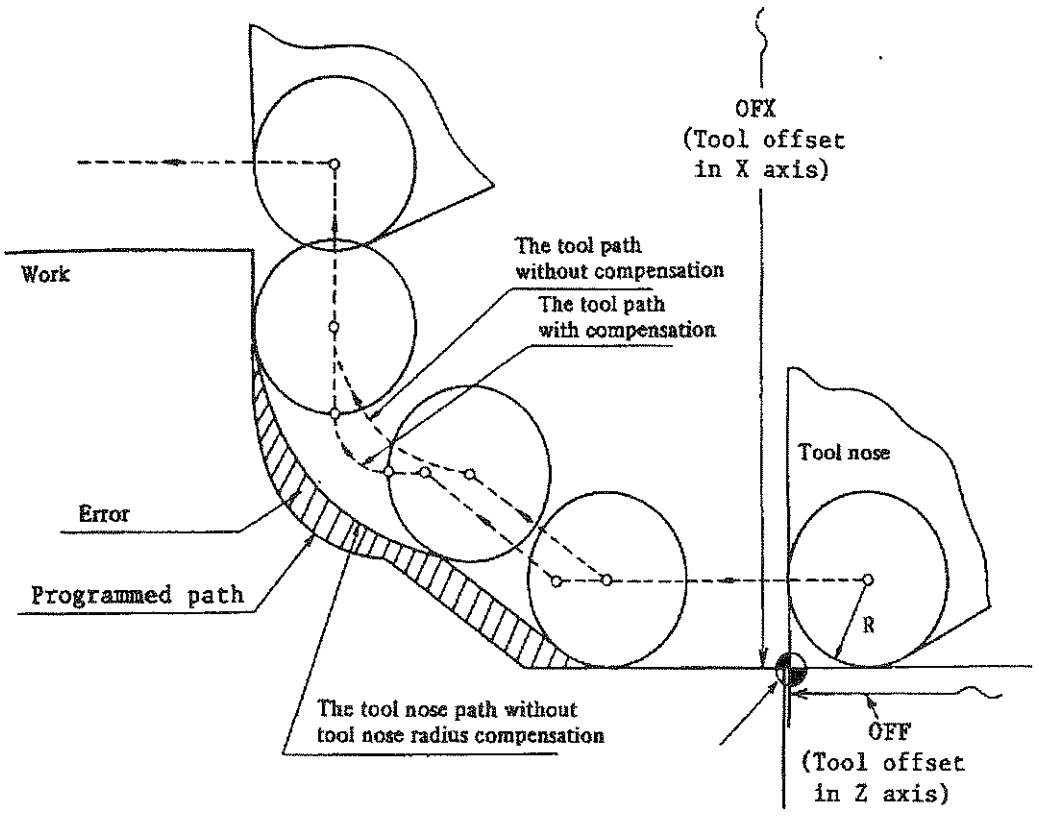
Tool offsets can be entered into the control's memory using the following:-

- X and Z = Tool Offset for X and Z. This is based on the incremental distance calculated by the software allowing for diameter programming.
- R = Tool Radius. This is the actual radius of the tool tip.
- T = Tool Location Code . This code represents the orientation of the tool radius - see imaginary tool nose, p 17.10, 17.11.

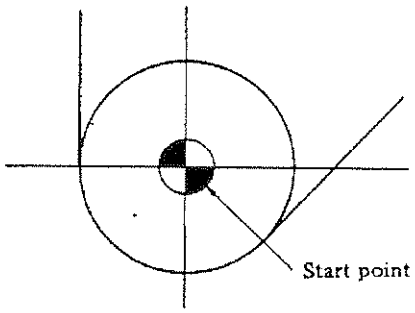


TOOL RADIUS COMPENSATION G40-G42

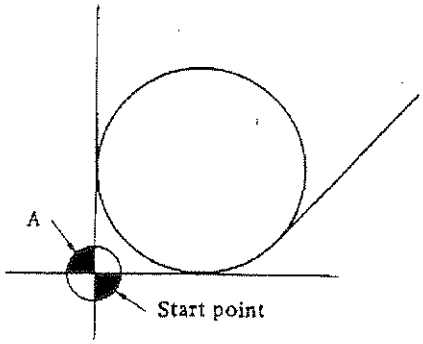
When producing parts which have tapers and arcs in the part profile, it becomes difficult to produce the part accurately when the tool offset function alone is used.This is because most tools have a radius at the cutting point of contact, rather than a sharp point. This radius must therefore be taken into account when producing parts of this nature. The TOOL NOSE RADIUS COMPENSATION automatically compensates for these errors.



IMAGINARY TOOL NOSE



When programmed using the tool nose center



When programmed using the imaginary tool nose

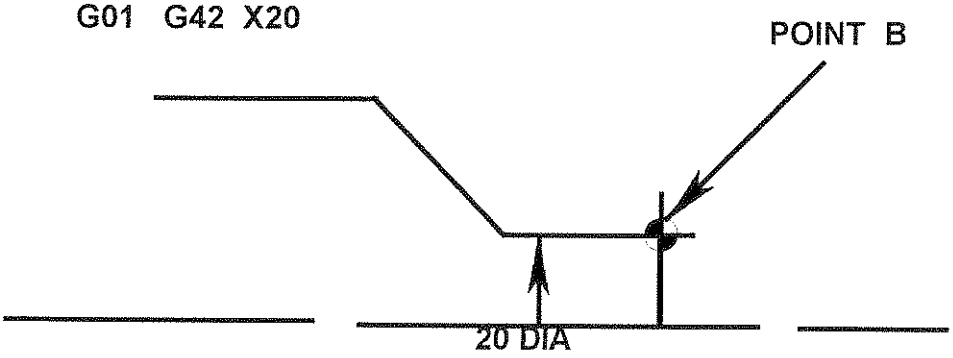
The tool nose at position A in the above figure does not, in actual fact, exist. The IMAGINARY tool nose is required because it is usually more difficult to set the tool nose centre to the start point. When the imaginary tool nose is used, the tool nose radius need not be considered during programming.

The positional relationship, when the tool is set to the start point is shown in the above figure.

Therefore the program to start at point B in the figure below would be:-

G00 X24 Z2

G01 G42 X20

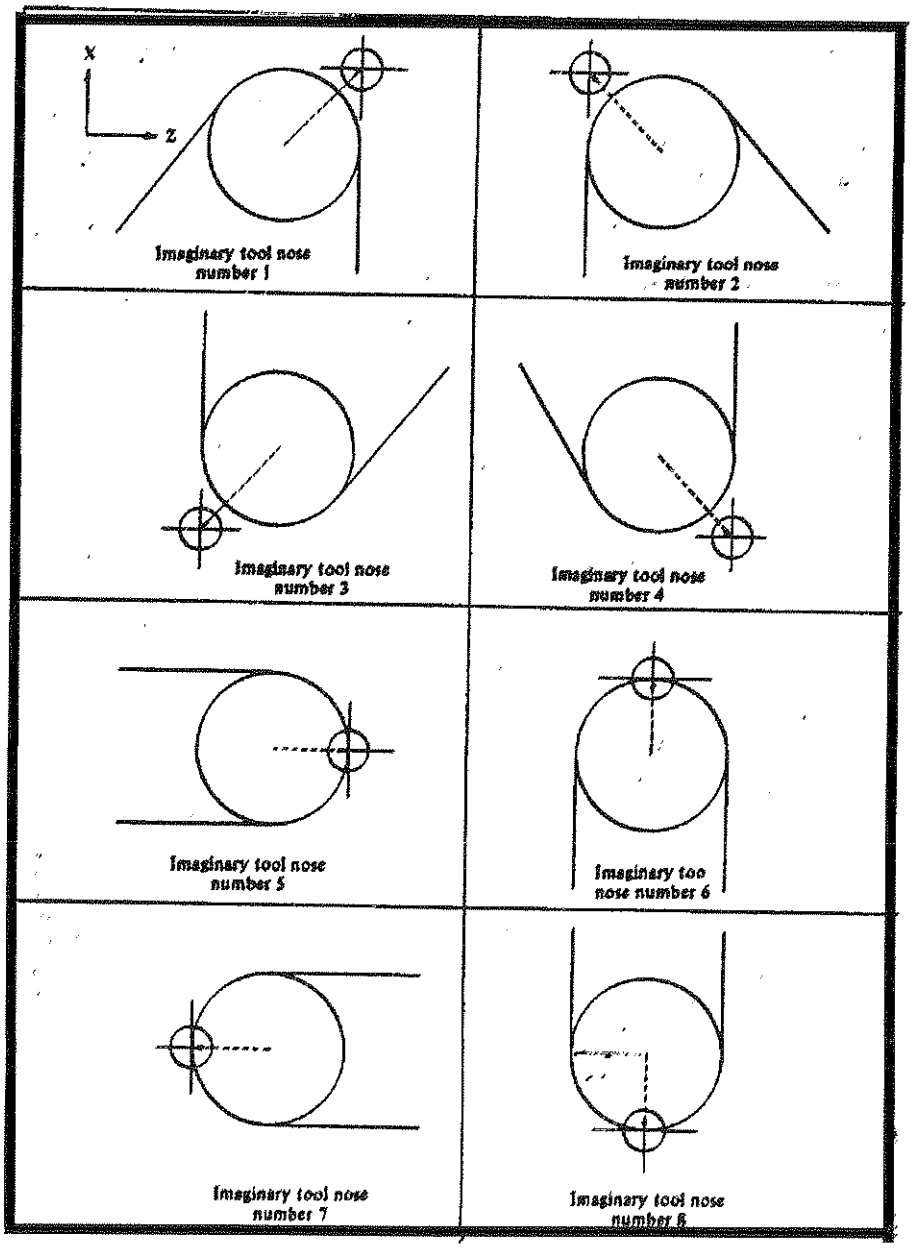


In the linear move before entering the TOOL NOSE COMPENSATION, always add on a value more than the radius of the tool.

After the tool nose radius has been applied and the particular move has been executed - i.e. an arc has been turned - then two linear straight line moves must be made before cancelling the tool radius compensation with a G40.

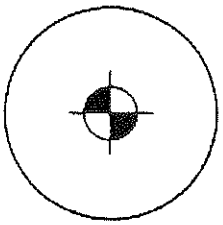
NB. The two linear moves must be greater than the tool radius compensation.

DIRECTION OF IMAGINARY TOOL NOSE.

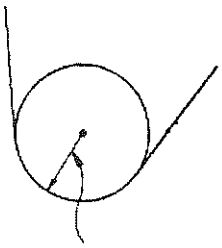


IMAGINARY TOOL NOSE

Imaginary tool nose numbers 0 and 9 are used when the tool nose centre coincides with the start point. Set the imaginary tool nose number for each tool offset number under the "T" value in the offset table.



Imaginary tool nose
number 0 or 9



Tool nose radius compensation value
(Tool nose radius value)

This value is set from the MDI according to the offset number.

Set the tool nose radius under the "R" value in the offset table for each offset number.

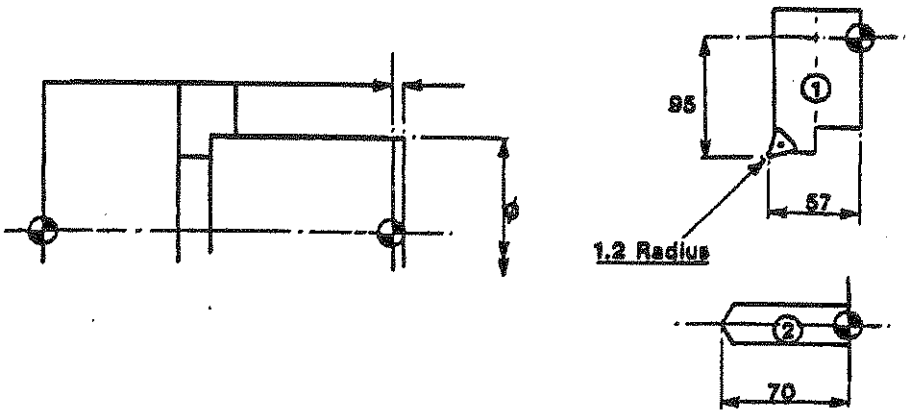
OFFSET TABLE

Offset No.	OFX Offset Amount on X Axis	OFZ Offset Amount on Z Axis	OFR Tool Nose Radius Comp. Amount	OFT Direction of Imaginary Tool Nose
1	0.030	0.020	0.020	1
2	0.060	0.030	0.030	2

The number of offsets is restricted to the number of tools used - i.e. One offset per tool.

TYPICAL PART PROGRAM

The component shown below is to be faced and drilled with the tools shown.



O0002 (TRAINING)

N10 G21 G97 G99 G40

N30 G28 U0 W0

N40 G50 S3000

N60 G96 S240

N80 M06 T0101

N90 M13 G00 X42 Z2

N100 G01 Z0 F0.75

N110 X0

N120 G00 X42 Z2

N130 G28 U0 W0

N140 M06 T0202

N150 G00 X0 Z3

N160 Z-10

N170 G00 Z2

N180 G28U0 W0

N190 M30

DWELL (G04)

A dwell is a programmed time delay, the duration of which is programmed in the "X", "U" or "P" Words.

For the slides to remain motionless for 1 second, the following example is programmed:-

N100 G01 Z10.0 F1.0	Feed to position.
N110 G04 X1.0	1 second duration.
N120 G01 X.....Z.....	Continue.

Alternatively, block N110 could read:-

N110 G04 U1.0	1 second duration.

or

N110 G04 P1000	1 second duration.

BAR FEED, BAR PULL:

Continuous Cycle:

For machines fitted with a bar feed/bar pull system the continuous cycle feature can be used. This feature is activated by:-

Continuous Cycle - N.... M99 P....

If it is programmed before the M30 the machine will automatically repeat the part program.

The continuous cycle can be stopped by:-

a) End of bar signal from the bar feed.

b) Pressing the"CYCLE STOP".

To enable either bar feed or pull operations the spindle must be stopped(M05 active).

a) BAR FEED

The program for bar feed with a standard chuck would be:-

N130 G28 U0 W0 M05.....	(safe start)
N135 M06 T0101	(tool change)
N140 X0 Z50.0 M06 T0101	(bar stop to position)
N160 M10	(chuck open and bar feed)
N170 G98 G04 X1.0	(dwell to ensure operation)
N180 G01 Z200.0 F3000.0	(move to required position)
N200 M11	(close chuck)
N210 G04 X1.0	(dwell to ensure operation)
N220 G28 U0 W0	(move to index position)

b) BAR PULL

The program for bar pull with a standard chuck would be:-

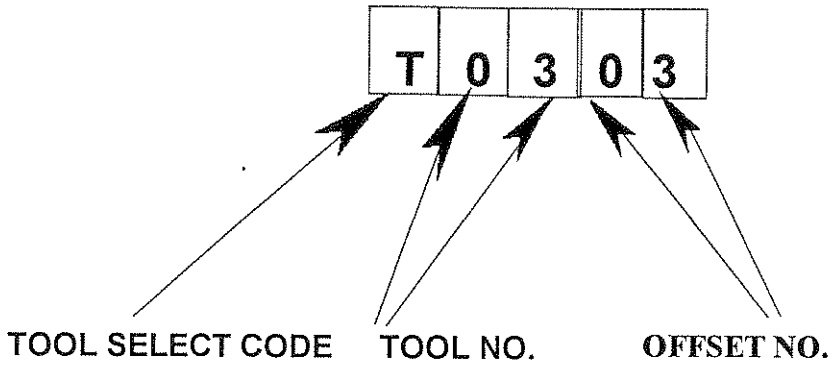
N130 G28 U0 W0 M05	(safe start)
N135 M06 T0101	(tool change)
N150 G00 X0 Z5.0	(move to position)
N160 G04 X1.0	(dwell to ensure spindle stop)
N170 G01 Z-20.0 F3000.0	(bar puller to position on bar)
N190 M10	(chuck open)
N200 G04 X1.0	(dwell to ensure operation)
N210 G01 W20.0	(move to position)
N230 M11	(close chuck)
N240 G04 X1.0	(dwell to ensure operation)
N250 G01 W30.0	(bar puller clear of bar)
N260 G28 U0 W0	(move to index position)

TOOL NUMBERS and OFFSET NUMBERS

When programming the tool number in a part program the rule is as follows:-

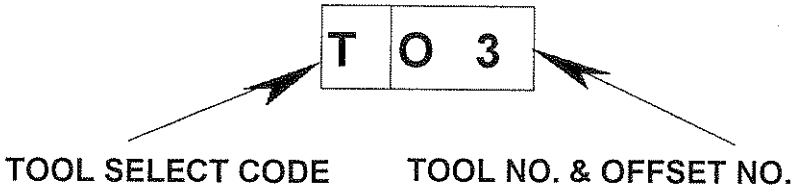
If a four digit number is programmed following the letter T, then the first two digits represent theTool number , and the second two digits represent the Tool Offset number.

EXAMPLE:-



If the digits following the Tool Selecton Code (T) is less than 100 i.e. two digits, then the Tool Number and the Offset Number are the same.

Example:-



	Tool	Offset
T3	3	3
T03	3	3
T103	1	3
T0103	1	3
T0613	06	13
T0100	1	0 (no offset)

The first two digits counting from the R H Side of the code is always the Offset Number.

18. SUBROUTINES, CYCLES AND SAMPLES

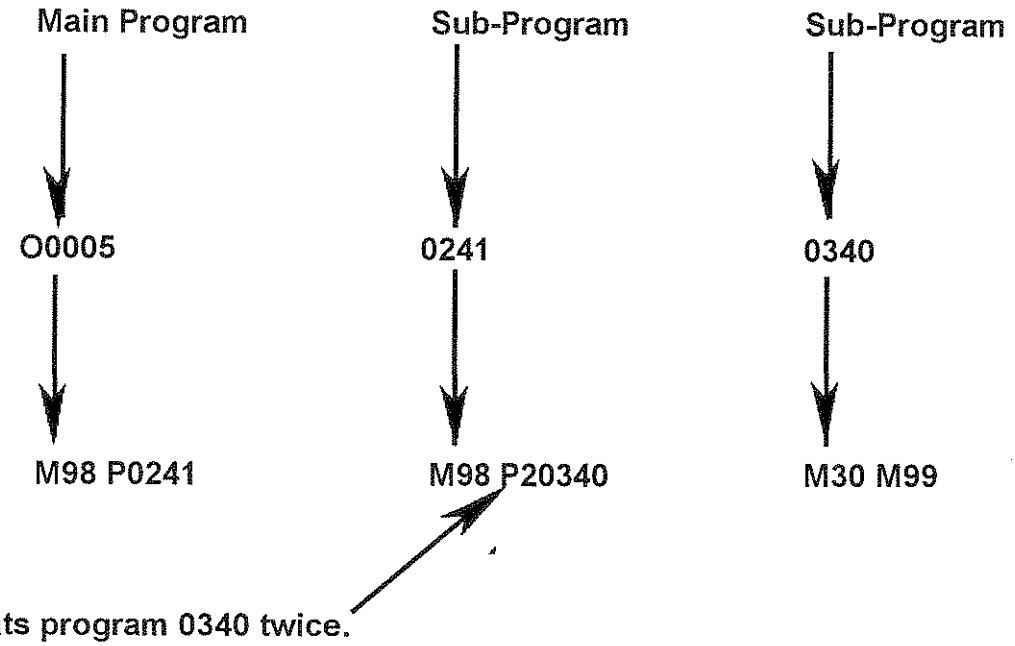
SUBPROGRAMS (SUBROUTINES)

By using the program jump functions, it is possible to simplyfy a machining program with repeated machining or function sequences.

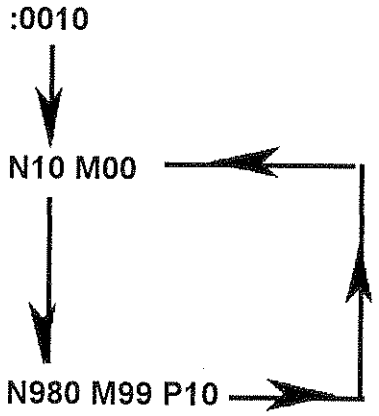
The machining sequences, which are repeated and can be used several times, are stored as subroutines and called up using the program jump functions.

M98 - Jump command to another program.

M99 - Return command.



M99 can be used to return to the start of the program.

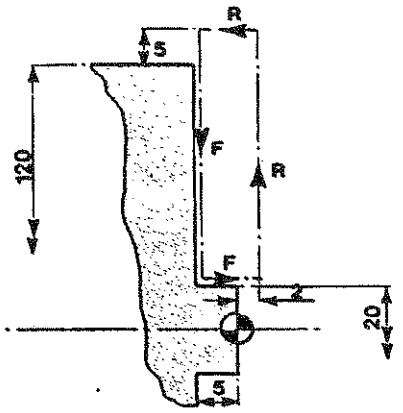


M99 generally indicates the end of a subroutine and allows the jump back to the main program. If it is used with "P" address, this indicates the "Jump To" block number.

The program will read the M99 P10 (GOTO N10)-i.e. automatic return to line ten.

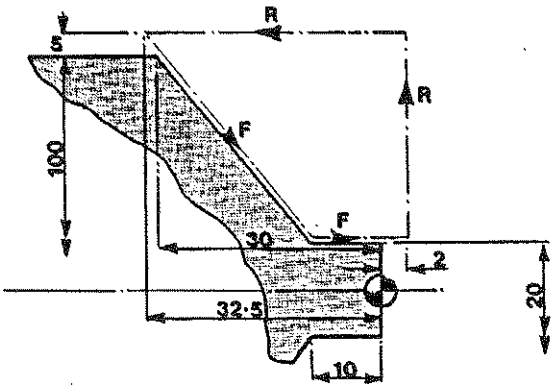
Line N10 must read M00 to stop the cycle for component loading. All information prior to N10-i.e. standard tool geometry-would not be read after the first cycle. Therefore M30 would not be programmed in this case.

G94 FACING CYCLE



N500 G00 X130.00 Z2.0

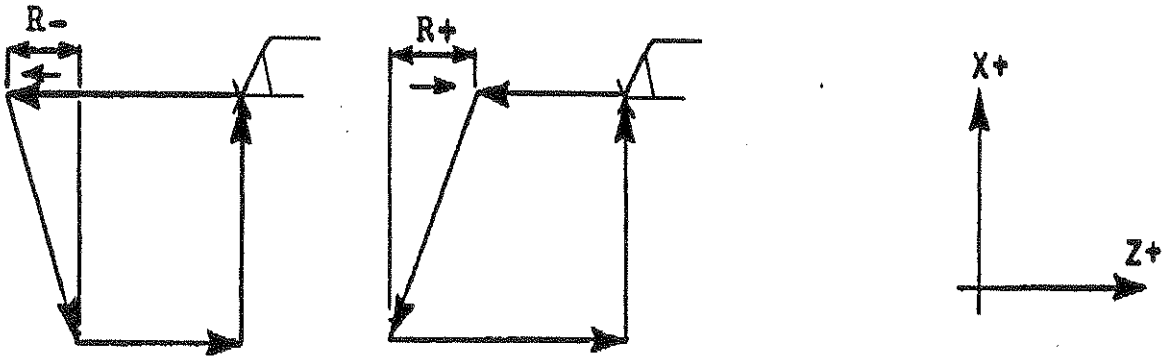
N510 G94 X20.0 Z-5.0F0.3



N400 G00 X130.00 Z2.0

N410 G94 X20.0 Z-10.0 R-22.5.F0.3

The sign of "R"depends on the direction of the taper.



The G94 function in the above figure is a single "box type" cycle. One line of program information will enable the tool to perform the 4 distinct moves.

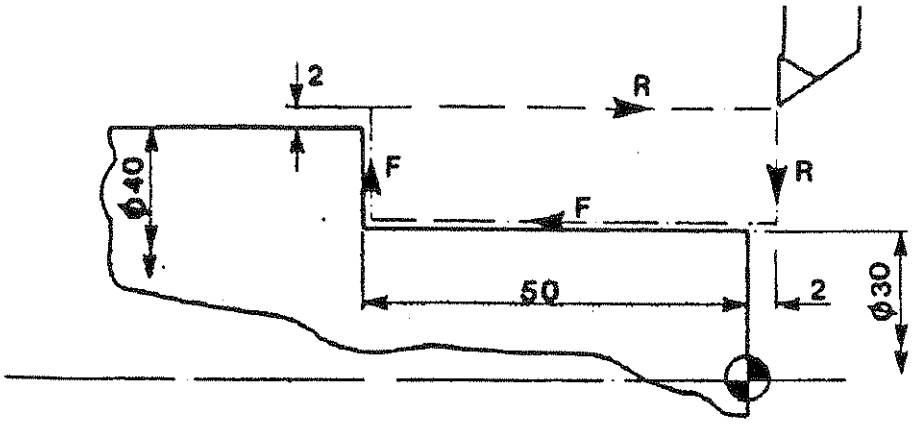
X - Final cutting position.

Z - Final cutting position.

R - Incremental distance to start of cut from final cutting position.

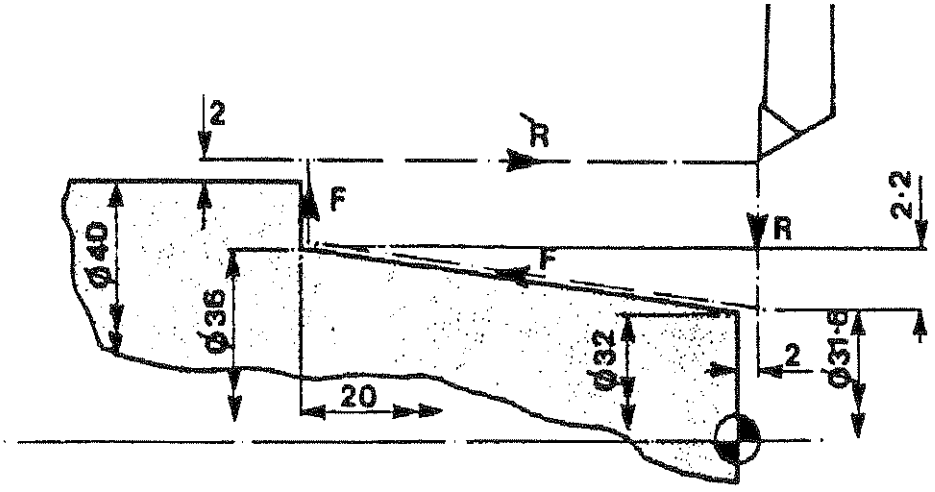
This R function can be omitted for cuts parallel to the X axis.

G90 TURNING CYCLE



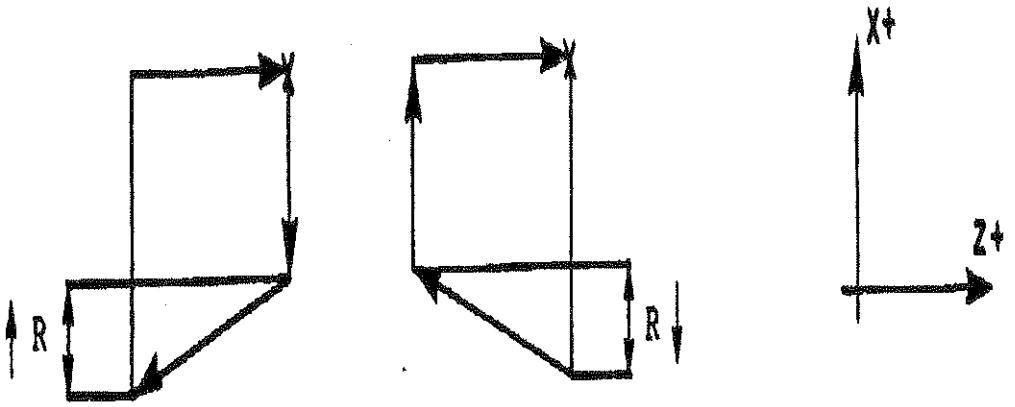
N600 G00 X44.0 Z2.0

N610 G90 X30.0 Z-50.0 F0.09



N700 G00 X44.0 Z2.0

N710 G90 X36.0 Z-20.0 R-2.2 F3



The G90 in the above figure is a single "box type" cycle. One line of program information will enable the tool to perform the four distinct moves.

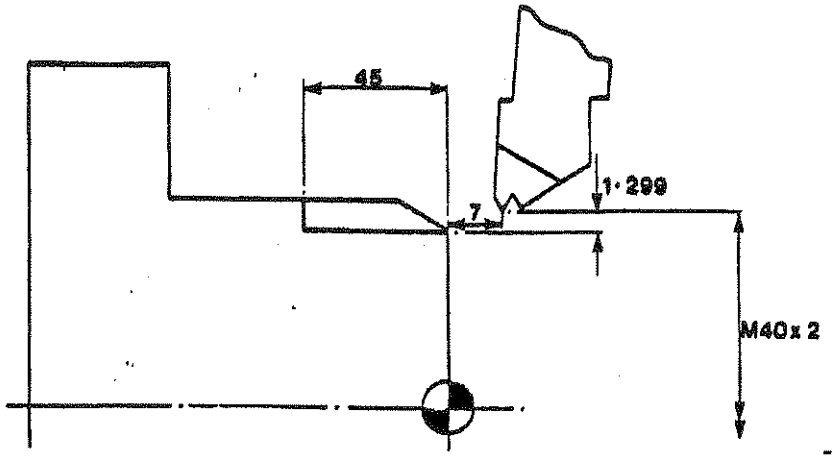
X - Final cutting position.

Z - Final cutting position.

R - Incremental distance to start of cut from final cutting position.

This R function can be omitted for parallel cuts.

CANNED CYCLE (G92) - THREADING



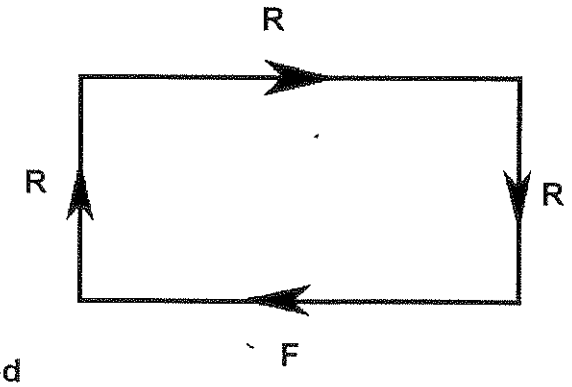
N690 M06 T0505

N700 M13 G00 X50.0 Z7.0

N710 G92 X39.35 Z-45.0 F2.0

N720 X38.954

N730 X38.65 etc.

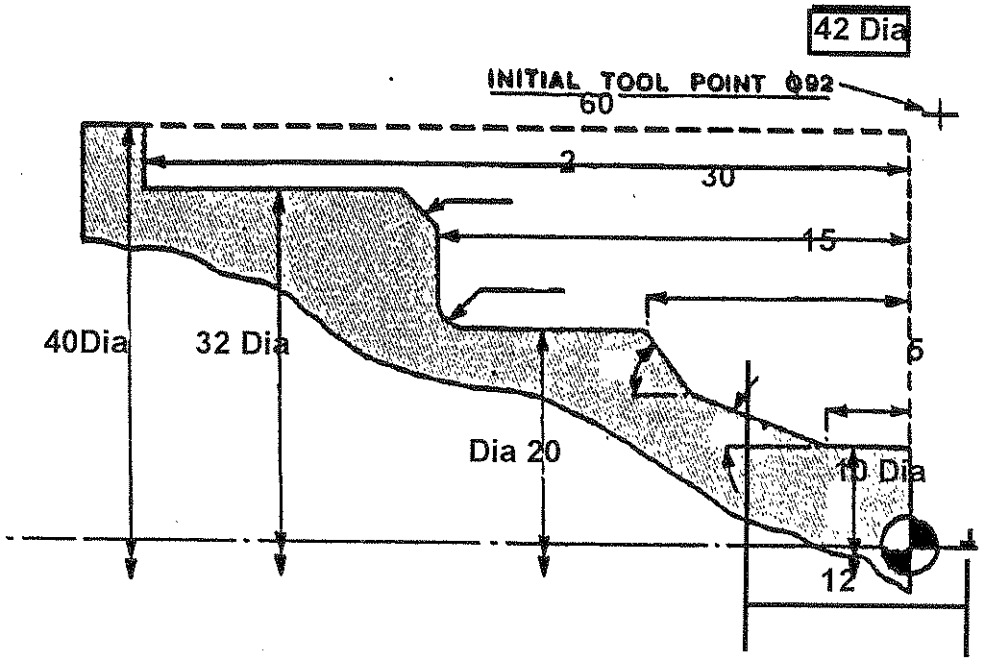


Tool path F = Feed

R = Return

G71 STOCK REMOVAL TURNING.

This feature generates cycles to remove material to a predefined contour. The contour is defined in part program blocks. The main application of this cycle is for bar stock or solid material. G71 is used when the major direction of cut is in the"X" axis.



N50 G00 X42.0 Z2.0

N55 G71 U1.5 R1

N60 G71 P70 Q160 U1.0 W.13 F0.075

N70 G01 X10

N80 Z-5 F0.1

N90 X15 Z-12

N100 X20 Z-15

N110 Z-27

N120 G02 X26 Z-30 R3

N130 G01X28

N140 X32 Z-32

N150 Z-60

N160 X40

N55 G71 .....etc.

U - Depth of cut

R - Tool relief amount

N60 G71 ..... etc.

P - Sequence number of the first block defining the contour.

U - X Axis stock allowance for finishing.

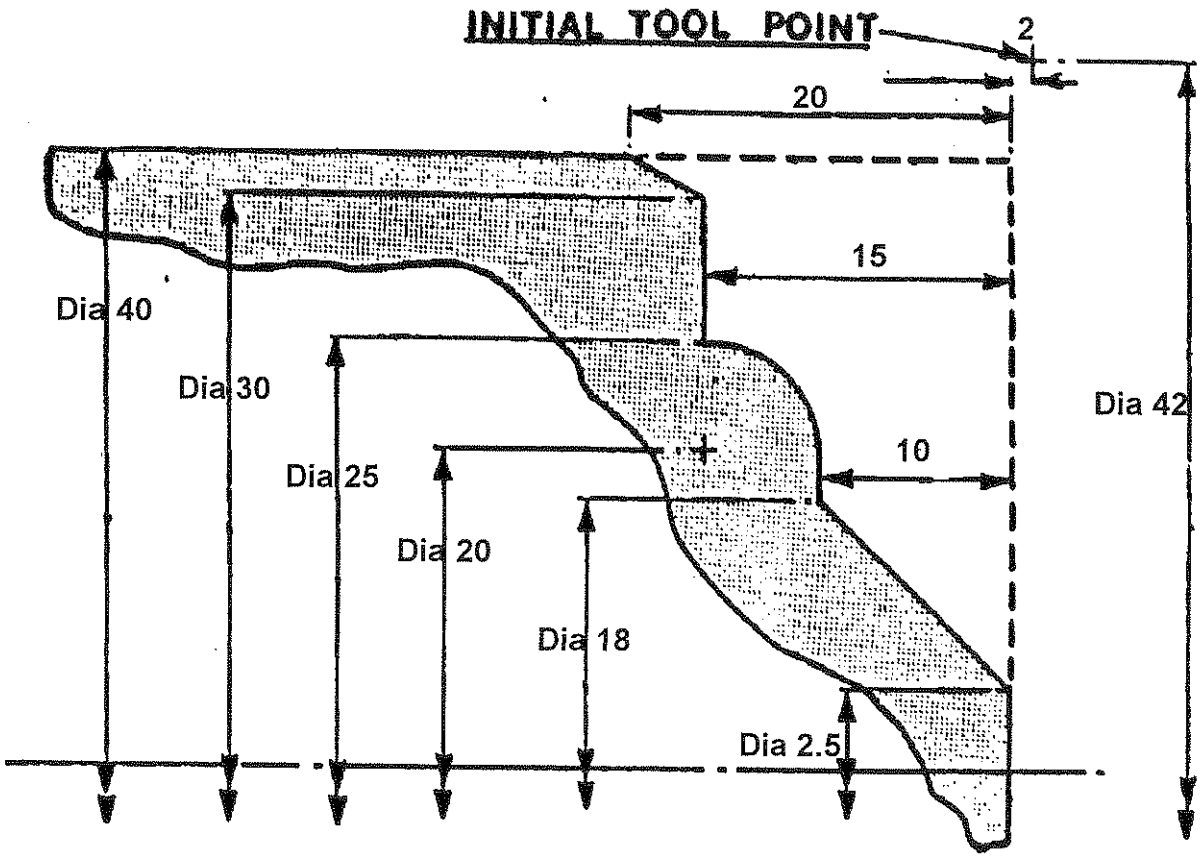
Q - Sequence number of the last block defining the contour.

W - Z axis stock allowance for finishing.

F - Feedrate for roughing cycle.

G72 STOCK REMOVAL FACING

This cycle is similar to G71 and is used when the major direction of cut is in the"Z" axis.



N40 M06 T0 808

N50 G00 X40 Z2.0

N55 G72 W1.0 R1

N60 G72 P70 Q130 U0.1 W1.0 F0.15

N70 G00 Z-20.

N80 G01 X40.0

N90 X30.0 Z-15

N100 X25.

N110 G02 X20.0 Z-10 R5

N120 G01 X18.0

N130 X2.5 Z0

G70 P70 Q130

N55 G72 .....etc.

W - Depth of cut

R - Tool relief amount

N60 G72 ..... etc

P - Sequence number of the first block defining the contour

Q - Sequence number of the last block defining the contour

U - X axis stock allowance for finishing (diameter).

W - Z axis stock allowance for finishing

F - Feed rate for roughing cycle.

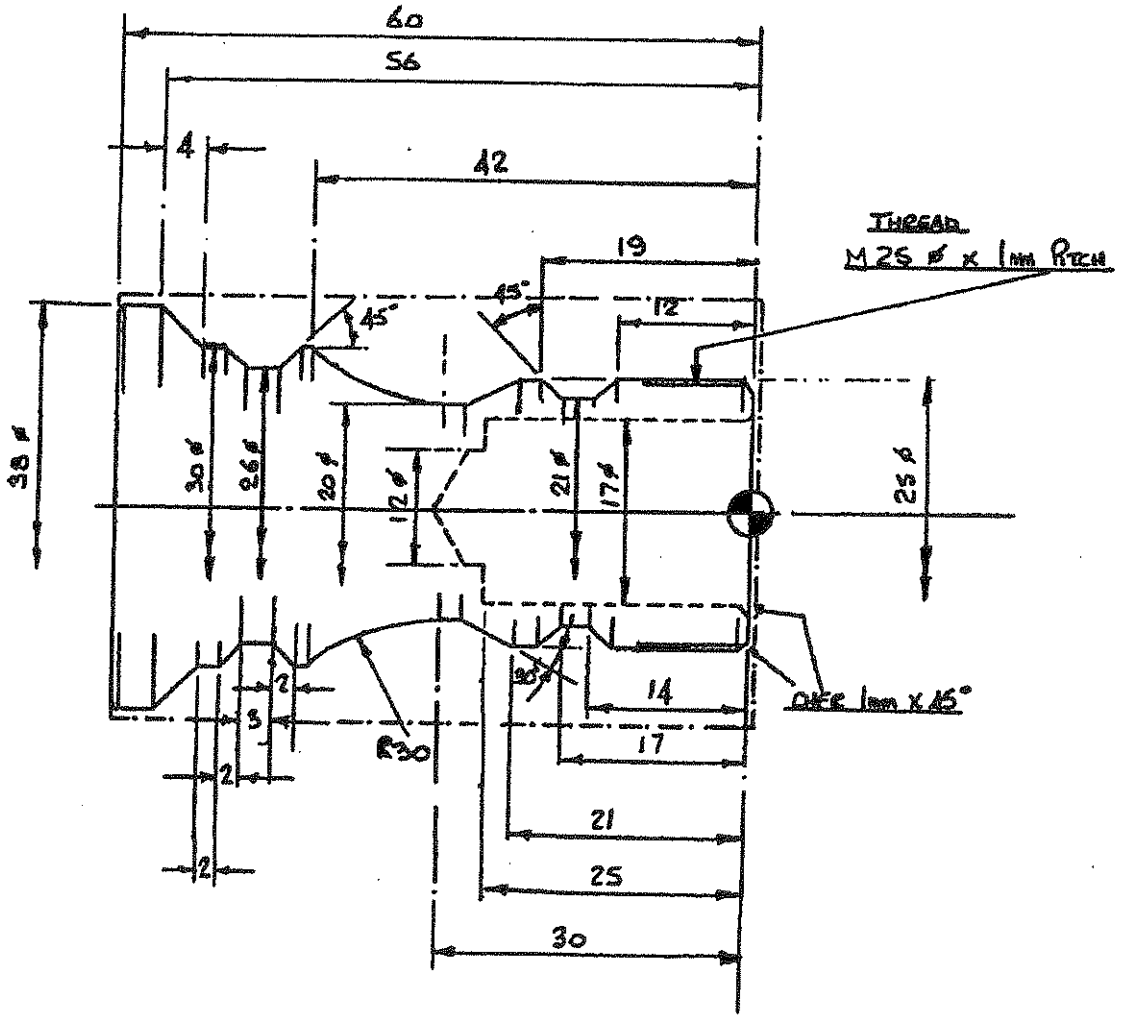
G70 FINISHING CYCLE

N200 G70 P70 Q120

P & Q - As above. Feed rates included in the blocks are active.

EXAMPLES

DENFORD TEST PIECE (BLANK SIZE: 40mm DIA x 80 LONG)



TOOLING SHEET : **TOOLING SHEET**

CUSTOMER	DENFORD LTD
COMPONENT	TEST PIECE
DRG.NO.	
OPN. 1	DATE
PREP. BY	

TOOLING DETAILS

FACE & TURN

CENTRE DRILL

DRILL

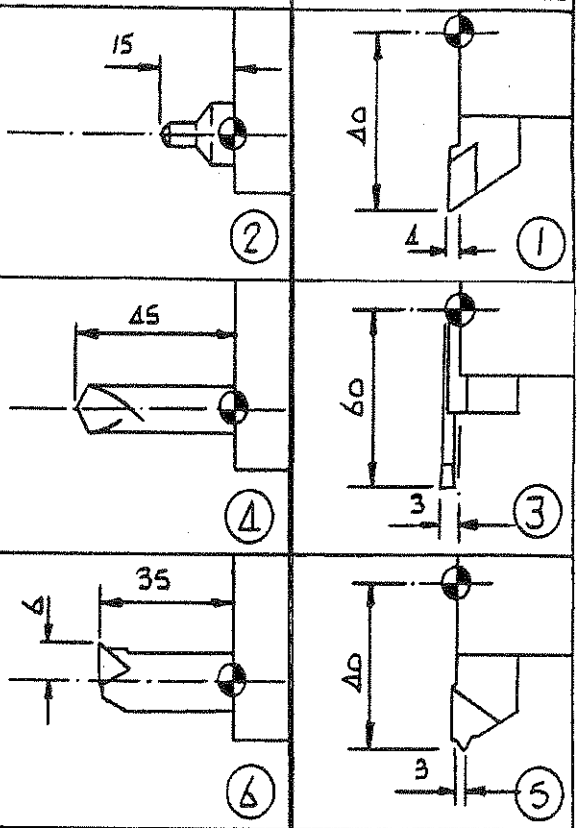
THREAD

BORE

PART OFF. CUT GRIP

INTERNAL

EXTERNAL



PROGRAMMING SHEET

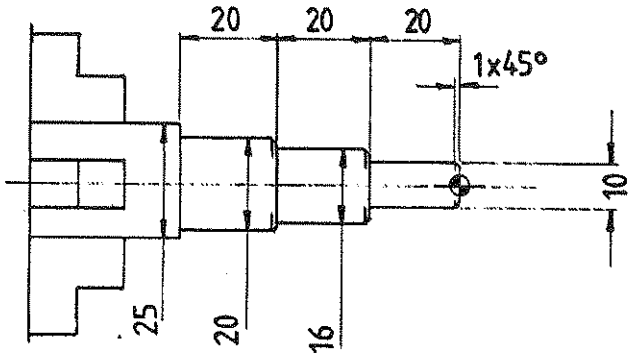
```
N10  G50 S2500
N20  G96 S200
N30  G21 G99 G40 M13
N40  M06 T01
N50  G00 X44 Z0
N60  G01 X-1 F0.15
N70  X44 Z2
N80  G71 U1.5 R1
N90  G71 P100 Q190 U0.5 W0.1 F0.1
N100 G01 X23 F0.4
N110 Z0
N120 X25 Z-1
N130 Z-21
N140 X20 Z-28
N150 Z-30
N160 G02 X30 Z-42 R30
N170 G01 Z-52
N180 X38 Z-56
N190 Z-64
N200 G70 P100 Q190
N210 G28 G97 U0 W0
N220 M06 T02 S2000
```

```
N230  G00 X0 Z2

N240  G01 Z-6 F0.1

N250  G00 Z5

N260  G28 U0 W0

N270  M06 T04 S1200

N280  G00 X0 Z2

N290  G01 Z-20 F0.15

N300  G00 Z2

N310  Z-19

N320  G01 Z-30

N330  G00 Z2

N340  G28 U0 W0

N350  M06 T05 S325

N360  G00 X25.25 Z5

N370  G76 P030060 Q100 R0.02

N380  G76 X23.773 Z-10 P0613 Q250 F1.0

N390  G28 U0 W0

N400  M06 T06 S2000

N410  G00 X11.75 Z2

N420  G71 U1.5 R0.5

N430  G71 P440 Q480 U-1 W0.1 F0.075

N440  G01 X19 F0.4

N450  Z0 F0.1
```

```
N460  X17 Z-1

N470  Z-25

N480  X11.75

N490  G70 P440 Q480

N500  G28 U0 W0

N510  M06 T03 S1500

N520  G00 X26 Z-17

N530  G01 X21 F0.07

N540  G00 X26

N550  Z-15

N560  G01 X25

N570  X21 Z-17

N580  G00 X26

N590  Z-19

N600  G01 X25

N610  X21 Z-17

N620  G00 X31

N630  Z-48

N640  G01 U-5

N650  G00 U5

N660  W2

N670  G01 U-1

N680  U-4 W-2
```

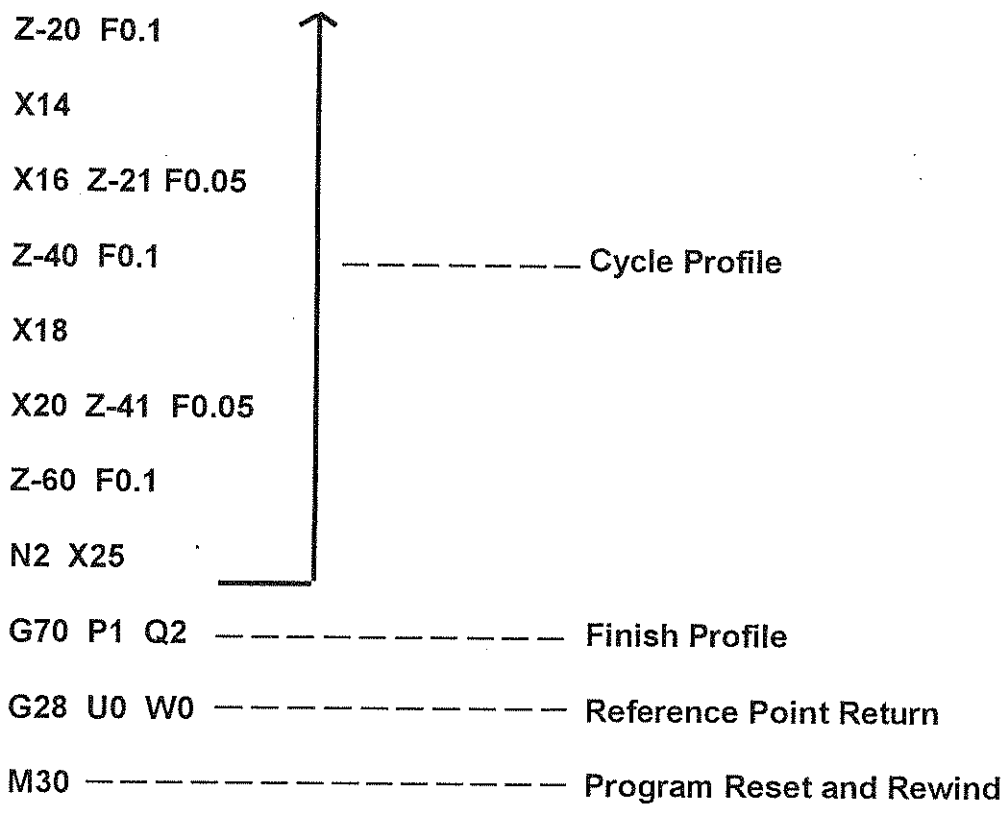
```
N690  G00 U5

N700  W-2

N710  G01 U-1

N720  U-4 W2

N730  G00 X40

N740  Z-63

N750  G01 X36 F0.08

N760  G00 X40

N770  Z-62.5

N780  G01 X38

N790  X37 Z-63

N800  X-0.5

N810  G28 U0 W0 M05

N820  M30
```

SAMPLE PROGRAMS

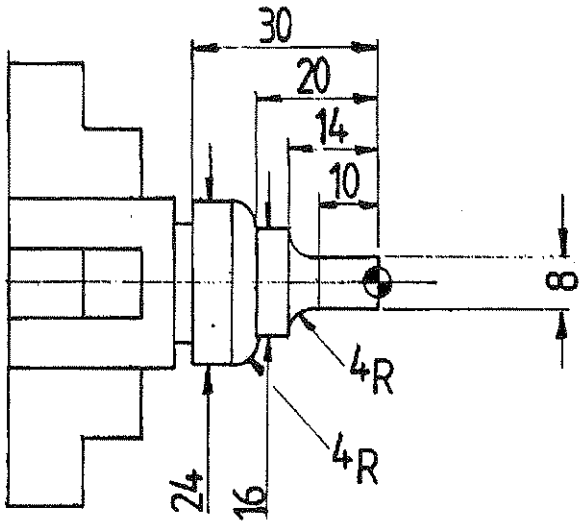


SAMPLE PROGRAM .....No. 1001

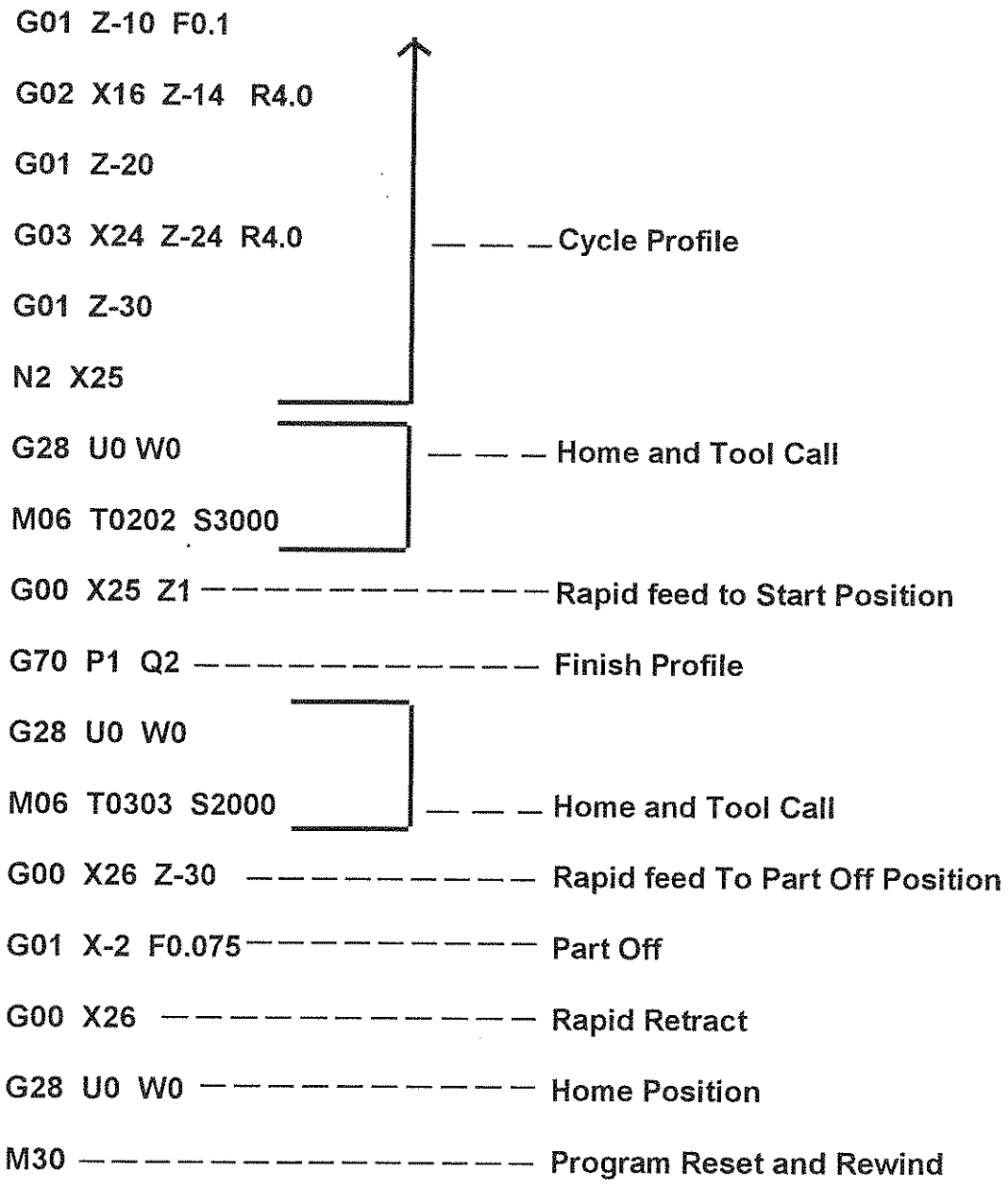
```
[BILLET X25 Z65. ---------- Billet Size
O 1001 ---------- Program Number
G 99 G21 G40 G97 S 2000 M13 --- Set Cuting Conditions
M06 T0101 ---------- Tool Call
G00 X26 Z2
G01 Z0 F0.3 ---------- Face and Retract
    X-1 F0.1
G00 X25 Z1
G71 U1.5 R0.5 ------ Set Parameters for
G71 P1 Q2 U1 W0.1 F0.125 Canned Cycles
N1 G00 X8
G01 Z0 F0.1
X10 Z-1 F0.05
```


Z-20 F0.1
X14
X16 Z-21 F0.05
Z-40 F0.1
X18
X20 Z-41 F0.05
Z-60 F0.1
N2 X25
Cycle Profile
G70 P1 Q2
Finish Profile
G28 U0 W0
Reference Point Return
M30
Program Reset and Rewind

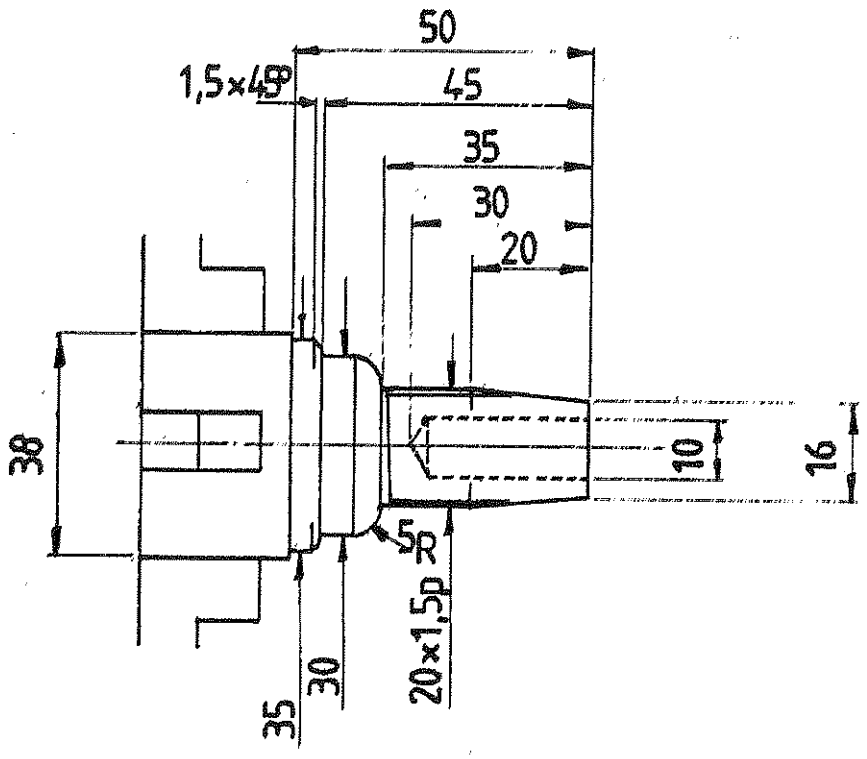
SAMPLE PROGRAM No 1002




```
[BILLET X25 Z35 ---------- Billet Size
O 1002 ---------- Program Number
G99 G21 G40 G97 S2500 M13 ---- Set Cutting Conditions
M06 T0101 ---------- Tool Call
G00 X26 Z2
G01 Z0 F0.2 ---------- Face and Retract
      X-1 F0.1
G00 X25 Z1
G71 U1.5 R0.5 ---- Set Parameters for Canned
G71 P1 Q2 U1 W0.1 F0.125 Cycle
N1 G00 X8
```

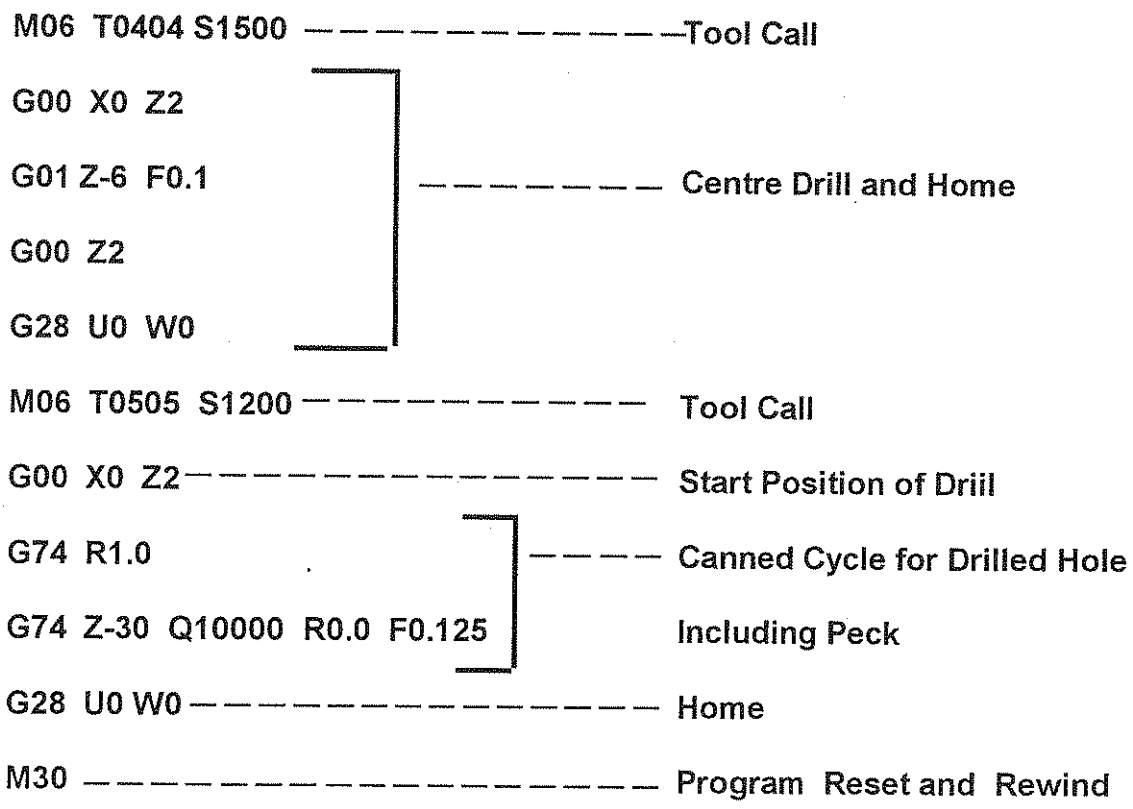

G01 Z-10 F0.1
G02 X16 Z-14 R4.0
G01 Z-20
G03 X24 Z-24 R4.0
G01 Z-30
N2 X25
Cycle Profile
G28 U0 W0
M06 T0202 S3000
Home and Tool Call
G00 X25 Z1
Rapid feed to Start Position
G70 P1 Q2
Finish Profile
G28 U0 W0
M06 T0303 S2000
Home and Tool Call
G00 X26 Z-30
Rapid feed To Part Off Position
G01 X-2 F0.075
Part Off
G00 X26
Rapid Retract
G28 U0 W0
Home Position
M30
Program Reset and Rewind

SAMPLE PROGRAM No. 1003




```
[ BILLET X38 Z55 ---------- Billet Size
O 1003 ------------------- Program Number
G99  G21 G40 G50  S3000---- Set cutting Conditions, Max RPM
G96  S150  M13  ---------- Constant  Surface Speed, Start
M06  T0101                 Spindle, Coolant on, Tool Call
G00  X39  Z2 ------------- Rapid feed to Start Position
G01  X-1 F0.125 ---------- Face and Retract
G00 X38  Z1
```

```
G71 U1.5 R0.5                    ]
                                 ]  ---- Set Parameters for Canned
G71 P1 Q2 U1 W0.1 F0.125         ]          Cycle
N1 G00 X16                       ]
G01Z0 F0.1                       ]
      X20 Z-20                   ]
      Z-35                       ]
G03 X30 Z-40 R5.0                ]  ---- Cycle Profile
G01 Z-45                         ]
      X32                        ]
      X35 Z-46.5                 ]
      Z-50                       ]
N2 X38                           ]
G28 U0 W0                        ]
                                 ]  ---- Home Position and Tool Call
M06 T0202                        ]
G00 X43 Z6                       ]
                                 ]  ---- Positioning and Applying
G01 G42 X38 Z1 FO.3              ]          Cutter Compensation
G70 P1 Q2 ---------------------- Finishing Cycle
G28 G40 U0 W0 ------------------ Cancel Cutter Compensation & Home
G97 M06 T0303 S900 ------------- Tool Call & Change Speed to RPM.
G00 X20.25 Z0 ------------------ Rapid to Thread Start Position
G76 P030060 Q100 R0.05           ]
                                 ]  ---- Threading Cycle
G76 X18.16 Z-33 P0920 Q250 F1.5  ]
G28 U0 W0 ---------------------- Home
```


M06 T0404 S1500
Tool Call
G00 X0 Z2
G01 Z-6 F0.1
G00 Z2
G28 U0 W0
Centre Drill and Home
M06 T0505 S1200
Tool Call
G00 X0 Z2
Start Position of Driil
G74 R1.0
G74 Z-30 Q10000 R0.0 F0.125
Canned Cycle for Drilled Hole
Including Peck
G28 U0 W0
Home
M30
Program Reset and Rewind

19. APPENDIX 1.
HELP FILES AND TOOLSHAPES.

HELP SCREENS

All the Help Screens which appear in the software can be configured, amended and added to by the end user. This is not, however, an easy task and care must be taken to produce good results.

All the Help is stored in two files on Disk:-

1 - The compiled version is "FANUCSLD" extension.

2 - The text version is "FANUC.TXT" extension.

The Text file can be loaded into any word processor, or even into the CNC Editor. Changes can be made and the new file can be saved to DISK.

IMPORTANT

Keep the original copy of FANUC.TXT, in case the changes you have made result in errors during compilation.

Once the changes have been made and a new file has been saved to DISK, it is possible to produce a compiled version. It is the compiled version which is integrated within the software- simply changing the Text file will not alter the Help used by the software.

HOW TO COMPILE A NEW HELP FILE.

On the Drivers Disk supplied with system, there is a file called "MESSAGES.EXE". This will have been copied over when you installed the system if you are on a Hard Drive.

COMPILING FROM HARD DRIVE.

Make sure that the new file is stored in the same Directory as the software, and that you are in that Directory. For example if you have created a Directory called "DENFORD" then type :-

CD\DENFORD

Then Type:-MESSAGESFANUCSLD

You will have to wait a few seconds before a compilation of the completed message appears. If you are successful run the software again to see the changes. If an error shows, read the following section on How Help Screens Work.

COMPILING FROM FLOPPY DRIVE

Make sure the new file is saved on the Drivers DISK where the MESSAGES.EXE file exists.

Type:-MESSAGES FANUCSLD

A new FANUC.MSG file will be created which can be copied over on to the main program DISK, overwriting the original Help.

HOW HELP SCREENS WORK

The Help file contains different types of information. to locate a particular entry, it is necessary to understand the structure of the Help file.

HELP FILE STRUCTURE

Comments example :- (THIS IS A COMMENT

Comments start with an open bracket and can be inserted anywhere in the Help file. They are ignored by the compiler, and are only there to help the end user document

the Help file.

Page Start Example |HELP PAGE1 A|

Example |G and M Code Errors|

Each Page Start begins with a tramline character followed by the page name. Actual Help Pages have a designated letter from A to Z. A page start can be :- help, start of error messages, or information text.

IMPORTANT

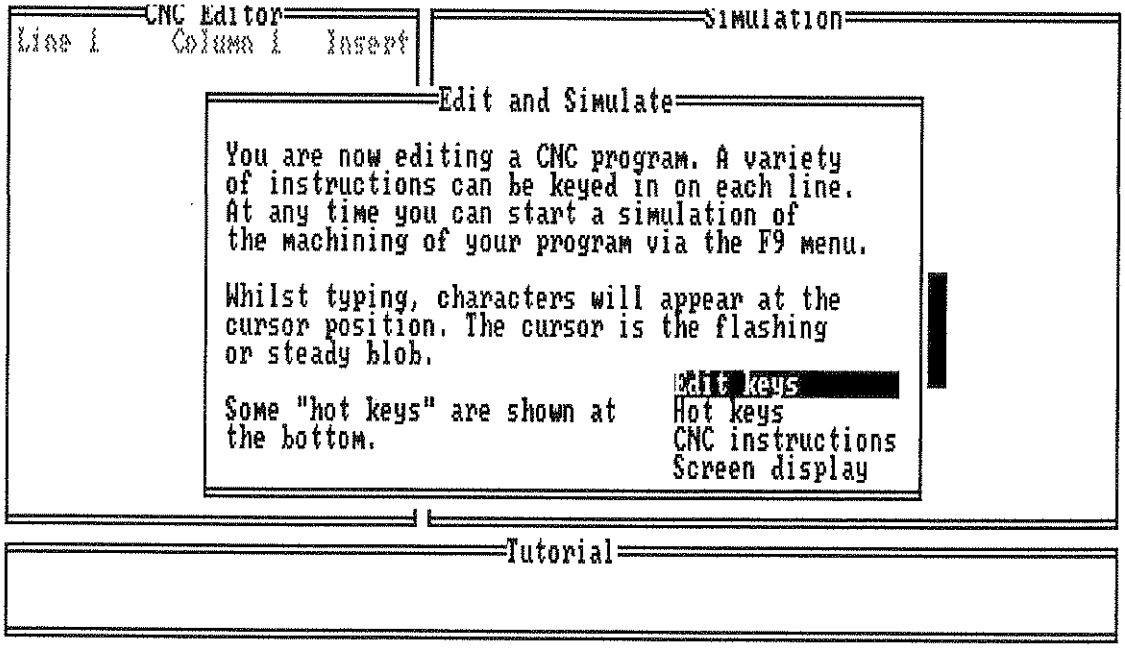
It is vital that the pages relating to error messages, fast key strips and text which are used directly by the software are not altered. The software relies on each line being in sequence. Help screens that are used with the F1 key can be placed anywhere, but we suggest you put them at the end of the program.

CONTEXT SENSITIVE HELP (FOR HELP SCREENS ONLY)

HOW IT WORKS

To insert a new Help Page you must first decide what type of Help you require and where in the program the Help should be placed.

For example, we might like a lesson on CNC programming to be available by pressing



the F1 key during edit mode. At the moment pressing the F1 key during edit displays:-

HOW TO ADD A CNC LESSON AS A SUB MENU:-

· 1 - Decide what the menu option should be called. In this case, we shall call it "CNC Lessons".

· 2 - Choose a name for the Help Page we want to display when we select CNC Lessons. In this case LESSON1.

· 3 - Load the Help File into the CNC Editor and locate the Help page displayed above. You will find the start of the page near to line 676. The page start is |EDIT AND SIMULATE|which is called up by pressing F1 when in the Edit and Simulate Mode.

· 4 - At the bottom of the Help Text, you will see the sub menus being referenced.

· 5 - Add #CNC Lessons#LESSON1 A to list.

· 6 - Go to the end of the file to write the Help screen for LESSON1.

· 7 - Type |LESSON1 A|.This is the start of Lesson1 and the page number is"A". Always use capital letters for Page Starts.

· 8 - On the next line, write a title for the Help Screen-try CNC Lesson One.

· 9 - Now you can write the actual text - making sure the lines are no longer than 47 characters.

· 10 - If you want more room for the CNC Lesson, you can start a new page by typing |LESSON1 B|

CNC Lesson one Page 2 - Followed by the Text.

· 11- If you want sub-menus to be displayed on your new Help Page, simply place the referencing commands at the bottom of the Help Page. Example #G and M code#HELP PAGE1 A

· 12 - Upon completion, save the new Help File and compile to Disk. Run the software to test the changes.

FAST KEYS

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

The Fast Keys change during menu changes and are constantly updated by the Key Strip.The following is a list of FAST KEYS:-

· F1	Get Context Sensative Help
· F2	Quick save CNC program
· F3	Quick load CNC program
· F5	Get information
· F9	Check/run CNC programs
· F10	Get main menu

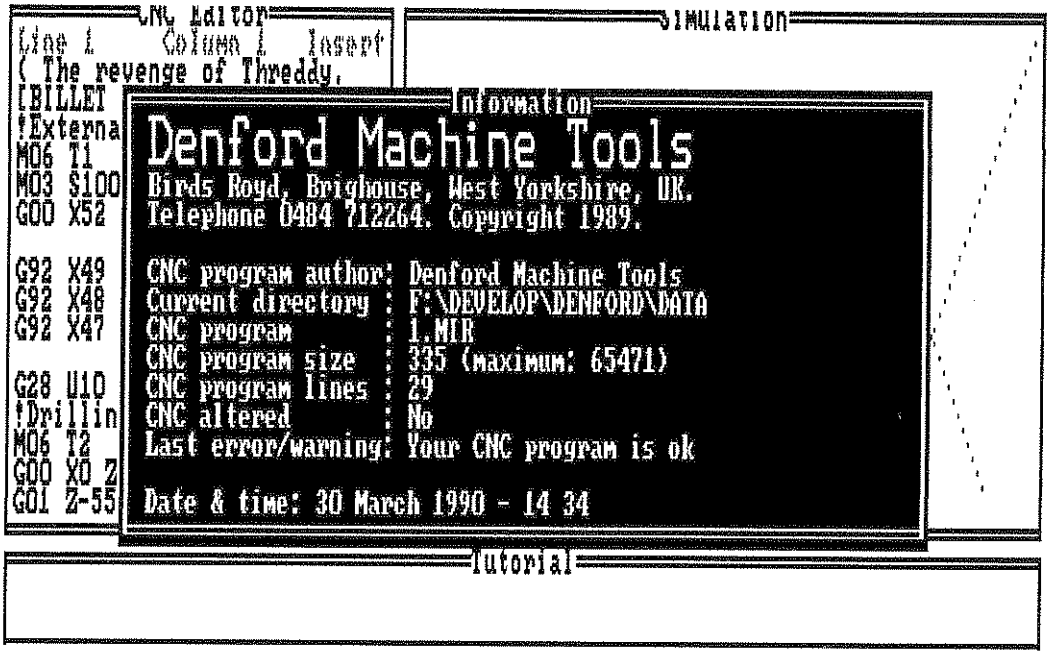
HELP SCREENS

Context sensitive Help is available at all times, with sub-related topics.

The following keys are used:-

· F1	To select Help
· Ctrl F1	To select G and M code Help.
· F1	To select Sub Related Help.
· Arrow Keys	To select from Help Options.
· PageUp/Down	To display Help Pages.
· RESET	To exit.

INFORMATION WINDOW



Press F5 to display the Information Window.

Press Page Down to view additional information window.

Press RESET key to clear.

DEFINING TOOLSHAPES.

Toolshapes can be written in text format or passed over to the system from CAD, using the DXF interface.

Each toolfile contains the following:-

Any number of comments, preceded by the open bracket "("; and a name for the tool (this can be a tool reference number, or a short description).

Two lines may be used for an information title. This will be displayed with the tool-shape. It is advisable to record the sizes of the toolshape, especially boring bars.

The geometry of the toolshape in absolute will follow. These are all line or arc moves. This format is compatible with DXF geometry. Simple shapes can be typed in. A drill, for instance, changes in diameter, so taking an existing drill shape and changing the diameter can be done in the editor.

CONVERTING TOOLSHAPES

Load FANUCL.GRT file into an editor and add a toolshape at any position in the file before or after an existing toolshape.

Save the file to disk.

DXF TO TOOLSHAPE FILE

To convert from DXF to a toolshape data, use the file DXF_TEXT.EXE. It is only possible to convert one shape at a time. Type DXF_TEXT.EXE followed by the Drive letter,Directory and DXF filename. Example :- DXF_TEXT.EXE C:\ACAD\TOOL1

Once the DXF file has been converted, it can be merged into the toolfile, and the toolshape name and information added.

NEW TOOL SHAPES

PROCEDURE FOR ADDING NEW TOOL SHAPES USING CAD

Draw the new tool shape using actual dimensions, positioning the tool datum point at X0 Y0.

Save as DXF format, eg., TOOL.DXF

Run DXF-TEXT, eg., DXF-TEXT TOOL

Run Denford Offline Software, eg., FANUCSLD

Load FANUCSLD.GRT

Position the cursor at the end of the file.

Load TOOL.GRT (answer YES when asked to merge)

Save as FANUCSLD.GRT

Run TEXT-SHP eg., TEXT-SHP FANUCSLD

The new tool is now included in the tool library.

THE FANUCSLD .GO FILE

FANUCSLD.GO is a file supplied on the DISK containing the list of "drivers" used by the software. The file indicates the order in which the drivers are loaded into the memory when the system first starts up.

DefaultFANUCSLD.GO

IBM.RS2 1

$2F8 3 MACHINE

DENFORD. KBD 1

18.2 $3F8 4

EGA.CRT 0

FANUCSLD.APP

Some drivers require a number of additional lines as parameters. The number of lines required by the driver is shown by a number written after the driver name.

eg., The IBM.RS2 driver loads COM1 at the address $3F8 using interupt 4 and MACHINE at the address $3EB using interupt 4.

The last line in the file executes the tutor software specified, ie., FANUCSLD.APP.

EDITING THE .GO FILE

In some circumstances drivers other than the defaults may be required for your computer setup. This file can be edited in any DOS text file editor.

Example:- To change from an EGA driver to a CGA driver, load theFANUCSLD.GO file into the editor. Change line 6 from EGA.CRT 0 to CGA.CRT 0. Save the changes made to DISK.

NB. Make sure that the file "CGA.CRT" can be found on the DISK that is being used, if not it can be copied from the "DRIVERS DISK".

Example:- To change the tutor connection from COM1 to COM2. Load the FANUCSLD.GO File into the editor. Change line 5 from 18.2 $3f8 4 TO 18.2 $2f8 3. save the changes made to DISK. When run, the software will now expect the tutor to be connected to COM2.

20. APPENDIX 2 CUSTOMISING SETTINGS FILE.

TYPICAL SETTINGS FILE

Most of the settings are self explanatory, but selecting from the Menu Options is easier than editing the file. Some settings options can only be changed by loading a Settings file, and adding the extra options. Because a control might have changed over the years, it is possible to change various settings, to make the software compatible with all controls.

The vast majority of end users won't have any modifications at all.

An example of a setting change that might be required can be seen below, where the HIGHSPINDLE has a value of 5000rpm. The same control might be on a machine with a faster spindle motor, so the top spindle speed will be higher. If the traverse distance on the "X" axis is different this can be changed by editing the MOVERANGEX value.

IMPORTANT

The parameters in the Settings file are always in Metric.

```
# Fanuc Lathe Settings File

UTILS_MENU_1 Dos Access

UTILS_GO_1 dos

AUTOERROR 0

LOWSPINDLE 50

HIGHSPINDLE 5000
```

MOVERANGEX 200

MOVERANGEZ 170

MACHINEEXTENTX 300

MACHINEEXTENTZ 190

BILLETX 40

BILLETZ 90

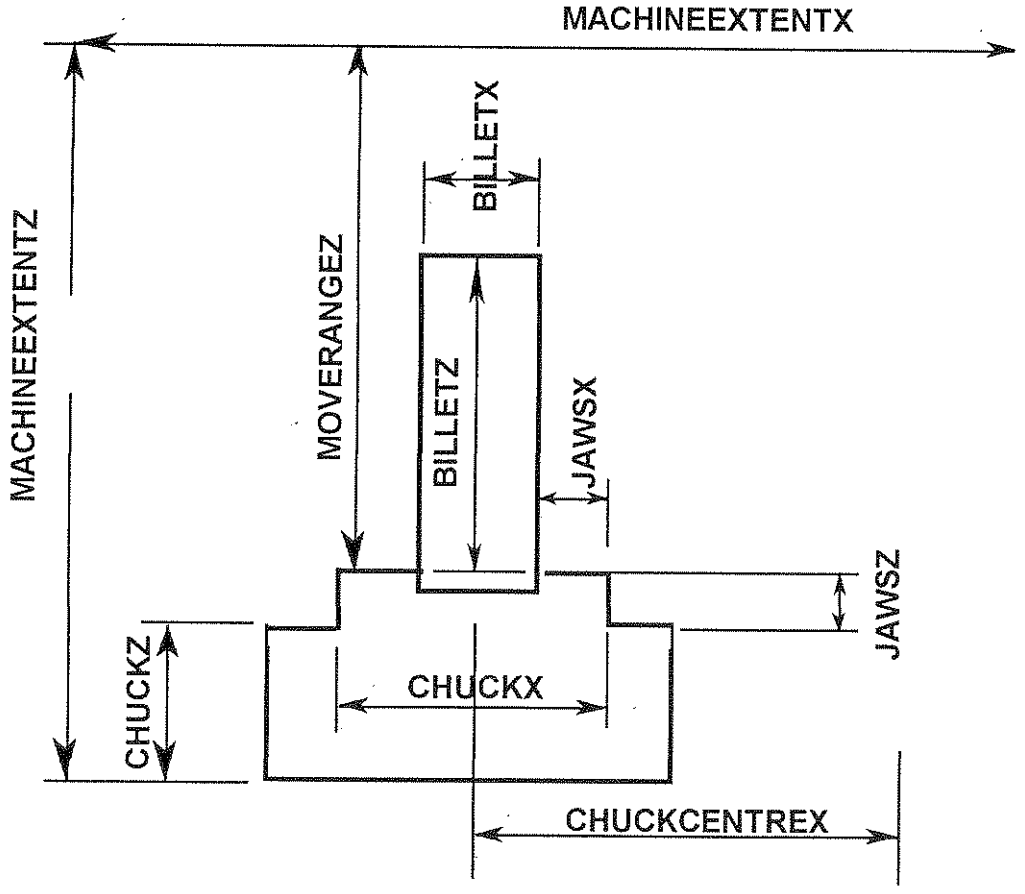
CHUCKX 80

CHUCKZ 30

JAWSX 10

JAWSZ 10

CHUCKCENTREX 200



SHAPE1 TOOL15

SHAPELR1 1

SHAPEFB1 1

SHAPE2 20MMDRL

SHAPELR2 1

SHAPEFB2 1

SHAPE3 TOOL26

SHAPELR3 1

SHAPEFB3 1

SHAPE4 TOOL25

SHAPELR4 1

SHAPEFB4 1

SHAPE5 TOOL14

SHAPELR5 1

SHAPEFB5 1

PRINT_DEVICE 0

PRINT_DDEVICE PRN

PRINT_BAUD 1200

PRINT_PARITY 2

PRINT_PROTOCOL 2

PRINT_STOPBITS 2

PRINT_USELF 1

PRINT_FFNULLS 0

PRINT_PW 80

PRINT_PL 64

PRINT_LM 10

PRINT_TM 20

PRINT_BM 10

PRINT_CRNULLS 0

COMPUTER_SDEVICE COM1

METRIC 1

DISPLAYSIZE 1

21. APPENDIX 3 GLOSSARY OF TERMS

ARC A portion of a circle

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

ASCII American Standard for Information Interchange.

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

AUXILIARY FUNCTION The function of a machine other than the co-ordinate commands.-eg:- F00, S100, T0100, M08.

AXIS X axis: Line perpendicular to the spindle centre- line (absolute).

U axis: Line perpendicular to the spindle centre -line (incremental)

Z axis: Line parallel with the spindle centre -line (absolute)

W axis: Line parallel with the spindle centre -line (incremental)

BINARY A system for describing numbers using only two digits.

BIT The smallest programmable unit (i.e 1 or 0) in machine code. 8 Bits = 1 Byte

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

BLOCK DELETE Is a feature which provides means for skipping blocks (which have been preceded by a slash/code) at the discretion of the operator.

BUFFER STORAGE Is an information storing area which is utilised to transfer the

stored data to active storage almost instantaneously.

CANNED CYCLE Is an automatic motion cycle which is held in buffer storage for the duration of the amount of cycle repeats programmed.

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

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

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

CNC Computer numerical control See SOFTWIRED N.C

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

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

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

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

C.P.U. Central Processing Unit.

DECODE The traslation from tape coded language to control recognisable language.

DRY RUN The use of this function enables the operator to run through the program replacing the programmed feed rates with the maximum jog feed, to test run the program and check tool clearance.

DWELL A programmed time delay.

EDIT Having put the program into memory, the machine can then be operated from memory and suspect blocks of data can then be displayed for examination. The faulty block can then be erased or correct information put in its place by means of MDI and the EDIT facility. The batch can then be run off entirely from the memory. The

program information can then be stored on disk.

EIA Electrical Industries Association has been responsible for setting many N.C standards, one being tape coding information.

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

END OF BLOCK A character inputted in the program which denotes the end of a blockof data.

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

FEED HOLD At any stage in the cycle, the operator can stop the machine movements using the cycle stop key.

FEEDRATE Is commanded in the C.N.C. program by the F word and used by the control to drive the slides. The rate of feed may be programmed as FEED/MIN or FEED/REV.

FEEDRATE Is an interrupt used to hold slide motion.

FEEDRATE OVERRIDE A manual function can be used by the operator to override the programmed feed rate within certain limits.

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

FORMAT The arrangement and overall pattern in which the input data is organised (formatted).

G CODE A preparatory code in the program which determines the control mode.

HARDWARE The physical components of a control system or computer.

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

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

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

INPUT The transfer of external information into the control system.

INTERGRATED CIRCUIT A complete circuit constructed within or on the surface of a silicon chip.

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

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

ISO International Standards Organisation.

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

M CODE (Miscellaneous Function) The M words are used by the machine tool builder to indicate certain auxiliary functions such as coolant on, turret index, speed range etc.

MANUAL DATA INPUT (MDI) A means of inserting data into the control system manually. The data entered is identical to that entered from a disk or stored program.

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

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

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

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

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

OPTIONAL STOP A miscellaneous command which is given by the programmer to stop the spindle, and feed at a specific point in the program at the discretion of the

operator.

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

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

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

RESOLUTION The smallest increment of distance that will be developed by the control system in order to command machine motion.

S ADDRESS A word used in commanding the spindle speed.

SEQUENCE NUMBER See N Word.

SOFTWIRED N.C. A control system that has been pre-programmed with software to enable it to drive a machine tool.

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

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

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

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

U WORD The U Word is used to command motion perpendicular to the spindle centre-line (incremental).

W WORD The W Word is used to command motion parallel to the spindle centre line (incremental).

WORD A combination of the letter address and digits.

ADDRESS	DIGITS	WORD
X	+002.0000	X+002.0000
F	1.9990	F1.9990

X WORD On 2 and 4 axis machines X Word is used to command motion perpendicular to the spindle centre line (absolute).

Z WORD On 2 and 4 axis machines Z Word is used to command motion parallel to the spindle centre line (absolute).

ZERO In absolute programming, zero is the point from which all other dimensions are referenced.

22. APPENDIX 4
LIST OF ABBREVIATIONS

G.B.T. NO.	=	Group Block Terminal
EXECUTE OR MDI	=	Manual Data Input
MSD	=	Machine Set Up Data (or Parameters)
RAM	=	Random Access Memory
ROM	=	Read Only Memory
PROM	=	Programmable Read Only Memory
EPROM	=	Erasable Programmable Read Only Memory
PCI	=	Program Controlled Interface (i.e. Ladder Diagram in Software).
PWM	=	Pulse Width Modulated G.E Servo Drive or Hi-Ak Drives.
SCR	=	Silicon Controlled Rectifier (i.e Thyristor Drives used on Spindle)
TRIAC	=	Solid State Relay
I.P.	=	Initial Position or Zero Ref. Point
BCD	=	Binary Coded Decimal
LED	=	Light Emitting Diodes
IC	=	Input Conditioner or Input Filter

OD	=	Output Driver
+VE	=	Positive
-VE	=	Negative
DVM	=	Digital Voltmeter
POT	=	Potentiometer or Variable Resistor
LSI	=	Large Scale Integration
PCL	=	Programmable Controller Language
MCL	=	Machine Control Logic

23. APPENDIX 5 TRIGONOMETRY WITH EXAMPLES

The following pages of trigonometrical formulae are intended as an aid to working out angles and distances when planning a CNC Program.

APPENDIX 5 TRIGONOMETRY WITH EXAMPLES

a) Tangent

In Fig. 2 AOB is an angle of say 30 Degrees. The triangles LPO, MQOand NRO are similar because they all share the common angle of 30 Degrees. Therefore a common ratio for the three triangles can be shown as:

$$\frac{LP}{OP} = \frac{MQ}{OQ} = \frac{NR}{OR}$$

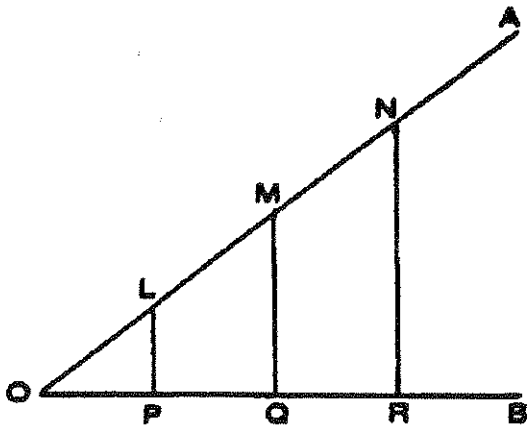


Fig. 2

If these sides were known, then dividing LP by OP and MQ by OQ and so on will produce the same common ratio.

If side LP is 5 mm and OP is 8.66 mm then LP/OP produces a ratio of .5774 or tangent value for 30 Degrees, a set of tangent tables are compiled in this way.

In the triangle LOP, side LP is opposite to the angle AOB, side OP is adjacent to the angle LOP, so to calculate a tangent ratio for the triangle LOP may be shown as:

$$\text{Tangent value} = \frac{\text{Opposite Side}}{\text{Adjacent side}}$$

b) Sine

In Fig. 2 the triangles LPO, MQO, NRO are similar because they all share the same angle (90 Degrees) at P, Q and R. Therefore a common ratio for three triangles can be shown as:

$$\frac{PL}{OL} = \frac{QM}{OM} = \frac{RN}{ON}$$

Therefore the ratio of PL/OL depends on the angle value of AOB only, and dividing these two values will always produce the same common ratio.

If side PL is 5 mm and OL is IO mm then PL/OL produces a ratio of .5000 or sine value for 30 Degrees, a set of sine tables are compiled in this way. In the triangle LOP, side PL is opposite to the angle AOB, side OL is the hypotenuse of the triangle LOP, so to calculate a sine ratio for the triangle LOP may be shown as:

$$\text{Sine Value} = \frac{\text{Opposite Side}}{\text{Hypotenuse}}$$

C) Cosine

It is also true that in Fig. 2 the ratios:

$$\frac{OP}{OL} = \frac{OQ}{OM} = \frac{OR}{ON}$$

Therefore the ratio of OP/OL depends on the angle value of AOB only and dividing these two values will always produce the same common ratio.

If side OP is 8.66 mm and OL is IO mm then OP/OL produces a ratio of .866 or cosine value for 30 Degrees, a set of cosine tables are compiled in this way. In the triangle LOP, side OP is adjacent to the angle AOB, side OL is the hypotenuse of the triangle LOP, so to calculate a cosine ratio for the triangle LOP may be shown as:

$$\text{Cosine Value} = \frac{\text{Adjacent Side}}{\text{Hypotenuse}}$$

SUMMARY

Tangent

$$TAN = \frac{\text{Opposite Side}}{\text{Adjacent Side}}$$

also Opposite Side = Tan x Adjacent Side

$$\text{Adjacent Side} = \frac{\text{Opposite Side}}{TAN}$$

Sine

$$SIN = \frac{\text{Opposite Side}}{\text{Hypotenuse}}$$

also Opp. Side = Sin. x Hypot.

$$\text{Hypotenuse} = \frac{\text{Opposite Side}}{SIN}$$

Cosine

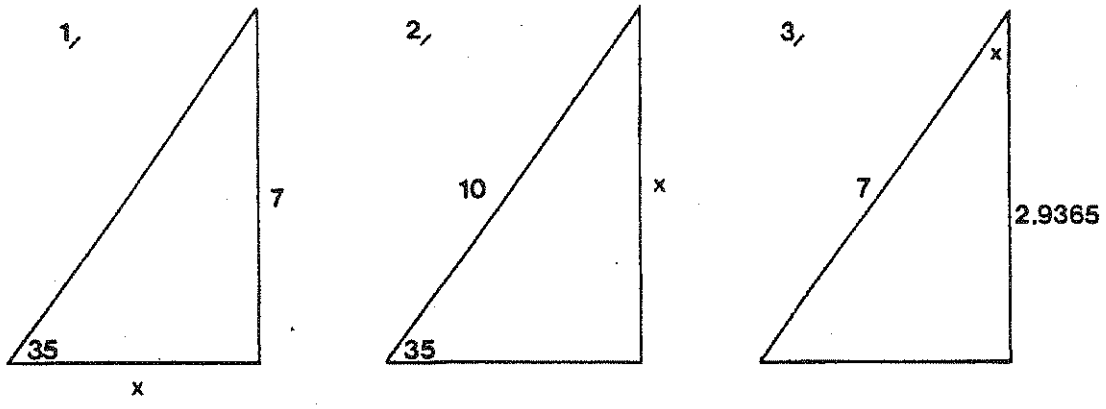
$$COS = \frac{\text{Adjacent Side}}{\text{Hypotenuse}}$$

also Adjacent Side = Cos x Hypotenuse

$$\text{Hypot.} = \frac{\text{Adjacent Side}}{COS}$$

EXAMPLES:

Determine X values



1) Solution

Find adjacent side knowing opposite.

$$\text{Adj} = \frac{\text{Opposite}}{\text{Tan 35 Degrees}}$$

$$\text{Adj} = \frac{7}{0.7002}$$

Adj. = 9.9971 = X

2) Solution

0pposite = Sin 35 Degree x Hypotenuse

0pposite = .5736 x 10

0pposite = 5.7360 = X

3) Solution

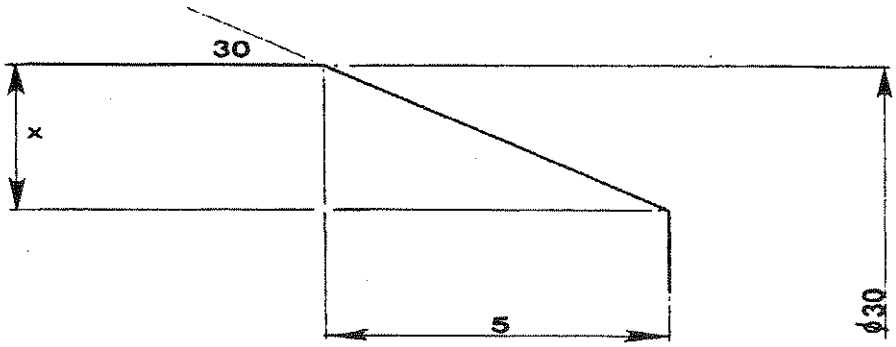
Find angle knowing adjacent side and hypotenuse.

$$\text{Cos} = \frac{\text{Adjacent}}{\text{Hypotenuse}}$$

$$\text{Cos} = \frac{2.9365}{7} = 0.4195 \text{ (as cosine ratio)}$$

0.4195 from cosine tables = 65 deg 12' = X

4/



4) Solution

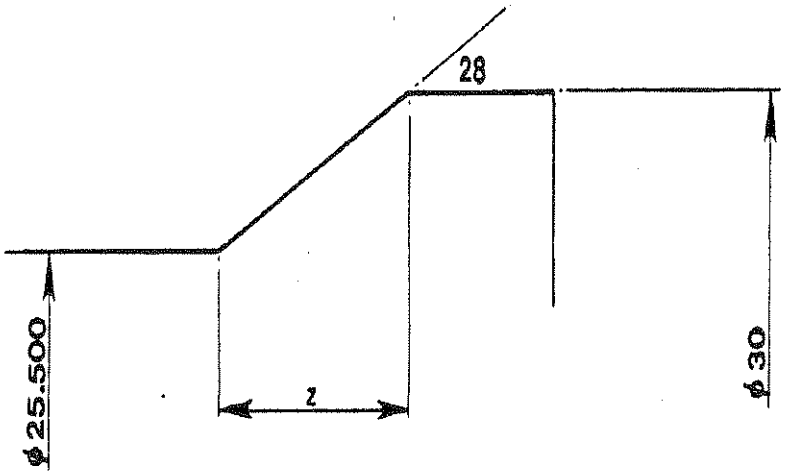
Find X distance on component chamfer

X = Tan 30 x 5

X = 2.887 mm

5) Solution

5/



Determine Z distance on thread relief.

$$Z = \frac{2.250}{\text{Tan } 28}$$

Z = 4.232 mm

GEOMETRY

DEFINITION Right Angle

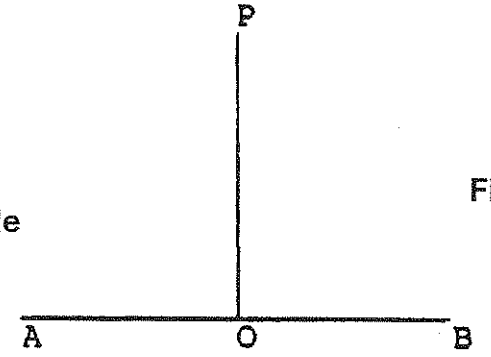


Fig. 1

If a straight line OP meets another straight line AOB as shown in Fig. 1, so as to make the adjacent angles POA, POB equal, each angle is called a right angle.

Acute Angle.

An acute angle is an angle less than a right angle.

Obtuse Angle.

An obtuse angle is an angle greater than a right angle but less than two right angles.

Reflex Angle.

A reflex angle is an angle between two and four right angles. Any two angles whose sum is two right angles are called supplementary. Two angles whose sum is one right angle are called complementary.

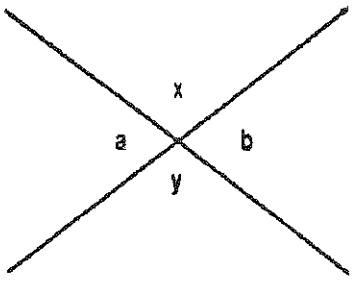


Fig. 2

If two straight lines intersect, the vertically opposite angles are equal. In fig. 2, where two lines intersect, a is equal to b and x is equal to y.

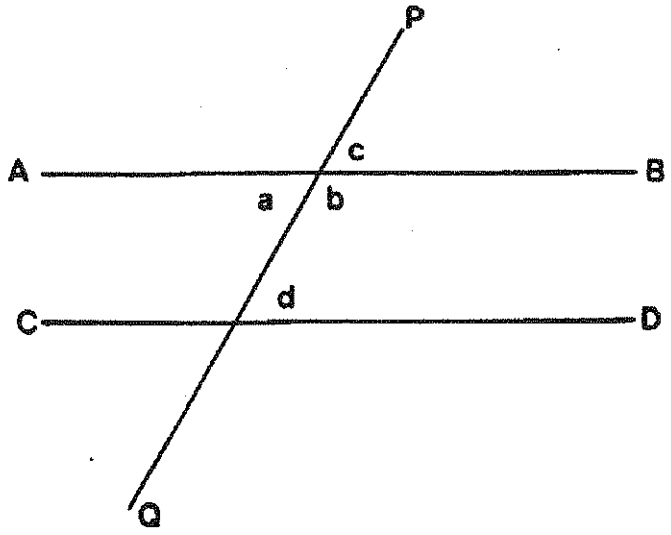


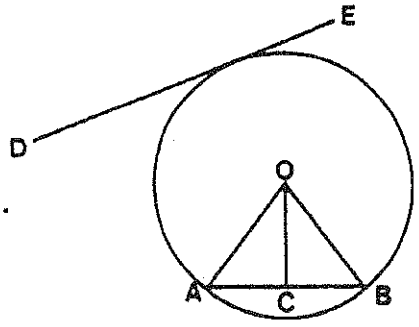
Fig.3

If two straight lines in the same plane do not meet they are called parallel lines. In fig. 3 the line is called a transversal line, angles a and d are called alternative angles; angles c and d are called corresponding angles; angles b and d are called interior angles.

Therefore if a transversal line PQ cuts two parallel lines, angles a and d are equal, angles c and d are equal.

Fig.4

Angle properties of the circle .



In figure 4, line D E is tangent to circle 1, because it only touches the circle at one point, line A B is a chord of circle 1, if O C is perpendicular to the chord A B from the centre of circle I then AC is equal to BC.

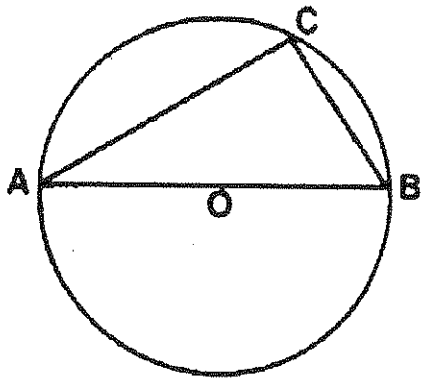
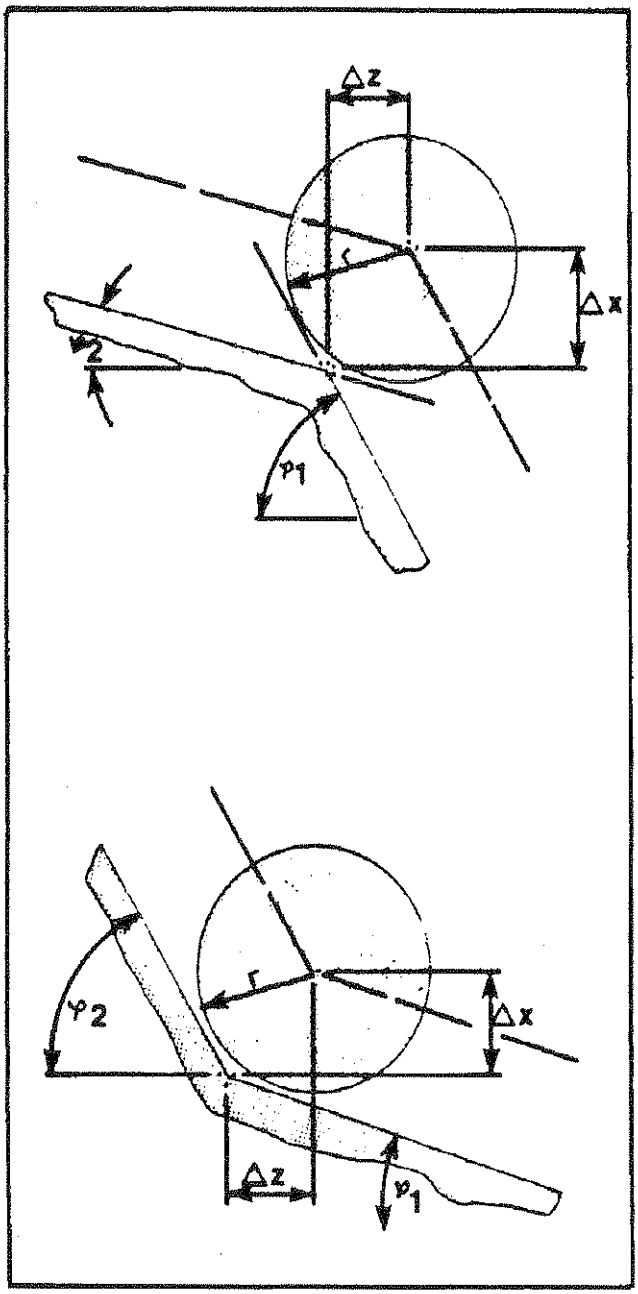


Fig. 5

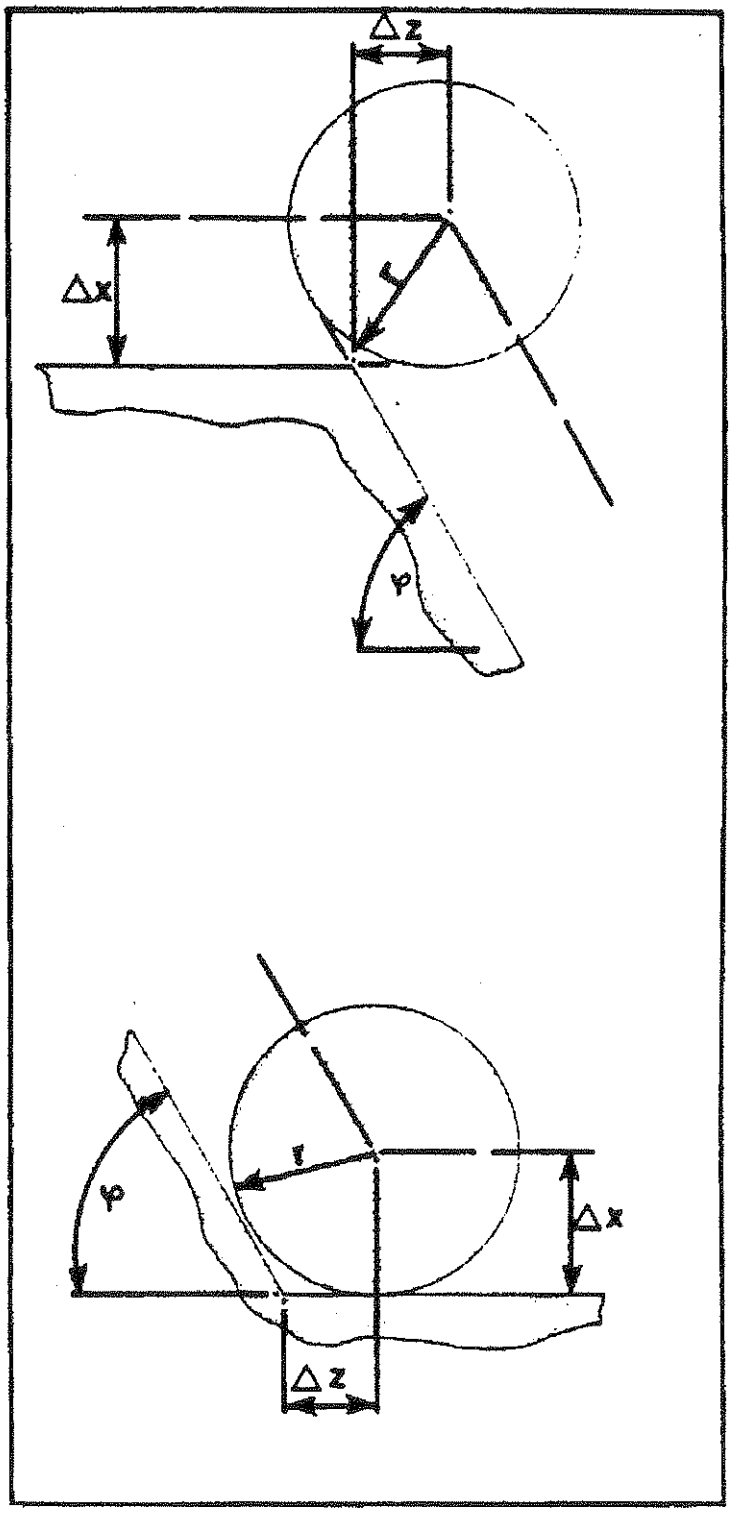
The angle in a semi circle is a right angle. In fig 5 line A B passes through the centre of circle 2 at O. Any point chosen within the semi circle at C will produce a right angle ACB.

.USEFUL FORMULAE FOR DETERMINING CONTOUR CHANGE POINTS



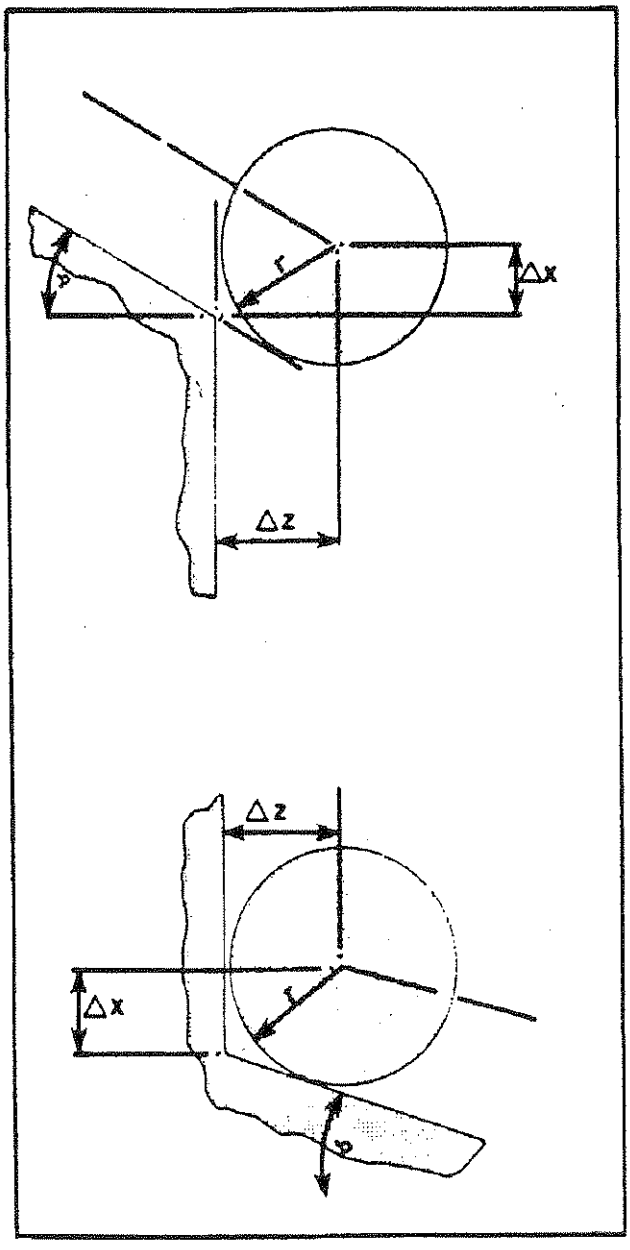
$$\frac{\Delta X = r.\cos\frac{\alpha 2+\alpha 1}{2}}{\cos\frac{\alpha 2-\alpha 1}{2}}$$

$$\frac{\Delta Z = r.\sin\frac{\alpha 2+\alpha 1}{2}}{\cos\frac{\alpha 2-\alpha 1}{2}}$$



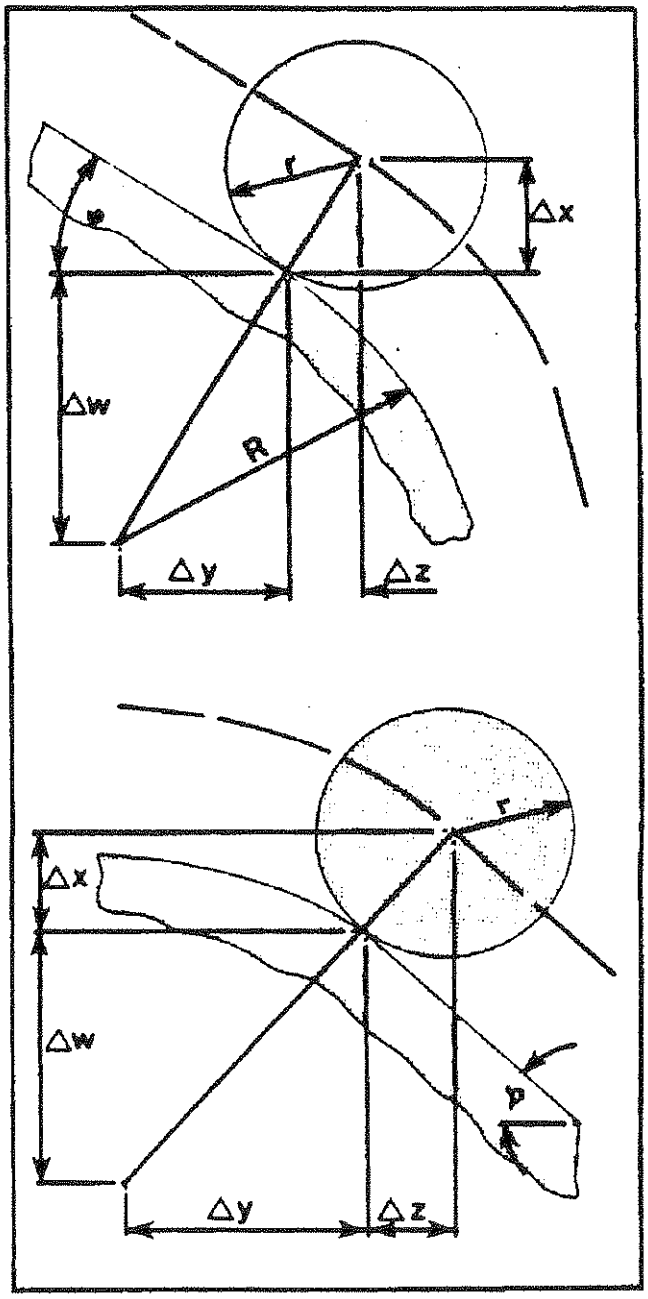
$\Delta X = r$

$\Delta Z = r . \tan\frac{\alpha}{2}$



$$\Delta X = r \cdot \tan\left(45 - \frac{\alpha}{2}\right)$$

$$\Delta Z = r$$

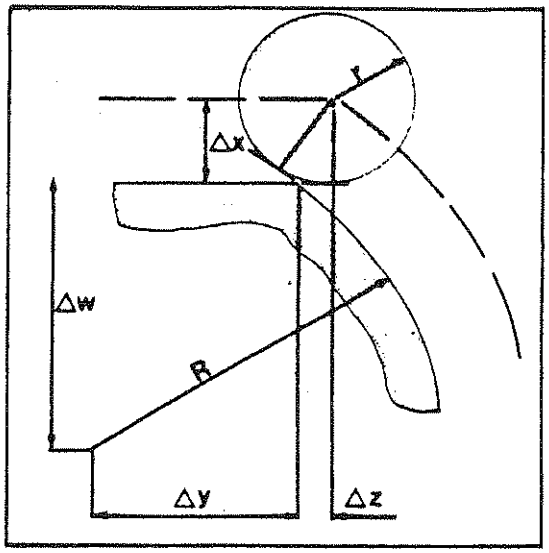


$\Delta W = R \cos \alpha$

$\Delta X = r \cos \alpha$

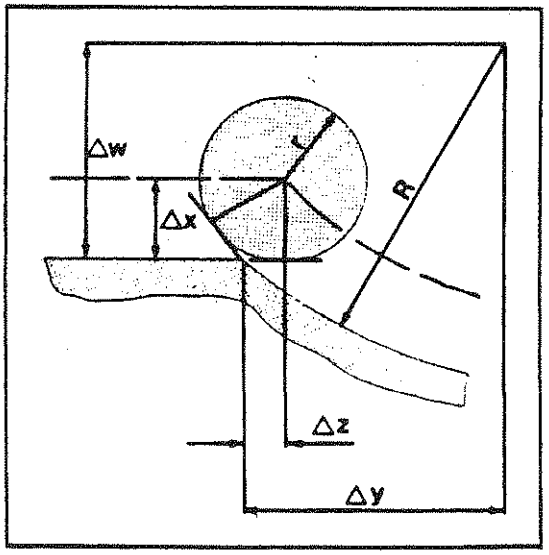
$\Delta y = R \sin \alpha$

$\Delta Z = r \sin \alpha$



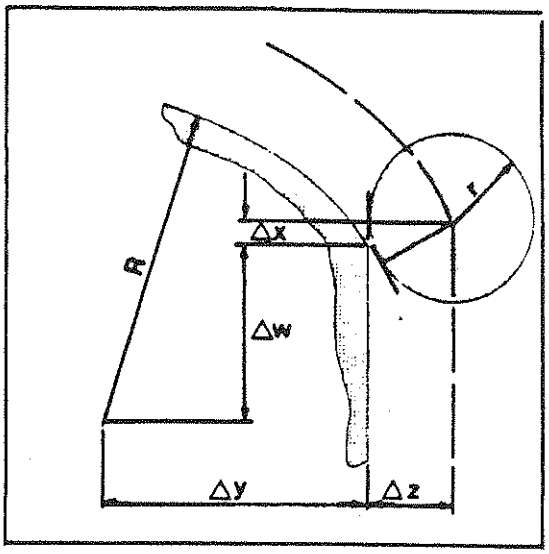
$$\Delta X = r$$

$$\Delta Z = \sqrt{[\,(R+r)^2 - (\Delta W + r)^2\,]} - \Delta y$$



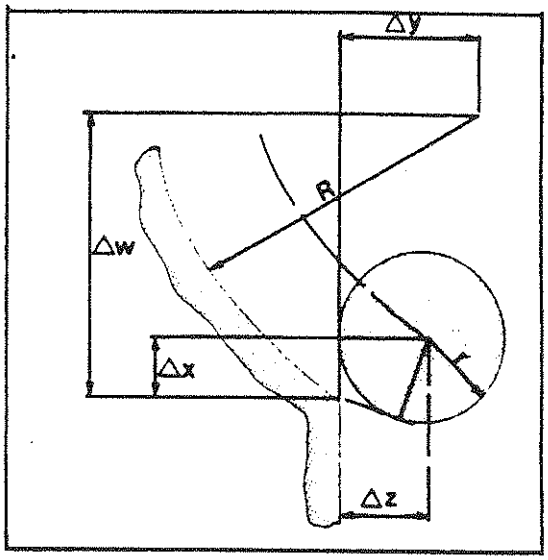
$$\Delta X = r$$

$$\Delta Z = \Delta y - \sqrt{[\,(R - r)^2 - (\Delta W - r)^2\,]}$$



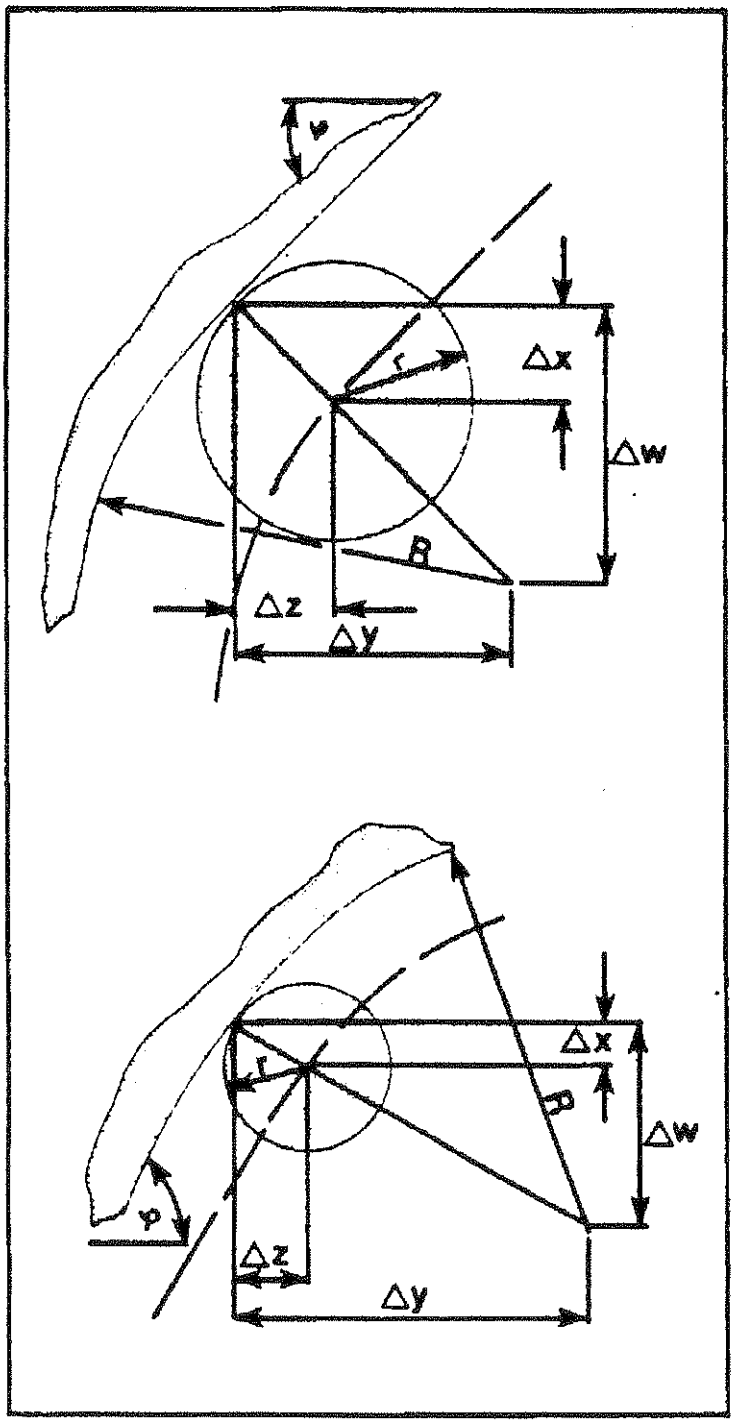
$$\Delta y = r$$

$$\Delta X = \sqrt{[(R + r)2 - (\Delta y + r)^2]} - \Delta W$$



$$\Delta Z = r$$

$$\Delta X = \Delta W - \sqrt{[(R - r)^2 - (\Delta y - r)^2]}$$



DW = R COS a

DX = R COS a

Dy = R SIN a

Dz = r SIN a