Harrison

TU150E

CNC LATHE

INSTALLATION OPERATION

&

MAINTENANCE MANUAL

THE 600 GROUP

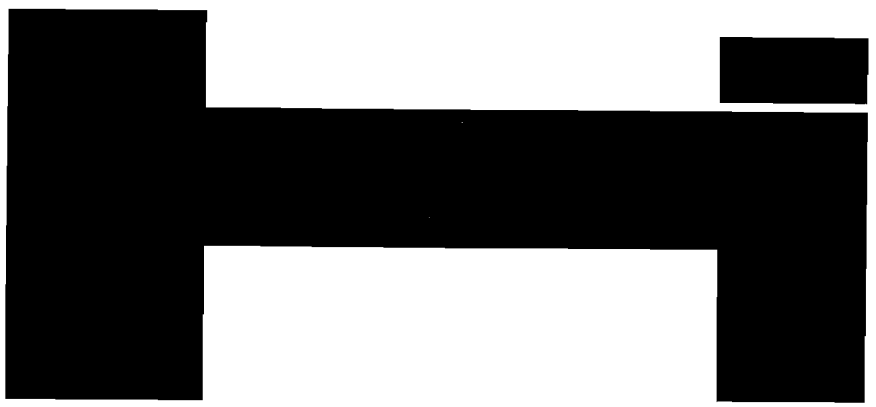
WARNING

WARRANTY ON THIS MACHINE WILL BE INVALIDATED IF ANY MODIFICATIONS, ADDITIONAL ANCILLIARY EQUIPMENT IS FITTED OR ANY ADJUSTMENTS MADE TO THE CONTROLLING DEVICES WITHOUT PRIOR NOTIFICATION FROM T.S. HARRISON & SONS LTD.

T.S. HARRISON & SONS

TU. 150E SERIES OF CNC MACHINES

COMPACT 2 AXIS CNC PRODUCTION LATHES



This manual applies only to the machine having the serial number shown; this MUST be quoted in all communications.

Machine Serial Number.....

Year of Manufacture .....

CE

Manufactured by

DENFORD MACHINE TOOLS LTD
FOR

600 LATHES LIMITED

Union Street, Heckmondwike, West Yorkshire, England

Printed in England

EC Declaration of Conformity

The Responsible Person — Mr. B. Kellett.

Business Name — DENFORD MACHINE TOOLS LTD.

Address — BIRDS ROYD
BRIGHOUSE
WEST YORKSHIRE
HD6 1NB ENGLAND

Declares that the machinery described :

1.Make — **DENFORD MACHINE TOOLS**

2. Model — **TU150E**

3. Serial Number

Conform to the following Directives: — EC MACHINERY DIRECTIVE 89/392/EEC AS AMENDED BY DIRECTIVE 91/368 EEC AND DIRECTIVE 93/44/EEC; AND CE MARKING DIRECTIVE 93/68/EEC

and the following standards:

and complies with the relevant health and safety requirements

Signature

(IF NOT SIGNED BY THE RESPONSIBLE PERSON, STATE HERE THE NAME OF THE PERSON SIGNING THE DECLARATION)

WORKS DIRECTOR

Position

Signed by

Signed at: DENFORD MACHINE TOOLS LTD.
BIRDS ROYD
BRIGHOUSE
WEST YORKSHIRE
HD6 1NB ENGLAND

Date:

1. STANDARD EQUIPMENT

STANDARD EQUIPMENT SUPPLIED

1 TU150E LATHE CNC LATHE

TU150E SOFTWARE DISC

COOLANT SUPPLY

FULLY INTERLOCKED SAFETY GUARDS

EXTERNAL PARALLEL PRINTER PORT

EXTERNAL RS232 SERIAL LINK

FITTED DISC DRIVE (3.5" DISC)

AUTOMATIC SLIDEWAY LUBRICATION

1 set INSTRUCTION / MAINTENANCE MANUALS

2. SPECIFICATION

SPECIFICATION SHEETS

MACHINE SPECIFICATION

1. STANDARD FEATURES - TU 150E

A) General:

1) Machine Dimension List:-

Length: 1145mm (45")

Width: 1100mm (43 .5")

Height: 700mm (27 .5").

B) Capacity:

MaximumTurned Length:	210mm (8.5")
Maximum Turned Diameter:	158mm (6 .25")
Swing Over Bed:	300mm (12")
Spindle Bore:	35mm (1.375")
Spindle Nose:	A2-3 type
"X" Axis Travel:	94mm (3.700")
"Z" Axis Travel:	210mm (8.267")

Bed: Hardened and Ground of slant design

C) Main Spindle:

Spindle Drive Motor	2.2 Kw.	Speed Range 100 - 5000 RPM
Spindle Nose :	A2-3 type	
Spindle Bore :	35 mm	

D) Saddle

Configuration:	45 degree Slant
Axis Motors:	Stepper Motors

E) Indexing Turret:

Turret Lead Type:	Drum
Number of Tool Stations:	6 or 8 (Optional)
Tool Size Square:	12mm x 12mm(0.5" x0 .5")
Round:	25 mm Diameter (1 .00")
Indexing Time:	One Station: Approximately 0.65 secs
One Revolution: Approximately:	1.75 secs

NOTE: Maximum Turning Diameter may be limited due to Tooling

F) Coolant (Cutting Fluid) Tank Capacity: 2.6 Imp. Gallons.

G) Machine Work Light : 110 Volts AC

H) Environment Conditions:

Power Supply:	3PH +Earth
Total Power Connected 15 amps	

2 OPTIONAL FEATURES

A) Pneumatic chuck.

B) Manual 125mm 3 Jaw Chuck

C) Pneumatic tailstock

D) Manual Tailstock

E)Pneumatic guard

F)Parts Catcher

G) Pendant Type C R T

H) Machine Bench

I) Desk top Tutor

J) CAD/CAM (Super-Draft)

K) CAD/CAM (Auto-Cad)

Tooling and work-holding equipment can be supplied to suit a customers requirements.

(FOR FURTHER DETAILS CONTACT HARRISONS SALES DEPARTMENT)

CONTROL SOFTWARE FEATURES

STANDARD FEATURES

- Simultaneously controlled axes X and Z.
- Full Circular interpolation.
- Combined Use of Absolute and Incremental in Same Block.
- Internal / External Screwcutting.
- Imperial or Metric Programming.
- Diameter or Radius Programming.
- Subprogram with Repeat Facility .
- Tool Nose Radius Compensation.
- Canned Turning Cycles.
- Roughing and Finishing Cycles.
- Output to Printer and Punch Tape.
- Tool offsets for Sixteen Tools.
- Dry- run Facility.
- Single Block and Auto Execution.
- Datum Shift.
- Zoomed views.
- Programs Stored on Floppy or Hard Drive(if fitted)
- Up to 6000 G Codes in Editor.
- Context Sensitive Help
- Overtravel Limits and Emergency Stop.
- Feed and Speed Rate Overide.

- Dwell Facility
- Comprehensive Toolpath Graphics Simulation Including 3D .
- D.N.C" Drip Feed" from RS232 or Floppy Disk.
- Multiple Repetitive Cycles.
- Display Actual Position and Distance to Go.
- Edit and Simulate Mode.
- Directory Listing.
- Merge Programs.
- Comprehensive Error Messages.
- Cycle Start \ Feed Hold.
- Tool Wear Compensation.
- Tool Nose Radius Compensation.
- Feed\Min or Feed\Rev Programming.
- Constant Surface Speed Programming.

3. SAFETY FUNCTIONS

- Key operated Isolator switch.
- Emergency Stop.
- Membrane Keyboard.
- Totally Interlocked Sliding guard.
- Axes Limit Switches.
- Graphics Facility for proving programs prior to machining.

NOTES:

Power Supply: Three phase 50 Hz AC.

Temperature: 0 - 45 degrees C.

Relative Humidity: Less than 75%.

4. INSTALLATION

INTRODUCTION

This section describes the correct procedures for installing your TU150E Lathe.

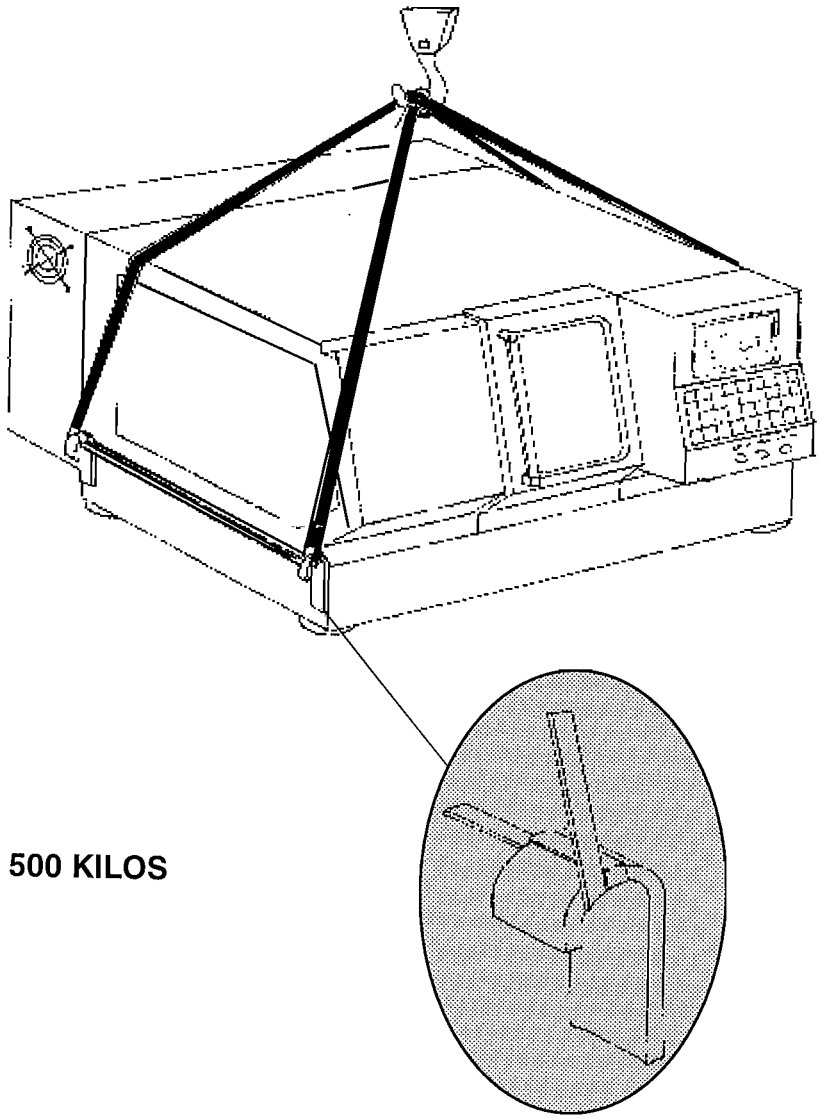
These procedures should be followed precisely to ensure your TU150E is not damaged in any way during the installation period.

During the installation period the protective coats and coverings applied prior to despatch should NOT be removed.

All installation work should be carried our by qualified personnel.

If in doubt contact HARRISONS Service Department for further details.

LIFTING THE TU150E



NET WEIGHT = 500 KILOS

The TU150E is lifted by the two slings wrapped around the four lifting hooks which are found at each corner of the machine, as shown in the above diagram.

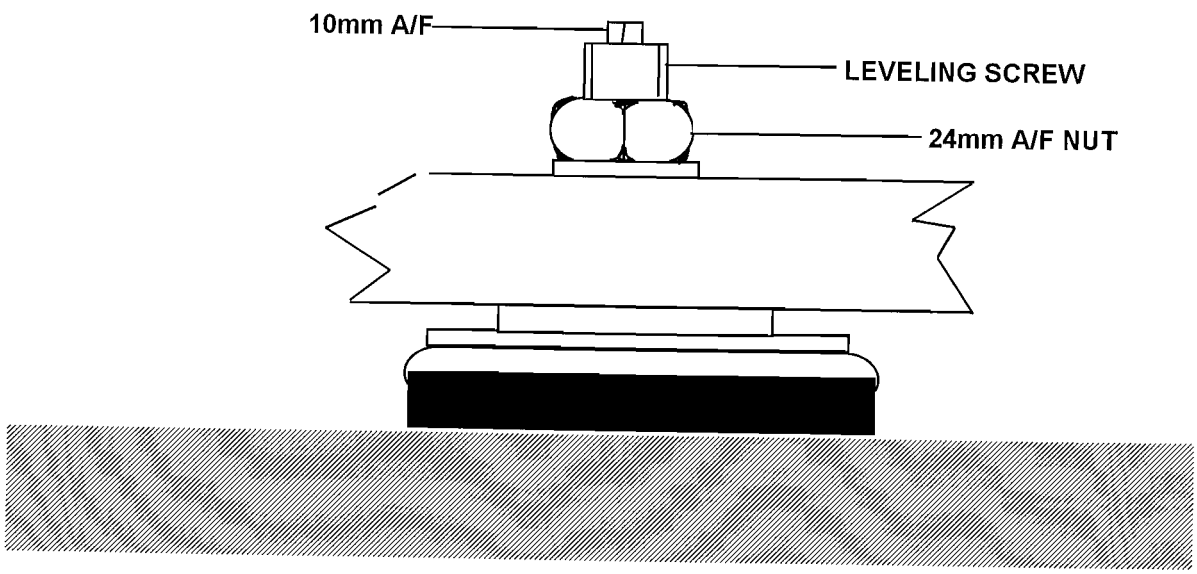
Ensure the elecrical cabinet at the back of the machine is closed and locked before lifting.

To prevent the slings slipping during lifting they should be wrapped around the hooks as shown in the diagram inset above.

Always check for balance before transporting the machine on the lifting slings.

LEVELING PROCEDURE

TU150E's machine bed is leveled to the cabinet during manufacture, hence it is only



necessary to level the cabinet during installation.

The machine stands on four aniti-vibration leveling feeet located at each corner of the machine base.

LEVELING FEET LOCATION

Before removing any covers ensure that the machine is electrically isolated.

The two feet at the headstock end of the machine are accessible by removing the headstock cover which is secured with eight M6 button head screws. The front foot at the tailstock end of the machine is accessible through the front sliding door. The rear tailstock end foot is reached by opening the oil pump door at the side of the machine(see tailstock end diagram page 4.8).

Tools required :- Spirit Level, !0mm A/F Spanner, 24mm A/F Spanner and 4 mm A/F Allen Key.

Open the oil pump door and the front sliding door,then remove the headstock cover plate, this makes all four adjustable feet accessible.

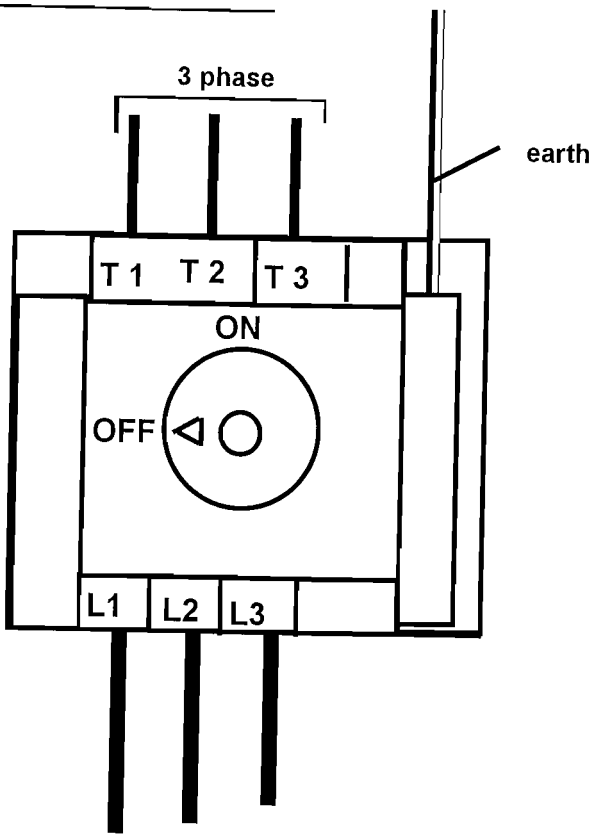
Place the Spirit Level on the top of the machine lenghthways, if the bubble is not aligned , adjust as follows:-

Release the 24mm A/F nut on the adjusting screw, holding the screw stationary with the 10mm A/F spanner. By turning the screw Clockwise the machine corner can be raised, and by turning Counterclockwise it can be lowered. When the bubble is aligned, turn the Spirit Level widthways and again adjust the feet to align the bubble checking at both ends of the cabinet. Once the bubble is aligned in both planes, tighten the 24mm A/F nuts taking care to hold the screw with the 10mm A/F Spanner whilst doing so.

ELECTRICAL CONTROL BOX

NB. THE ELECTRICAL CONTROL BOX IS INSPECTED THEN SEALED WITH A YELLOW SEAL, IF THIS SEAL IS BROKEN ON DELIVERY INFORM THE SUPPLIERS IMMEDIATELY. THE SEAL SHOULD ONLY BE BROKEN FOR THE INITIAL MAINS POWER CONNECTION.

ELECTRICAL CONNECTION



Cable Required:-3core(3phase) & Earth- 2.5mm per phase.

Tools Required:- Phillips Head Screwdriver and Crimping Pliers.

CONNECTION PROCEDURE:-

·Unlock and open electrical control box (back of machine)

· Isolator is found centre top.

·Put crimp connectors on to the mains,neutral and earth wires

·Connect the mains wires into the top of the contact block, as shown.

For Export Market - Contact HARRISON Service Department.

MACHINE PREPARATION

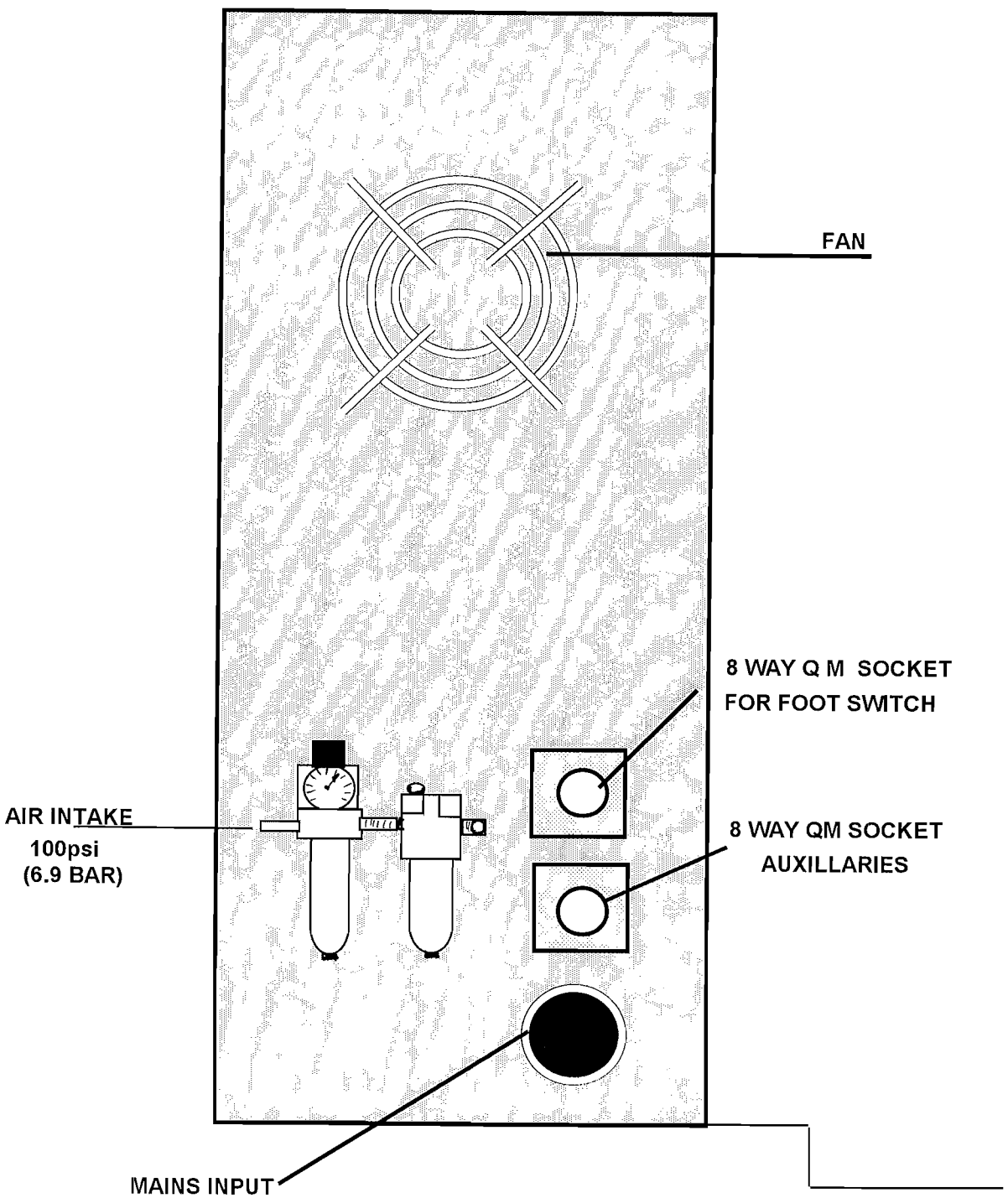
Once the machine has been sited and connected electrically, the protective coatings must be removed to prepare the machine for running.

The protective coating applied to the slideways and bright surfaces can be removed using a kerosene based solvent. The coating must be removed from the slideways before any attempt is made to move them, or operate the machine.

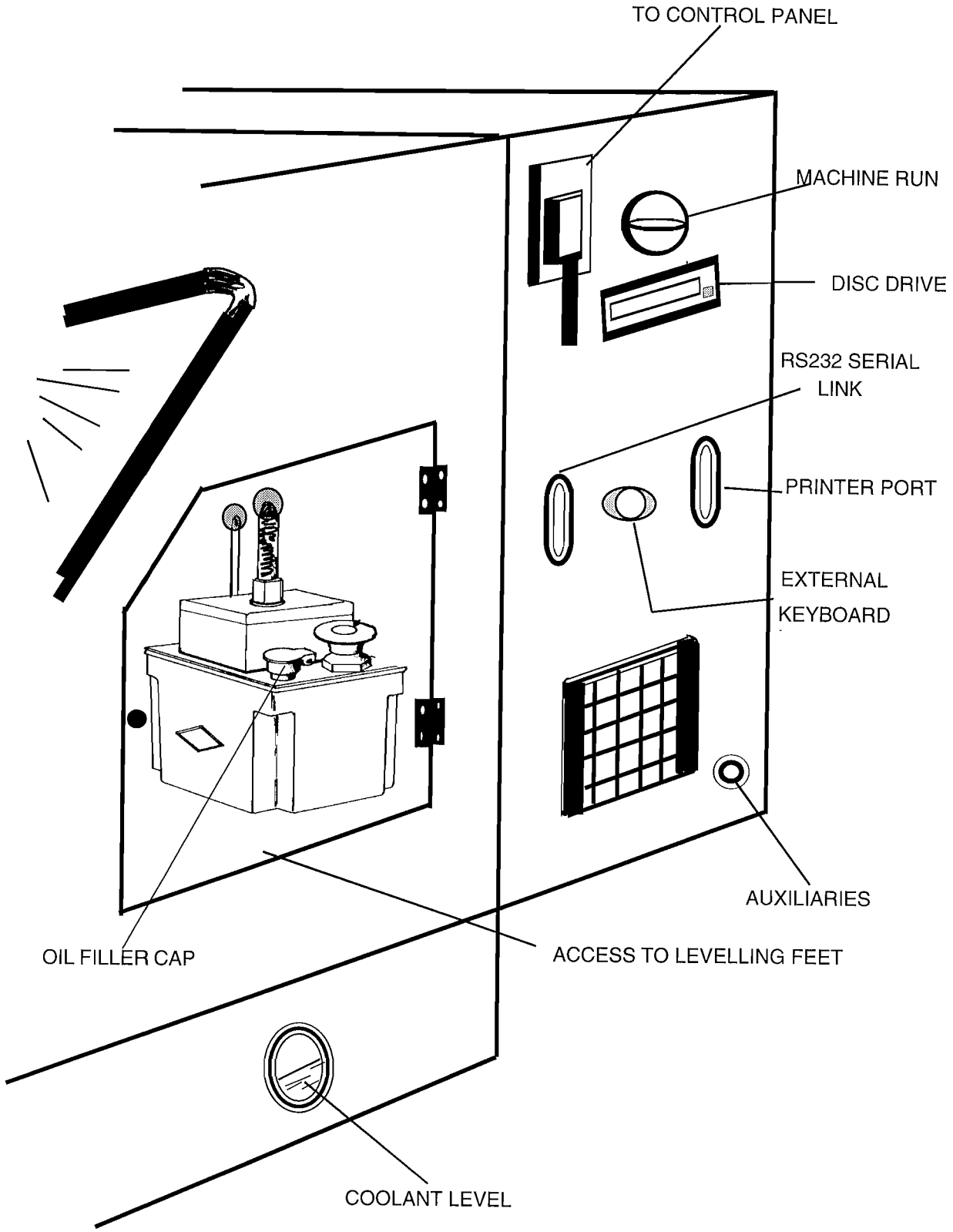
Once the protective coat is removed, all untreated surfaces should be coated with a light smear of machine oil (i.e. BP. : CS 68).

The protective plastic sheet on the windows should be removed and the glass and perspex should then be cleaned with an anti-static cleaner.

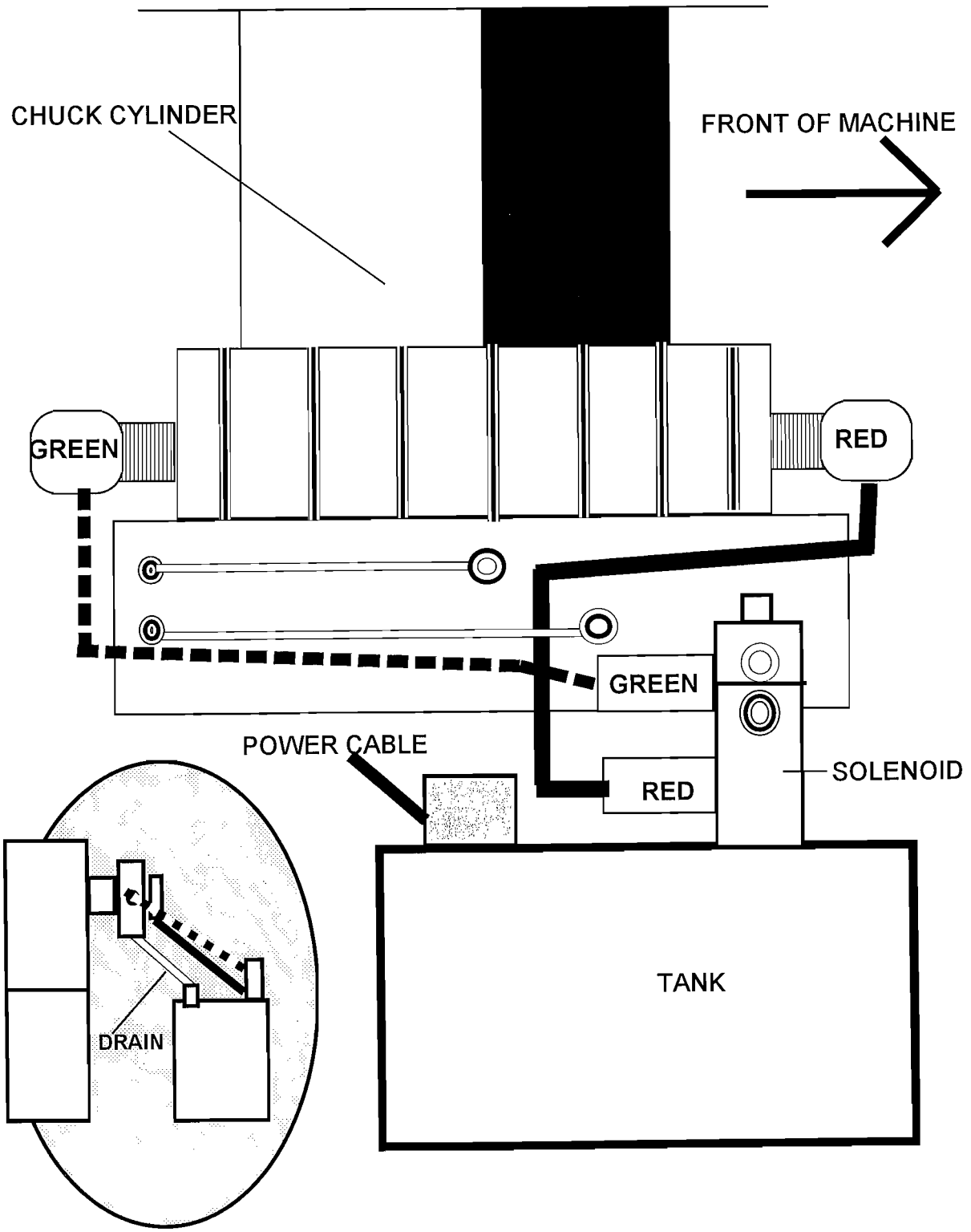
HEADSTOCK END DIAGRAM



TAILSTOCK END DIAGRAM



CONNECTION DIAGRAM FOR HYDRAULIC CHUCK



WARNING

COMPATIBILITY OF CHUCKS AND WORK HOLDING EQUIPMENT MUST BE APPROVED BY 600 LATHES LTD.

ONLY HIGH SPEED CHUCKS SHOULD BE USED.

DO NOT USE FAULTY OR DAMAGED CHUCKS AS THIS CAN LEAD TO SERIOUS INJURY.

INSTALLATION AND MAINTENANCE INSTRUCTIONS FOR 5-C & 16-C DEAD LENGTH PICK OFF CHUCKS

DESCRIPTION

This chuck operates in conjunction with standard 5-C or 16-C collets. Unlike conventional drawback chucks in which the chuck body is static and the collet moves axially rearward to clamp. These chucks permit static fixing of the collet relative to the chuck body. The collet closing sleeve is pushed forward to engage the collets closing taper via bridging bolts and spacers driven by the threaded rear actuating plate. this gives several advantages :-

1.) In second operation work, shouldered components can be stopped against the collet face.

2.) Both collet types are internally threaded in the rear end to house either a backstop or ejector if required.

3.) Drawback spring collets achieve the highest possible concentric accuracy, however,

they cannot be used in the conventional mode for second operations where a tight control on component lengths is required.

4.) The axial movement of drawbar collets in conventional chucks imposes undue loads on machine ballscrews when used in pick-off applications. These problems are overcome by the new design.

5.) When used for picking-off, extremely short components may be gripped.

COLLET CHUCK REMOVAL

* CAUTION - READ THOROUGHLY PRIOR REMOVAL

1.) Turn off the air pressure and remove any collet from the collet chuck

2.) Remove the three 7/16" UNC screws from the collet chuck body.

3.) Insert tube holding /driver and turn CCW , this will remove the chuck body and the draw tube together out of the air cylinder, they can then be removed from the machine.

4.) Remove the three 7/16 UNC screws from the A2/3 - A2/5 adaptor, this can then

be removed from the spindle nose. Once the spindle nose has been revealed and cleaned an alternative holding device may be fitted.

LOADING COLLETS TO THE DEAD LENGTH PICK - OFF CHUCK

With the chuck sleeve fully advanced, slacken the collet key actuating screw approximately one full turn. Mark the collet face with an ink marker in line with the keyway position.

Screw the collet fully into the chuck with the insertion tool provided and then back off only sufficiently to advance and tighten the collet key

* CAUTION

The maximum drawbar force and maximum speed indicated on the chuck must not be exceeded.

* MAINTENANCE

Remove the collet, clean and replace occasionally during long production runs.

Lubricate sparingly every 50 (fifty) operating hours with MOLYCOTE TR42 or LITHIUM GREASE. For further information contact CRAWFORD COLLETS LTD.

STANDARD OPERATIONAL FUNCTIONS

During the operation of the TU150E it will become neccessary from time to time to perform operational functions which follow a general keypress sequence. To aid in the learning of these functions the following pages give a step by step flowchart illustration of a number of these sequences.

1. GRAPHICS - SEQUENCING TOOL GRAPHICS TO TOOL STATION NUMBERS

2. GRAPHICS - TO LOAD OR SAVE GRAPHICS TO A PART PROGRAM.

3. TO LOAD A PROGRAM TO DISC

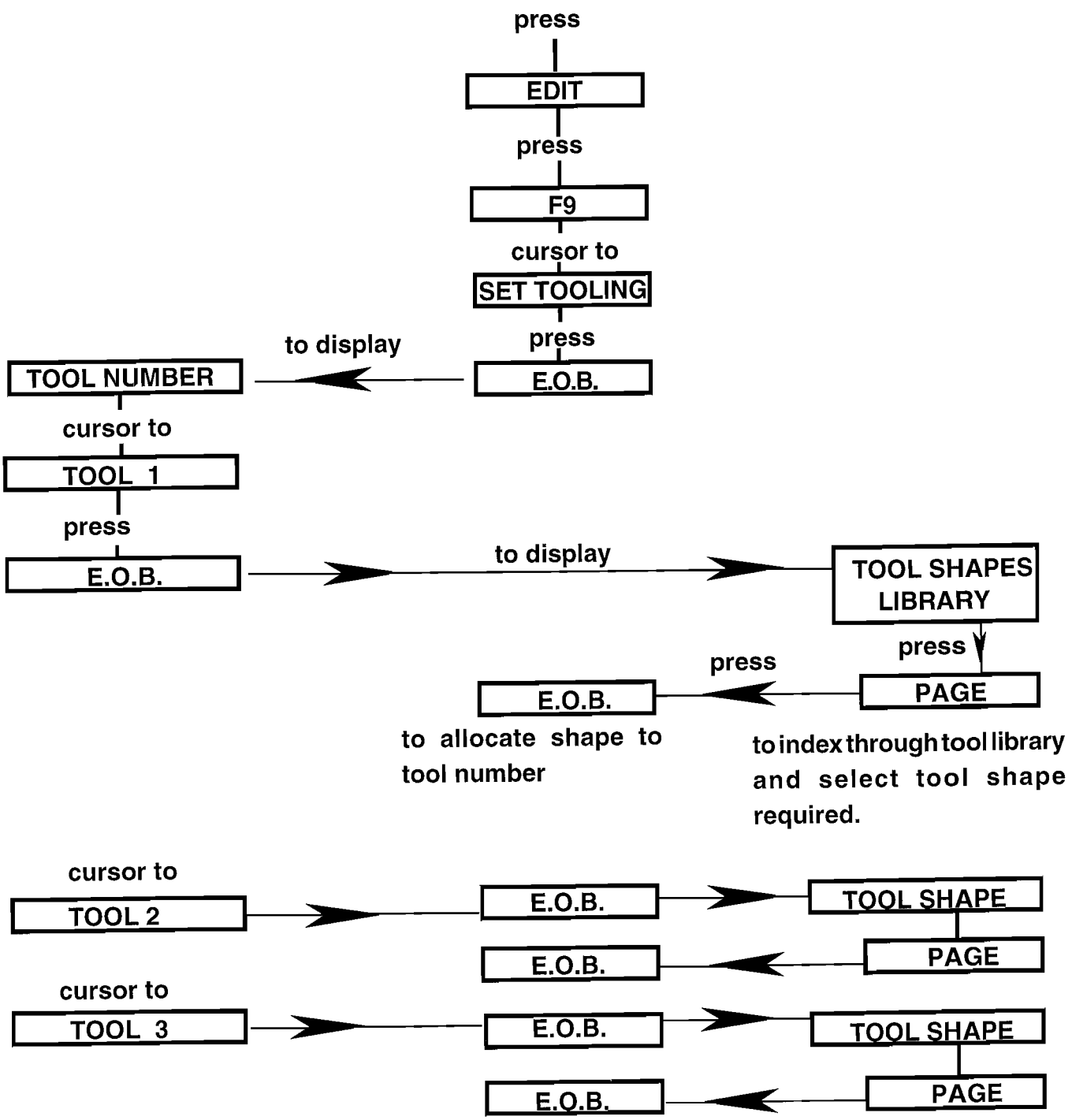
4. TO SAVE A PROGRAM TO DISC

5. TO CALL UP A CNC LIBRARY

6. TO SAVE, LOAD AND DELETE TOOL OFFSETS

GRAPHICS

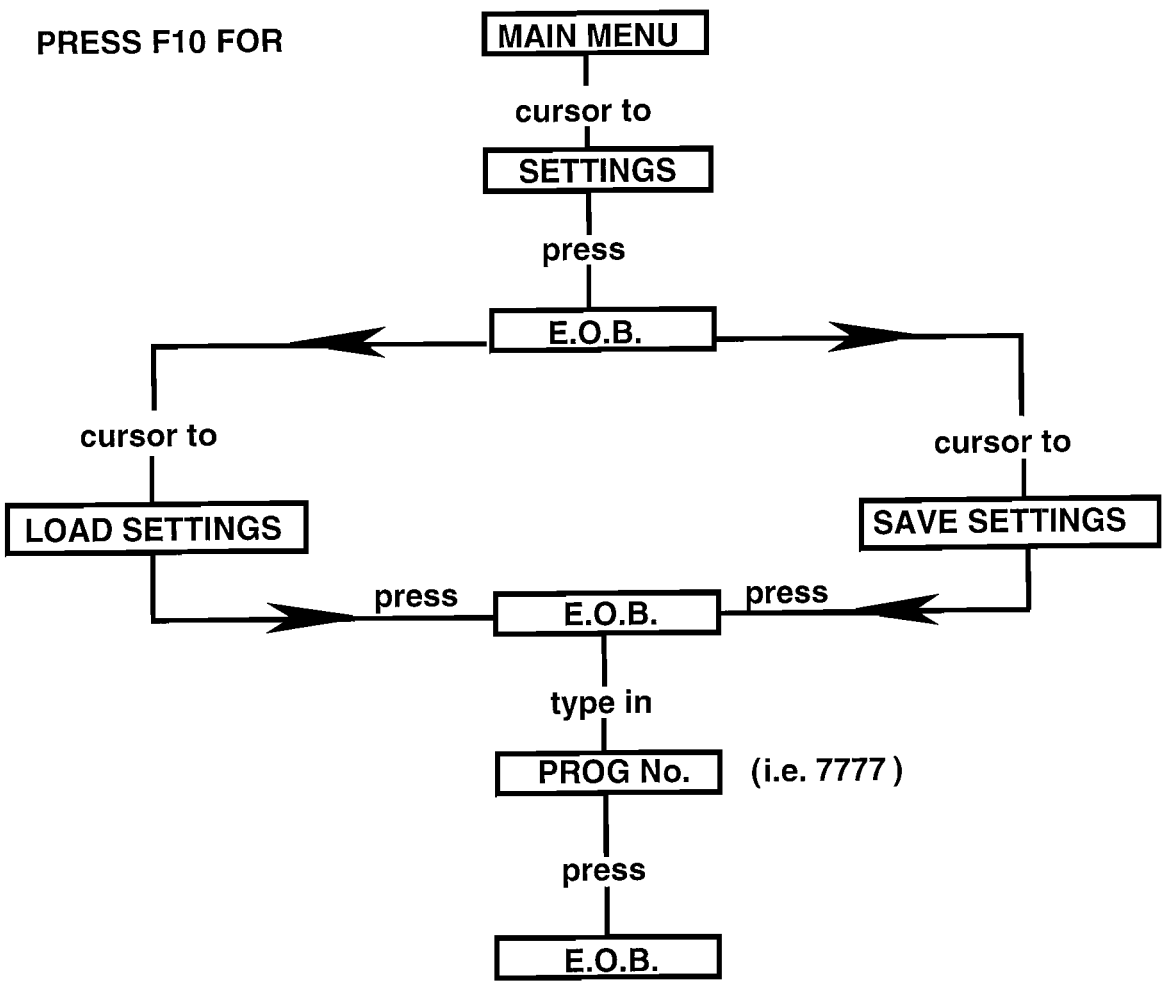
TO SEQUENCE TOOL GRAPHICS TO TURRET STATION NUMBERS FOR ANY PROGRAM :-



GRAPHICS

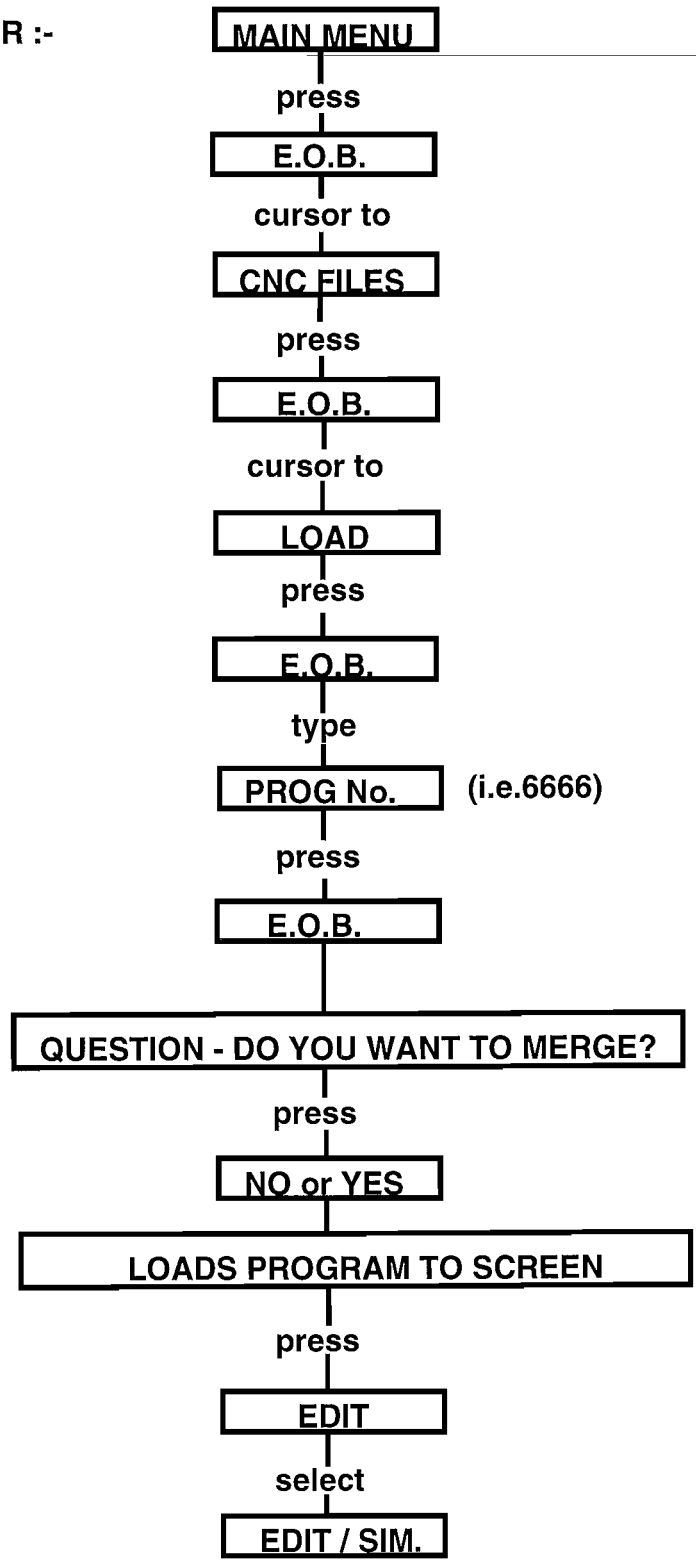
TO LOAD OR SAVE GRAPHICS FOR A PART PROGRAM:-

NB. THE MACHINE MUST FIRST BE HOMED



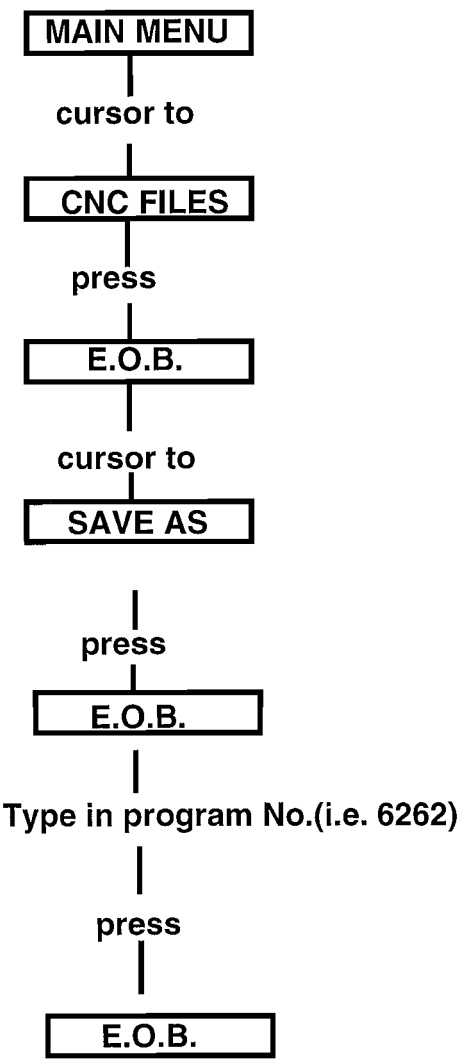
TO LOAD A PROGRAM FROM DISC:-

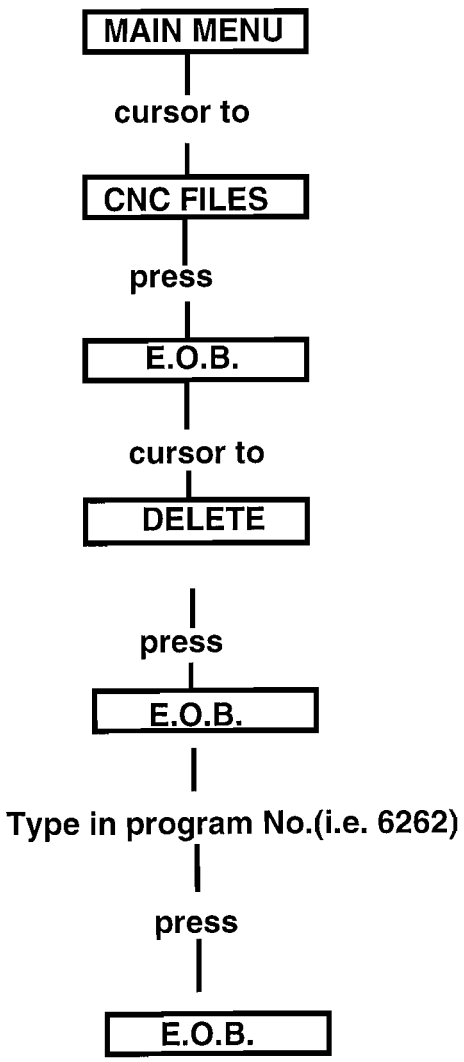
PRESS F10 FOR :-



TO SAVE OR DELETE A PROGRAM TO DISC

PRESS F10 FOR

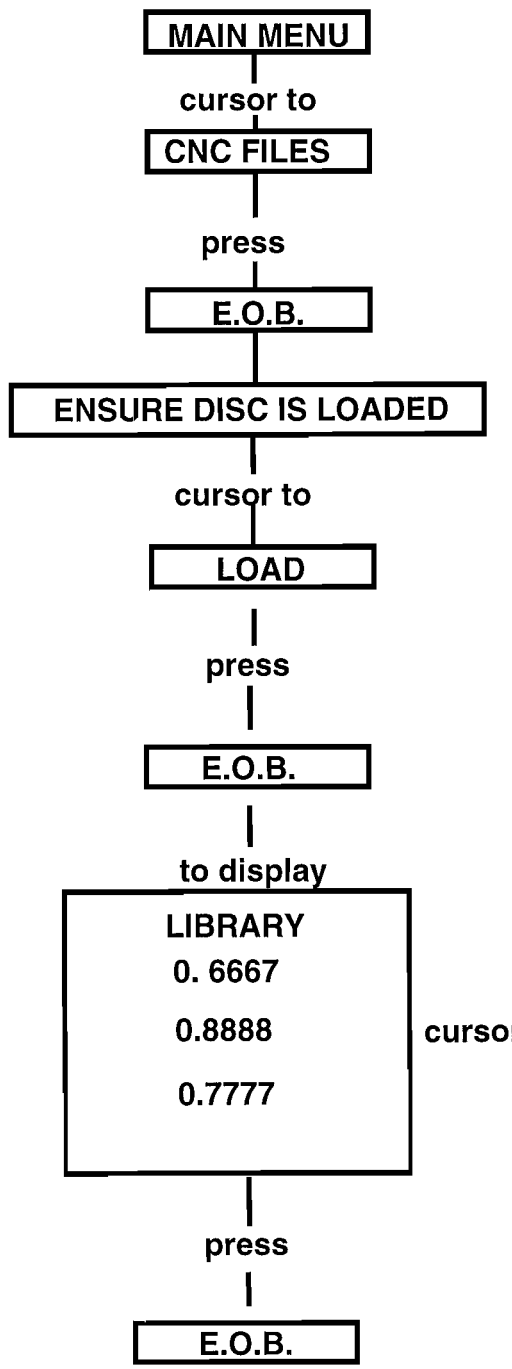




TO CALL UP THE CNC PROGRAM LIBRARY

NB. THE MACHINE MUST FIRST BE HOMED

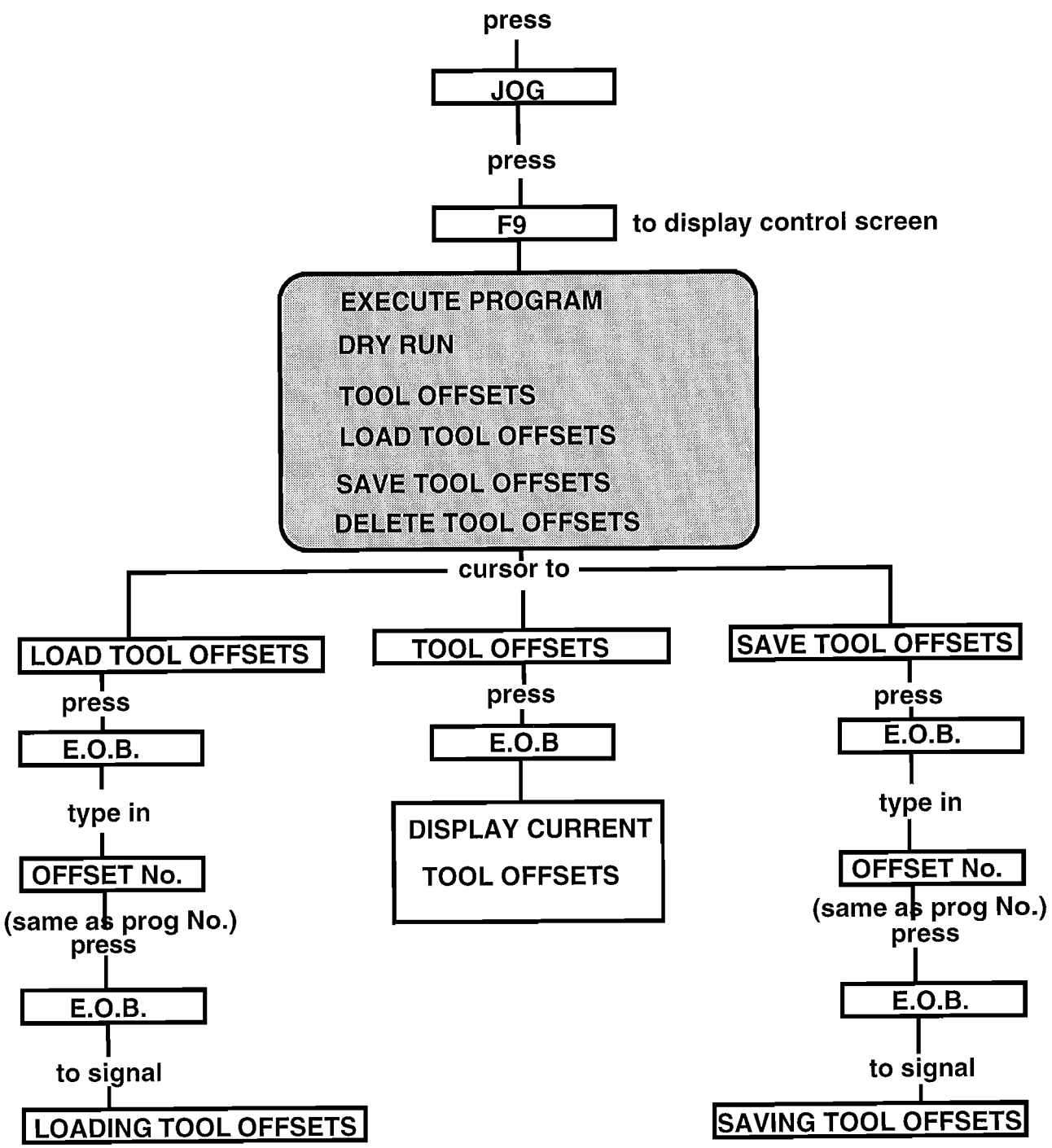
PRESS F10 FOR



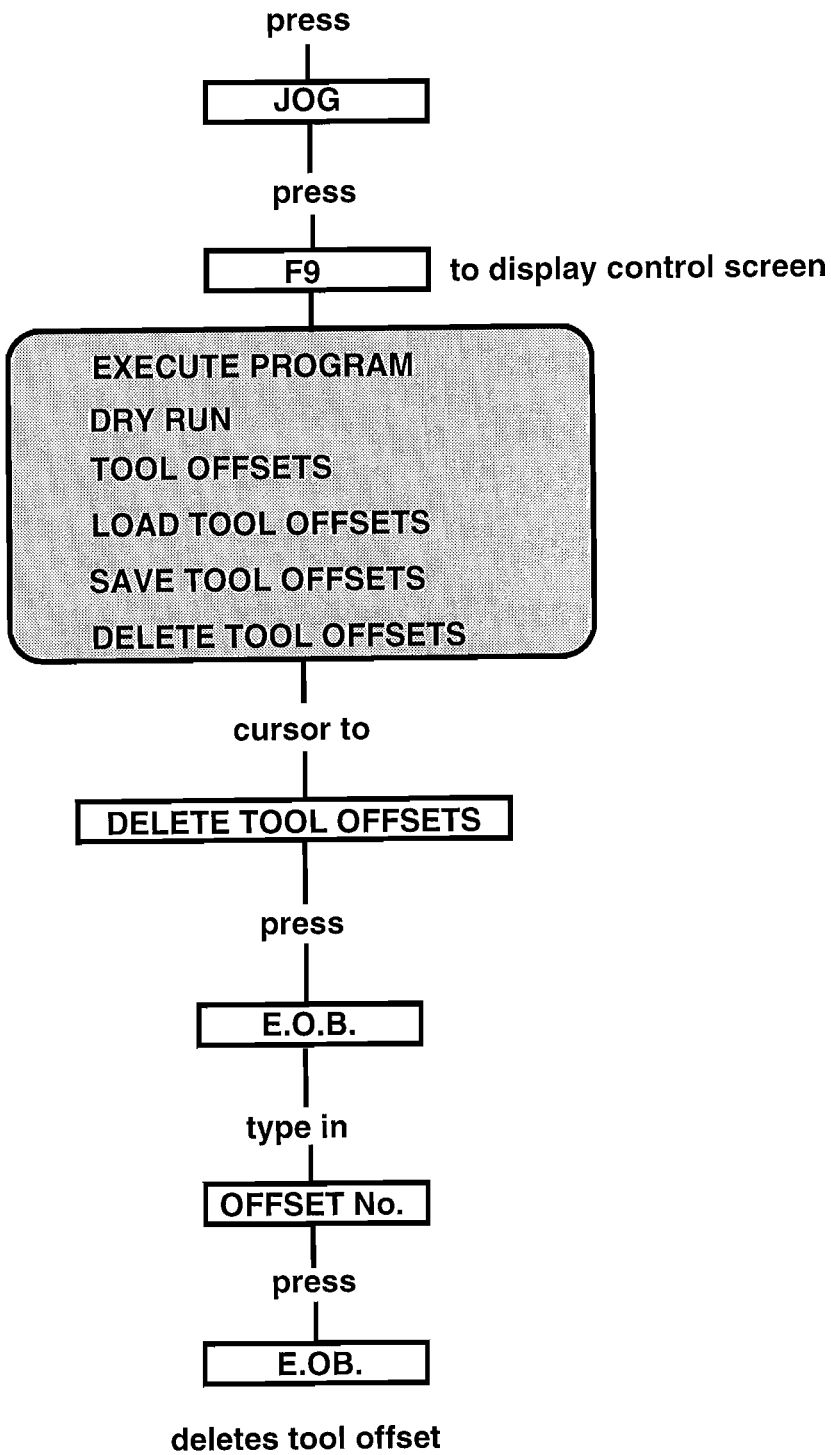
TO SAVE OR LOAD TOOL OFFSETS FOR A GENERATED PROGRAM

1. LOAD TOOL OFFSETS **2. DISPLAY TOOL OFFSETS** **3. SAVE TOOL OFFSETS**

N.B. THE MACHINE MUST FIRST BE HOMED

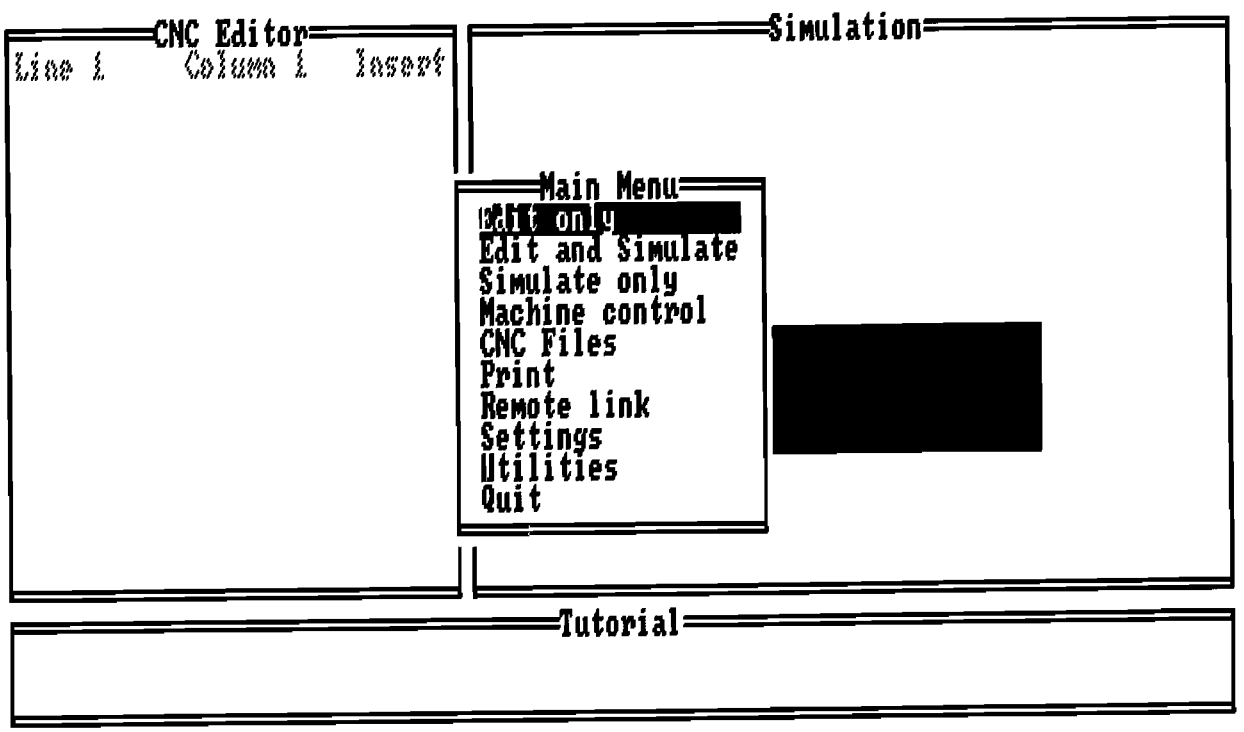


TO DELETE A TOOL OFFSET



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 the TU150E 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.

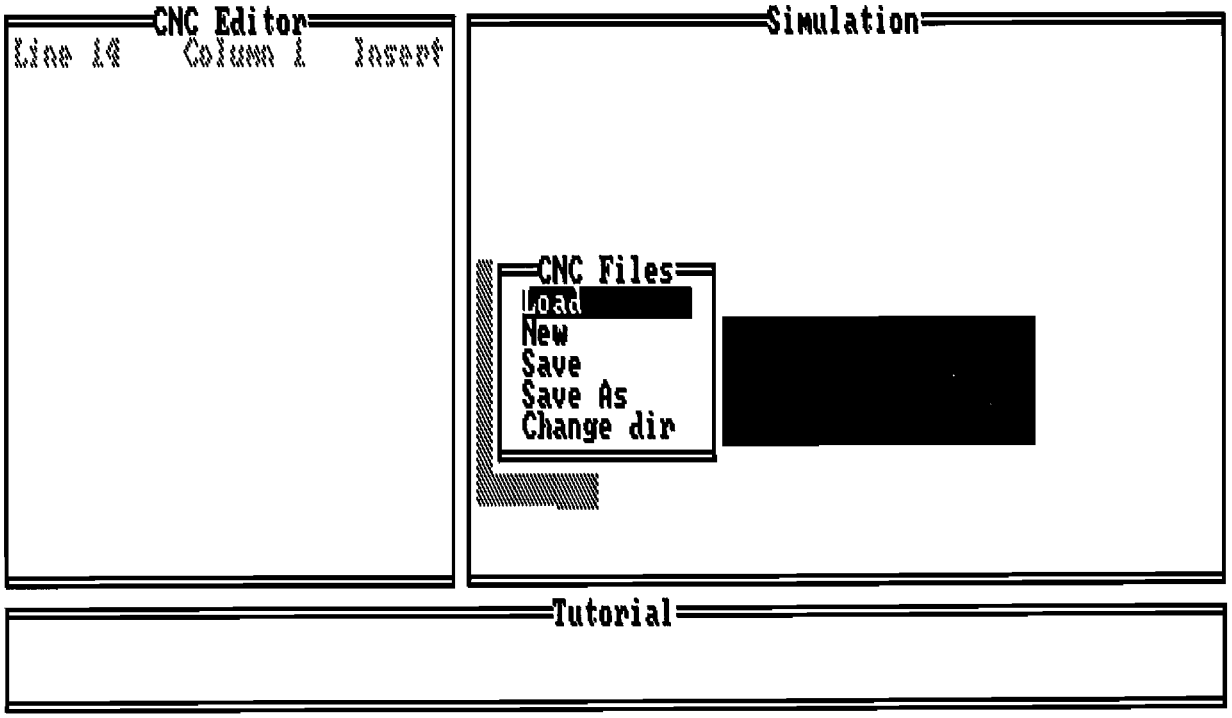
b) Use the cursor keys to position the cursor.

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

d) All Files are listed alphabetically in a window.

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

f) Press EOB to confirm or RESET to Exit



LOAD

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

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 Disc 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 ".HAR".

If you want to save to Disc 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.

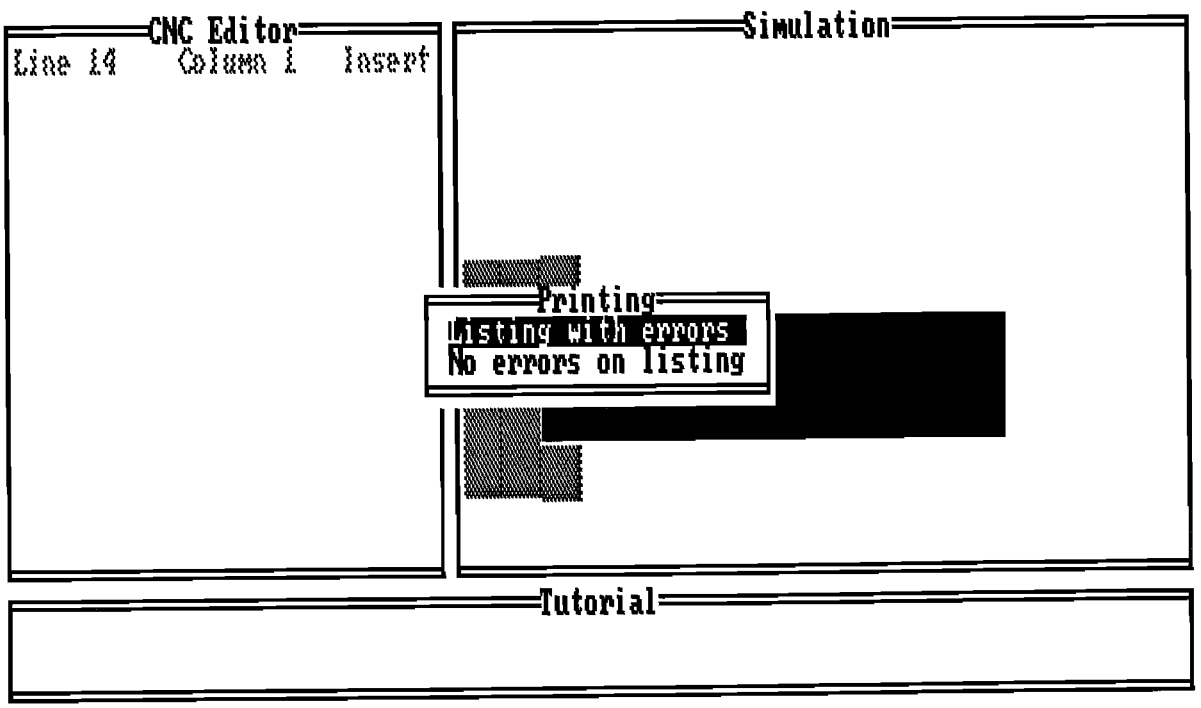
CHANGE DIR

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?

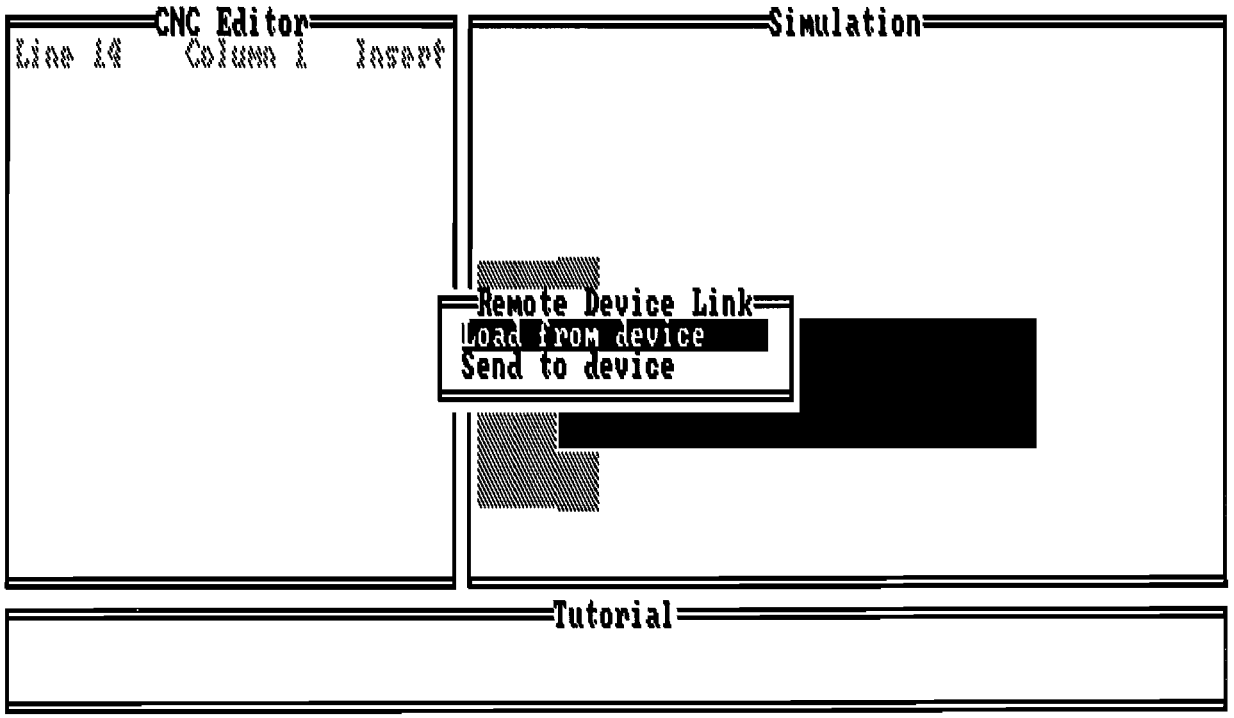
Have you set the correct parameters in the Settings Menu for"Print Device".?

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 machine and at the back of the Remote Device. A 25pin RS232 cable is required to link the machine.

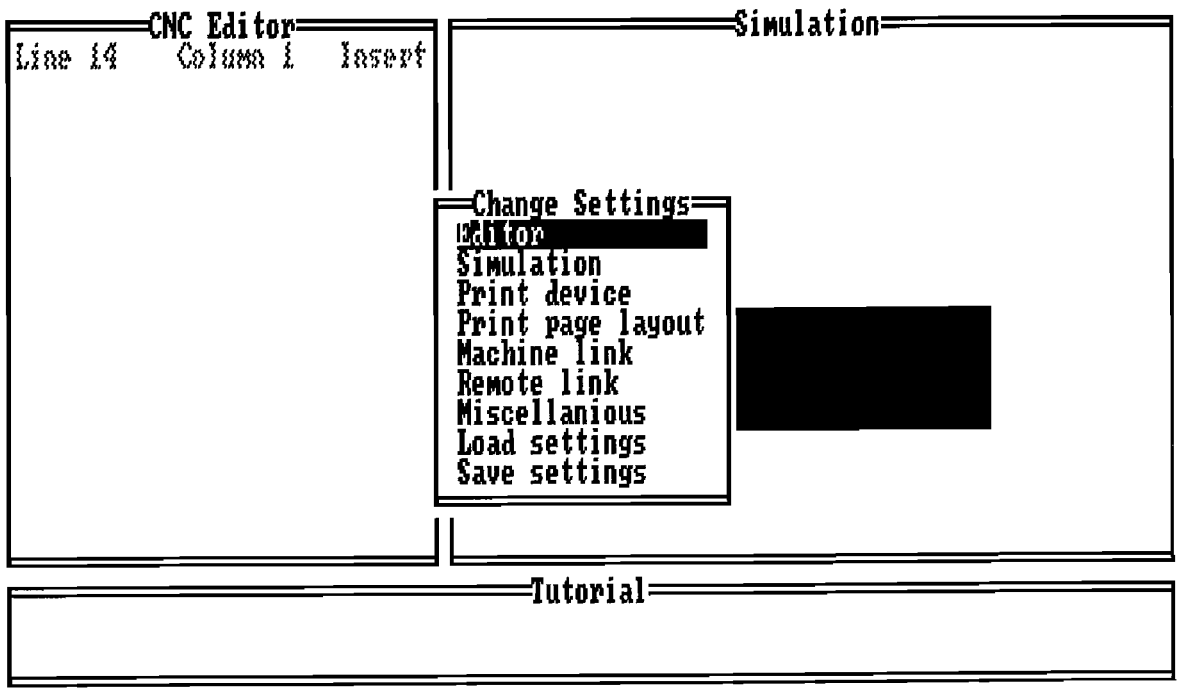
b) Whether the cable connections are faulty.

c) That the Remote device is set "Ready To Receive" or "Send", before sending or 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.

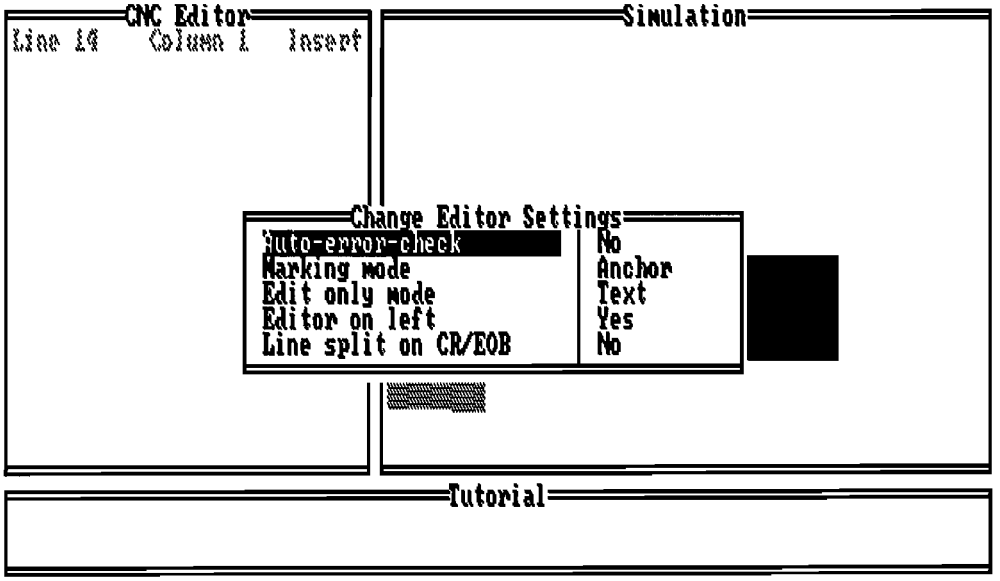
NOTE:- The RS232 connector on the side of the machine is named "COM1".

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 theTU150E machine and is only displayed for compatability with other HARRISON 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

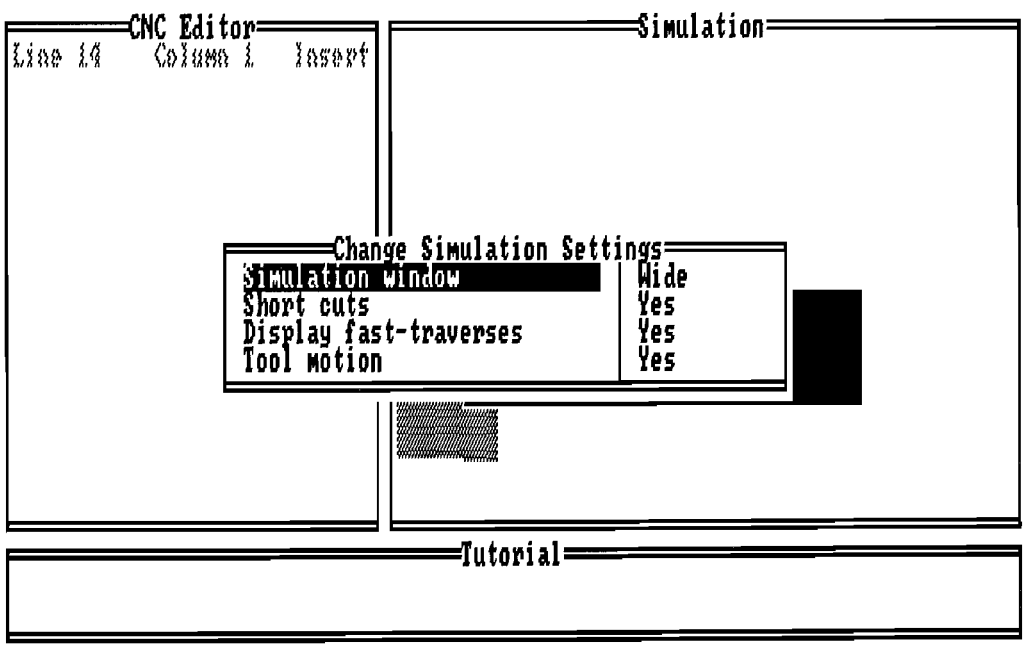
Splits the line at the cursor position after the Return key if set to ON.

NOTE:- Inefective on TU150E compatability.

CHANGE SIMULATION SETTINGS

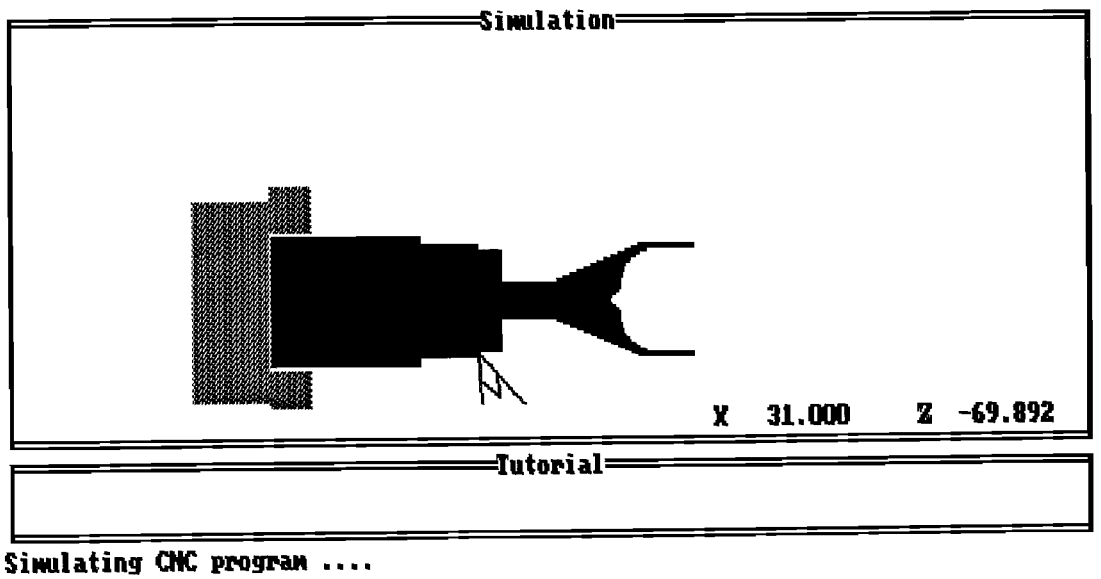
SIMULATION WINDOW

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.

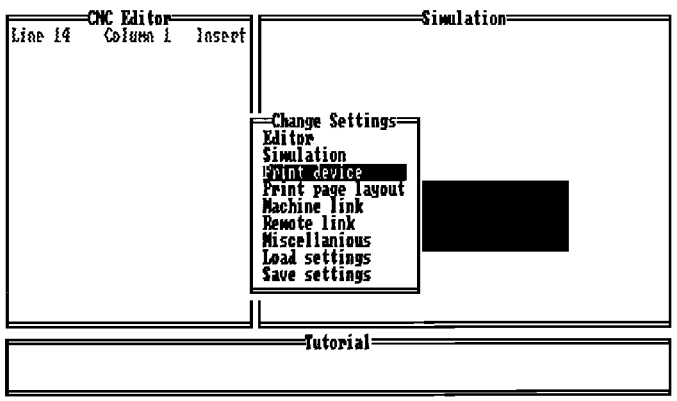
DISPLAY FAST TRAVERSE

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

TOOL MOTION

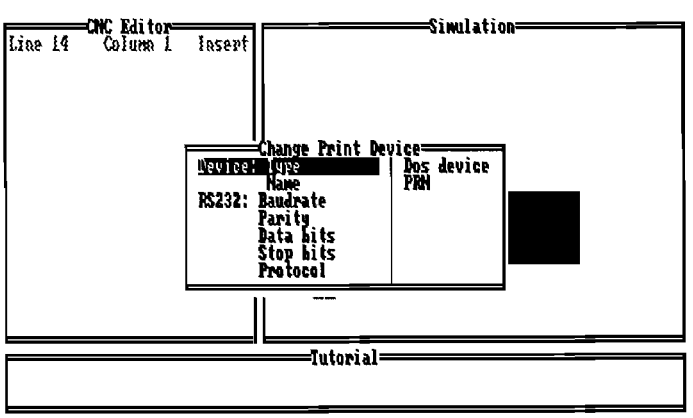
Toggles Tool Display ON and OFF. If it is set to ON, the Tool Shape is displayed. If set to OFF, a Toolpath Plot only is displayed.

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

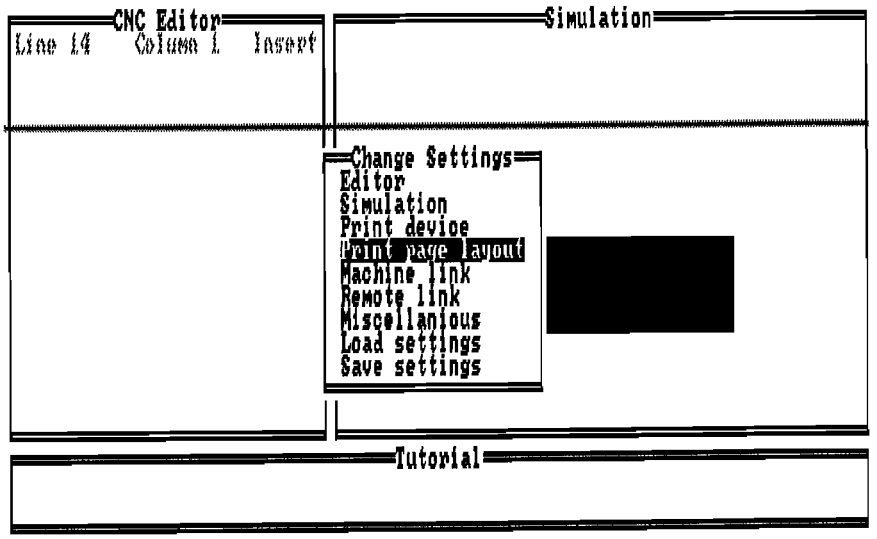
Protocol

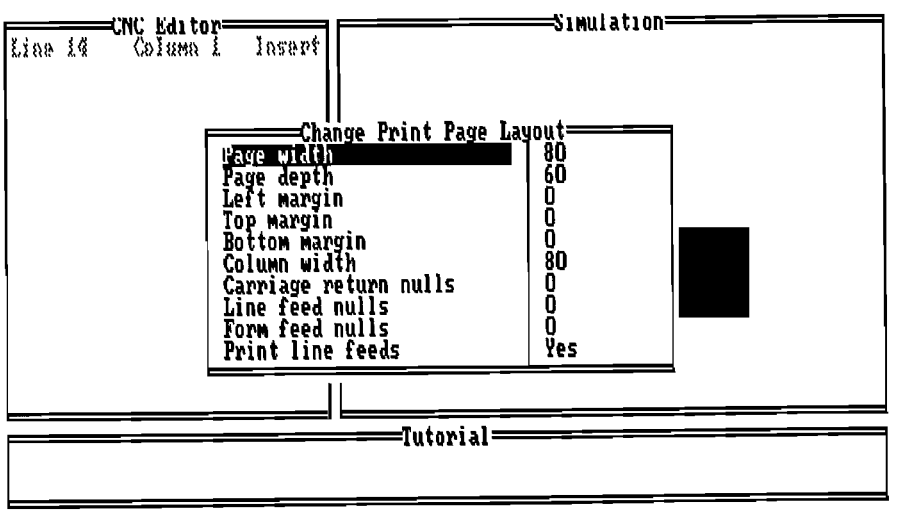
To change any of the above parameters, make sure that the Device is set to Serial. Highlight the option with the cursor keys, and press the RETURN key to scroll between the different options.

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 ".HAR".

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

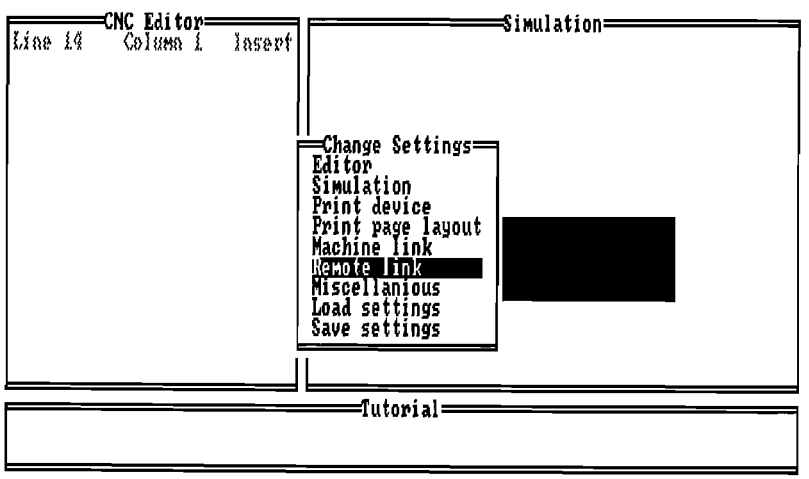
Line feeds and form feeds depend on the Printer's settings. If you are not sure, try different options until a correct printout is obtained.

10. CHANGE SETTINGS

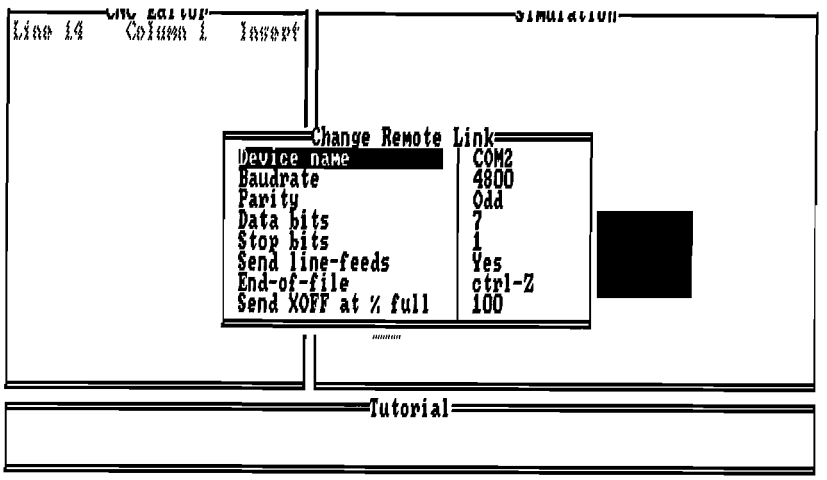
CHANGE MACHINE LINK (The machine link is always through the SERIAL PORT)

This option is ineffective on the TU150E and should therefore never be altered.

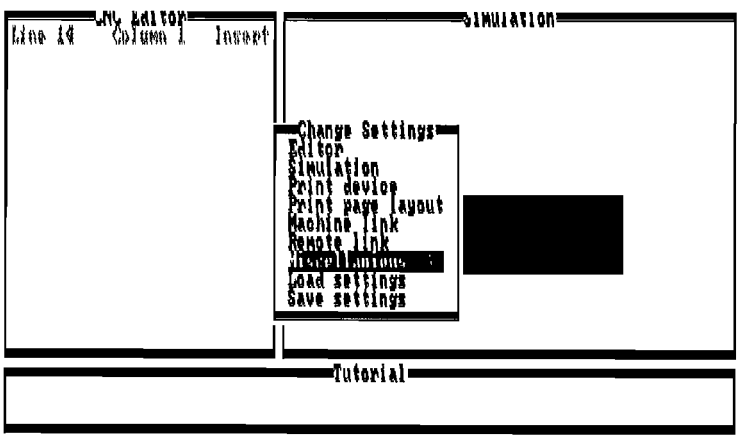
REMOTE LINK

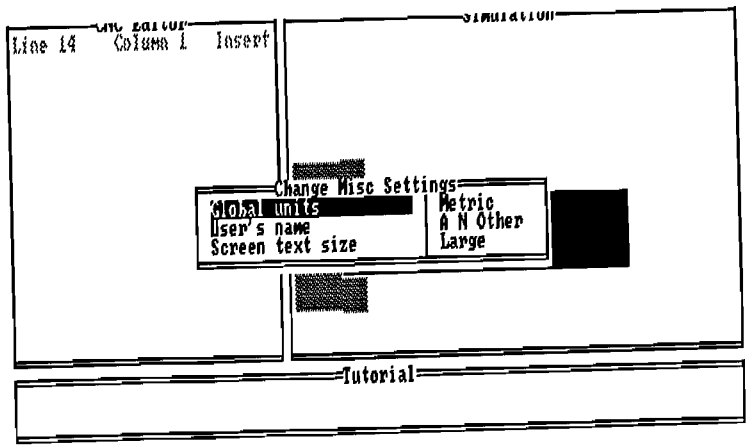


The settings for the remote link are for the Serial Port. This sets the protocol when communicating to an external device, such as a remote computer or paper tape punch.



CHANGE MISCELLANEOUS SETTINGS





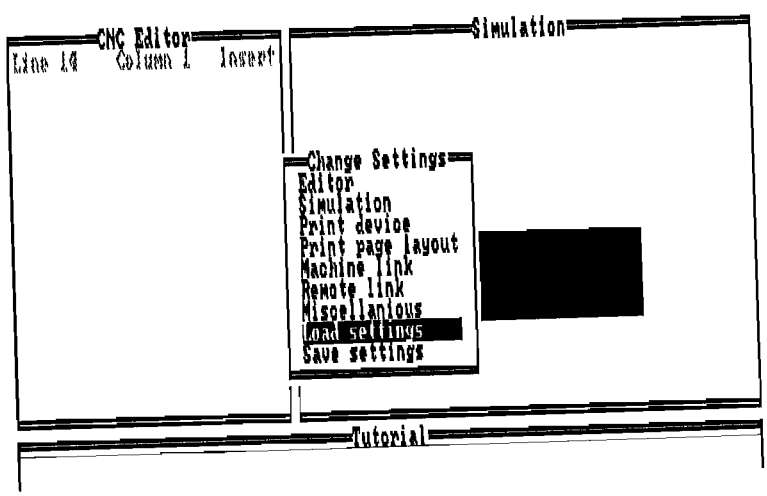
GLOBAL UNITS - USERS NAME - SCREEN TEXT SIZE

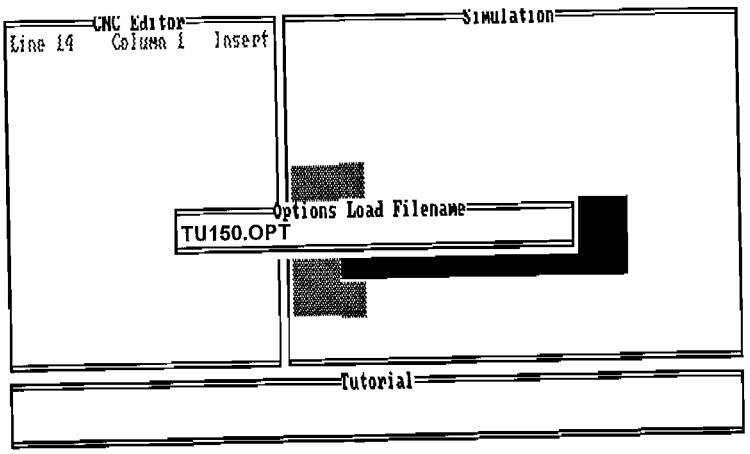
Global Units toggles between Imperial or Metric programming as the Default setting.

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

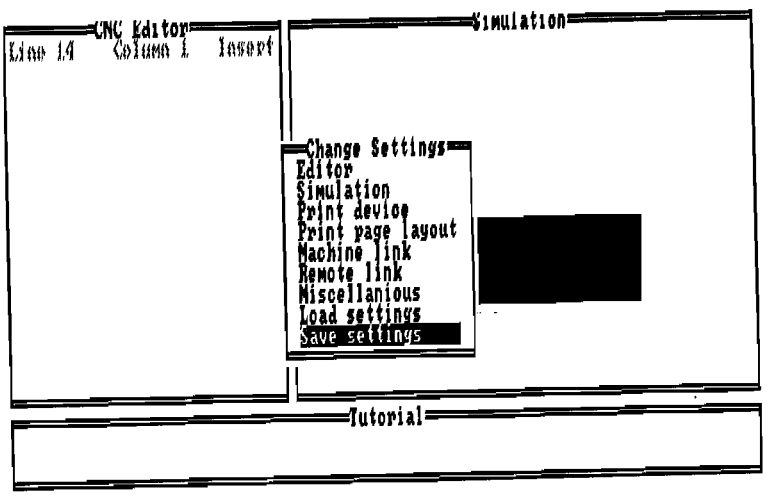


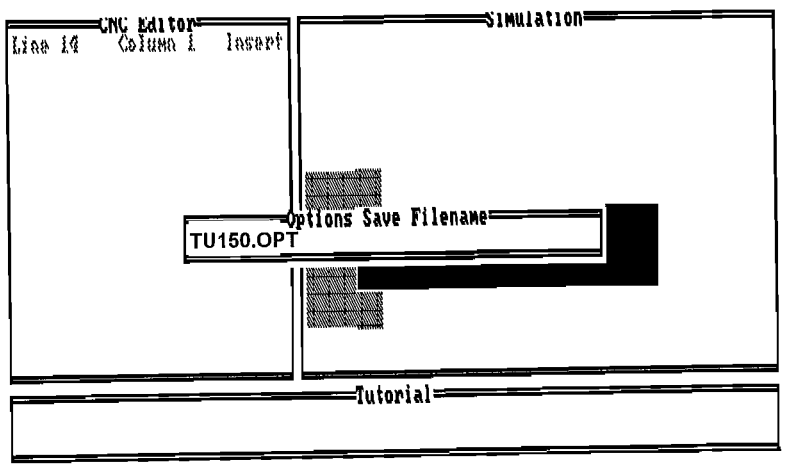


Any number of settings files can stored to Disk with the Extension ".OPT".

Select the Load Settings option, and the Default Settings File appears in the Edit Window. The default filename will be TU150.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

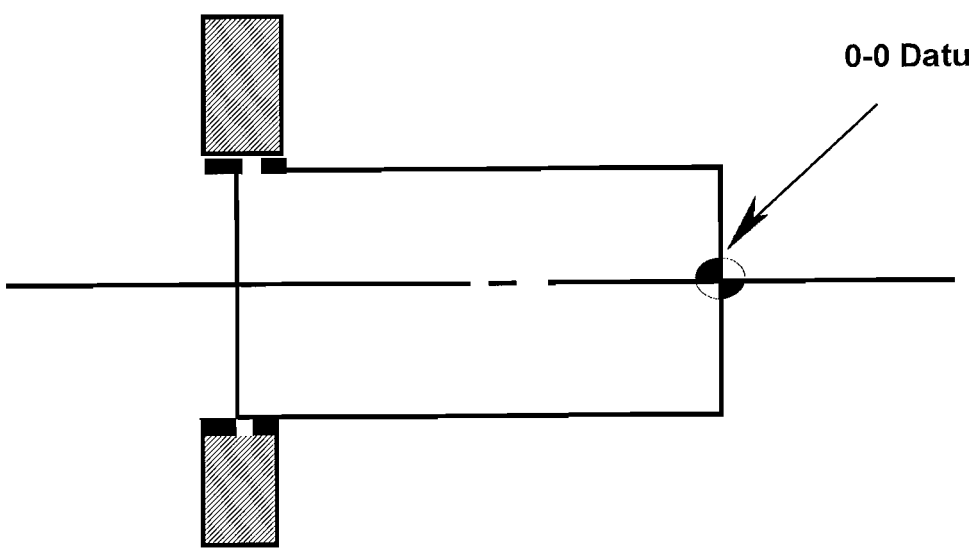
a program, and are ignored when it is executed.

Example: (Entering circular cycle

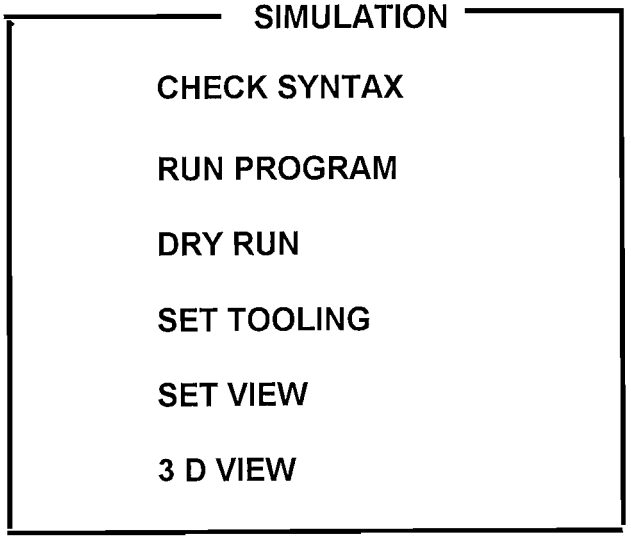
13. SIMULATION MENU

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).

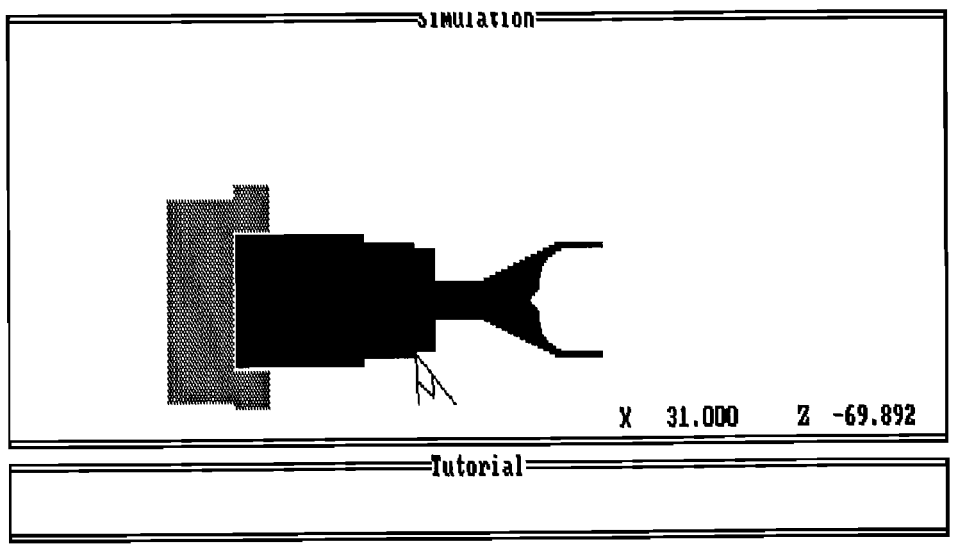
SET_VIEW

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.

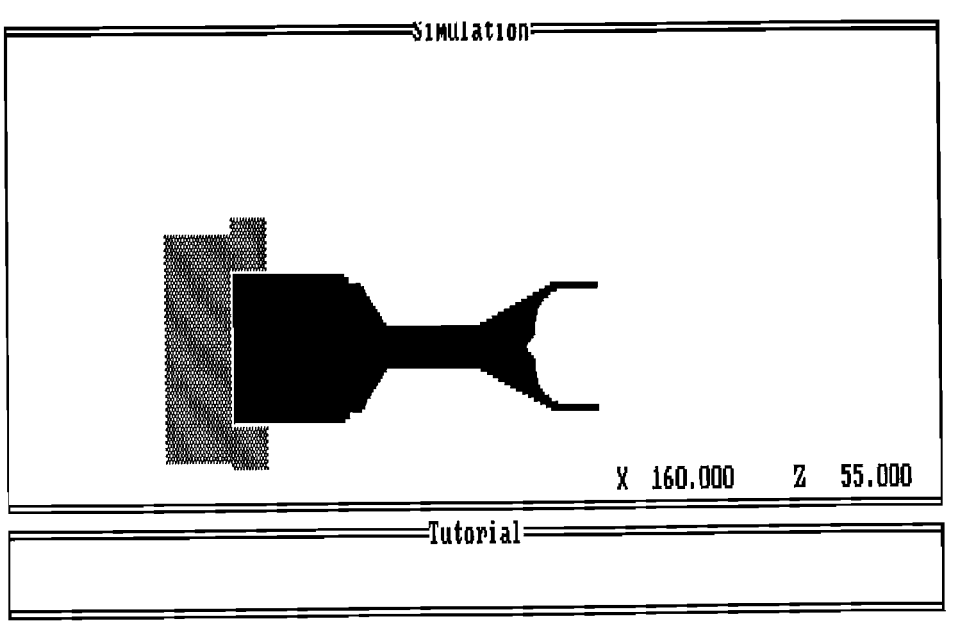
3D VIEW

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).

15. WORD DETAILS & AXIS COMMANDS

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 HARRISON 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
	0	G28	Reference Point Return
	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
	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	G76	Thread Cutting Cycle
1	G81	Drillng
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 HARRISON 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

M08 COOLANT "A" ON

* M09 COOLANT OFF

M10 CHUCK OPEN

M11 CHUCK CLOSE

M13 SPINDLE FORWARD & COOLANT ON

M14 SPINDLE REVERSE & COOLANT ON

M25 QUILL EXTEND

M26 QUILL RETRACT

M30 PROGRAM RESET & REWIND

M38 DOOR OPEN

M39 DOOR CLOSE

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

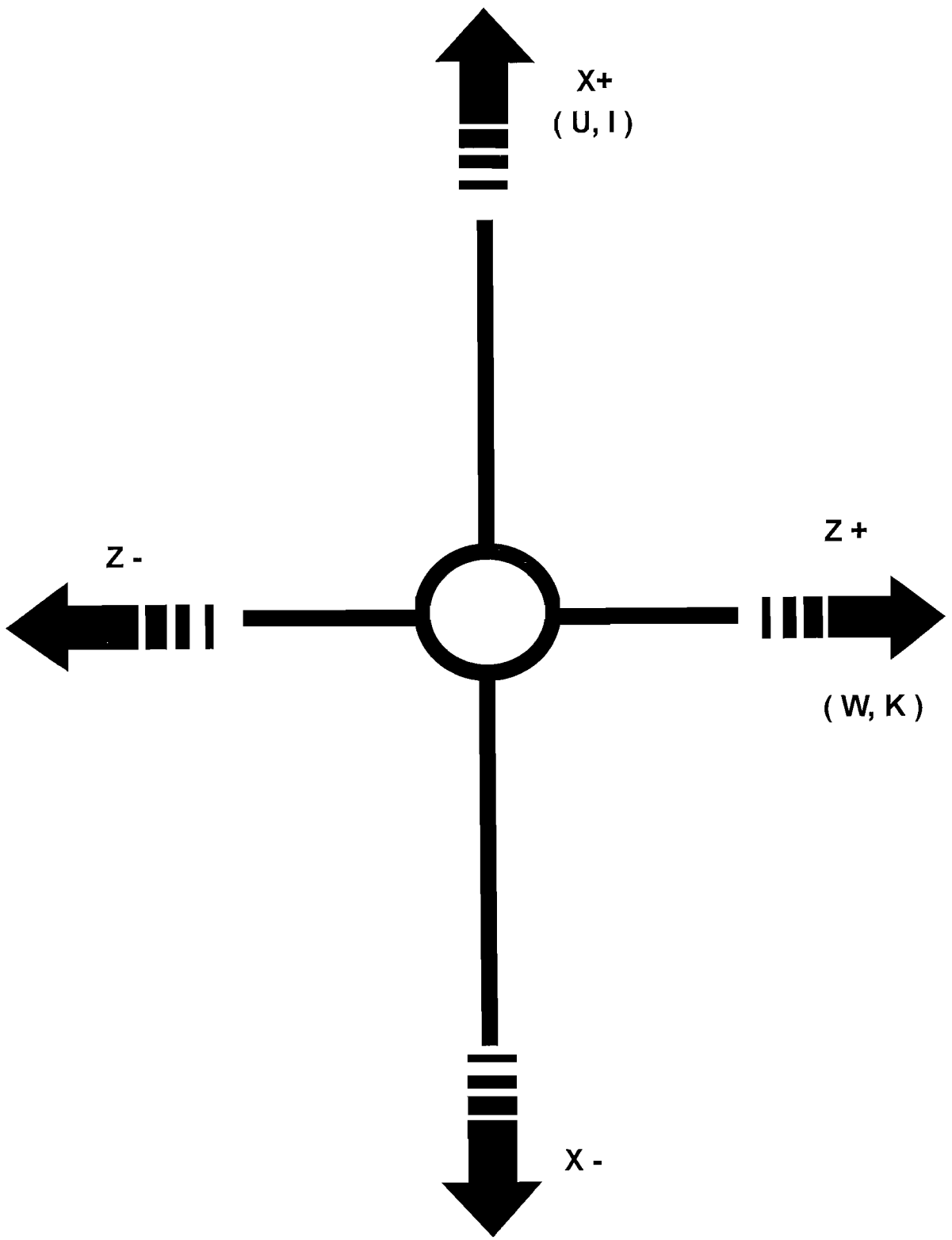
M76 WAIT FOR INPUT 1 TO GO LOW (from revision C)

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

M98 SUB PROGRAM CALL

M99 SUB PROGRAM END

16. BASIC MOVEMENTS



BASIC MOVEMENTS

STRAIGHT LINE MOTION (G00, G01)

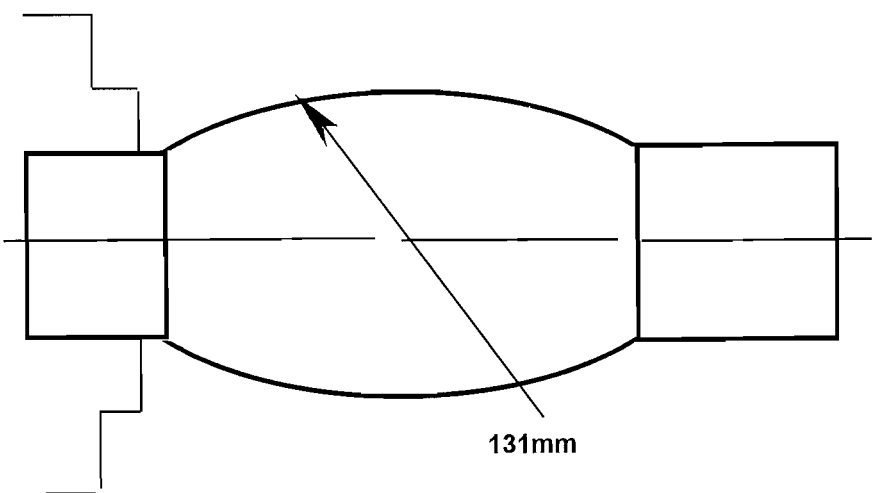
Almost all motion commands will be programmed as straight line motion for moving to, or cutting along diameters, faces and angles. During cutting statements, it is only necessary to change this mode of operation when an arc is to be generated.

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.

MAXIMUM ARC



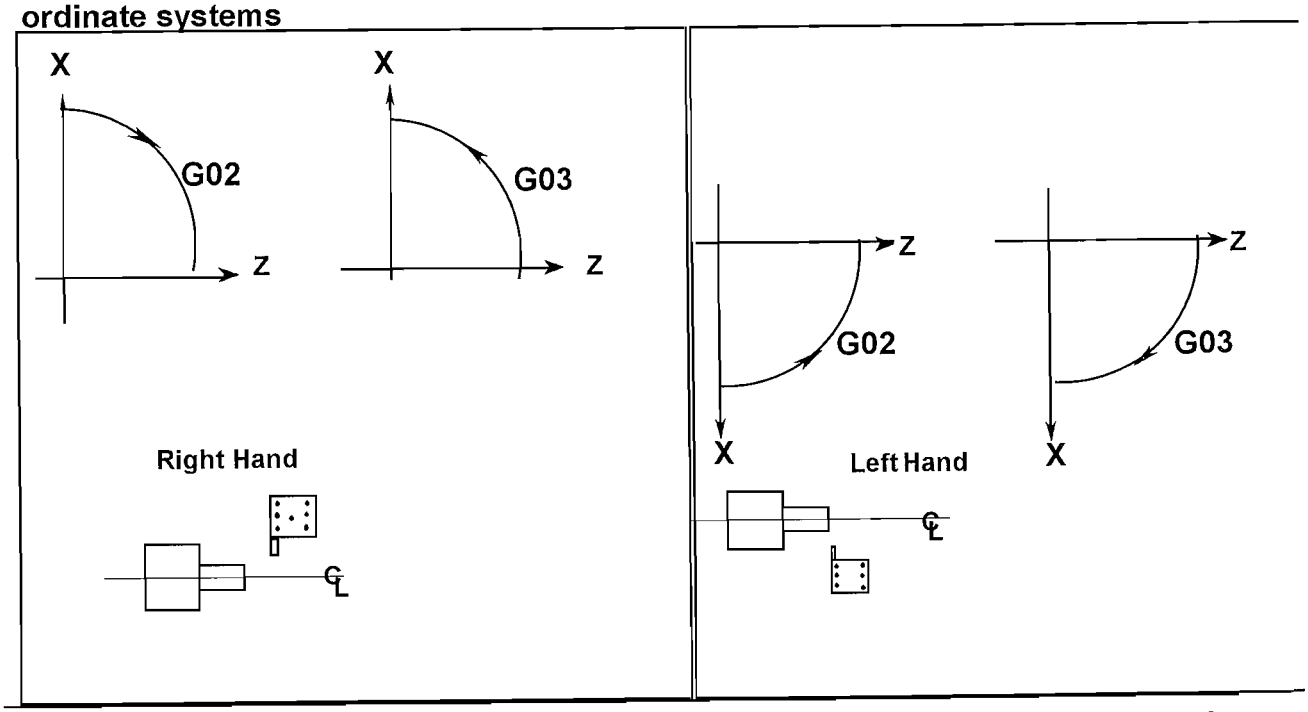
NB. The maximum arc that can be generated is 131 mm

CIRCULAR INTERPOLATION (G02 AND G03)

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

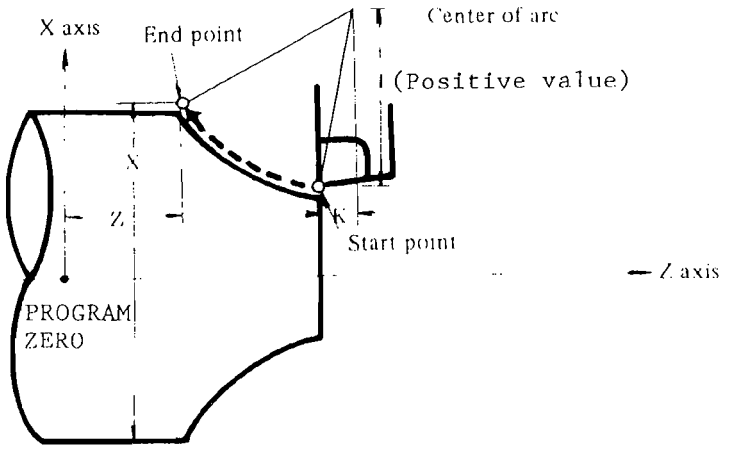
	Data to be given		Command	Meaning
1	Rotation Directiuon		G02	Clockwise Direction (CW)
			G03	Counterclockwise Direction(CCW)
2	End Point Position	Absolute Command	X, Z	Endpoint Position in the Work Co-Ordinate Position
		Incremental Command	U,W	Distance from Start Point to End Point
3	Distance from Start 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



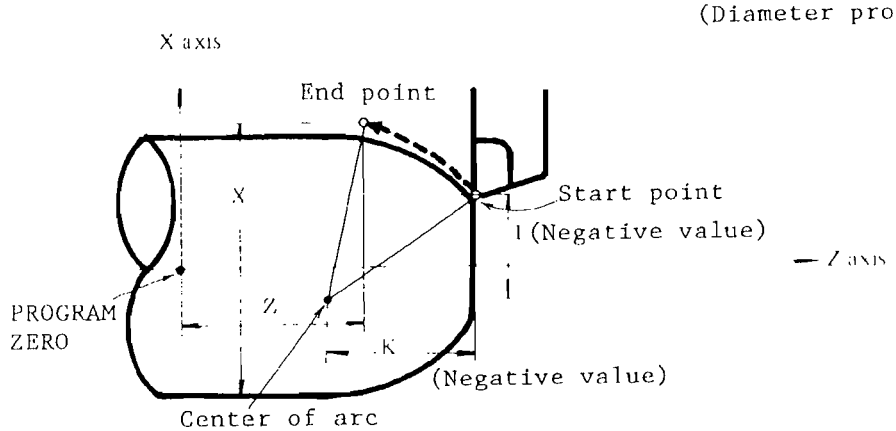
G02X ---- Z ---- I ---- K ---- F ----;
(absolute programming)

(Diameter programming)



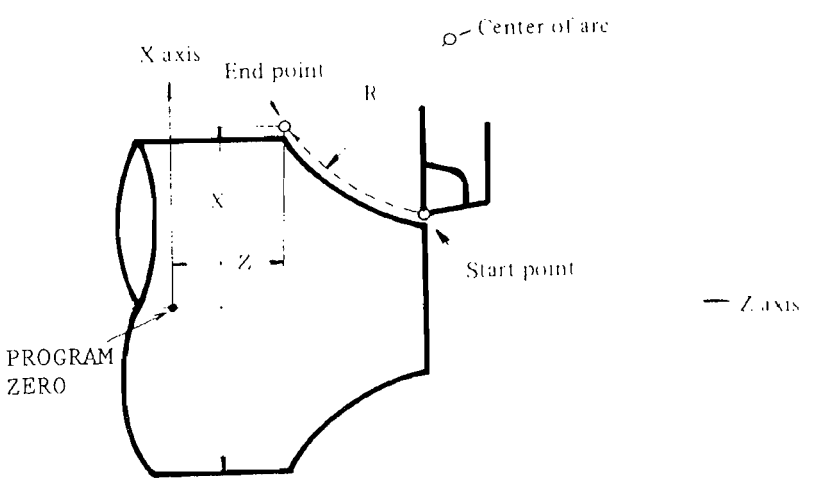
G03X ---- Z ---- I ---- K ---- F ----;
(absolute programming)

(Diameter programming)



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

(Diameter programming)



Program Example (Absolute Command)

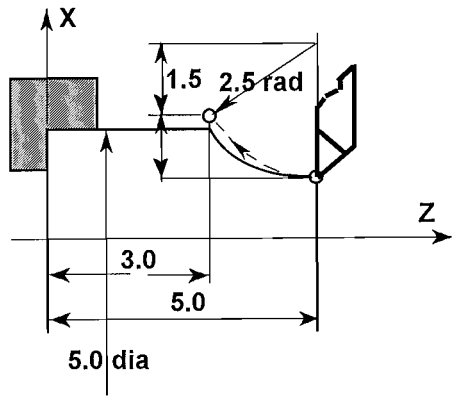
(Diameter Programming)

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;

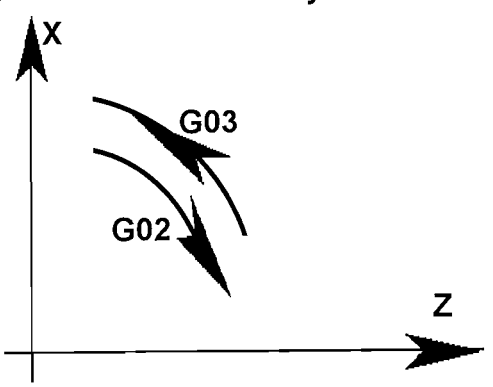
or G02 U2.0 W-2.0R2.5 F0.03;



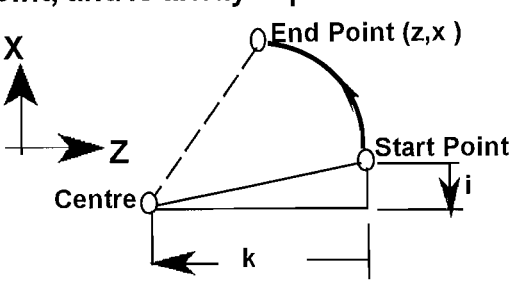
The feedrate for circular interpolation is specified by the F Code. The feedrate along an arc (tangent to an arc) is controlled to maintain the specified feedrate.

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:-

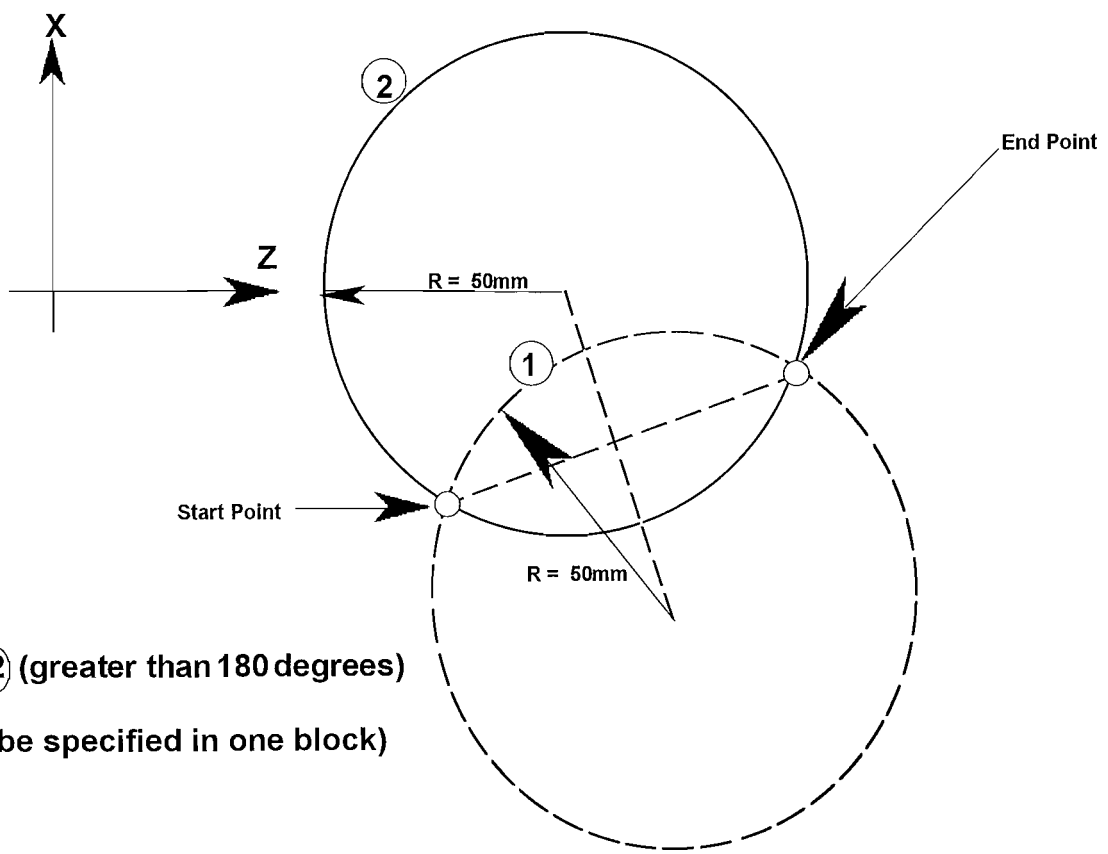
G02 }
    } X .....Z..... R..... ;
G03 }

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.

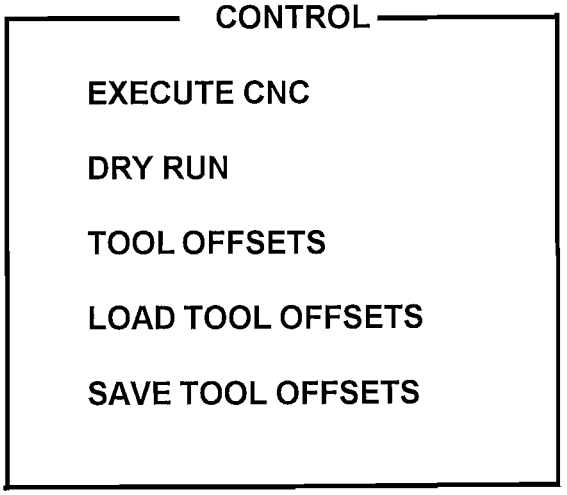
· 3. Draw tooling layout sheet showing tools to be used in the program and indicate the station numbers for each tool.

· 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.

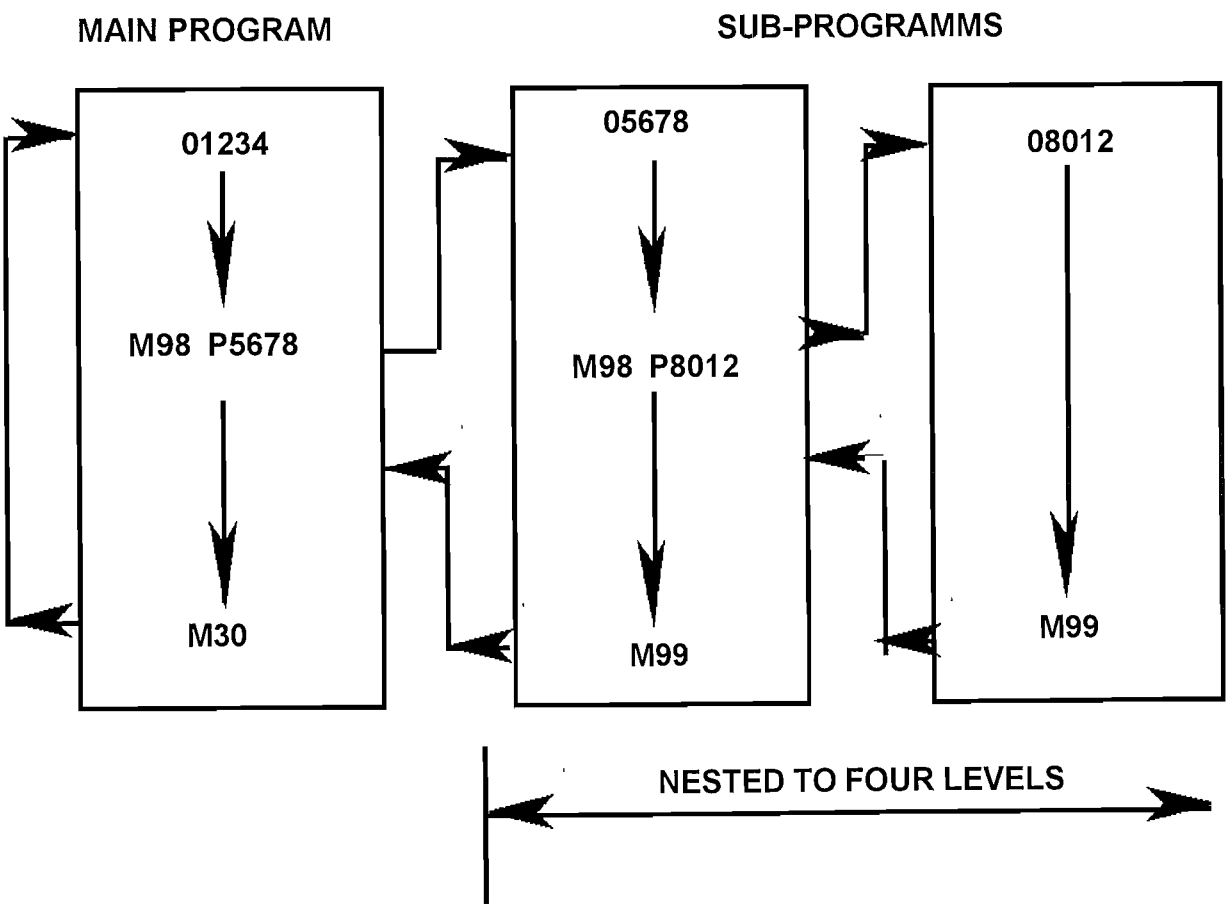
SAVE TOOL OFFSETS

Saves current offsets to disk.

PROGRAM AND SUBROUTINE IDENTIFICATION

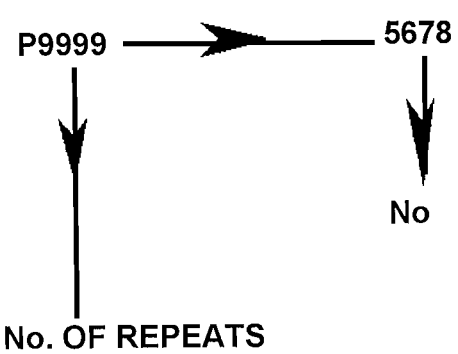
The first block of a program/subroutine must contain a program number "O".

The program would be as follows:-



M98

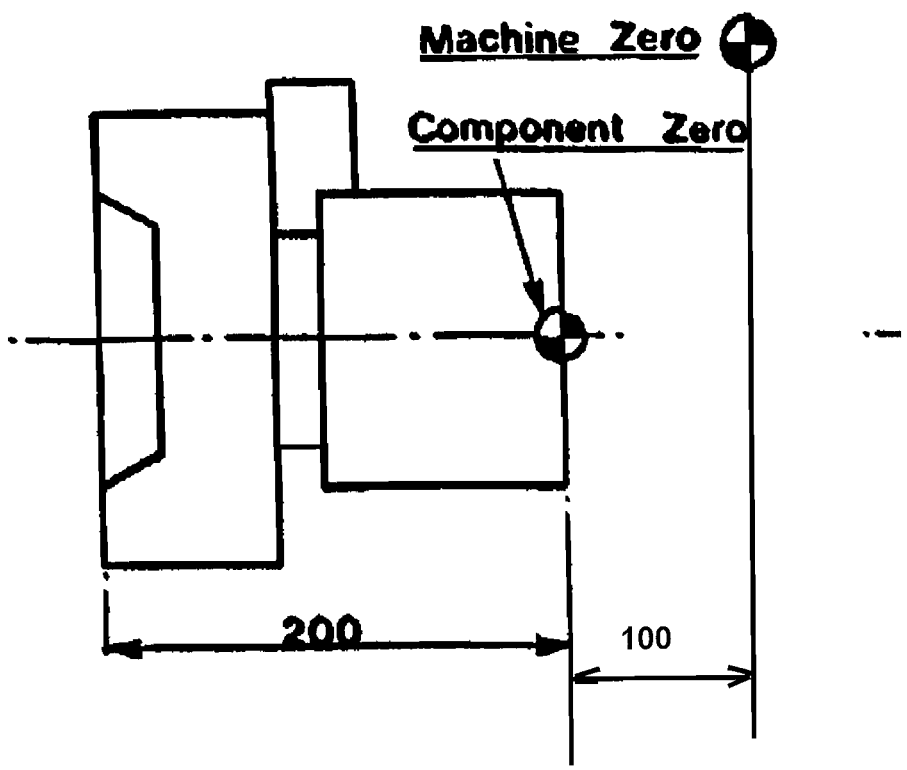
SUB PPROGRAM CALL



See page 18.1 for further details.

COMPONENT ZERO

During the Programming Procedure step 2 (Page 17.2) 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 :-

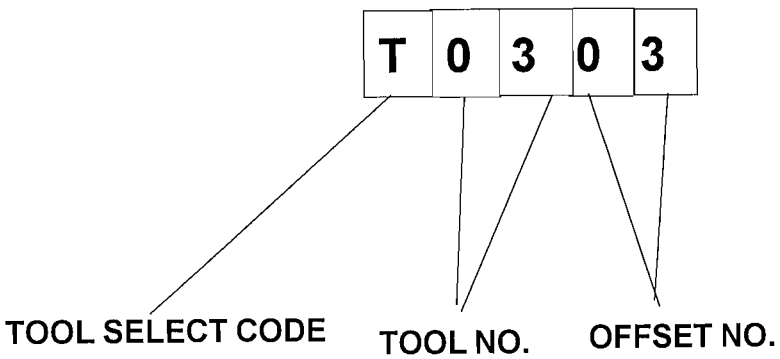
- Datum the machine
- Any offsets already in the file can be overwritten.
- Select tool in MDI,(JOG mode).
- Press T, then tool number required, then EOB. The correct tool will now be selected.
- Start the Spindle - Press S, then a spindle speed, then press EOB.
- Press SPINDLE CW.
- Using AXIS JOG keys, touch on the end of the bar.
- Press the MENU\OFFSET Key.
- Press MZ 0 EOB Z now reads zero.
- Press MENU OFFSET 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 MENU OFFSET 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.

TOOL NUMBERS and OFFSET NUMBERS

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

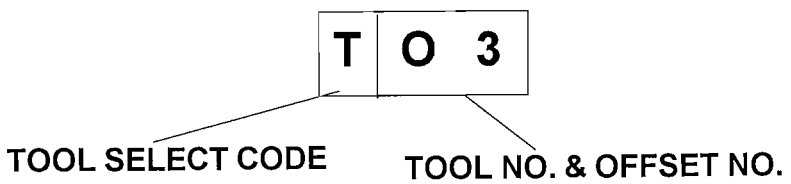
If a four digit number is programmed following the letter T, then the first two digits represent the Tool number, and the second two digits represent the Tool Offset number.

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.

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

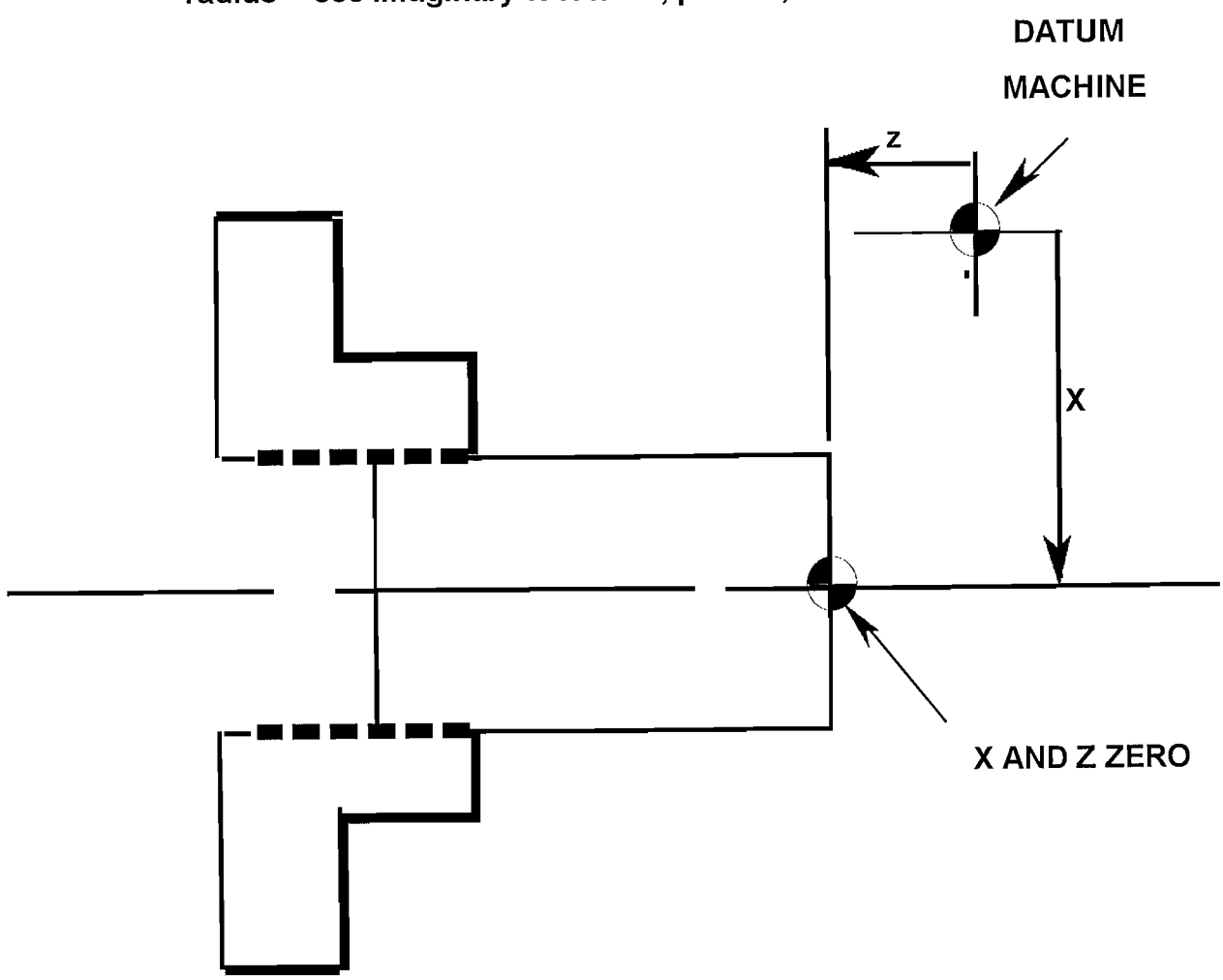
N60 G01 X.... Z.... F....

TOOL DATA

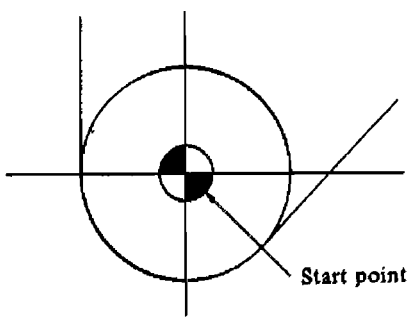
By using an OFFSET FILE, the program can be prepared using a common reference point.

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

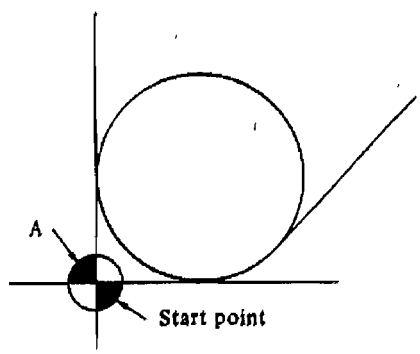
- **X and Z = Tool Offset for X and Z. This is based on the incremental distance calculated by the software allowing for diameter programming.**
- **R = Tool Radius. This is the actual radius of the tool tip.**
- **T = Tool Location Code . This code represents the orientation of the tool radius - see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



IMAGINARY TOOL NOSE



When programmed using the tool nose center



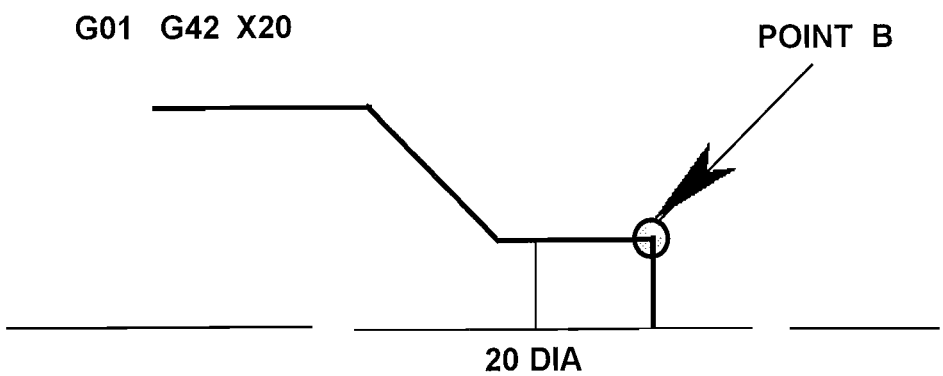
When programmed using the imaginary tool nose

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
```

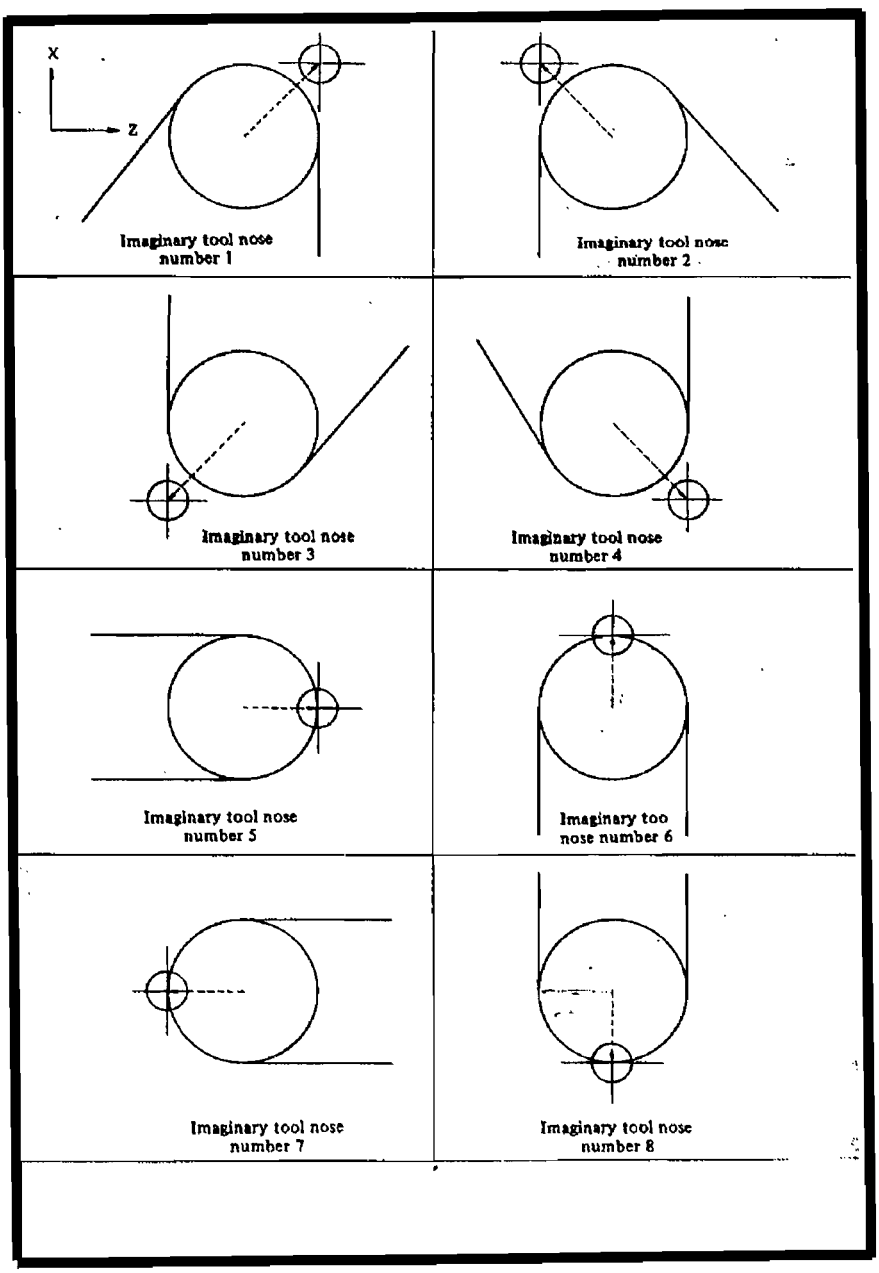


In the linear move before entering the TOOL NOSE COMPENSATION, always add on a value more than the radius of the tool.

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.

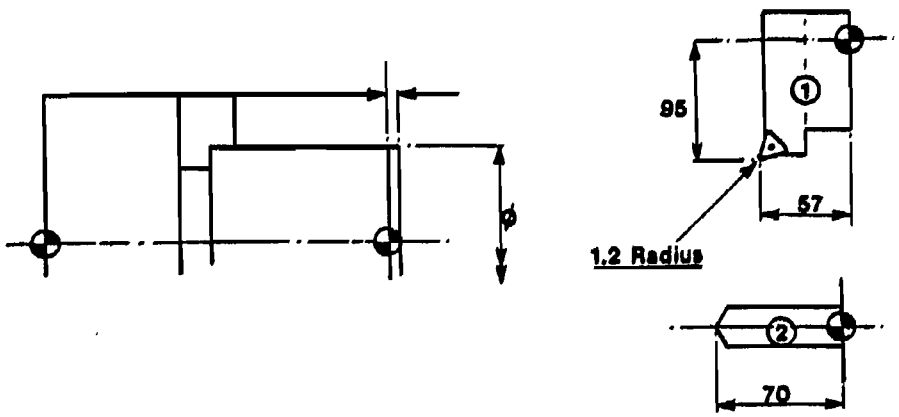
NB. The two linear moves must be greater than the tool radius compensation.

DIRECTION OF IMAGINARY TOOL NOSE.



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)

18. SUBROUTINES, CYCLES AND SAMPLES

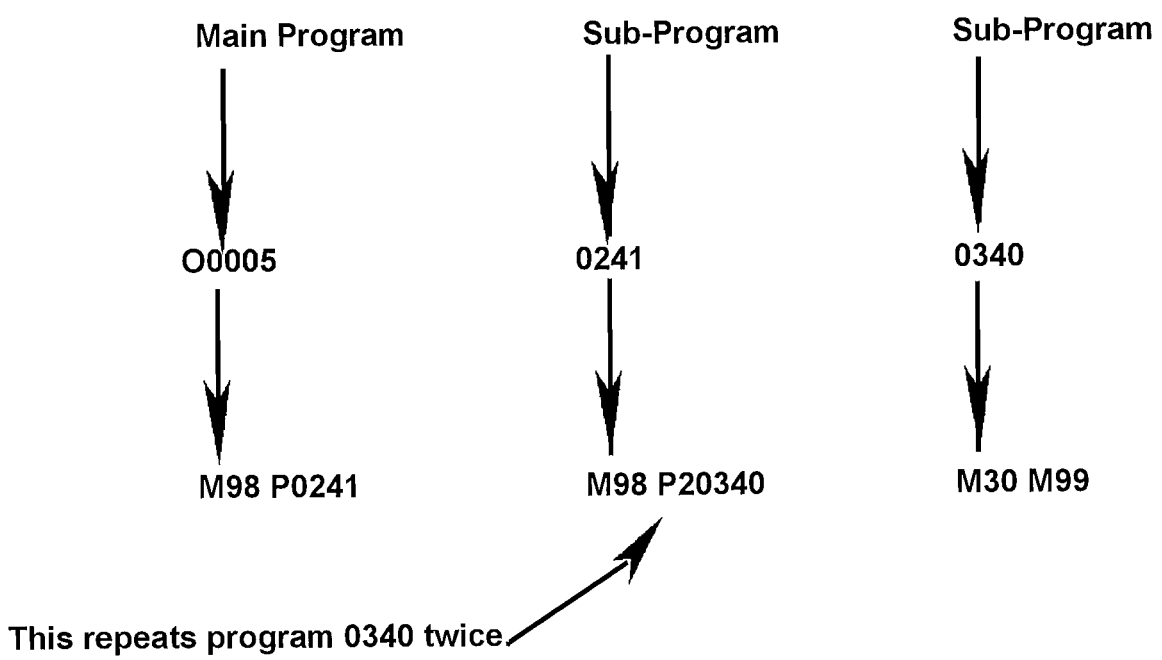
SUBPROGRAMS (SUBROUTINES)

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

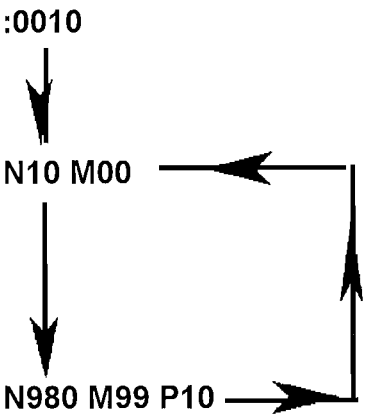
The machining sequences, which are repeated and can be used several times, are stored as subroutines and called up using the program jump functions.

M98 - Jump command to another program.

M99 - Return command.



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

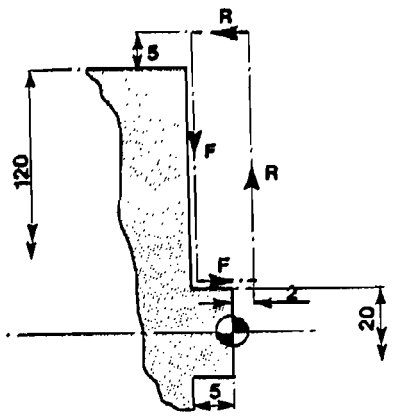


M99 generally indicates the end of a subroutine and allows the jump back to the main program. If it is used with "P" address, this indicates the "Jump To" block number.

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

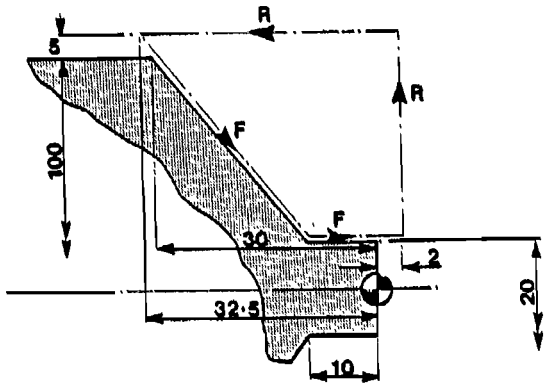
Line N10 must read M00 to stop the cycle for component loading. All information prior to N10-i .e. standard tool geometry-would not be read after the first cycle.Therefore M30 would not be programmed in this case.

G94 FACING CYCLE



N500 G00 X130.00 Z2.0

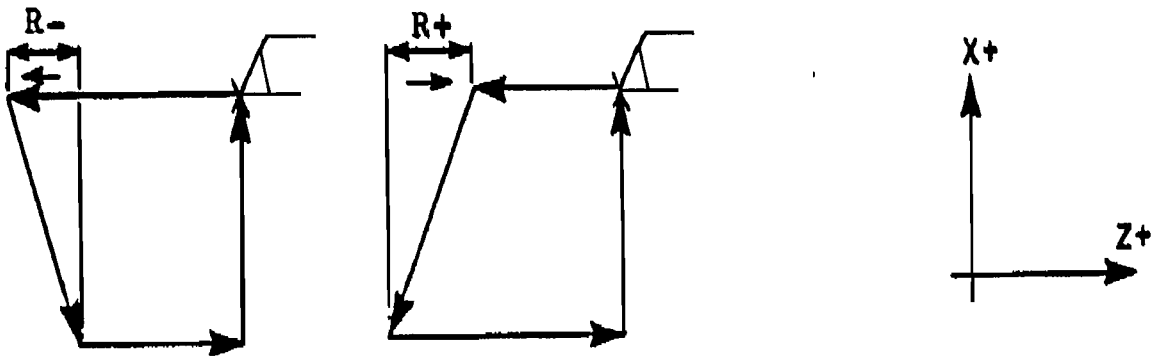
N510 G94 X20.0 Z-5.0F0.3



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.

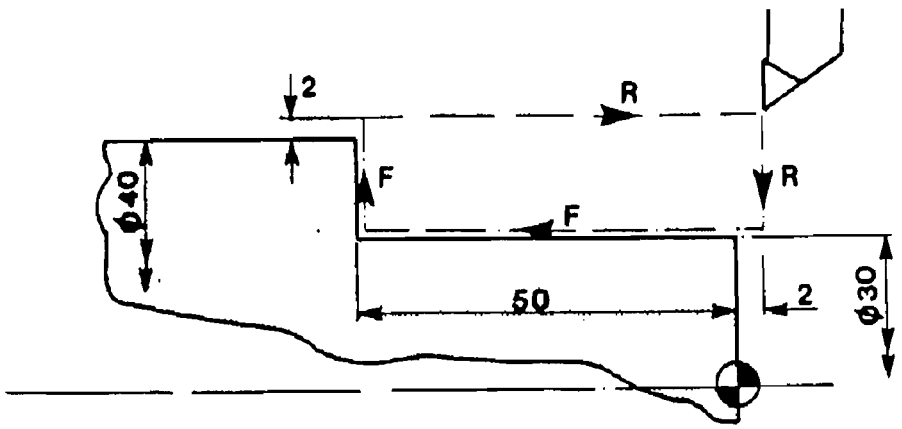
X - Final cutting position.

Z - Final cutting position.

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

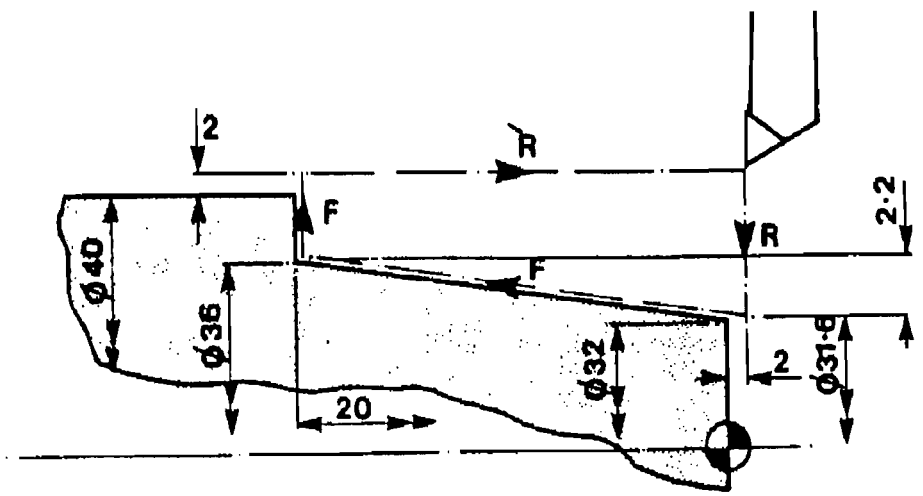
This R function can be omitted for cuts parallel to the X axis.

G90 TURNING CYCLE



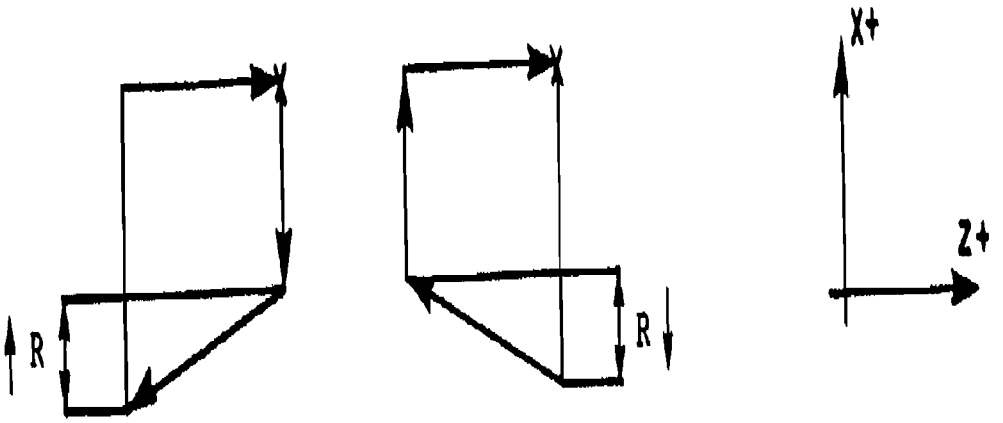
N600 G00 X44.0 Z2.0

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.

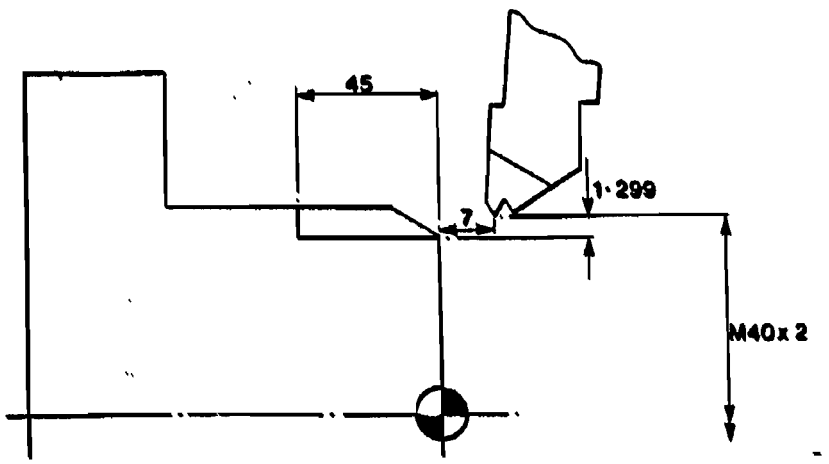
X - Final cutting position.

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

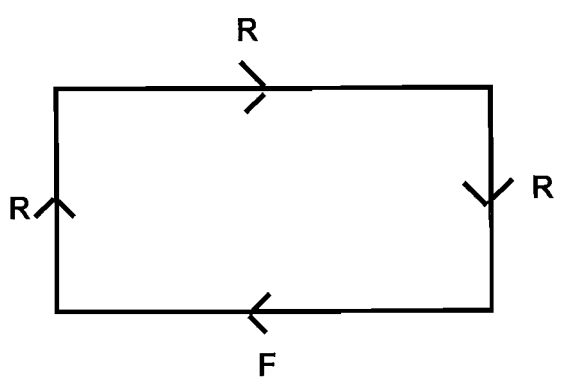
N700 M13 G00 X50.0 Z7.0

N710 G92 X39.35 Z-45.0 F2.0

N720 X38.954

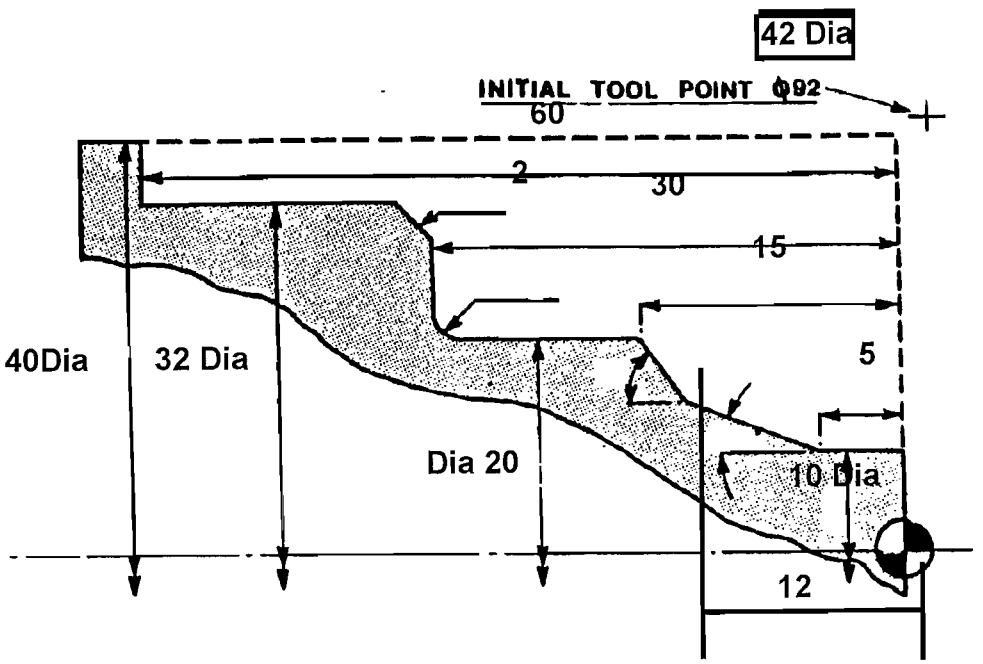
N730 X38.65 etc.

.



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"Z" 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.

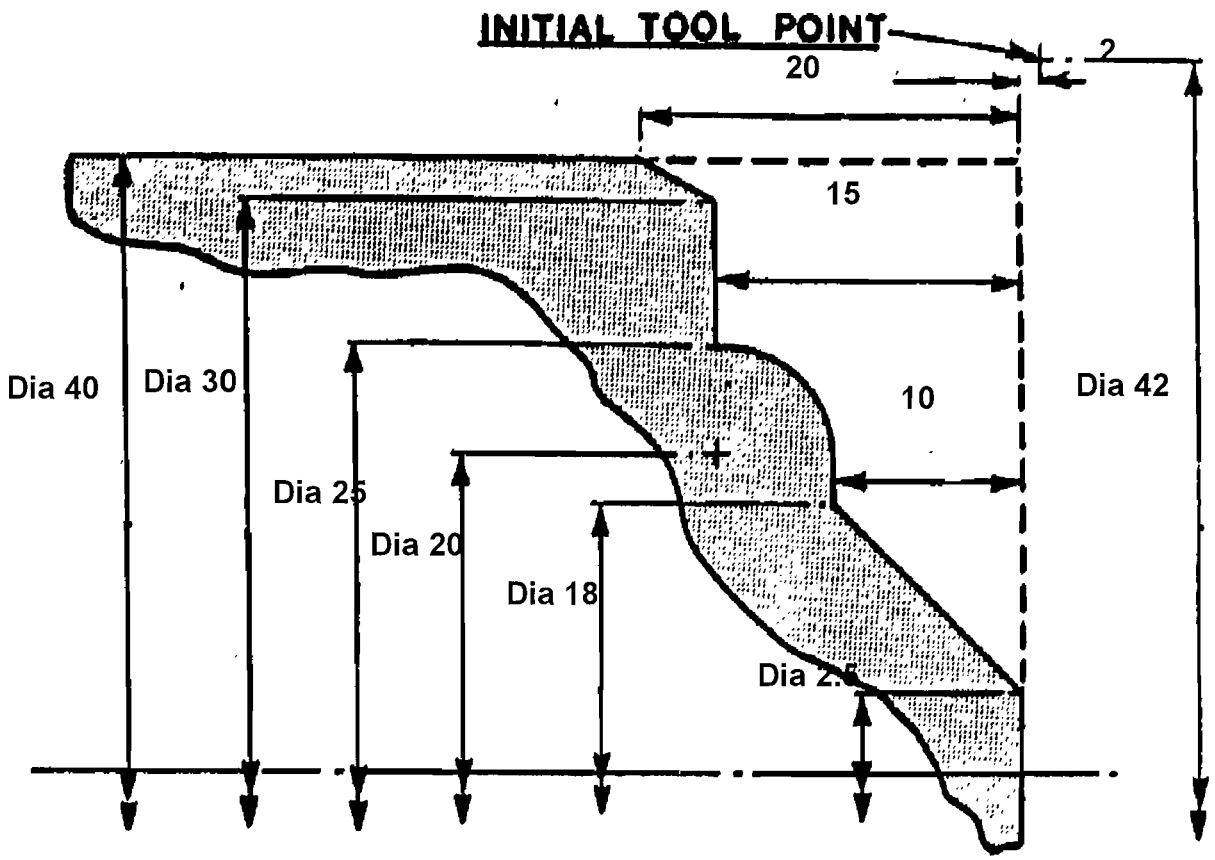
Q - Sequence number of the last block defining the contour.

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"X" 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

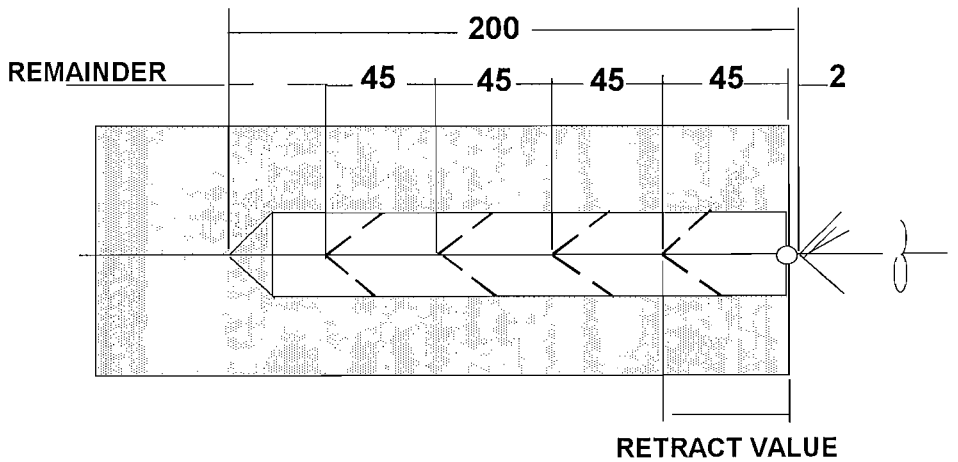
F - Feed rate for roughing cycle.

G70 FINISHING CYCLE

N200 G70 P70 Q120

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

G74 PECK DRILLING



N700 G00 X0 Z2.0 T020 M08

N705 G74 R45

N710 G74 Z -200.0 Q45000 F.15

N705 G74

R - Retract Value

N710 G94

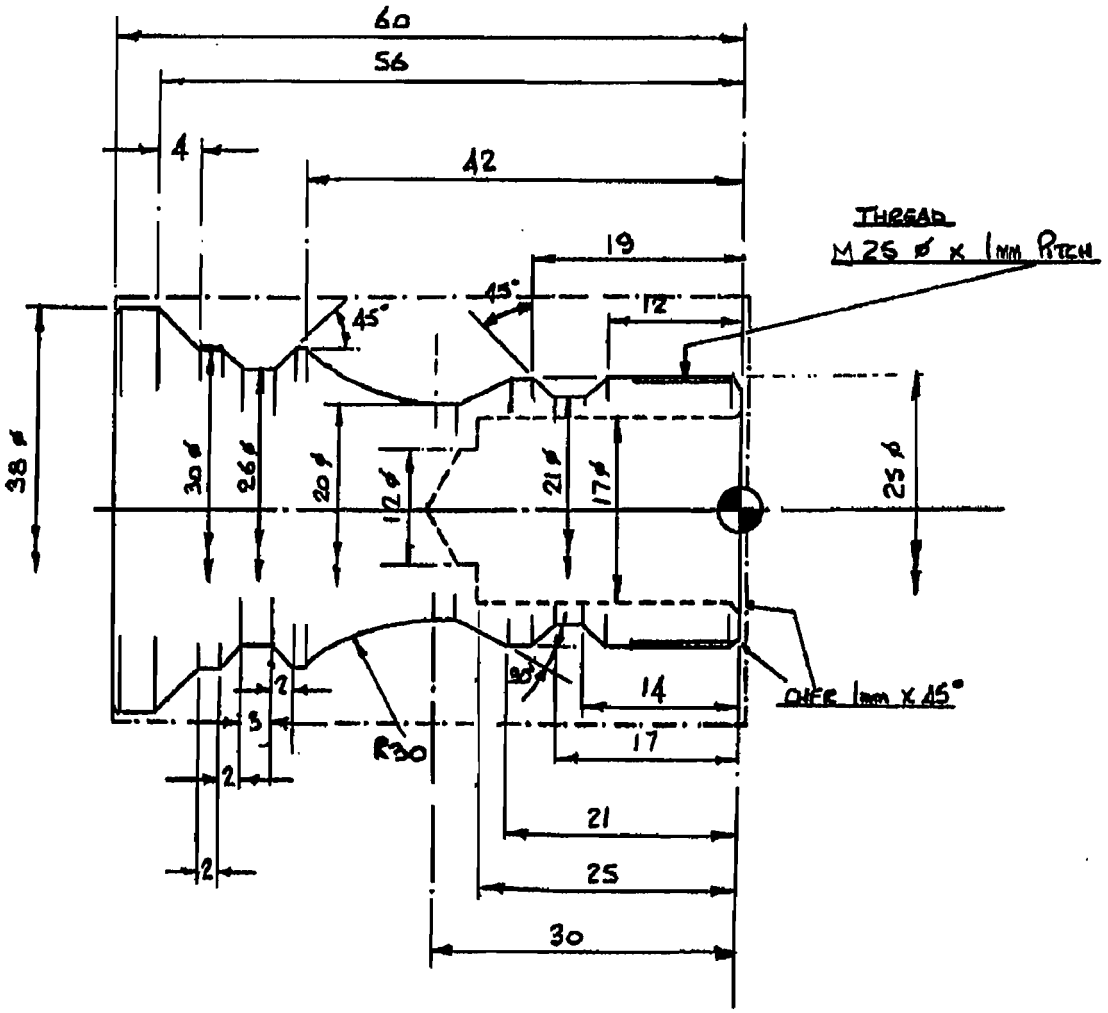
Z - Final Hole Depth

Q - Peck Increment, measured in microns, (No Sign Needed)

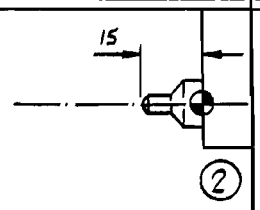
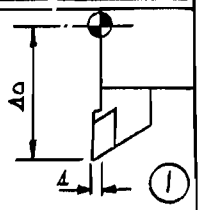
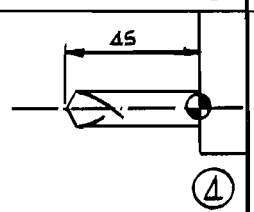
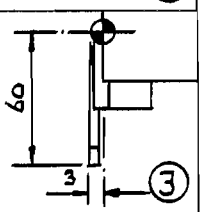
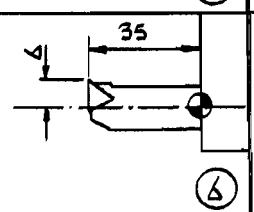
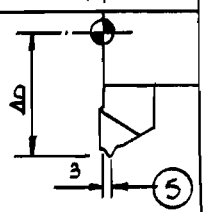
F - Feed Rate

EXAMPLES

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



TOOLING SHEET

TOOLING SHEET			
CUSTOMER	DENFORD LTD	INTERNAL	EXTERNAL
COMPONENT	TEST PIECE HARRISON		
DRG.NO.			
OPN. 1	DATE		
PREP. BY			
TOOLING DETAILS			
FACE & TURN			
CENTRE DRILL			
DRILL			
THREAD			
BORE			
PART OFF. CUT GRIP			

PROGRAMMING SHEET

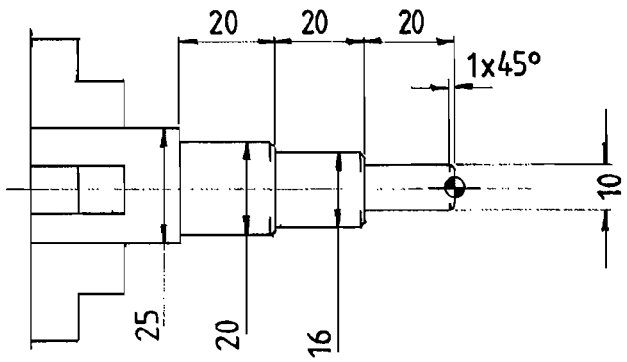
```
N10  G50 S2500

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
```

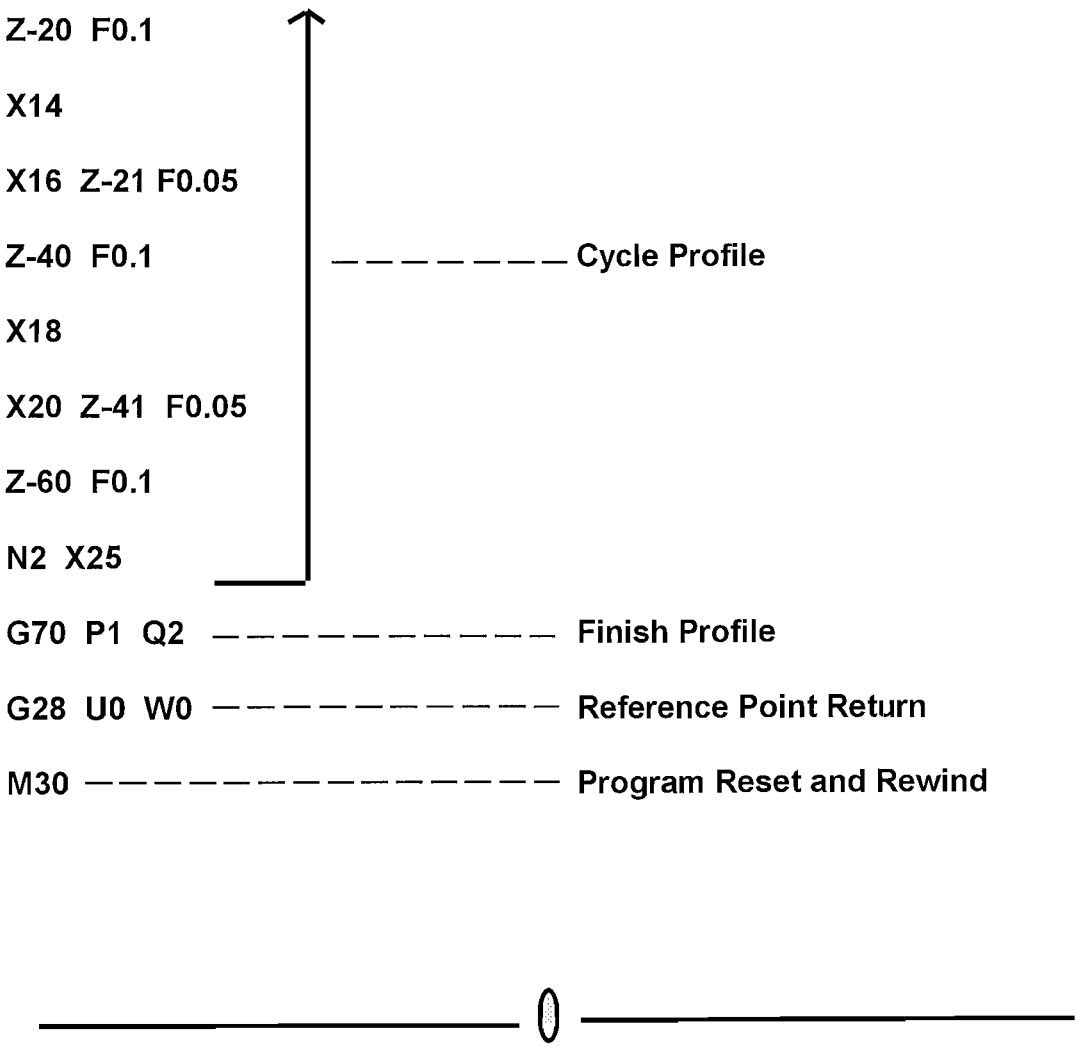
```
N690  G00 U5

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

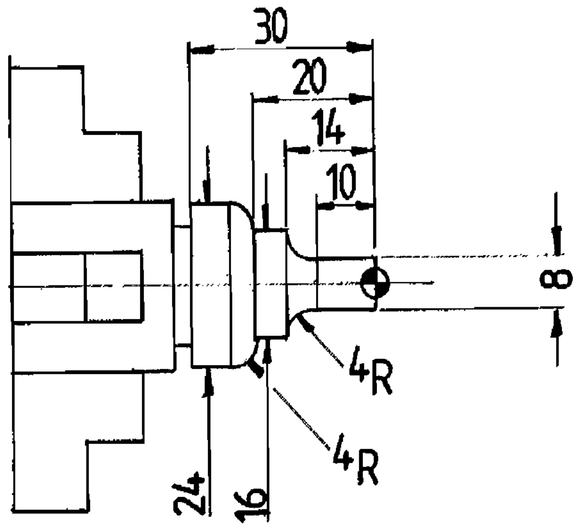


SAMPLE PROGRAM .....No. 1001

```
[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

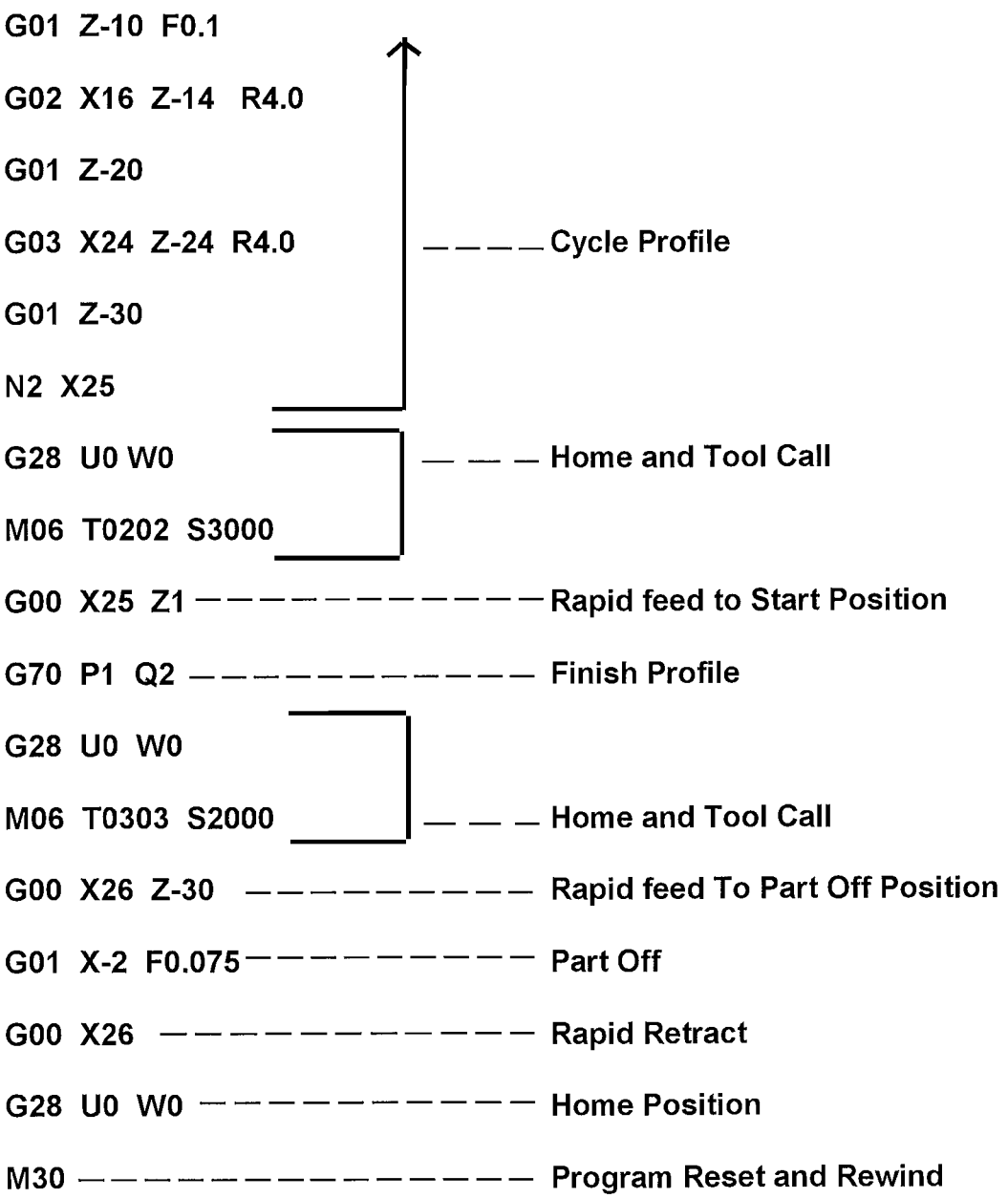
G01 Z0 F0.2 —————————— Face and Retract

X-1 F0.1

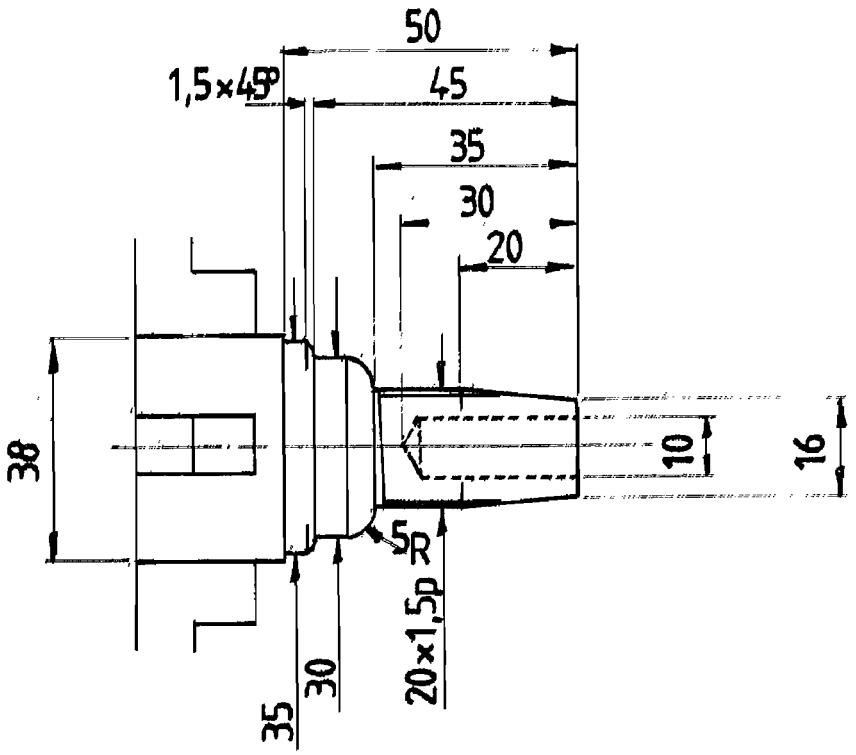
G00 X25 Z1

G71 U1.5 R0.5 ———— Set Parameters for Canned Cycle
G71 P1 Q2 U1 W0.1 F0.125

N1 G00 X8

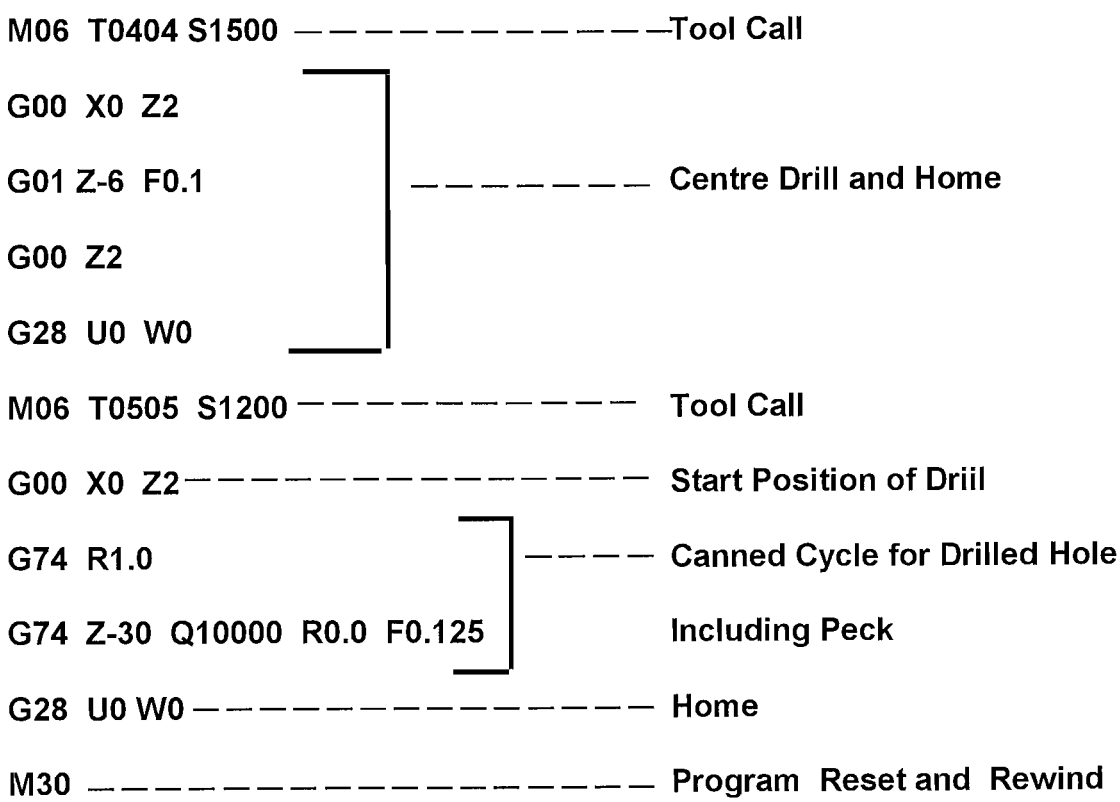

G01 Z-10 F0.1
G02 X16 Z-14 R4.0
G01 Z-20
G03 X24 Z-24 R4.0
G01 Z-30
N2 X25
Cycle Profile
G28 U0 W0
M06 T0202 S3000
Home and Tool Call
G00 X25 Z1
Rapid feed to Start Position
G70 P1 Q2
Finish Profile
G28 U0 W0
M06 T0303 S2000
Home and Tool Call
G00 X26 Z-30
Rapid feed To Part Off Position
G01 X-2 F0.075
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 X-18.16 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 "TU150.MSG" extension.

2 - The text version is "TU150.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 TU150.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 "HARRISON" then type :-

CD\HARRISON

Then Type:-MESSAGES TU150

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 TU150

A new TU150.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

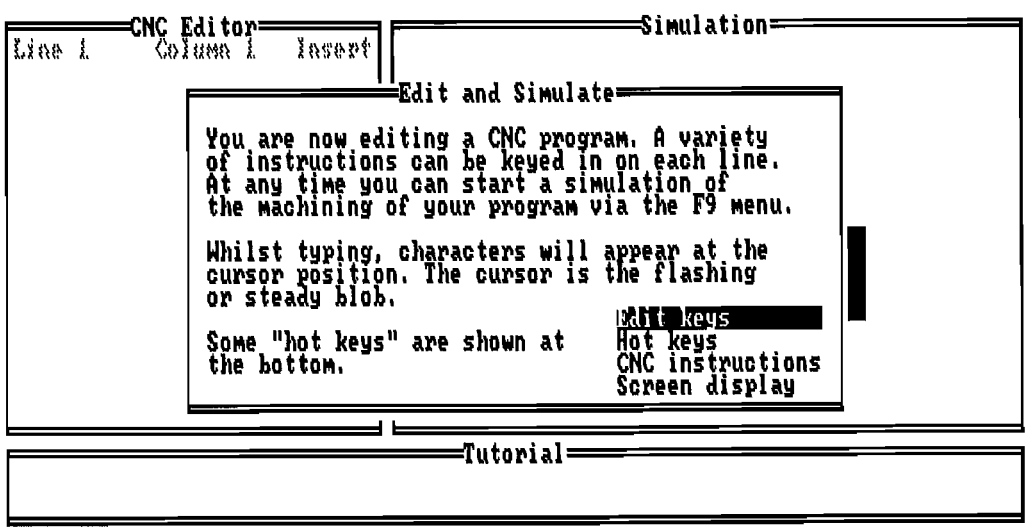
It is vital that the pages relating to error messages, fast key strips and text which are used directly by the software are not altered. The software relies on each line being in sequence. Help screens that are used with the F1 key can be placed anywhere, but we suggest you put them at the end of the 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 TU150.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.

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 100

HIGHSPINDLE 5000

MOVERANGEX 82.5

MOVERANGEZ 178

MACHINEEXTENTX 130

MACHINEEXTENTZ 202

BILLETX 70

BILLETZ 200

CHUCKX 170

CHUCKZ 20

JAWSX 15

JAWSZ 10

CHUCKCENTREX 75

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

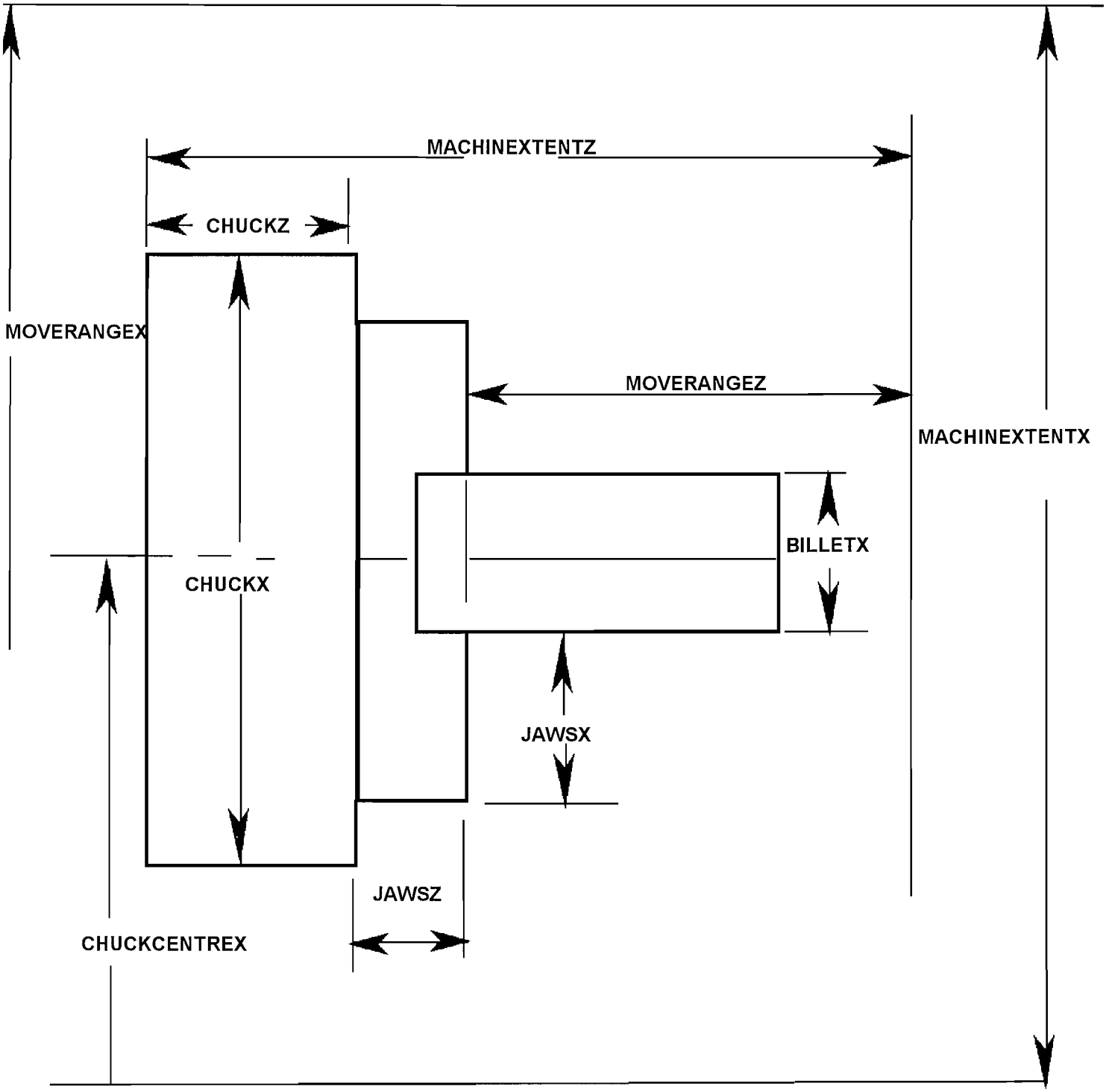
PRINT_BM 10

PRINT_CRNULLS 0

COMPUTER_SDEVICE COM1

METRIC 1

DISPLAYSIZE 1



OPTIONS FOR LATHE SIMULATION

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:

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

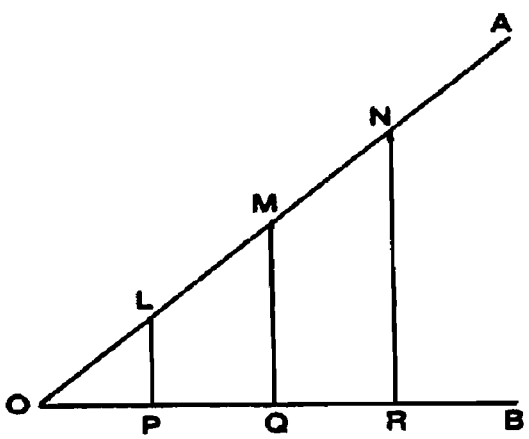


Fig. 2

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

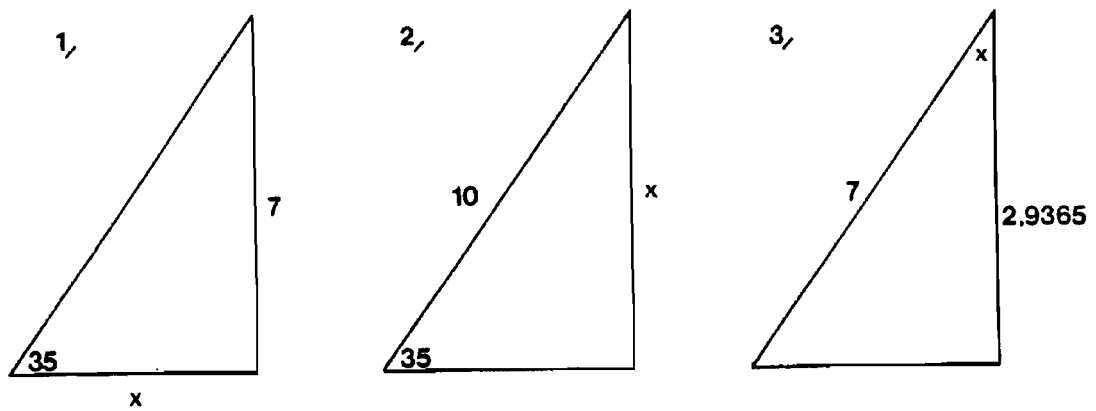
$$COS = \frac{\text{Adjacent Side}}{\text{Hypotenuse}}$$

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

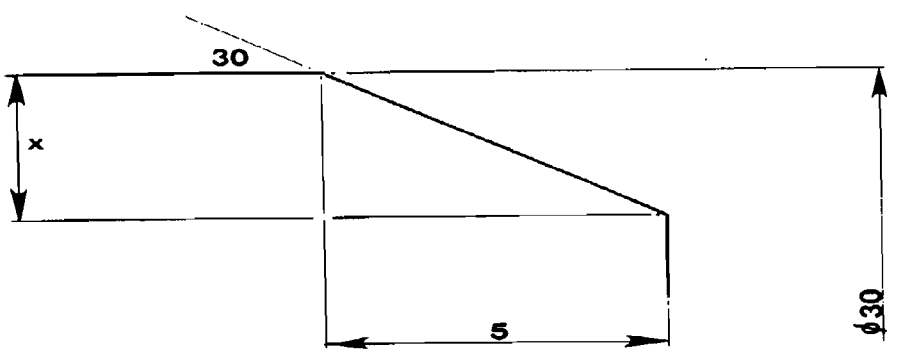
Find angle knowing adjacent side and hypotenuse.

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

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

0.4195 from cosine tables = 65 deg 12' = X

4/



4) Solution

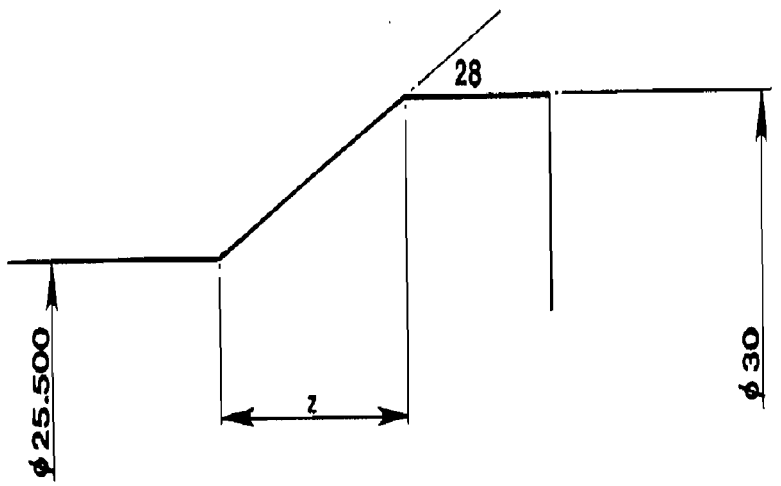
Find X distance on component chamfer

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

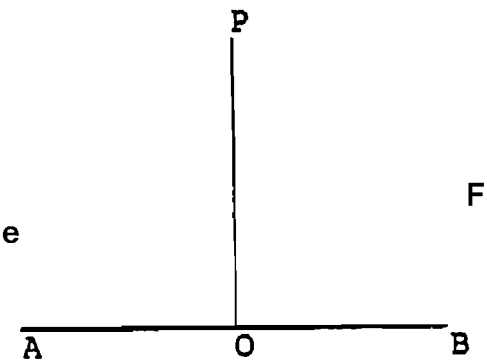


Fig. 1

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.

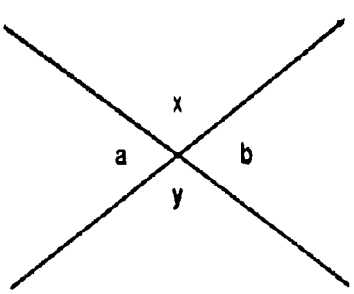


Fig. 2

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.

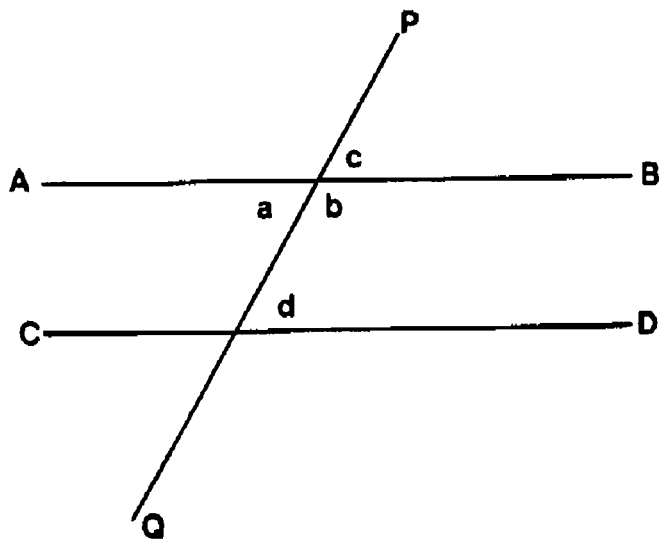


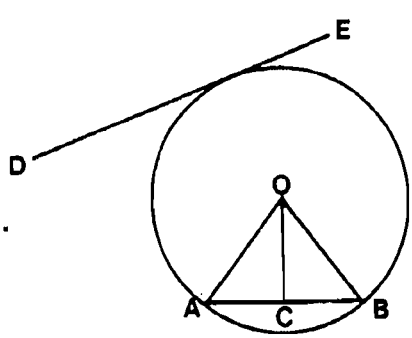
Fig.3

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.

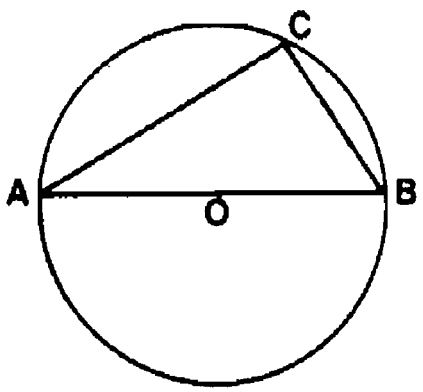
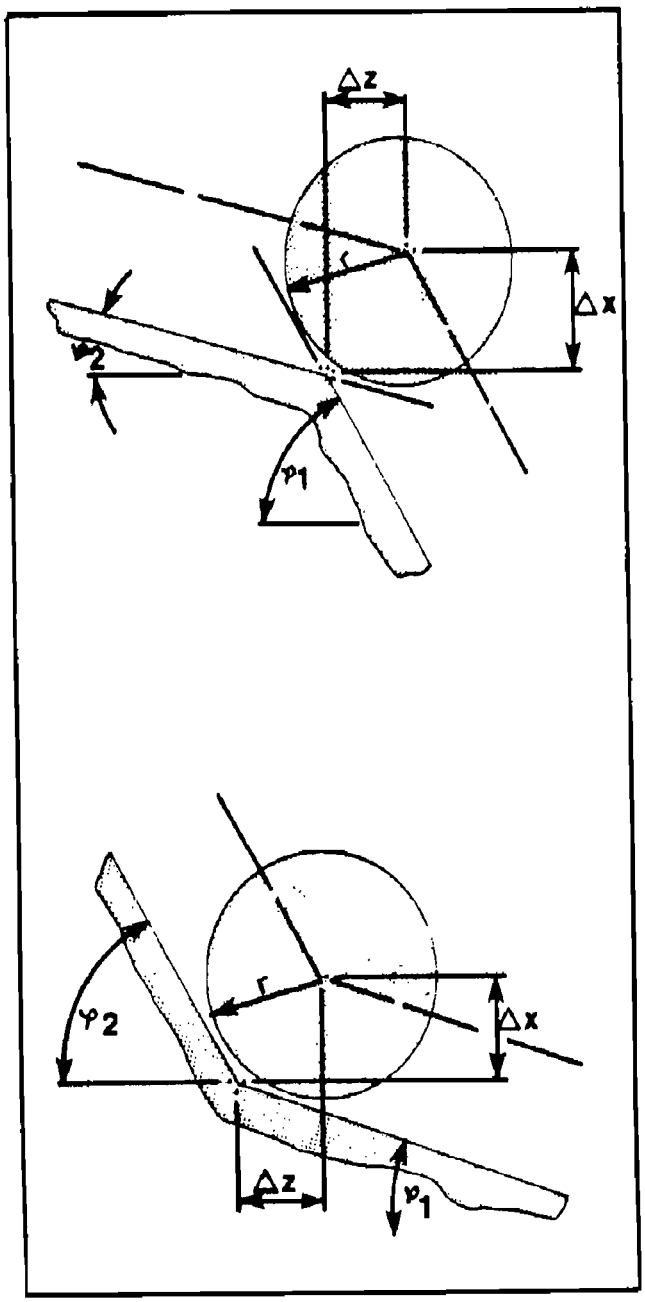


Fig. 5

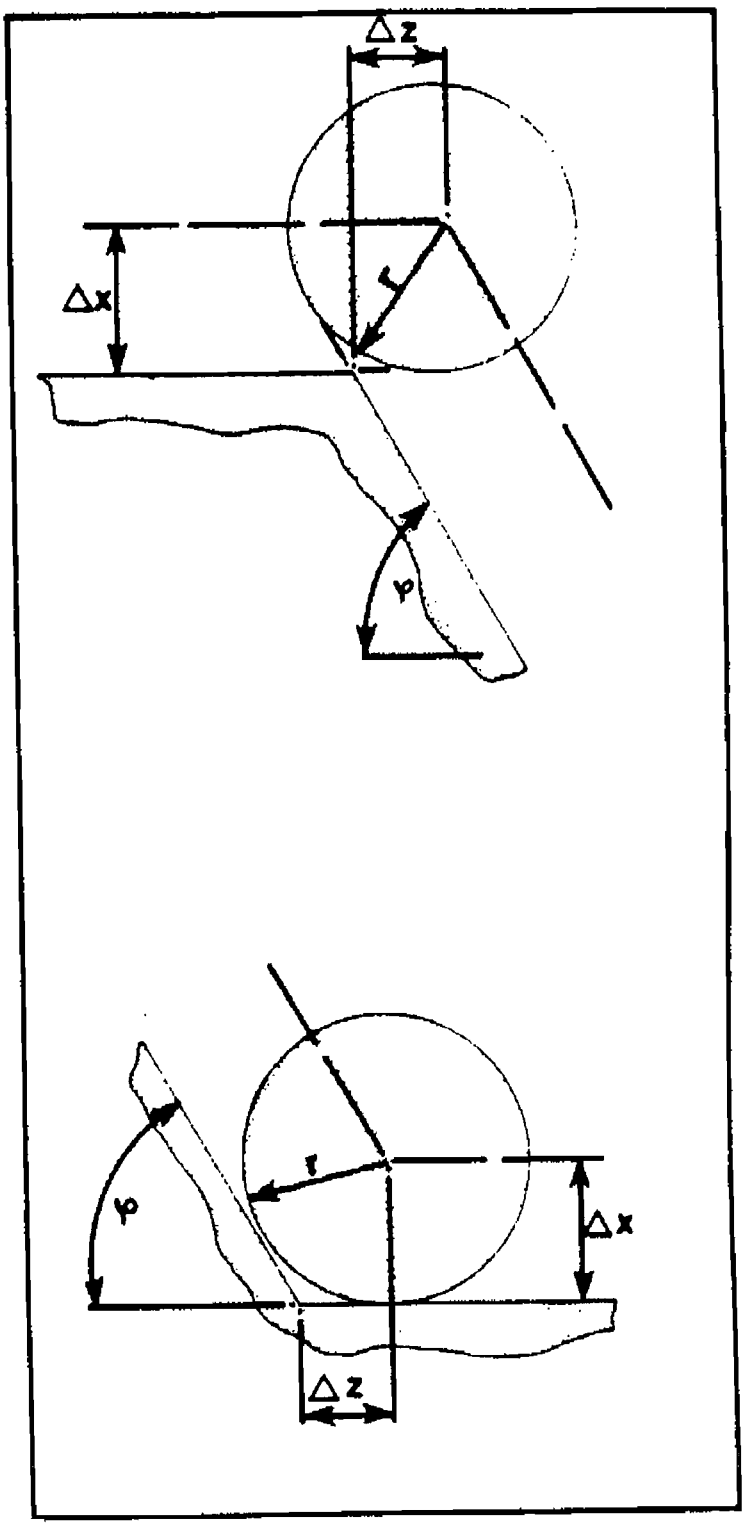
The angle in a semi circle is a right angle. In fig 5 line A B passes through the centre of circle 2 at O. Any point chosen within the semi circle at C will produce a right angle ACB.

.USEFUL FORMULAE FOR DETERMINING CONTOUR CHANGE POINTS



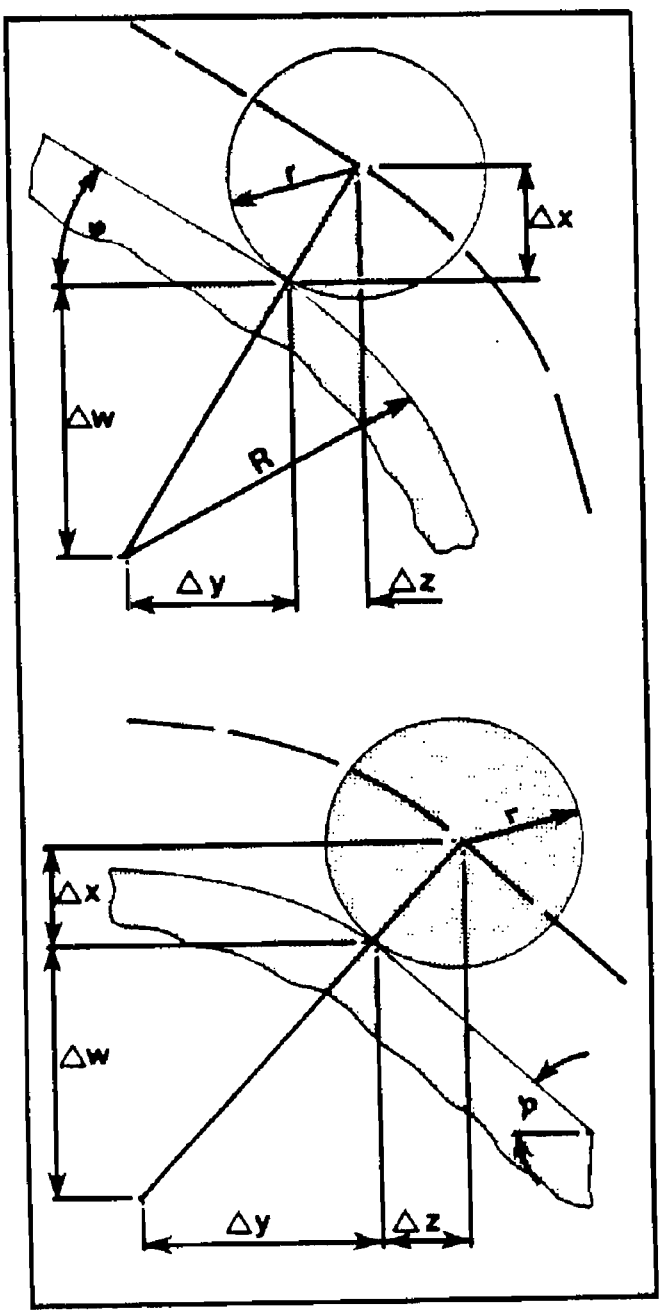
$$\frac{\Delta X = r.\cos\frac{\alpha 2+\alpha 1}{2}}{\cos\frac{\alpha 2-\alpha 1}{2}}$$

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



$\Delta X = r$

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

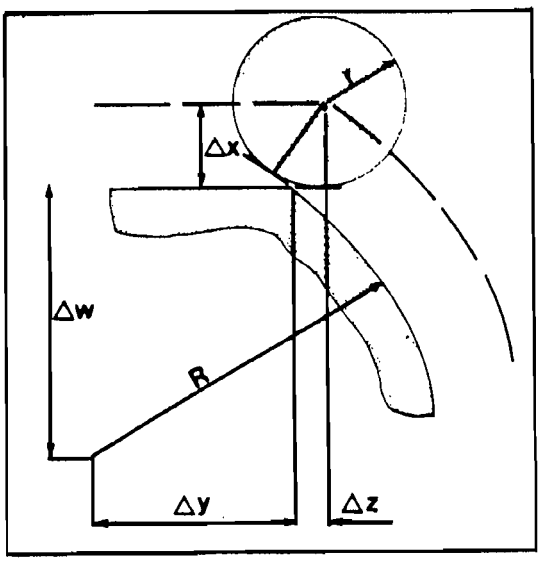


$\Delta W = R \cos \alpha$

$\Delta X = r \cos \alpha$

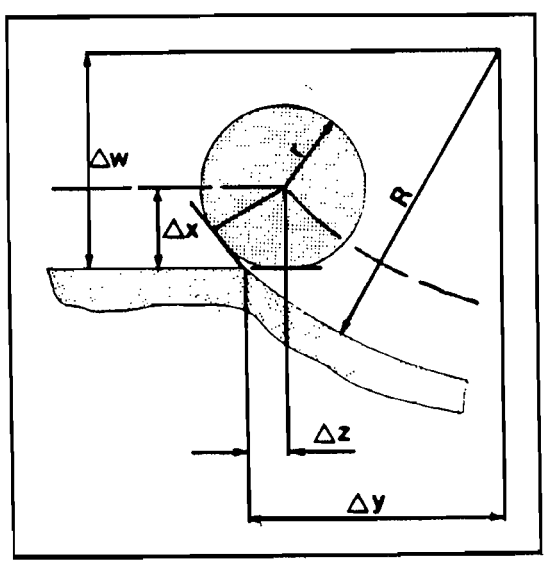
$\Delta y = R \sin \alpha$

$\Delta Z = r \sin \alpha$



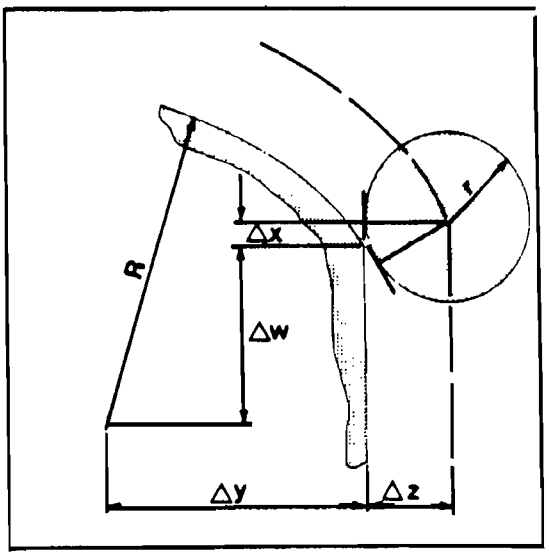
$\Delta X = r$

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



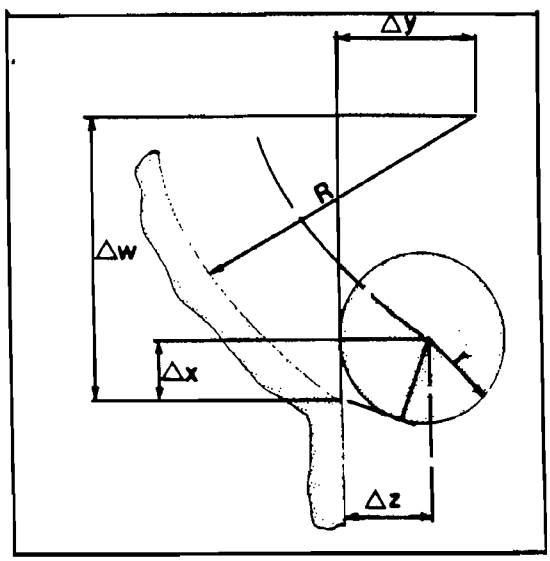
$\Delta X = r$

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



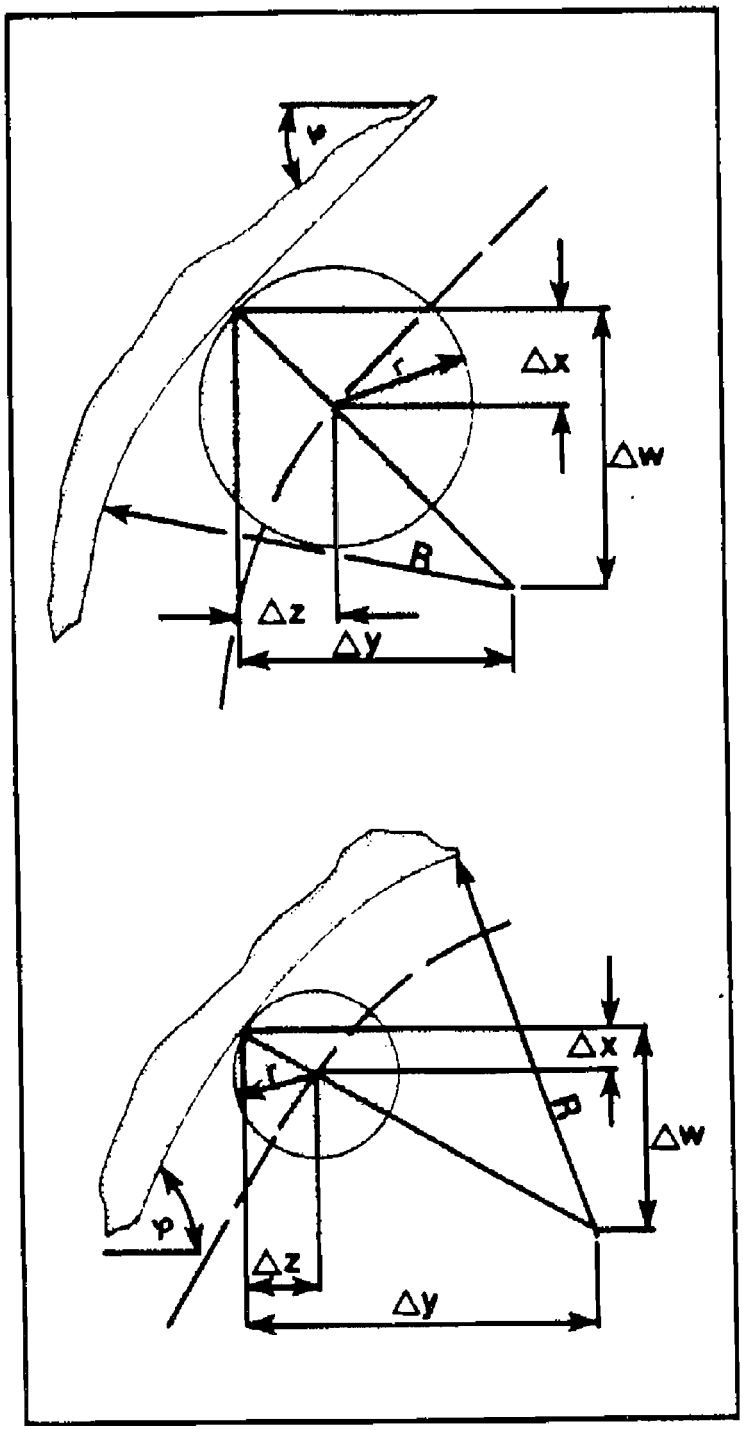
$$\Delta y = r$$

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



$$\Delta Z = r$$

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



DW = R COS a

DX = R COS a

Dy = R SIN a

Dz = r SIN a

PLANNING PROCEDURE FOR MAINTENANCE WORK

When carrying out maintenance work, pay special attention to the following items to ensure safe and correct procedures.

1. Before starting any maintenance work, define the task and obtain the information relevant to carry out the maintenance to a successful conclusion.

2. Prior to commencing any maintenance task, define the work period to do the neccessary work, obtain the correct tools, order the spare parts neeeded to complete the task.

3. During the maintenance work period put up a notice in a place easily seen , to the effect that the machine is under maintenance and should not be used until the notice is removed.

4. Safety should be a priority when carrying out any maintenance, covers and safety guards that are removed during the maintenance period should be replaced after the work is completed and all interlocks and micro-switches reset.

5. All maintenance work should be carried out by suitably qualified personnel.

6. When replacing electrical components ensure that they are the ones specifed.

7. All maintenance work done on the machine whether progressive or preventative should be logged so that a complete service record can be kept for future referal.

8. When the maintenance is completed, check that the replaced and serviced parts are working correctly, and that the machine runs efficiently.

MAINTENANCE SCHEDULE

Routine inspection and maintenance of the machine should be carried out to the the following schedule.

DAILY

Check oil level in the reservoir (see diagram and lubrication chart)

Clean out swarf.

WEEKLY

Clean machine thoroughly.

Check coolant level and top up or change as neccessary.

BI-ANNUALLY

Check adjustment of saddle strips.

Check adjustment of crosslide strip.

ANNUALLY

Grease axis bearings (see lubrication chart).

Check machine alignments and accuracy.

Check headstock bearing adjustment. (NOTE., Lock-nut is LH).

Check spindle drive belt.

Check axis drive belt for wear.

Change air filters.

IF IN DOUBT ABOUT ANY OF THE ABOVE PROCEDURES, CONTACT T. S. HARRISON & SONS LTD. FOR ASSISTANCE.

COOLANT FILLING AND DRAINING

Capacity : 25 Litres

Recommended Type: Cincinnati Millacron Simcool C 60

FILLING

Mix the coolant as prescribed by the manufacturer.

Pour in through the work area at the front of the machine.

Watch the coolant sight glass at the RH side of the machine (see disagram - tailstock end).

Stop filling when the coolant level is reached.

DRAINING

The drain plug is found on the RH Side of the machine beneath the coolant sight glass. See RH end of machine diagram.

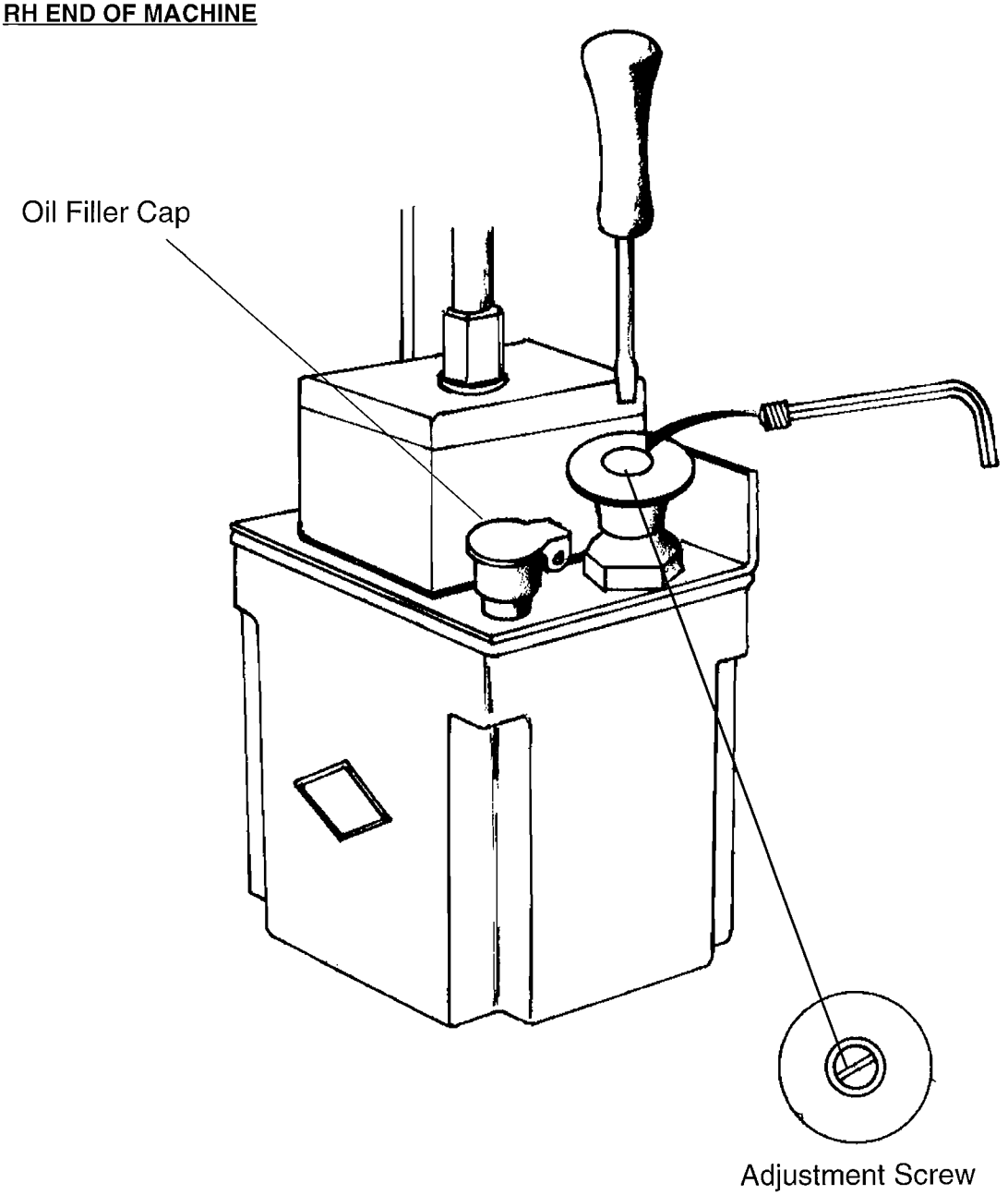
To drain,remove the drain plug after first placing a receptacle beneath the drain point and allow to drain off. Remember to reseal the plug after refilling.

NOTE :- The recomended dilution for Simcool C 60 is 2.5 - 5% , any increase in the dilution percentage could have a detrimental effect on the paintwork and seals.

LUBRICATION PUMP

The lubrication reservoir and pump are found at the tailstock end of the machine

RH END OF MACHINE



ADJUSTMENT OF OIL FLOW

The oil flow from the lubrication pump can be altered by first removing the allen screw (see diagram on the previous page) then turning the cross head screw clockwise will lessen the flow of oil from the pump.

NB. The pump is only active when the spindle the spindle is running.

LUBRICATION CHART

LUBRICATION POINT	LUBRICATING SYSTEM	FREQUENCY	RECOMMENDED OIL / GREASE	QTY
SLIDE WAYS AND BALLSCREWS	AUTO PUMP UNIT	AS REQUIRED	BP : CS 68 SHELL : VITREA 68 CASTROL : PERFECTO NN	0,5 litre
HEADSTOCK	GREASE SEAL	ON MAINTENANCE OF HEADSTOCK	KLUBER ISOFLEX NBU 15	4 cc/BEARING
AXIS BEARINGS	GREASE SEAL	ONCE A YEAR	BP : LS 3 SHELL : ALVANIA No.3	2cc/BEARING
COOLANT	ELECTRIC PUMP		CINCINNATI MILLACRON SIMCOOL C60	25 litre

MECHANICAL TROUBLE SHOOTING

PROBLEM : Poor surface finish.

Corrective Action:

1. Ensure that the tool tip is in good condition and that the grade is suitable for the material being cut.

2..Check feeds and speeds, are they compatible with the material being cut and the grade of carbide being used for cutting?.

3. Ensure that the tool tip, tool body, tool turret or toolpost are rigid and secure. Do not overhang the tool too far from the turret or toolpost body.

4. Ensure that the axis gib strips are correctly adjusted.

5. Ensure the spindle drive belt is not worn and is correctly adjusted for tension.

PROBLEM : Coolant not flowing.

Corrective Action :

1. Has the coolant ON been programmed - M08 - COOLANT ON, or M13 COOLANT ON and SPINDLE FORWARD.

2. Check the level of the coolant in the tank and top up if necessary.

3. Check that the pump is working - observe flow in pipe connected to the pump.

4. Check all pipes for leakages and replace if necessary. If accumulated swarf is causing an obstruction remove the pipe and clear the obstruction and replace the pipe ensuring a good seal without leaks.

ISOLATE MACHINE BEFORE MAINTENANCE COMMENCES

MECHANICAL CORRECTIVE PROCEDURES

1. PROCEDURE FOR ADJUSTMENT OF SPINDLE DRIVE BELT TENSION.

Tools required: - Tension rod - 8mm dia, 8mm A\F Allen Key, 4mm A\F Allenkey.

a) Remove eight M6 button head screws from headstock end cover and remove the cover.

b) Release the three M10 Allen cap head screws holding the motor plate to the lathe bed.

c) Place the tension rod through the top RH slot and lever plate towards the back of the machine to tension the spindle drive belt.

d) Whilst still under tension tighten the bottom RH screw then release and remove the tension rod, tighten the remaining M10 screws.

e) Run the spindle and observe the belt to ensure there is no excessive slack, fluctuation or bulging away from the drive pulley.

f) Replace the headstock end cover and secure with the eight M6 button head screws.

2. PROCEDURE FOR ADJUSTING THE THE GIB STRIPS.

Z AXIS

Tools required:- Long reach screw driver,

a) Remove eight M6 button head screws from the headstock end cover and remove the cover.

b) Remove M6 knurled headed screw from lubrication pump door at the tailstock end of the machine and open the door to full extent.

c) Release the gib strip lock screw at the headstock side of the saddle. This is accessible through the headstock housing and under the bed guards.

d) Adjust the gib strip by turning the lock screw which is accessible through the lubrication pump door.

e) Turn the screw clockwise to tighten the strip. Do not overtighten.

f) Lock the strip with the lock screw at the headstock side of the saddle.

g) Move the saddle up and down the bed to ensure smooth and continuous movement.

h) When correctly adjusted replace headstock end cover and close lubrication pump door and secure with appropriate screws.

X AXIS

a) Remove the crosslide cover by releasing the two M8 grub screws at the rear of the indexing turret base.

b) Release with a screwdriver the gib strip lock screw at the rear of the crosslide.

c) Using the same screwdriver turn the screw at the front of the crosslide clockwise to tighten the gib strip. Do not overtighten.

d) Lock the strip in position by tightening the lock screw at the rear of the crosslide.

e) Move the crosslide up and down the saddle to ensure a smooth and continuous movement.

f) When correctly adjusted replace the crosslide cover.

3. PROCEDURE FOR CHECKING LIMIT SWITCH

Tools required: 4mm A\F Allen key.

a) Z Axis limit switches are accessible through the headstock end cover and lubrication pump door. Remove and open these covers - as previously described.

b) X Axis limit switches are accessible by removing the cross slide cover - as previously described.

c) Ensure that the limit switches are clean and dry, and the switch pillars are not sticking.

d) If the switch is found to have failed contact HARRISON SERVICE DEPT.

AIR FILTERS

The air filters on the side of the machine control box, should be changed either annually of when "black" in colour.

When ordering quote reference :

Air Inlet Filter, 120mm Ref: RS 507-876

LED STATUS AND FAULT DISPLAY

Display	Meaning
-	Servo Power Off
8	Servo Power Up and Idle
c	Cam Profiling
c	Cam Table (Superscript)
C	Circular Interpolation
E	Encoder Following Mode
F	Flying Shear (No Flashing Dot)
H	Homing (Datuming)
J	Jogging (Velocity) Mode
o	Offset Mode
P	Linear Positional Move
q	Torque Control Mode
S	Stop Asserted
U	Pulse Following Mode

LED STATUS DISPLAY - ERRORS

Errors are shown for all axes with a flashing dot

Display	Meaning
E	External Error
E	Software Abort or Interpreter Error
F	Maximum Following Error Exceded
L	Limit Switch Open

See rack assembly drawing for position of LED Display

ASSEMBLY DRAWINGS

&

PARTS LISTS

TU150E HEADSTOCK AC7/100

Part No.	Description	Quantity
AC2/101	HEADSTOCK	1
102A	MAIN SPINDLE	1
103	GREASE RING	1
104	BEARING FRONT PLATE	1
105A	BEARING BACKPLATE	1
106A	SPACING SLEEVE	1
107	REAR BEARING SPACER	1
108	REAR COVER PLATE	1
109A	SPINDLE PULLEY	1
110	PRE-LOAD OUTER SHIM	1
113	SPINDLE LOCKNUT LH	1
504	SPINDLE ENCODER PULLEY	1
BI 00102H	FRONT BEARING	2
BI 00108B	REAR BEARING	2
BI 00186K	TAB WASHER	1

TU150E BED & SADDLE DRIVE AC9/200

Part No.	Description	Quantity
AC2/201	BED	1
202	BEARING HOUSING	1
203	BEARING SUPPORT BLOCK	1
204	Z AXIS BALLSCREW	1
206	SPACER	1
211	BALLNUT SHIM	1
806	DOWEL PIN	2
ZT2/310	GIB STRIP LOCKING SCREW	2
AC8/205	SADDLE DRIVE BRACKET	1
207	MOTOR PULLEY 10T	1
AC7/208A	AXIS PULLEY 20T	1
BI 00102A	BEARING	2
BI 00110	BEARING	2
BI 00421 D	STEPPER MOTOR MHY	1
BI 00186A	LOCKNUT	1
BI 00186B	TAB WASHER	1
BI 00646	TIMING BELT	1
BI 00381G	MICRO-SWITCH	1
BI 00381H	MICRO-SWITCH	1
BI 00178A	BALLNUT	2
1400/10MM	CIRCLIP	1

TU150E SADDLE & CROSS SLIDE AC9/300

Part No.	Description	Quantity
AC2/301	SADDLE	1
302A	CROSS SLIDE	1
303A	TOOL TURRET BASE	1
304	Z BALLSCREW BRACKET	1
306	BALLSCREW	1
305	BEARING HOUSING	1
307	SADDLE GIB STRIP	1
308	SADDLE GIB STRIP	1
309	SADDLE TAPER STRIP	1
310A	GIB STRIP	1
311	CROSSLIDE DRIVE BRACKET	1
312	CAM TRIP BAR	1
313	KNOCK OFF CAM	3
314A	DRIVE COVER	1
316	DATUM TRIP BAR	1
319	MICRO-SWITCH BRACKET	2
320	PLUG	1
321	CROSS SLIDE FRONT COVER	1
323	GIB STRIP SCREW	1
325	FLANGED BEARING HOUSING	1

Part No.	Description	Quantity
AC4/315	CROSS SLIDE REAR COVER	1
AC4/316A	TERMINAL BOX BRACKET	1
AC4/317	TERMINAL BOX	1
AC4/318	TERMINAL BOX SPACER	1
AC7/208A	X AXIS MOTOR PULLEY 20T	1
310	CROSS SLIDE WIPER	1
AC8/207	MOTOR PULLEY 10T	1
AC8/311	CROSSLIDE DRVE BRACKET	1
TR1/110	JIB STRIP LOCK	
BI 00421D	STEPPER MOTOR	1
BI 00102A	BEARING	2
BI 00106	BEARING	2
BI00178A	BALLNUT 1605UF	2
BI 00186A	LOCKNUT	1
BI 00186B	TAB WASHER	1
BI 00646	TIMING BELT	1
BI 00380C	MICRO-SWITCH	2
BI 01443D	CABLE CARRIER	1
1400/10MM	CIRCLIP	1
BI 00364E	KLIPPON TERM AKZ4	20
BI 00364G	END PLATE 36336	1
BI 00364F	EW15 TERM 038268	1
BI 00406C	GLANDS NC16/M16/D	1
BI 00439D	ECG/1 COMP GLAND	1

Part No.	Description	Quantity
BI 00439E	ECG/3 COMPRESSION GLAND	1
RS 507-933	HOUSING SEALED	1

TU150E CONTROL BOX AC9/400B

Part No.	Description	Quantity
AC2/401	CONTROL BOX	1
402A	CONTROL BOX DOOR	1
403B	BACKPLATE	1
408	CONTOL BOX SUPPORT	2
411	SUPPORT BUSH	2
413	REAR DOOR CABLE	2
416	CONTROL BOX BACKPLATE	1
AC4/403	THERMAL SWITCH BRACKET	1
ZT1/647	ISOLATOR SUPPORT	1
BI 00339Z	SD 15 STEPPER DRIVE	2
BI 00362A	RELAY BASE 8 PIN	2
BI 00362C	RELAY 24V DC 11 PIN	2
BI 00362D	RELAY BASE 11PIN	2
BI 00362E	RELAY 24V DC 8 PIN	2
BI 00362H	RELAY MODULE 24V DC	12
BI 00362II	I/FACE UNIT 54206	2
BI 00365T	FAN 109S074UL	1
BI00366A	ISOLATOR DETL 40D1	1
BI 00365U	FINGER GUARD 109-091C	2
BI 00366X	C/BREAKER C60 HD304	1

BI 00366Z	C/BREAKER C60 HD210	4
BI 00374A	TRANSFORMER 300VA	1
BI 00374G	TRANSFORMER 600VA	1
BI 00374J	TRANSFORMER FOR SD15	1
BI 00381M	CONTACTOR SUPP 20101	4
BI 00381N	100A09ND3 A.BRADLEY	3
BI 00381U	O/LOAD 2,8/4.2A A/B	2
BI 00381W	193BPM A/B O/LOAD	1
BI 00381X	CONTACTOR 100A18ND3	1
BI 00391	140-MN1000 6.0/10.0	1
BI 00391A	140-MN0100 0.6/1.0	2
BI 00392A	3 WAY COMBI 140-L13	1
BI 00393	TERM BLOCK 140-L2	1
BI 00415K	TRANSFORMER FOR 55500415J	1
BI 00425D	CA-10 A369	1
BI 00430C	SUPPRESSOR 23006	2
BI 00430D	SUPPRESSOR 23110	1
BI 00432L	COMBIVERT 2.2KW KEB	1
BI 00432M/24	AUXILLIARY INPUTS	1
BI 00432N	UNIVERSAL STEPPER	1
BI 004492	56-506-9053 EARTH	2
BI 00450	57.503.0053 WK2.5/32	31
BI 00450A	57.910.5053 FUSE FIT	1
BI 00450B	Z5.522.7453 END STOP	3

BI 00450C	07.311.0153 END SECT	8
BI 00450D	56.504.9053 EARTH	16
BI 00450F	07.311 0353 END TERMINAL	1
BI 00450G	57.510 0053 MAIN TERMINAL	3
BI 00450J	07.311.4153	1
BI 00463	PRO CARD EC386-1S20	1
BI 00463A	PT-1 STE-BUS	1
BI 00463B	EC386-BP15	1
BI 00463C	PSU-SMP8171	1
BI 01229	HINGE E6-10-501-20	4
BI 01229B	LOCK AND KEY	1
RS 239-056	ABSORBER 6.35mm	3
RS 466-208	SUB MIN SOCKET	1
RS 467-194	CHASSIS SHELL 8WAY	2
RS 469-566	'D' CONNECTOR HOOD	1
RS 470-910	CONNECTOR 25 WAY PLUG	1
RS 472-837	CONNECTOR 9 WAY PLUG	1
RS 477-876	PLUG DIN 5 WAY 180DEG	1
RS 478-273	SOCKET 5 WAY 180DEG	1
RS 507-905	FILTER UNIT FAN (1)	1

TU150E OPERATORS PANEL AC9 / 400A

Part N0.	Description	Qty
BI 00356B	12" MONITOR VGA	1
BI 00356C	12" CTR BEZEL	1
BI 01811	MULTI-PURPOSE TUTOR	1
BI 00365D	EMERGENCY STOP KEY OPER	1
BI 00364S	HINGED GUARD 593032	1
BI 00364W	SWITCH BODY 503008	2
BI 00364X	CONT BLCK 593560. 100	2
BI 00364Y	CAP WHITE 463022	1
RS 481-465	SKT HOUSING 15 WAY	1
RS 481-493	SOLDER PLUG	10
BI 00357M	HART 09160243001	1
BI 00357P	HART 0930060301	1

TU150E SPINDLE DRIVE AC8/500

Part No.	Description	Quantity
AC2/503	SPINDLE ENCODER BRACKET	1
505	ENCODER PULLEY	1
AC4/501A	SPINDLE MOTOR PLATE	1
502	MOTOR PULLEY	1
506	PULLEY RETAINING PLATE	1
AC5/501	ENCODER BRG. HOUSING	1
502	BEARING SHAFT	1
503	ENCODER ADAPTOR PL	1
BI 00102	BEARING 7203B FAG	2
BI 00442S	ENCODER HARR CYCLONE	1
BI 00319	SPINDLE MOTOR 2.5 KW	1
BI 00628N	SPINDLE DRIVE BELT 460J16	1
BI 00645A	TIMING BELT 225LO50	1
BI 00185	LOCKNUT FU02SS	1
BI 00186	MB2 LOCK WASHER	1

TU150E CABINET ASSEMBLY AC8/600

Part No.	Description	Quantity
AC2/601C	CABINET BASE	1
605B	REAR PANEL	1
606B	TOP HOUSING	1
608A	CONTROL PANEL HOUSING	1
609	SLIDING DOOR	1
610	END DRIVE COVER	1
612	LIFTING HOOK BRACKET	4
613A	ACCESS PANEL	1
614A	BED COVER EXTENSION	1
615	BED COVER	1
616	SADDLE SIDE PLATE (RH)	1
617	SADDLE SIDE PLATE (LH)	1
618	BED COVER HOUSING	1
619	INSPECTION WINDOW	1
620	DOOR WINDOW	1
621	FOOT SUPPORT	3
621A	FOOT SUPPORT	1
622	COOLANT PUMP BRACKET	1
625	HEADSTOCK RING	1
629A	MOUNTING PLATE	1

Part No.	Description	Quantity
AC2/ 638	REAR PAD	1
630A	MOUNTING PLATE	1
633A	COOLANT DRAIN	1
636	COOLANT PIPE BLOCK	1
639	GUARD WIPER	1
641	ACCESS PANEL SCREW	1
642	DIVIDING WALL	1
643	TOP GUARD WIPER	1
644A	DOOR WIPER	1
644B	DOOR WIPER	1
645	DOOR GUARD RAIL	2
646	DOOR GUARD ROD	6
650A	DOOR SLIDE ANGLE	1
651	COOLANT LEVEL TUBE	1
652	DOOR SLIDER	2
AC3/631	BED FILL IN PLATE	1
AC7/646	TOP RAIL SPACER	1
648	SLIDE GUARD	1
649	BOTTOM SLIDE ANGLE	1
AC9/610	INSPECTION PANEL	1
TR5/513	PENDANT HOUSING	1
514A	REAR PANEL	1
514B	REAR PANEL	1
527	PENDANT ARM BRACKET	1

SP 183	PLUNGER	1
183A	PLUNGER	1
191	1/2" TAIL PIECE ADAPTOR	1
201A	CYCLONE HANDLE	1
202	HANDLE SCREW CYCLONE	2
203	FIXING BLOCK CYCLONE	2

BI 00381J	**DOOR SWITCH**	**1**
BI 00398	COOLANT PUMP	1
BI 00448	CABLE CLEAT N08	4
BI 01102C	1/2" COOLANT PIPE	1
BI 01102D	COOLANT NOZZLE	1
BI 01116C	COOLANT TAP	1
BI 01102E	COOLANT PIPE ADAPTOR	1
BI 01202	DOOR CATCH	1
BI 01425A	BED COVER WIPERS	2
BI 01431D	RAIL & SLIDERS	1
BI 01435B	WINDOW SEAL	2.3 metres
BI 01435C	CHROME INSERT	2.3 metres
BI 01451Z	34-0321-27	
NIPPLE 3/8"	NIPPLE 3/8" BSP HEX	1
BI 01115B	OIL SIGHT	1
BI 01428B	ADJUSTABLE FEET No. 2	4
BI 01229	HINGE E6-10-501-20	4

TU150E TAILSTOCK (PNEUMATIC) AC7/700

Part No.	Description	Quantity
AC7/701	TAILSTOCK BED	1
702	SHOE	1
703	BODY	1
704	LOCKING BOLT	1
705	CLAMP	1
AC2/704A	BARREL	1
707A	LINK	1
BI 01131	AIR CYLINDER	1
BI 01131A	REGULATOR	1
BI 01131C	REGULATOR BASE	1
BI 01131F	FLOW CONTROL VALVE	1
BI 01129Y	SOLENOID VALVE	1
BI 01128U	1/8 BSP ADAPTOR	1
BI 01128T	SILENCER	1
BI 01128N	4MM-1/8 BSP ADAPTOR	1
BI 01451Y	PRESSURE SWITCH	1
BI 01123S	BULKHEAD ADAPTOR	2
BI 01123Q	LOCKNUT	1
BI 01126G	BRAIDED PIPE	1
BI 01126F	NYLON PIPE 4MM	4.0 metres

TU150E LUBRICATION SYSTEM AC2/1200

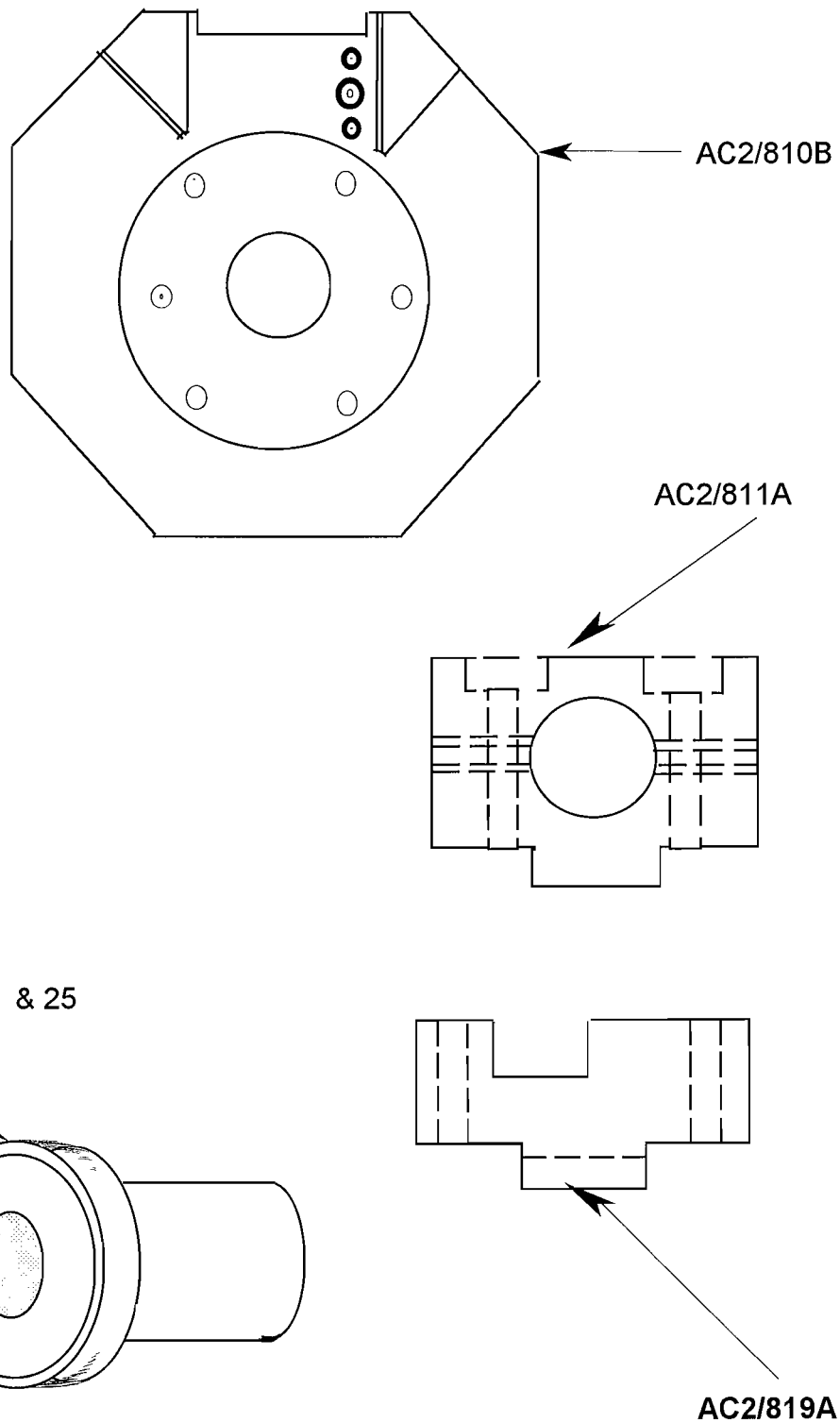
Part No.	Description	Quantity
BI 01120	OIL PUMP	1
BI 01123E	METER UNIT IB21	5
BI 01123F	METER UNIT IM21	1
BI 01123H	SLEEVE NUT	9
BI 01123J	COPPER PIPE	0.5 metres
BI 01123N	BLANKING PLUG	2
BI 01123V	JUNCTION 8 WAY	1
BI 01125	PIPE CLIP SINGLE	6
BI 01126F	NYLON PIPE	3.0 metres
BI 01123I	CONES	12

TU150E PARTS CATCHER AC7/1550

Part No.	Description	Quantity
AC2/1505	PIVOT BUSh	2
1506	SPIGOT	1
1507	BEARING HOUSING	1
1508	COLLAR	1
1509	TRIP CAM	1
1510	DRIVE SHAFT	1
1511	SUPPORT BRACKET	1
1512	DRIVE ARM	1
1513	LINK	1
1514	WASHER	2
1517	ANCHOR PLATE	1
1519	CYLINDER STOP	1
1520	CYLINDER SUPPORT	1
AC2/1551	BUCKET	1
1552	BUCKET ARMS	2
1553	ARM SUPPORT	1
AC2\1557	ROD	1
1558	TRIP CAM SPACER	2
1559	COLLECTION BOX	1

AC2/1560	COLLECTION BOX LID	1
1561	COLLECTION BOX PLATE	1
BI 01128J	AIR CYLINDER	1
BI 01415C	SPRING	2
BI 00195C	OILITE BUSH	3
BI 01229A	HINGE	2

TU150E TOOLING (8STATION) AC8/800



TU150E TOOLING (8 STATION) AC8/800

Part No.	Description	Quantity
AC2/802	TOOL CLAMP	8
810B	TOOL TURRET	1
811A	TOOL SLEEVE HOLDER	4
812	BORING BAR HOLDER 6 DIA	1
813	BORING BAR HOLDER 8 DIA	1
814	BORING BAR HOLDER 10 DIA	1
815	BORING BAR HOLDER 12 DIA	1
816	BORING BAR HOLDER 16 DIA	1
817	BORING BAR HOLDER 20 DIA	1
818	CENTRE DRILL HOLDER	1
819A	AXIAL TOOL HOLDER	1
825	DRILL CHUCK HOLDER	1
BI 00968B	INDEXING TOOLPOST	1