Harrison

TU150E

CNC LATHE

INSTALLATION OPERATION

&

MAINTENANCE MANUAL
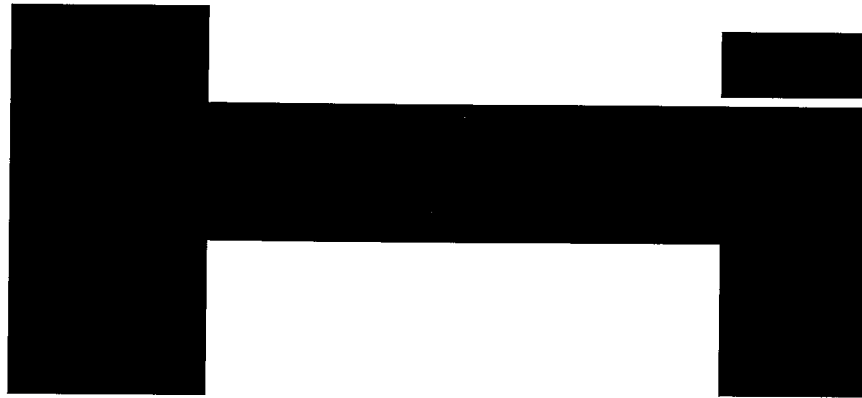
THE *600* GROUP

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   ....

$C\mathcal{E}$

Manufactured by

DENFORD MACHINE TOOLS LTD
FOR

600 LATHES LIMITED

Union Street, Heckmondwike, West Yorkshire, 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:

Harrison

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

Instruction Manual   1.1   TU150E

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
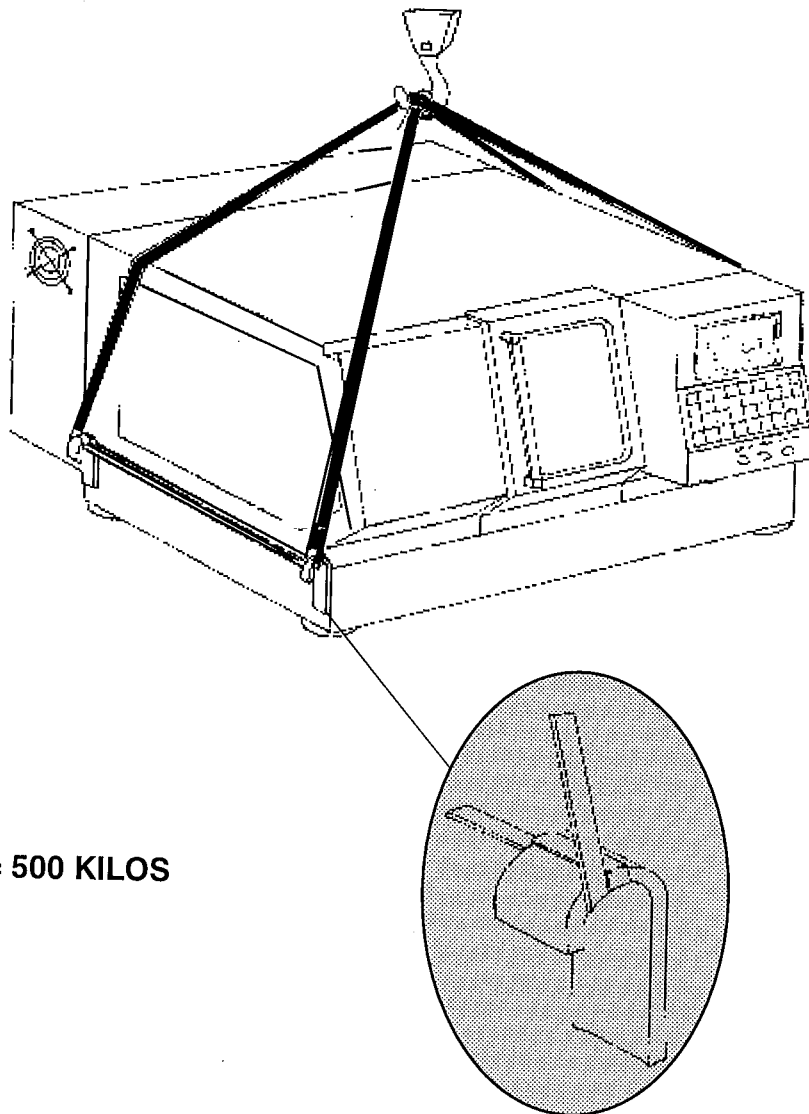
NET WEIGHT = 500 KILOS

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

10mm A/F
LEVELING SCREW
24mm A/F NUT
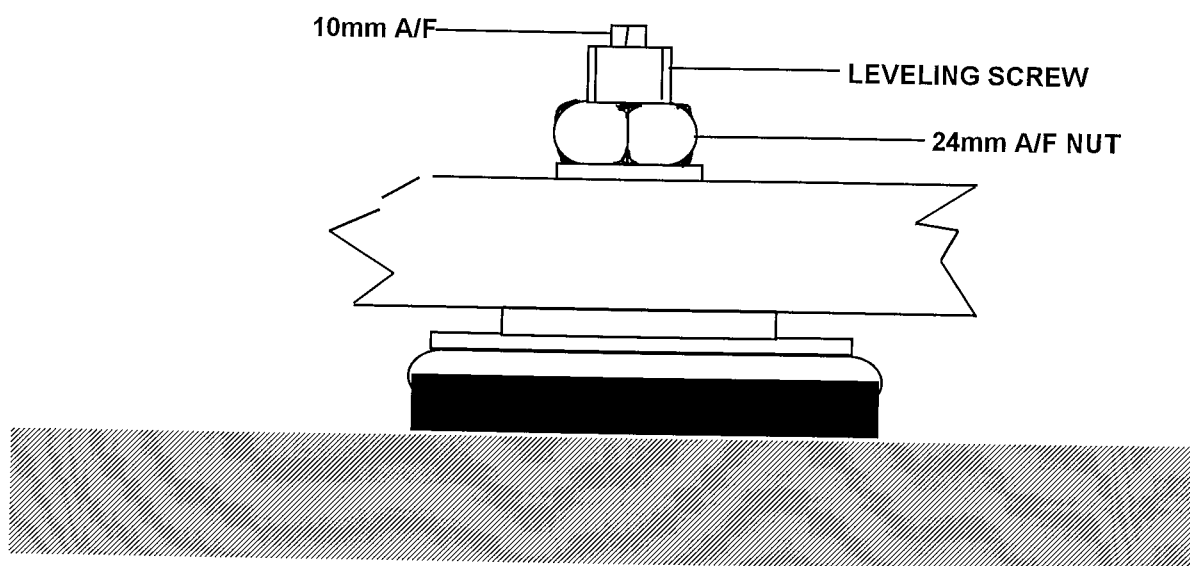
necessary to level the cabinet during installation.

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

3 phase

earth

```
T1   T2   T3

        ON

OFF  ◁ O
```

L1   L2   L3
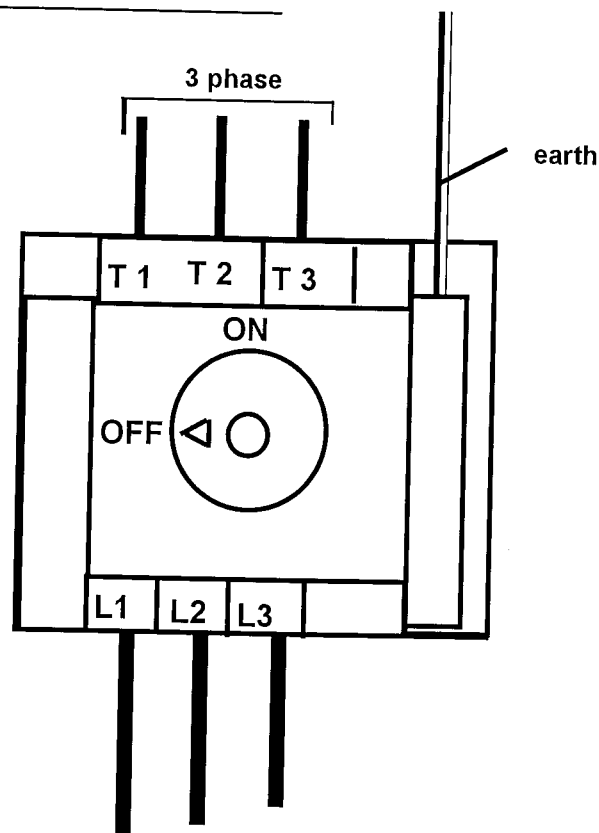
Cable Required:-3core(3phase) & Earth- 2.5mm per phase.

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

FAN

8 WAY Q M SOCKET
FOR FOOT SWITCH

AIR INTAKE
100psi
(6.9 BAR)

8 WAY QM SOCKET
AUXILLARIES

MAINS INPUT

TAILSTOCK END DIAGRAM

TO CONTROL PANEL

MACHINE RUN

DISC DRIVE

RS232 SERIAL LINK

PRINTER PORT

EXTERNAL KEYBOARD

AUXILIARIES

OIL FILLER CAP

ACCESS TO LEVELLING FEET

COOLANT LEVEL
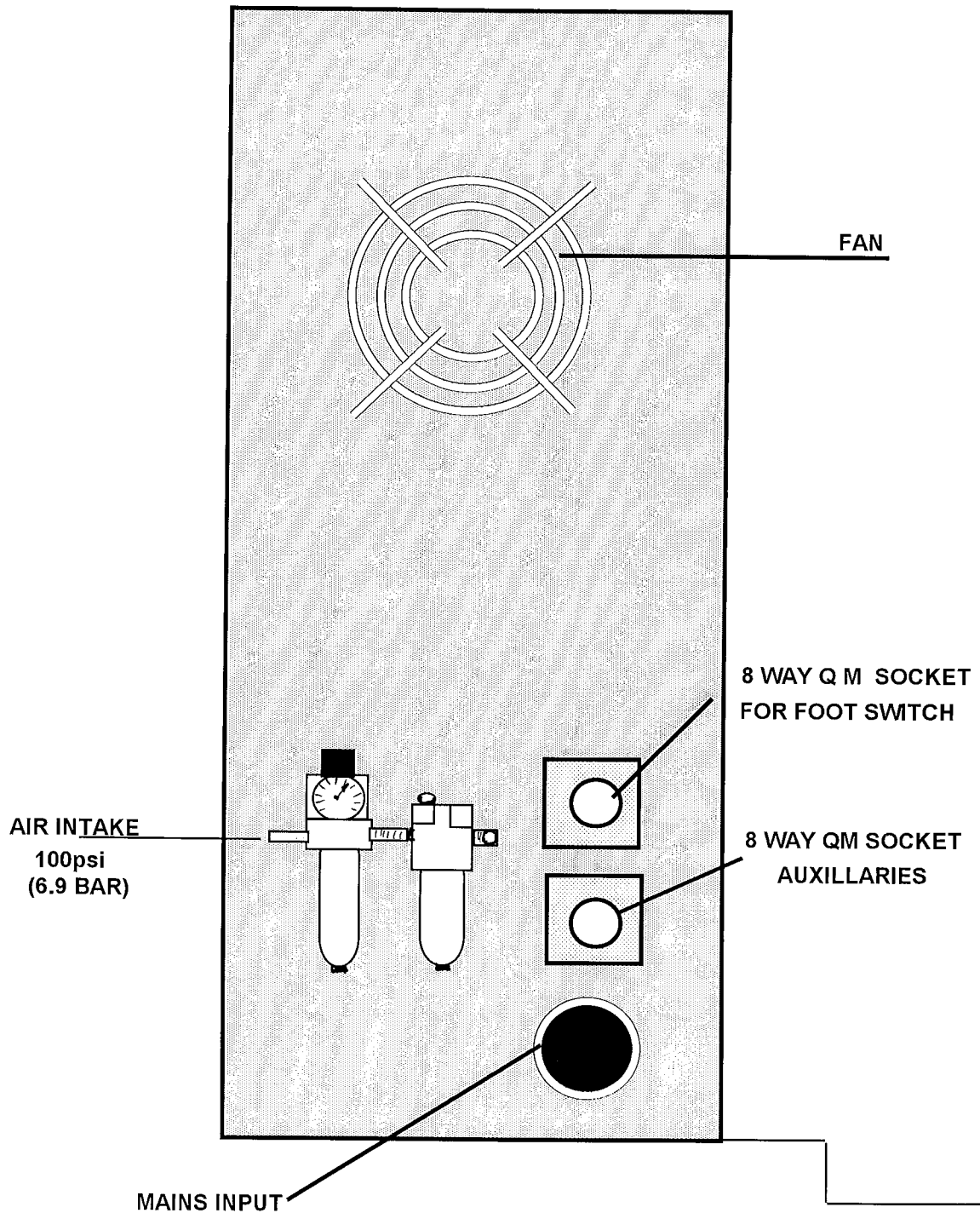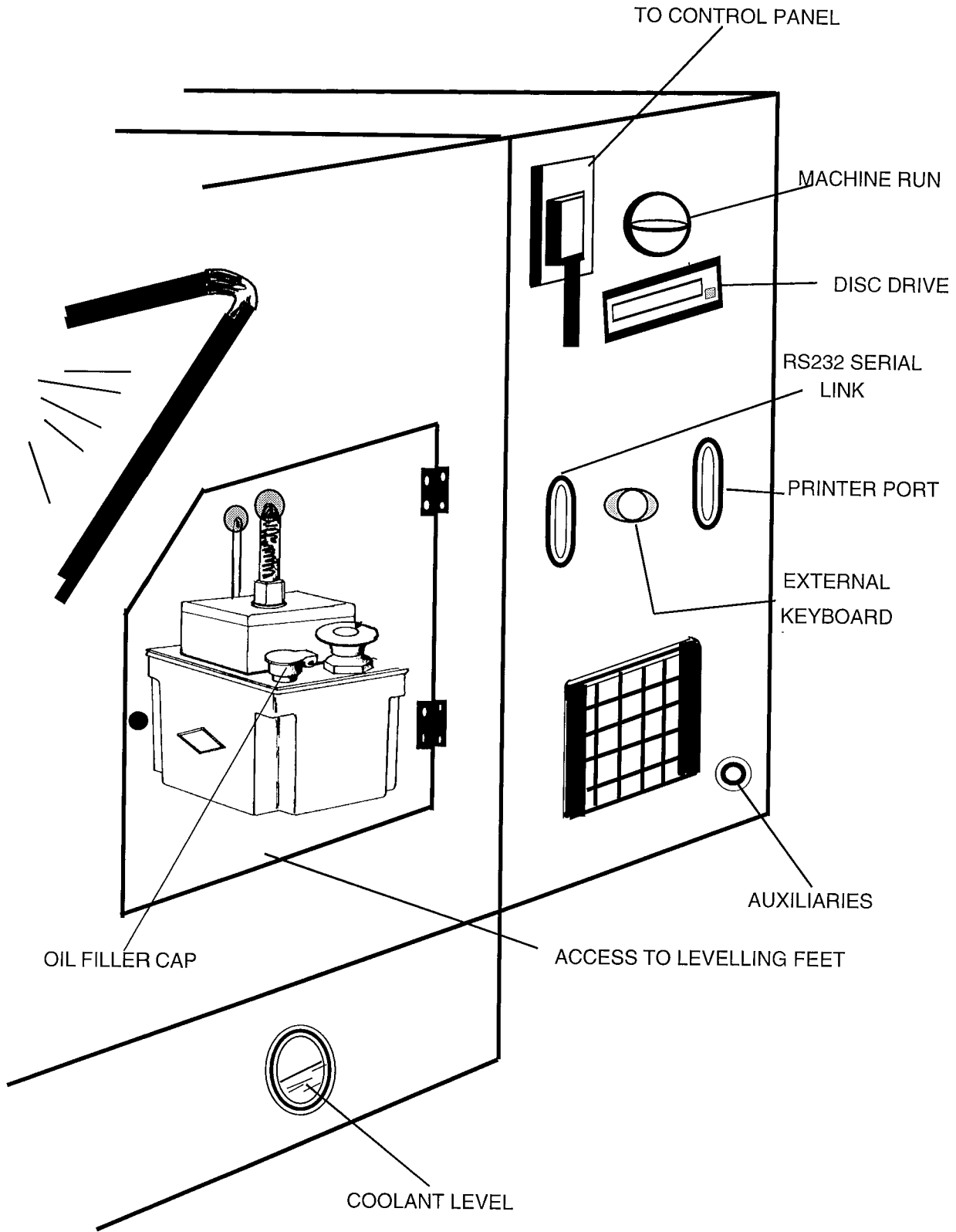
CONNECTION DIAGRAM FOR HYDRAULIC CHUCK

CHUCK CYLINDER

FRONT OF MACHINE

GREEN

RED

GREEN

POWER CABLE

RED

SOLENOID

DRAIN

TANK
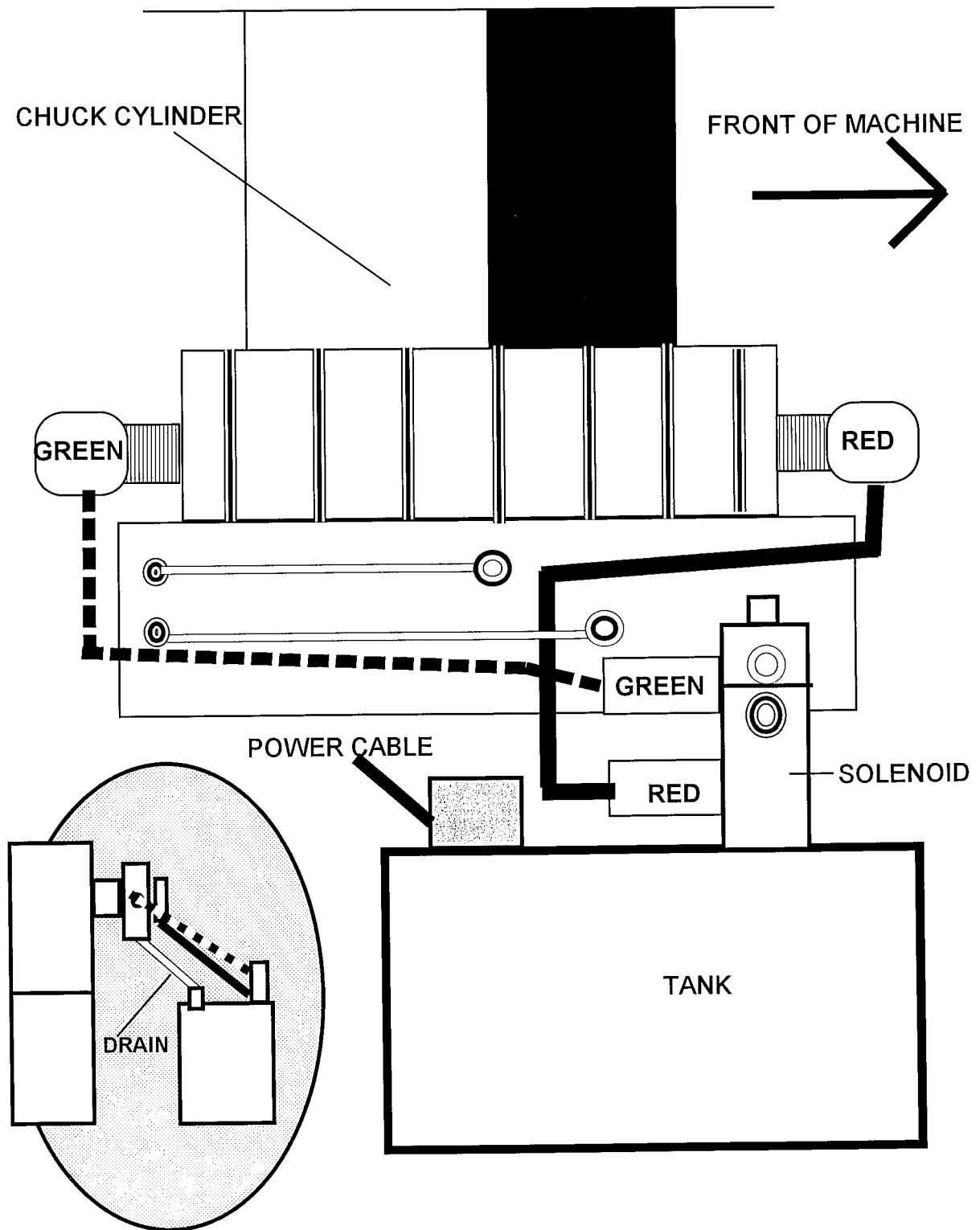
WARNING

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

Harrison

GRAPHICS

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

press

| EDIT |

press

| F9 |

cursor to

| SET TOOLING |

press

| TOOL NUMBER | ← to display | E.O.B. |

cursor to

| TOOL 1 |

press

| E.O.B. | → to display → | TOOL SHAPES LIBRARY |

press ↓

| E.O.B. | ← press | PAGE |

to allocate shape to tool number

to index through tool library and select tool shape required.

cursor to

| TOOL 2 | → | E.O.B. | → | TOOL SHAPE |

| E.O.B. | ← | PAGE |

cursor to

| TOOL 3 | → | E.O.B. | → | TOOL SHAPE |

| E.O.B. | ← | PAGE |
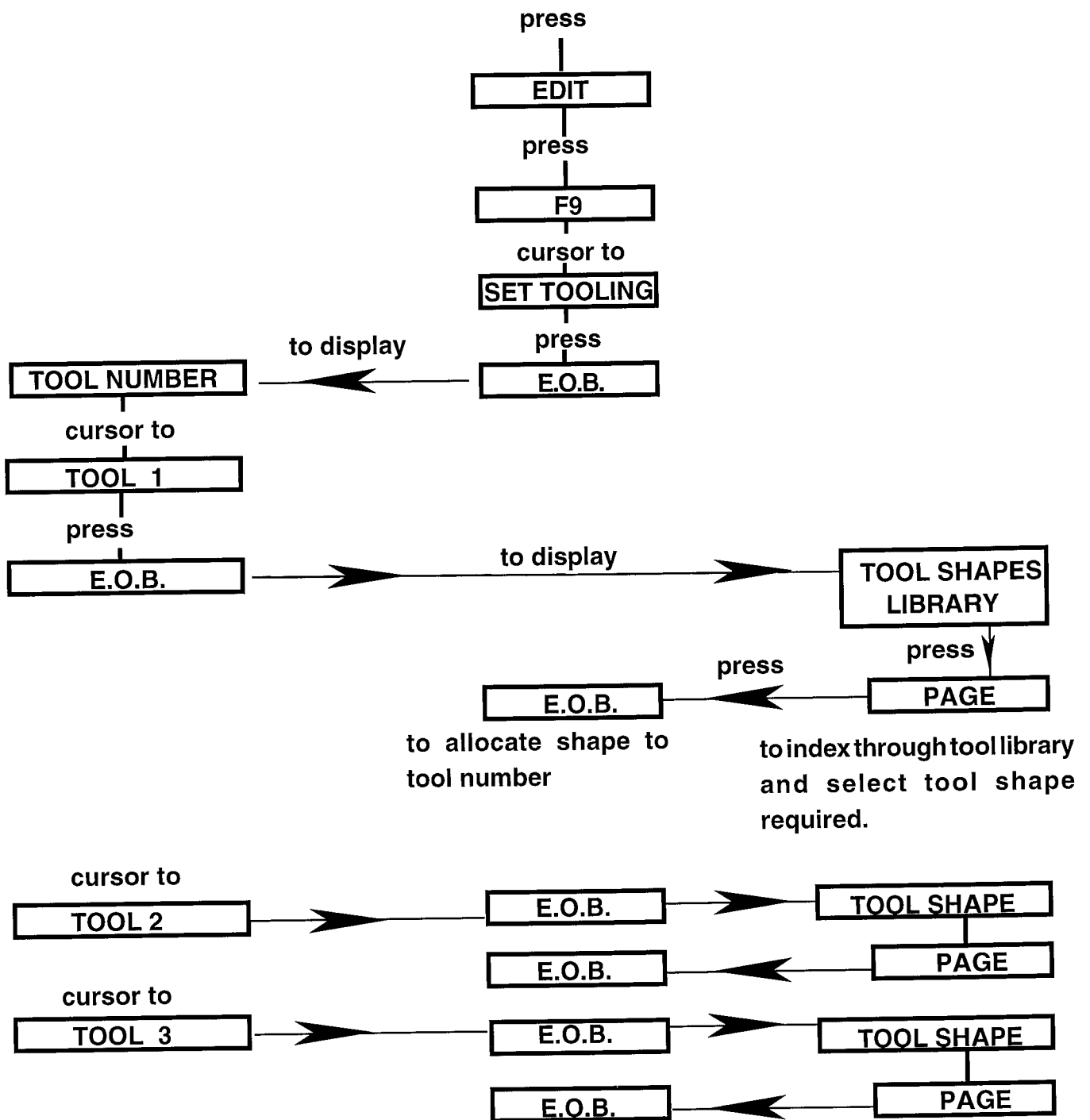
GRAPHICS

TO LOAD OR SAVE GRAPHICS FOR A PART PROGRAM:-

NB. THE MACHINE MUST FIRST BE HOMED

PRESS F10 FOR

| MAIN MENU |

cursor to

| SETTINGS |

press

| E.O.B. |

cursor to

| LOAD SETTINGS |

cursor to

| SAVE SETTINGS |

press   press

| E.O.B. |

type in

| PROG No. | (i.e. 7777)

press

| E.O.B. |
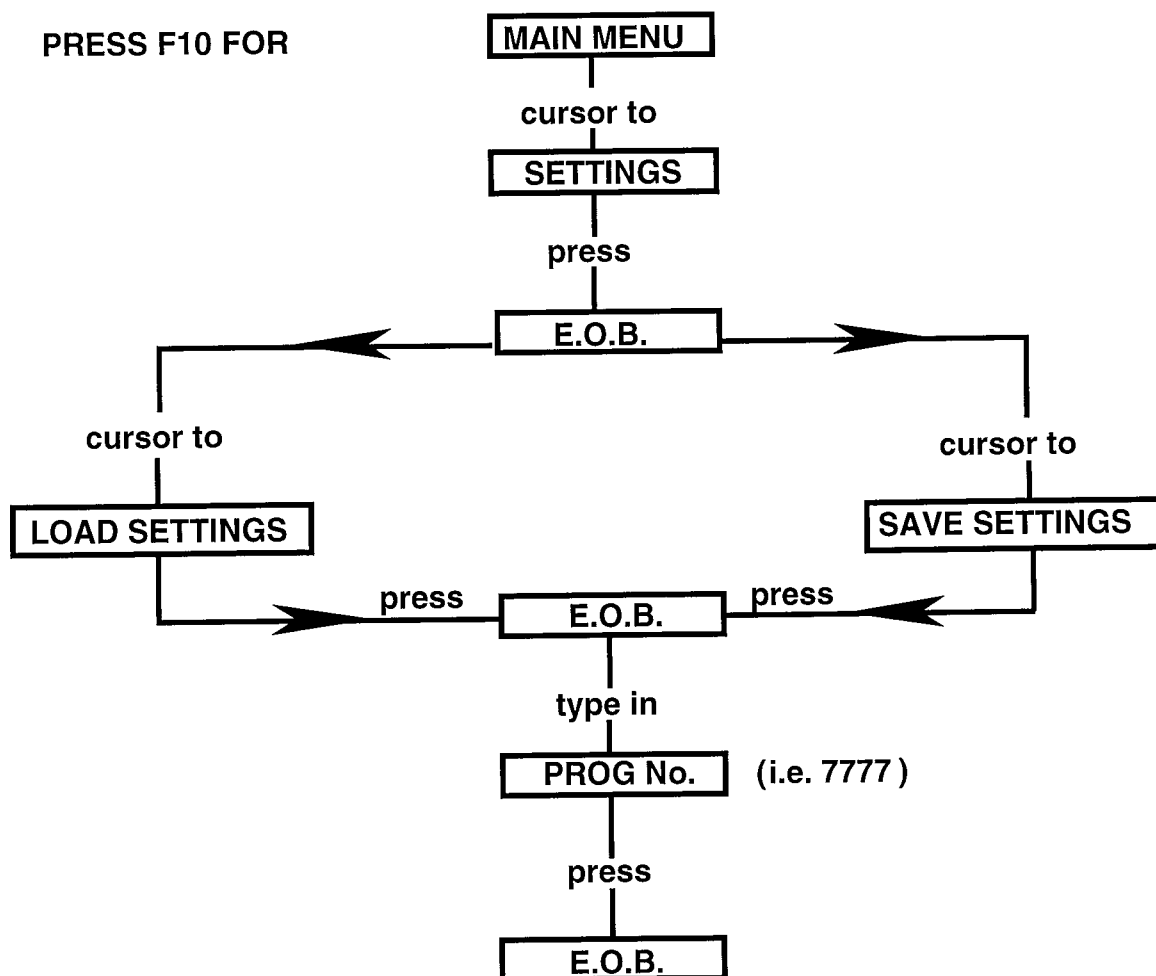
TO LOAD A PROGRAM FROM DISC:-

PRESS F10 FOR :-

| MAIN MENU |

press

| E.O.B. |

cursor to

| CNC FILES |

press

| E.O.B. |

cursor to

| LOAD |

press

| E.O.B. |

type

| PROG No. | (i.e.6666)

press

| E.O.B. |

| QUESTION - DO YOU WANT TO MERGE? |

press

| NO or YES |

| LOADS PROGRAM TO SCREEN |

press

| EDIT |

select

| EDIT / SIM. |
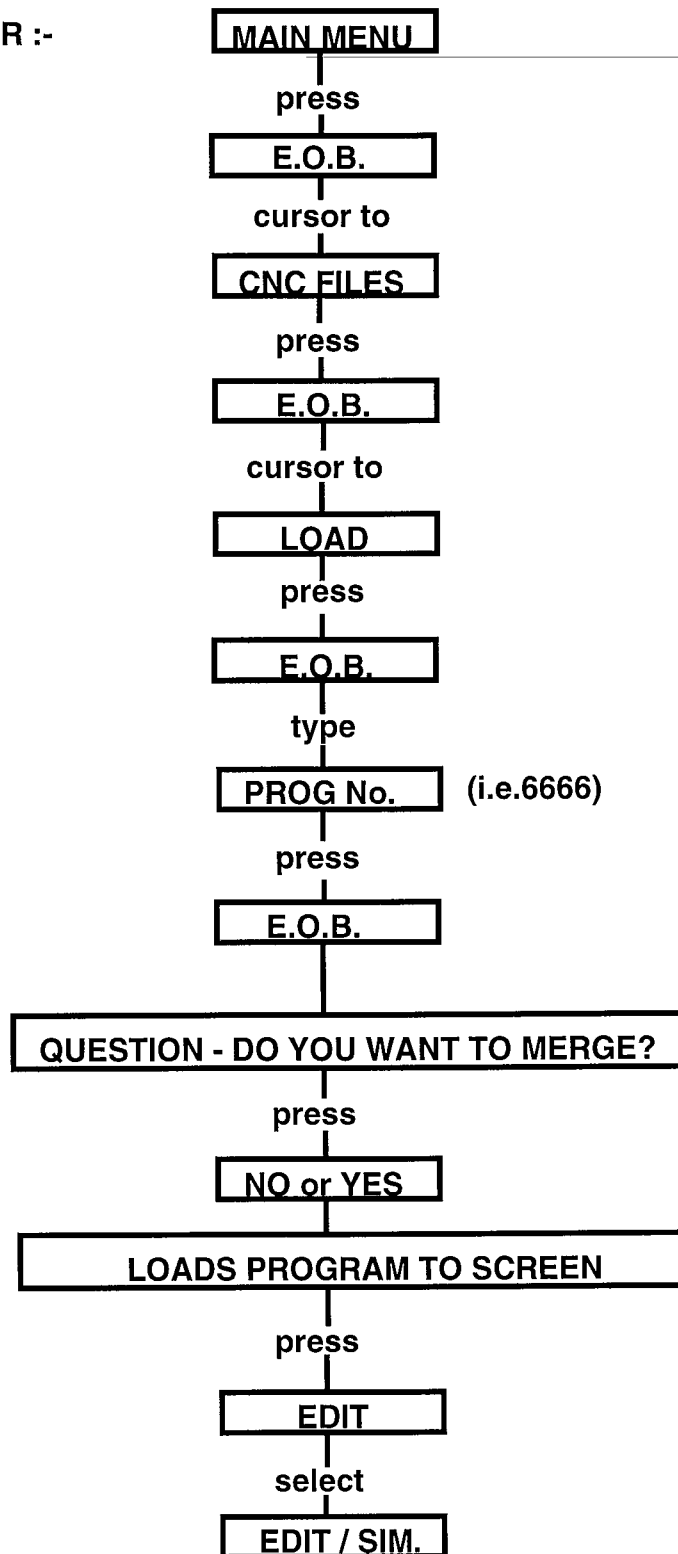
TO SAVE OR DELETE A PROGRAM TO DISC

PRESS F10 FOR

MAIN MENU

cursor to

CNC FILES

press

E.O.B.

cursor to

SAVE AS

press

E.O.B.

Type in program No.(i.e. 6262)

press

E.O.B.

MAIN MENU

cursor to

CNC FILES

press

E.O.B.

cursor to

DELETE

press

E.O.B.

Type in program No.(i.e. 6262)

press

E.O.B.
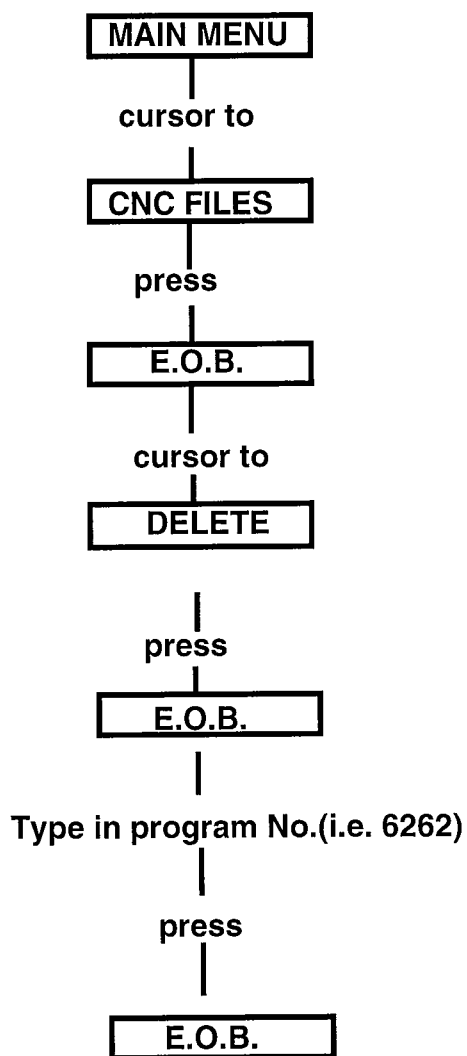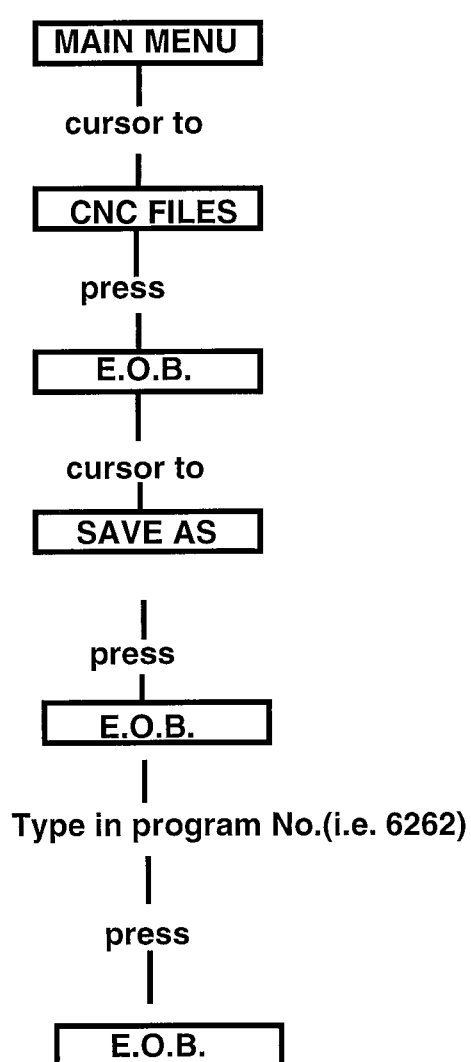
TO CALL UP THE CNC PROGRAM LIBRARY

NB. THE MACHINE MUST FIRST BE HOMED

PRESS F10 FOR | MAIN MENU |

cursor to

| CNC FILES |

press

| E.O.B. |

| ENSURE DISC IS LOADED |

cursor to

| LOAD |

press

| E.O.B. |

to display

| **LIBRARY**
0. 6667

0.8888   **cursor to selected program**

0.7777 |

press

| E.O.B. |
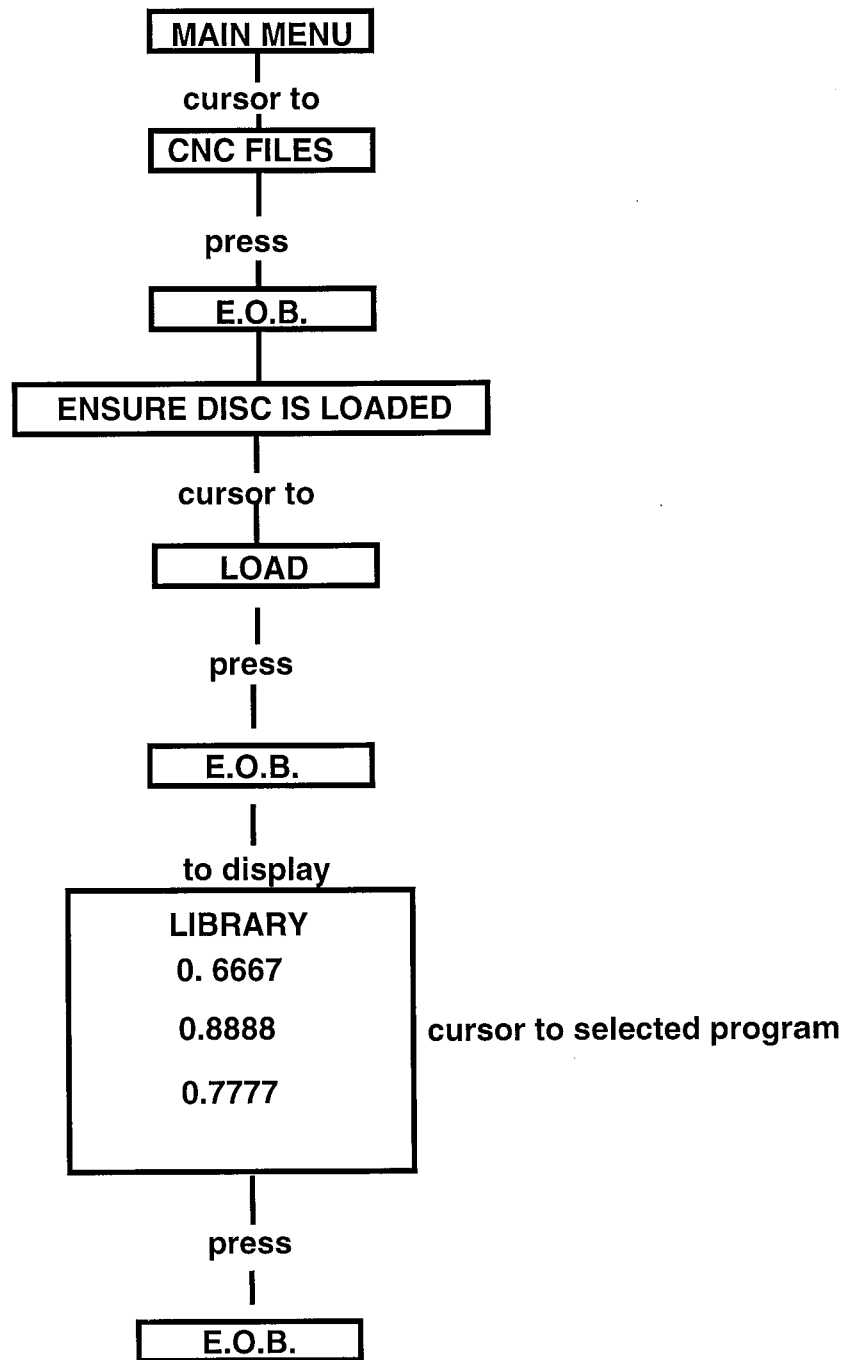
TO SAVE OR LOAD TOOL OFFSETS FOR A GENERATED PROGRAM

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

N.B. THE MACHINE MUST FIRST BE HOMED

press

JOG

press

| F9 | to display control screen
|---|

> EXECUTE PROGRAM
>
> DRY RUN
>
> TOOL OFFSETS
>
> LOAD TOOL OFFSETS
>
> SAVE TOOL OFFSETS
>
> DELETE TOOL OFFSETS

———— cursor to ————

LOAD TOOL OFFSETS		TOOL OFFSETS		SAVE TOOL OFFSETS
press		press		press
E.O.B.		E.O.B		E.O.B.
type in				type in
OFFSET No.		DISPLAY CURRENT TOOL OFFSETS		OFFSET No.
(same as prog No.) press				(same as prog No.) press
E.O.B.				E.O.B.
to signal				to signal
LOADING TOOL OFFSETS				SAVING TOOL OFFSETS

TO DELETE A TOOL OFFSET

press

JOG

press

| F9 | to display control screen

> EXECUTE PROGRAM
> DRY RUN
> TOOL OFFSETS
> LOAD TOOL OFFSETS
> SAVE TOOL OFFSETS
> DELETE TOOL OFFSETS

cursor to

DELETE TOOL OFFSETS

press

E.O.B.

type in

OFFSET No.

press

E.OB.

deletes tool offset

6. MAIN MENU

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

```
┌──────CNC Editor──────────┐  ┌─────────────Simulation════════════┐
│ Line 1     Column 1  Insert│  │                                    │
│                          │  │                                    │
│                          │  │                                    │
│                          │  ┌──────Main Menu══════┐               │
│                          │  │ Edit only           │               │
│                          │  │ Edit and Simulate   │               │
│                          │  │ Simulate only       │  ████████████ │
│                          │  │ Machine control     │  ████████████ │
│                          │  │ CNC Files           │  ████████████ │
│                          │  │ Print               │  ████████████ │
│                          │  │ Remote link         │               │
│                          │  │ Settings            │               │
│                          │  │ Utilities           │               │
│                          │  │ Quit                │               │
│                          │  └─────────────────────┘               │
│                          │  └────────────────────────────────────┘
└──────────────────────────┘
┌═══════════════════Tutorial═══════════════════════════════════════┐
│                                                                   │
└───────────────────────────────────────────────────────────────────┘
```
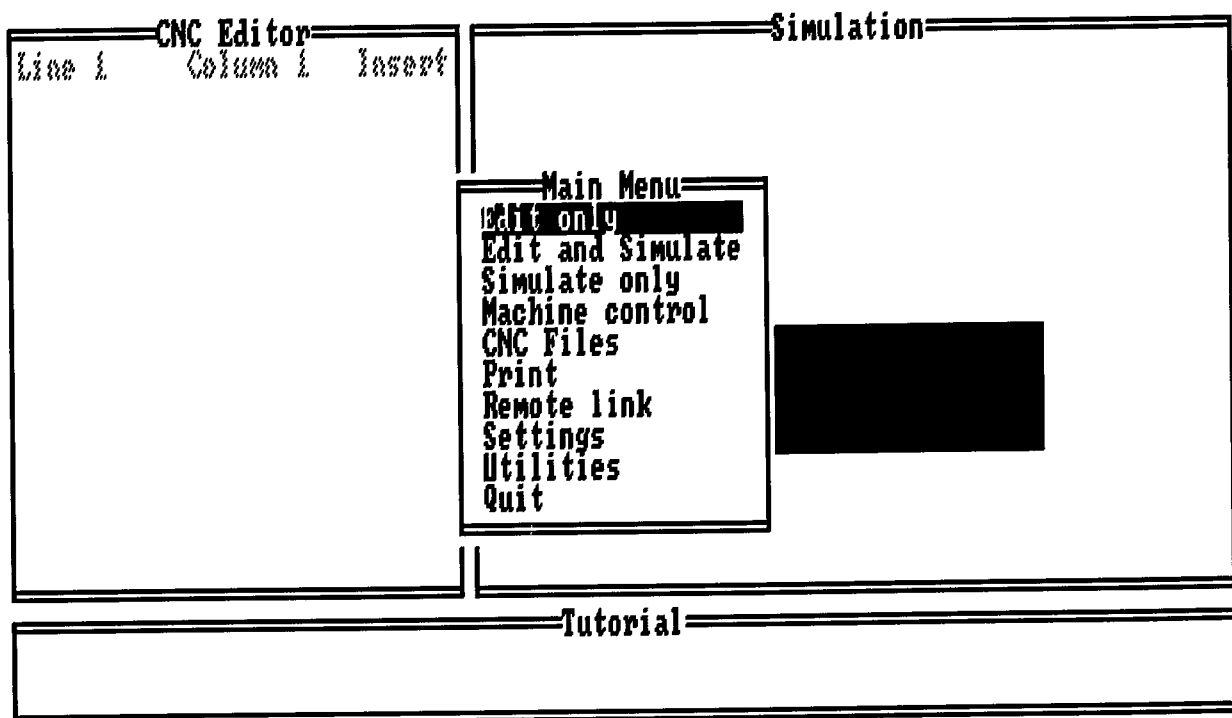
EDIT ONLY

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.
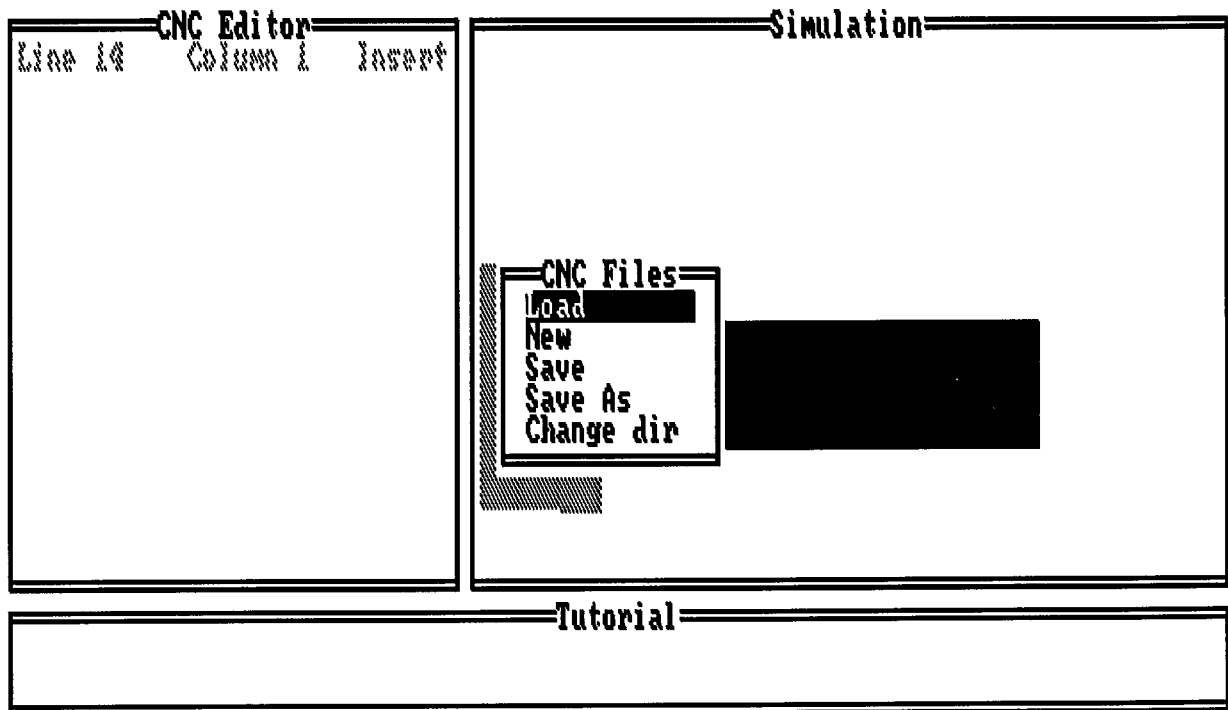
MACHINE CONTROL

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

```
┌──────CNC Editor──────┐ ┌─────────Simulation─────────┐
│ Line 10   Column 1  Insert │ │                            │
│                      │ │                            │
│                      │ │                            │
│                      │ │                            │
│                      │ │  ┌──CNC Files──┐           │
│                      │ │  │ Load        │           │
│                      │ │  │ New         │  ████████ │
│                      │ │  │ Save        │  ████████ │
│                      │ │  │ Save As     │  ████████ │
│                      │ │  │ Change dir  │  ████████ │
│                      │ │  └─────────────┘           │
│                      │ │                            │
└──────────────────────┘ └────────────────────────────┘
      ┌──────────────────Tutorial──────────────────┐
      │                                             │
      └─────────────────────────────────────────────┘
```

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.
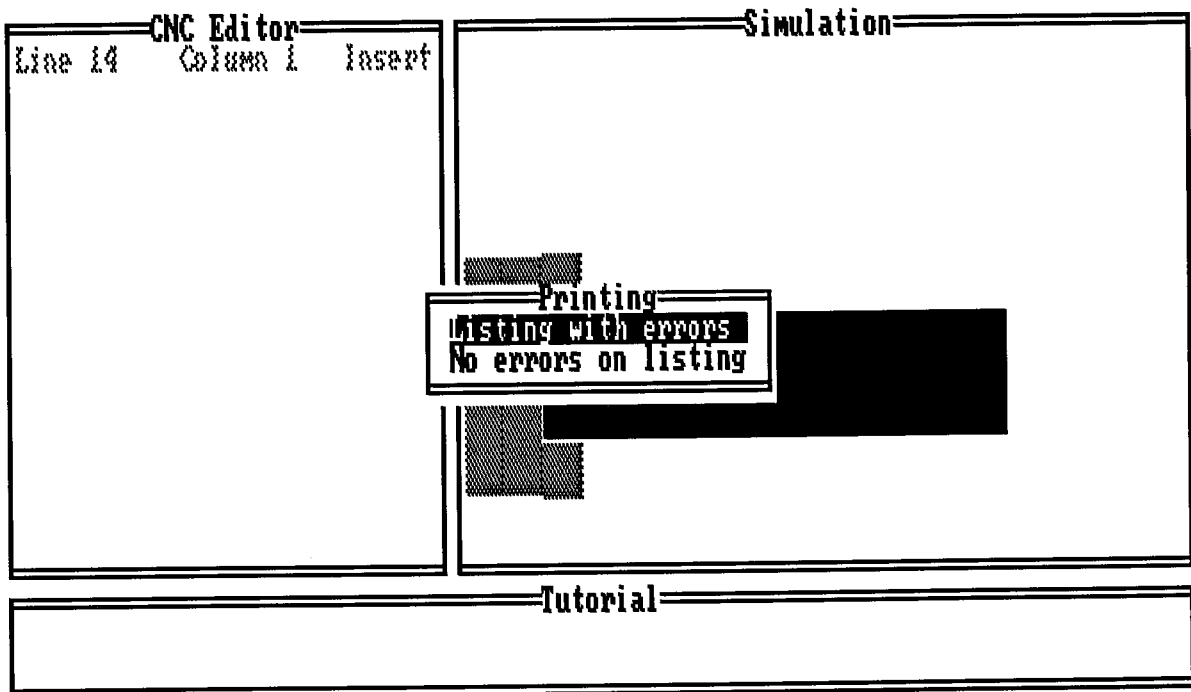
CHANGE DIR

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─────────┐  ┌───────────Simulation───────────┐
│Line 14   Column 1  Insert│  │                                 │
│                          │  │                                 │
│                          │  │                                 │
│                          │  │                                 │
│                   ┌─────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.

```
┌────CNC Editor────┐   ┌──────Simulation══════════┐
│ Line 14   Column 1   Insert │ │                          │
│                  │   │                          │
│                  │   │                          │
│                  │   │                          │
│           ┌─Remote Device Link─┐                │
│           │ Load from device   │█████████       │
│           │ Send to device     │                │
│           └────────────────────┘                │
│                  │   │                          │
│                  │   │                          │
└──────────────────┘   └──────────────────────────┘
┌══════════════════════Tutorial══════════════════┐
│                                                 │
└─────────────────────────────────────────────────┘
```
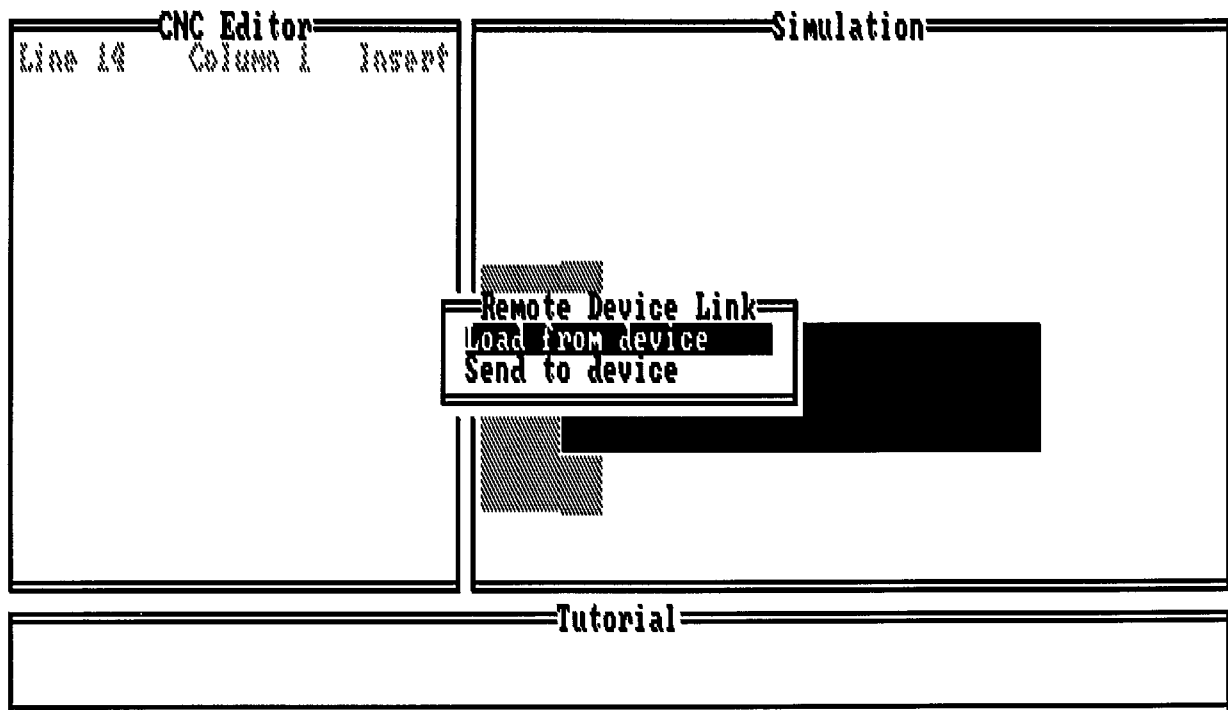
REMOTE DEVICE LINK

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.
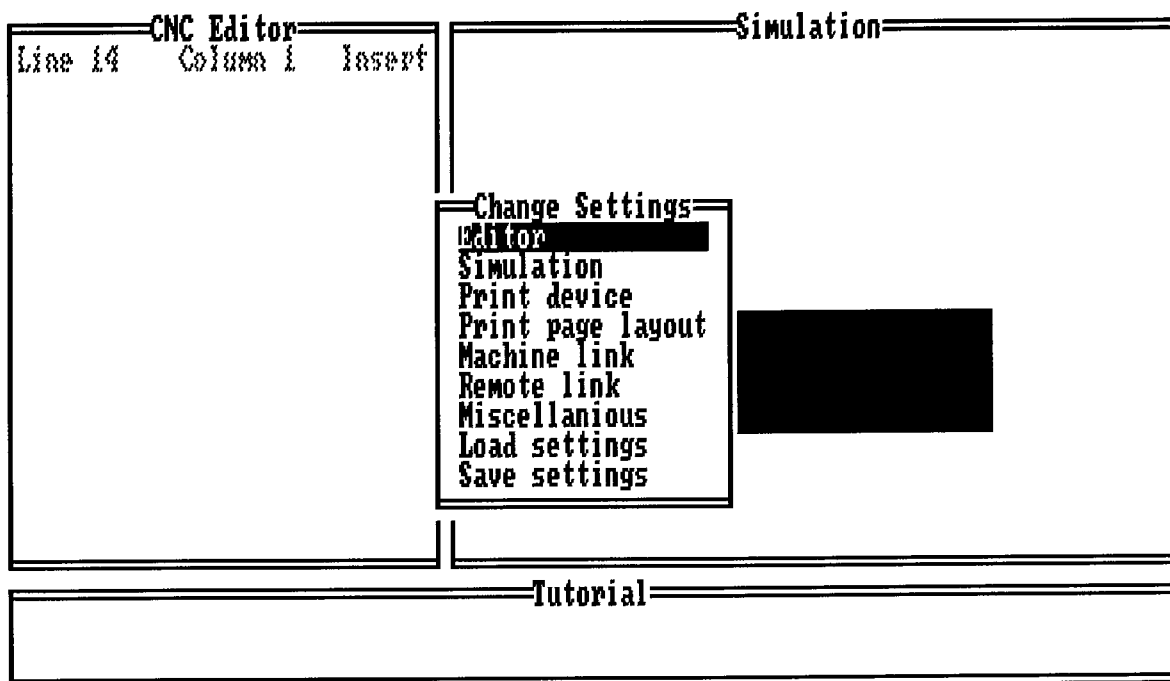
COMMUNICATIONS ERRORS

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.

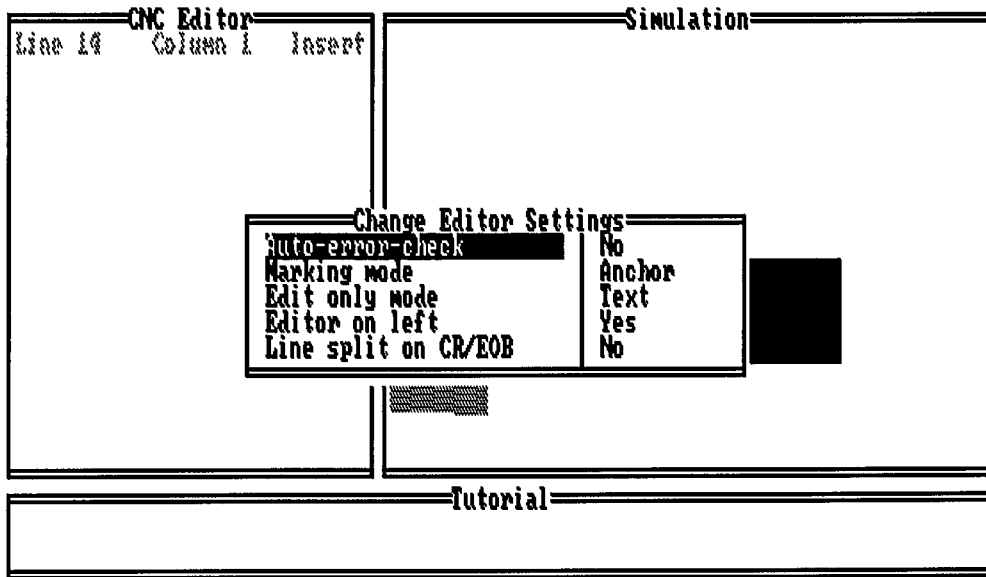d) Ensure that the cable is plugged into the correct Port of the Computer.

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

![Harrison logo]

8. SETTINGS

```
┌────────CNC Editor────────┐ ┌───────────────Simulation═══════════════┐
│Line 14    Column 1  Insert│ │                                         │
│                          │ │                                         │
│                          │ │  ┌─Change Settings─┐                    │
│                          │ │  │Editor           │                    │
│                          │ │  │Simulation       │   ███████████      │
│                          │ │  │Print device     │   ███████████      │
│                          │ │  │Print page layout│   ███████████      │
│                          │ │  │Machine link     │   ███████████      │
│                          │ │  │Remote link      │                    │
│                          │ │  │Miscellanious    │                    │
│                          │ │  │Load settings    │                    │
│                          │ │  │Save settings    │                    │
│                          │ │  └─────────────────┘                    │
└──────────────────────────┘ └─────────────────────────────────────────┘
        ┌═══════════════════════Tutorial═══════════════════════┐
        │                                                        │
        └════════════════════════════════════════════════════════┘
```

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.

```
┌────CNC Editor────────┐  ┌──────────────Simulation──────────────┐
│Line 14   Column 1  Insert│  │                                   │
│                          │  │                                   │
│              ┌───────Change Editor Settings───────┐             │
│              │Auto-error-check            No       │             │
│              │Marking mode                Anchor   │  ███████    │
│              │Edit only mode              Text     │  ███████    │
│              │Editor on left              Yes      │  ███████    │
│              │Line split on CR/EOB        No       │             │
│              └────────────────────────────────────┘             │
│              │   ▓▓▓▓▓▓▓▓                          │             │
│              │                                     │             │
└──────────────────────────────Tutorial──────────────────────────┘
│                                                                 │
└─────────────────────────────────────────────────────────────────┘
```

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
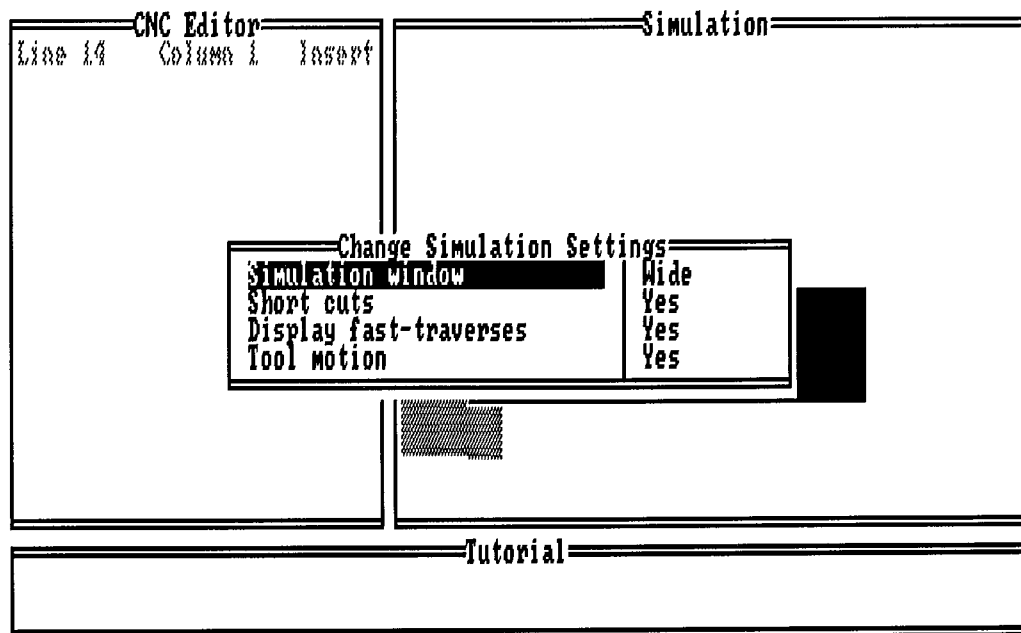
larger view of the Simulation, if the Wide option is selected - see screen on following page(8.5).

EXAMPLE SIMULATION ON WIDER WINDOW SETTING

=Simulation=

X   31.000   Z -69.892

=Tutorial=
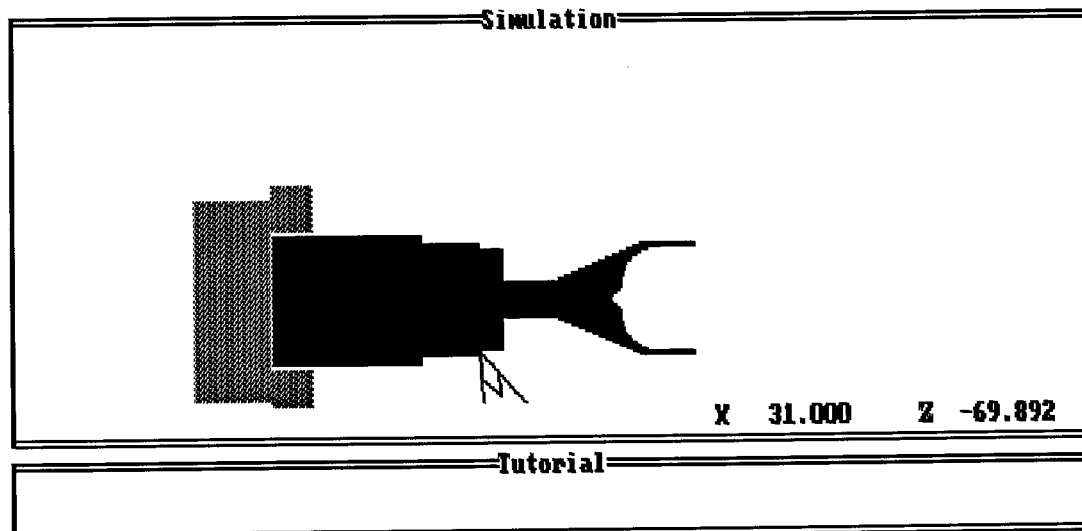
Simulating CNC program ....

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

```
┌──────CNC Editor──────┐  ┌──────────────Simulation──────────────┐
│Line 14   Column 1  Insert│                                      │
│                      │  │                                       │
│                      │  │                                       │
│                      │  │  ┌──Change Settings──┐                │
│                      │  │  │Editor             │                │
│                      │  │  │Simulation         │  ┌───────────┐ │
│                      │  │  │Print device       │  │███████████│ │
│                      │  │  │Print page layout  │  │███████████│ │
│                      │  │  │Machine link       │  │███████████│ │
│                      │  │  │Remote link        │  │███████████│ │
│                      │  │  │Miscellanious      │  └───────────┘ │
│                      │  │  │Load settings      │                │
│                      │  │  │Save settings      │                │
│                      │  │  └───────────────────┘                │
│                      │  │                                       │
└──────────────────────┘  └───────────────────────────────────────┘
                  ┌─────────────Tutorial─────────────┐
                  │                                   │
                  └───────────────────────────────────┘
```
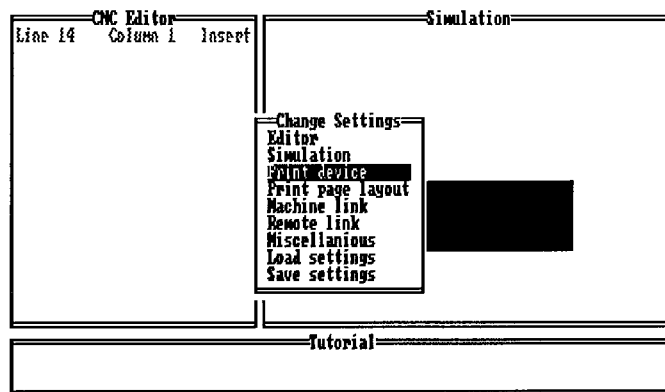
CHANGE PRINT DEVICE

DEVICE : TYPE This can be set to :-

```
┌──────CNC Editor──────┐  ┌──────────────Simulation──────────────┐
│Line 14   Column 1  Insert│                                      │
│                      │  │                                       │
│                      │  │                                       │
│                      │  │  ┌──Change Print Device──┐            │
│                      │  │  │Device: Type   │Dos device          │
│                      │  │  │        Name   │PRN                 │
│                      │  │  │RS232: Baudrate│          ┌───────┐ │
│                      │  │  │       Parity  │          │███████│ │
│                      │  │  │       Data bits│         │███████│ │
│                      │  │  │       Stop bits│         └───────┘ │
│                      │  │  │       Protocol │                   │
│                      │  │  └────────────────┘                   │
│                      │  │                                       │
└──────────────────────┘  └───────────────────────────────────────┘
                  ┌─────────────Tutorial─────────────┐
                  │                                   │
                  └───────────────────────────────────┘
```
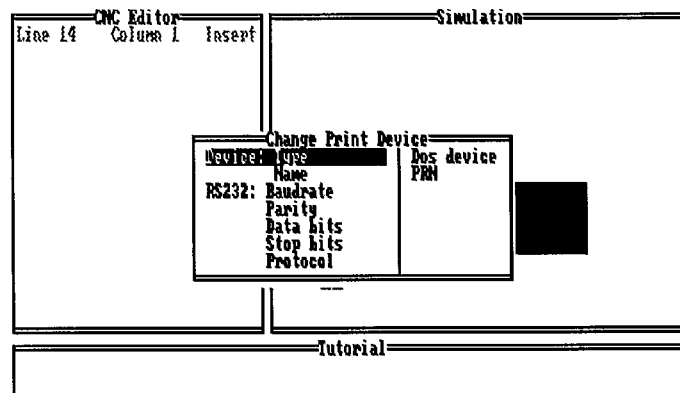
DOS DEVICE

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

```
CNC Editor                          Simulation
Line 14    Column 1   Insert


                    Change Settings
                    Editor
                    Simulation
                    Print device
                    Print page layout
                    Machine link
                    Remote link
                    Miscellanious
                    Load settings
                    Save settings


                         Tutorial
```

```
CNC Editor                          Simulation
Line 14    Column 1   Insert


                Change Print Page Layout
                Page width            80
                Page depth            60
                Left margin           0
                Top margin            0
                Bottom margin         0
                Column width          80
                Carriage return nulls 0
                Line feed nulls       0
                Form feed nulls       0
                Print line feeds      Yes


                         Tutorial
```

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

```
┌──────CNC Editor──────┐  ┌─────────────Simulation─────────────┐
│Line 14   Column 1   Insert│                                    │
│                      │  │                                     │
│                      │  │   ┌──Change Settings──┐             │
│                      │  │   │Editor             │             │
│                      │  │   │Simulation         │             │
│                      │  │   │Print device       │             │
│                      │  │   │Print page layout  │ ███████████ │
│                      │  │   │Machine link       │ ███████████ │
│                      │  │   │Remote link███████ │ ███████████ │
│                      │  │   │Miscellanious      │             │
│                      │  │   │Load settings      │             │
│                      │  │   │Save settings      │             │
│                      │  └───┴───────────────────┴─────────────┘
└──────────────────────┘  
  ┌──────────────────────────Tutorial──────────────────────────┐
  │                                                             │
  └─────────────────────────────────────────────────────────────┘
```
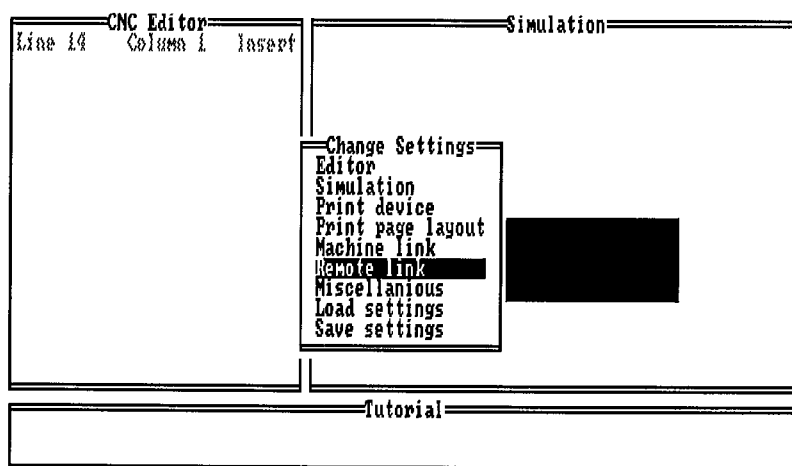
The settings for the remote link are for the Serial Port. This sets the protocol when communicating to an external device, such as a remote computer or paper tape punch.

```
┌─────────────────────┐ ┌──────────────────────────────┐
│ Line 14  Column 1  Insert │ │          SIMULATION          │
│                     │ │                              │
│         ┌═════Change Remote Link═══┐                │
│         │ Device name       │ COM2 │                │
│         │ Baudrate          │ 4800 │                │
│         │ Parity            │ Odd  │                │
│         │ Data bits         │ 7    │  ████████      │
│         │ Stop bits         │ 1    │  ████████      │
│         │ Send line-feeds   │ Yes  │  ████████      │
│         │ End-of-file       │ ctrl-Z│               │
│         │ Send XOFF at % full│ 100  │               │
│         └═══════════════════════┘                  │
│                                                      │
└─────────────────────┘ │                              │
┌═══════════════════════Tutorial═══════════════════════┐
│                                                      │
└──────────────────────────────────────────────────────┘
```
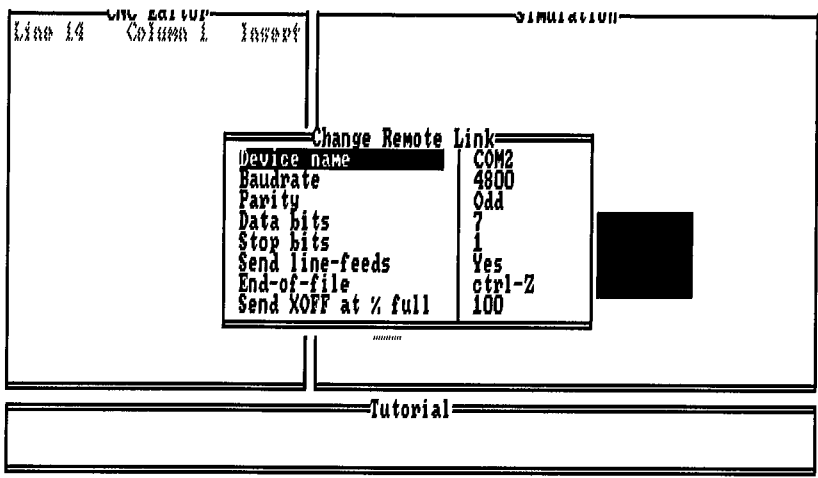
CHANGE MISCELLANEOUS SETTINGS

```
┌─────────────────────┐ ┌──────────────────────────────┐
│ Line 14  Column 1  Insert │ │          SIMULATION          │
│                     │ │                              │
│         ┌═Change Settings═┐                          │
│         │ Editor          │                          │
│         │ Simulation      │                          │
│         │ Print device    │                          │
│         │ Print page layout│  ████████               │
│         │ Machine link    │  ████████               │
│         │ Remote link     │  ████████               │
│         │ Miscellaneous   │                          │
│         │ Load settings   │                          │
│         │ Save settings   │                          │
│         └═════════════════┘                          │
│                     │ │                              │
└─────────────────────┘ │                              │
┌═══════════════════════Tutorial═══════════════════════┐
│                                                      │
└──────────────────────────────────────────────────────┘
```
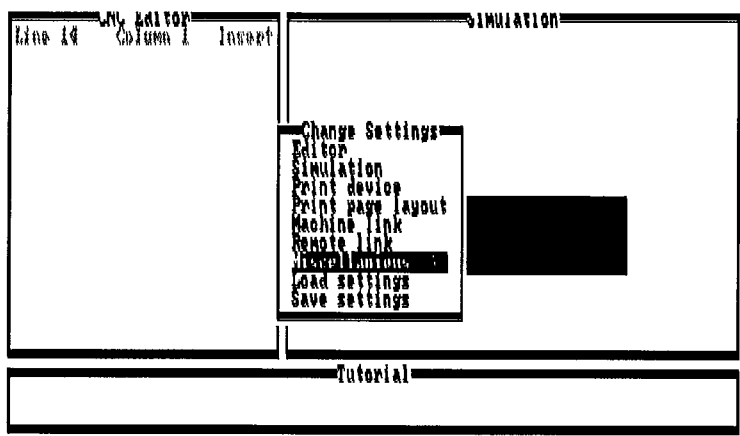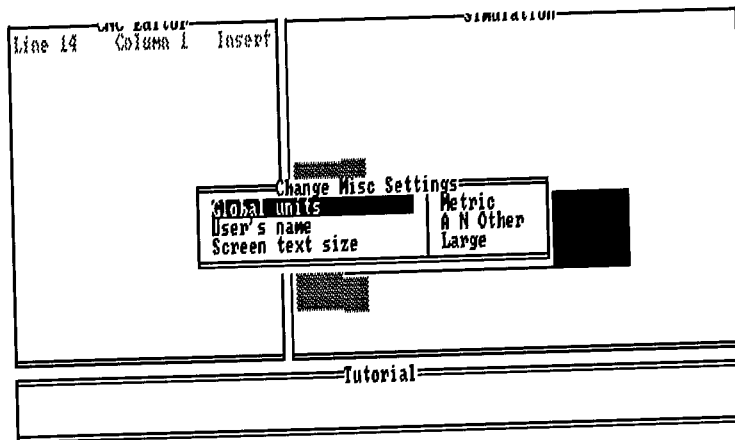
GLOBAL UNITS - USERS NAME - SCREEN TEXT SIZE

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

```
┌──────CNC Editor──────┐ ┌──────────Simulation──────────┐
│Line 14   Column 1  Insert│                              │
│                      │ │                              │
│                      │ │                              │
│                      │ │    ▓▓▓▓▓                      │
│             ┌────Options Load Filename────┐ ███████    │
│             │TU150.OPT                    │ ███████    │
│             └─────────────────────────────┘ ▓▓▓▓▓     │
│                      │ │                              │
└──────────────────────┘ └──────────────────────────────┘
         ┌══════════════Tutorial══════════════┐
         │                                     │
         └─────────────────────────────────────┘
```
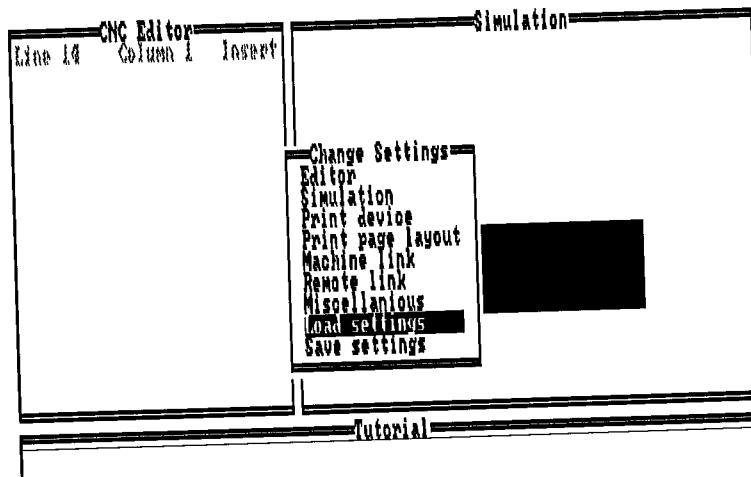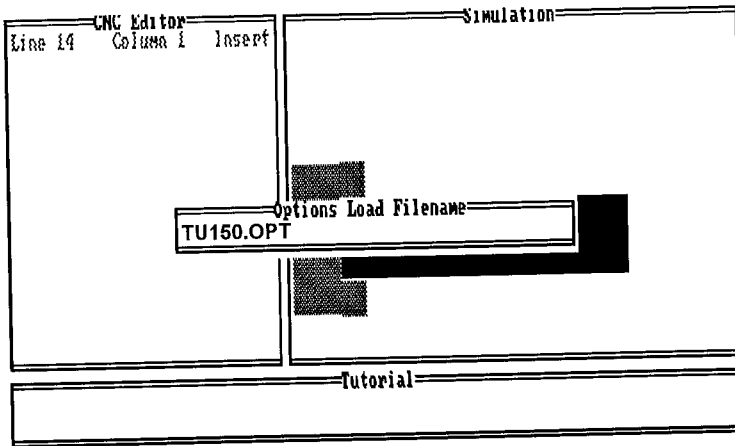
Any number of settings files can stored to Disk with the Extension ".OPT".

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

```
┌──────CNC Editor──────┐ ┌──────────Simulation──────────┐
│Line 14   Column 1  Insert│                              │
│                      │ │                              │
│                      │ │                              │
│                      │ │ ┌─Change Settings─┐           │
│                      │ │ │Editor           │           │
│                      │ │ │Simulation       │ ███████   │
│                      │ │ │Print device     │ ███████   │
│                      │ │ │Print page layout│ ███████   │
│                      │ │ │Machine link     │ ███████   │
│                      │ │ │Remote link      │           │
│                      │ │ │Miscellanious    │           │
│                      │ │ │Load settings    │           │
│                      │ │ │Save settings    │           │
│                      │ │ └─────────────────┘           │
└──────────────────────┘ └──────────────────────────────┘
         ┌══════════════Tutorial══════════════┐
         │                                     │
         └─────────────────────────────────────┘
```

```
┌══CNC Editor══┐   ┌═══════Simulation═══════┐
│Line 14  Column 1  Insert│
│              ┌═Options Save Filename═══│
│              │TU150.OPT│
│              │
│              │
└──────────────┘   └────────────────────────┘
        ═══════Tutorial═══════
```
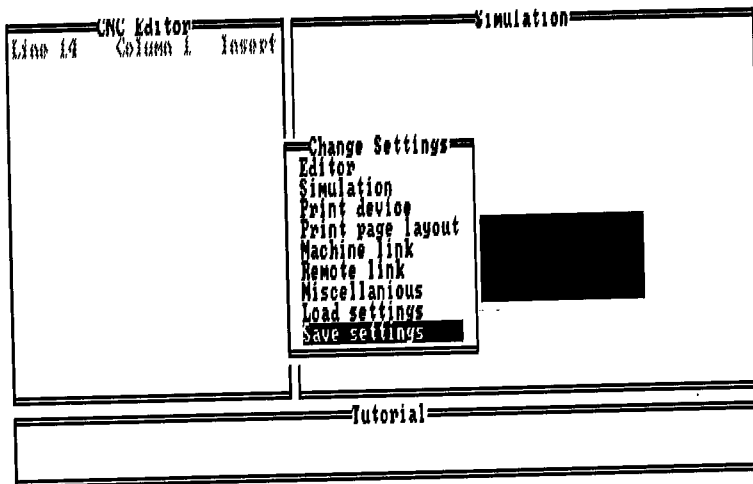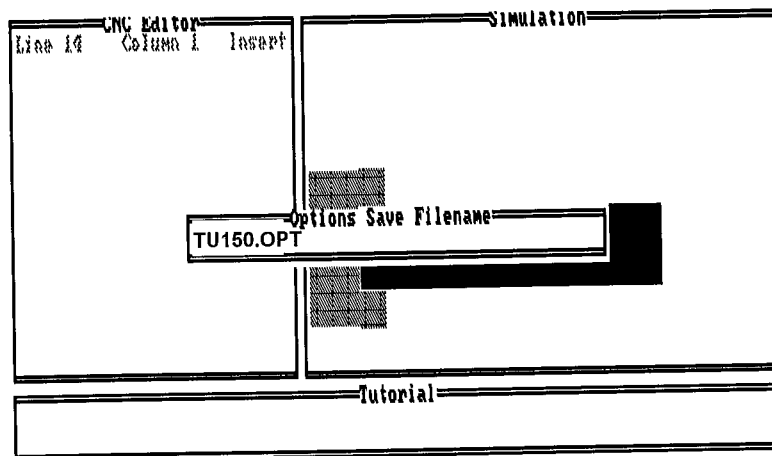
Select Save Settings option:

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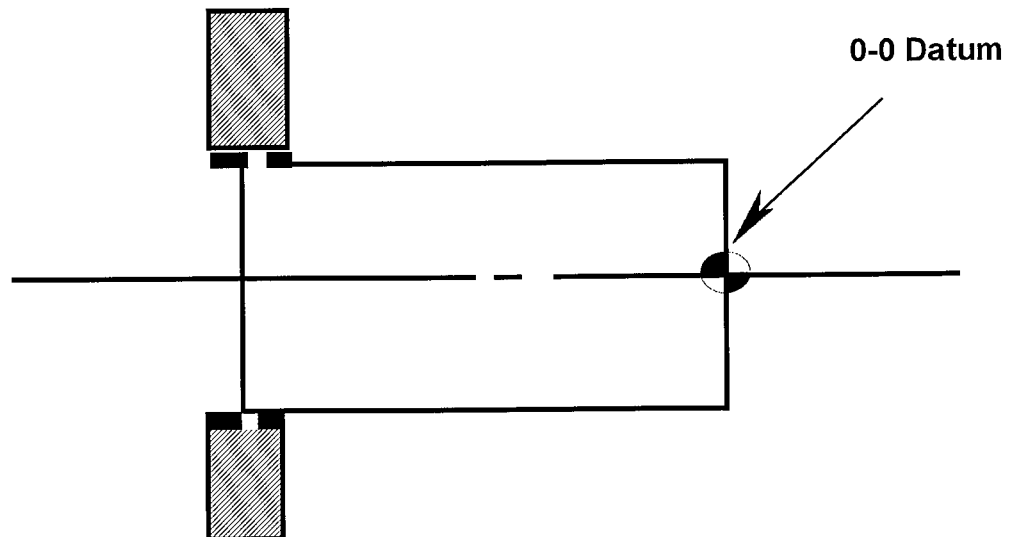
By pressing F9 the SIMULATION MENU is brought up on screen.

```
┌─────────── SIMULATION ───────────┐
│                                   │
│          CHECK SYNTAX             │
│                                   │
│          RUN PROGRAM              │
│                                   │
│          DRY RUN                  │
│                                   │
│          SET TOOLING              │
│                                   │
│          SET VIEW                 │
│                                   │
│          3 D VIEW                 │
│                                   │
└───────────────────────────────────┘
```
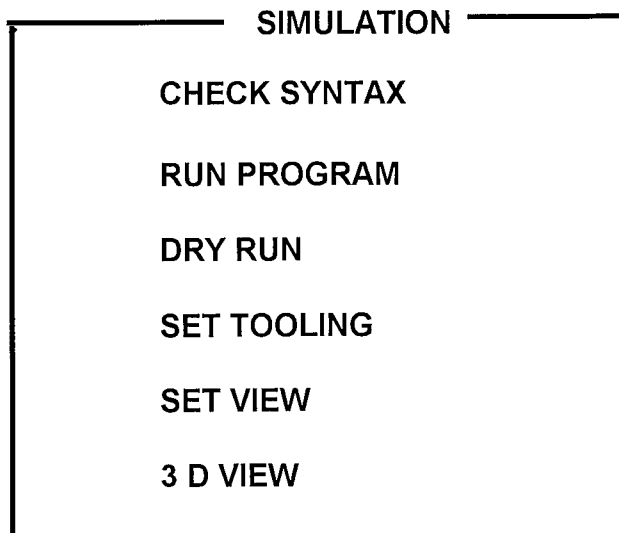
CHECK SYNTAX

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.

```
==============================Simulation=============================




                                                        X   31.000    Z  -69.892
==============================Tutorial==============================


```

Simulating CNC program ....

```
==============================Simulation=============================




                                                        X  160.000    Z   55.000
==============================Tutorial==============================


```
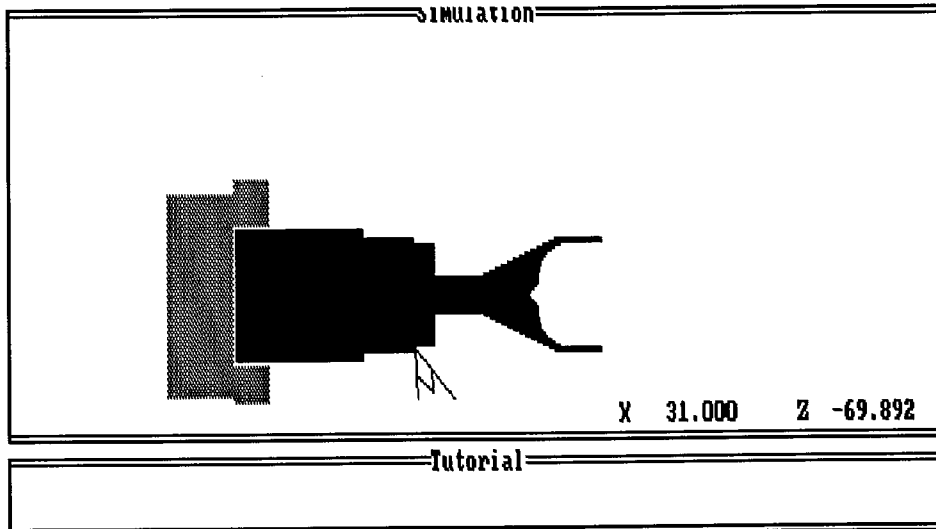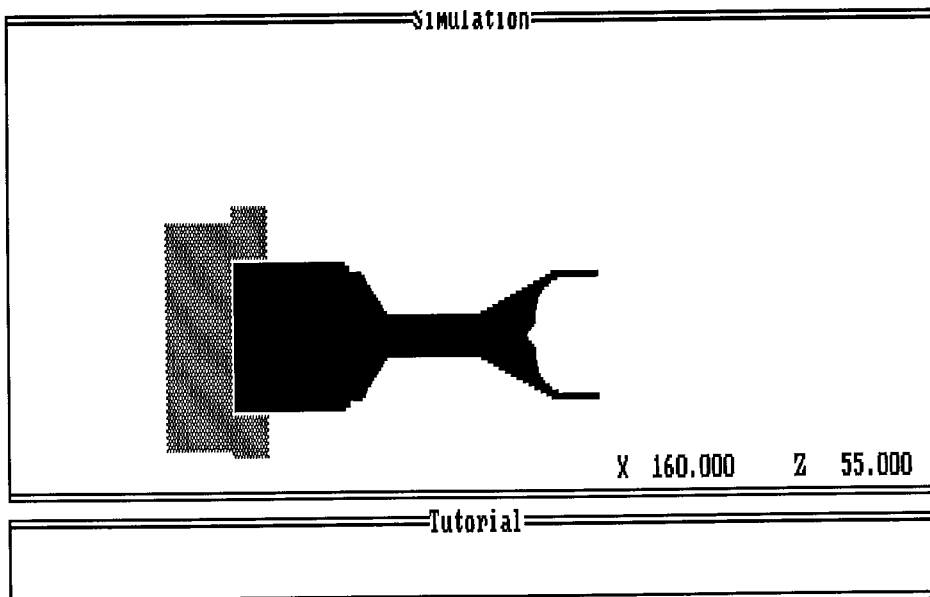
14. EDITING

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 <u>G40 G41</u> 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			
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

X+
(U, I)

Z +

Z -

(W, K)

X -
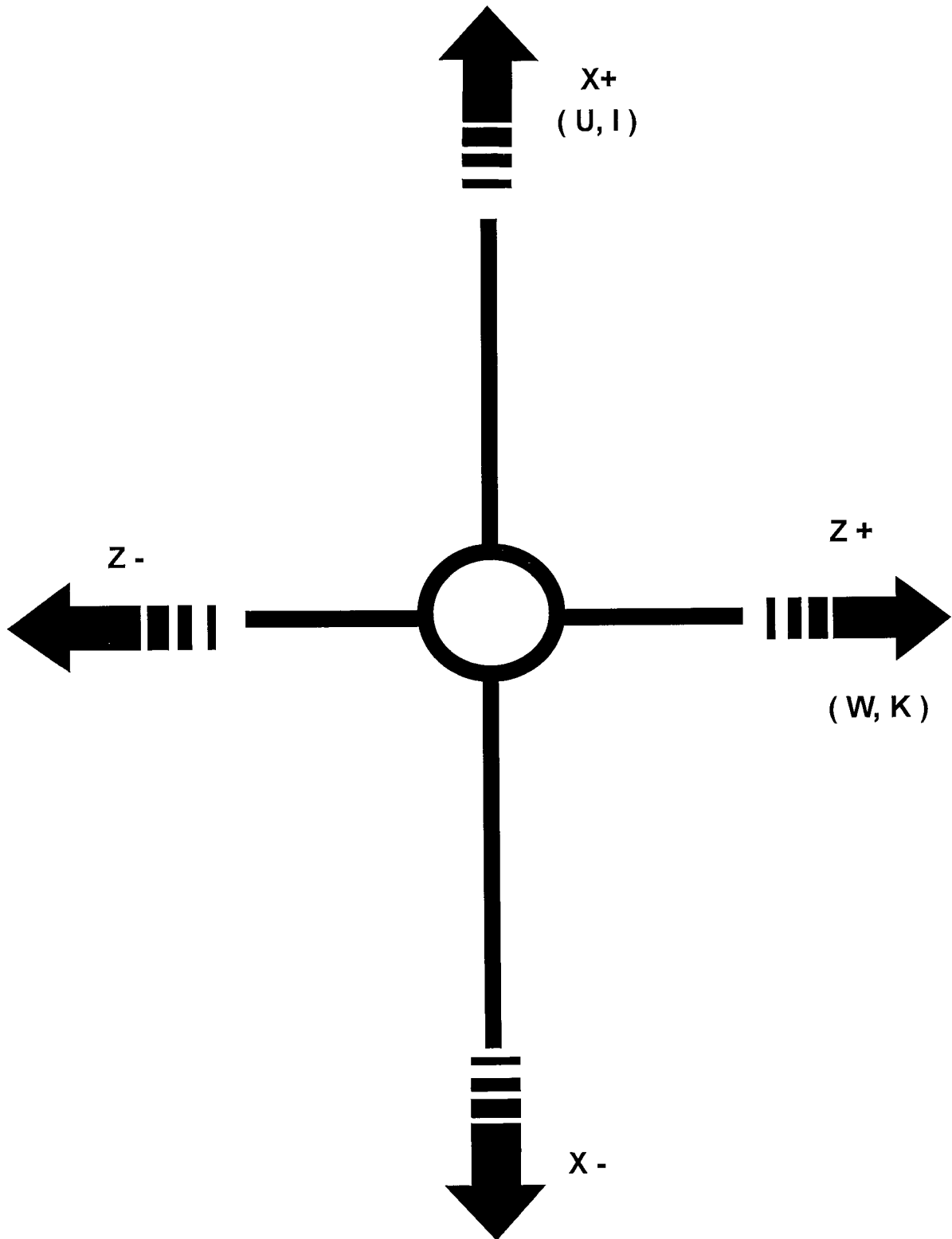
BASIC MOVEMENTS

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
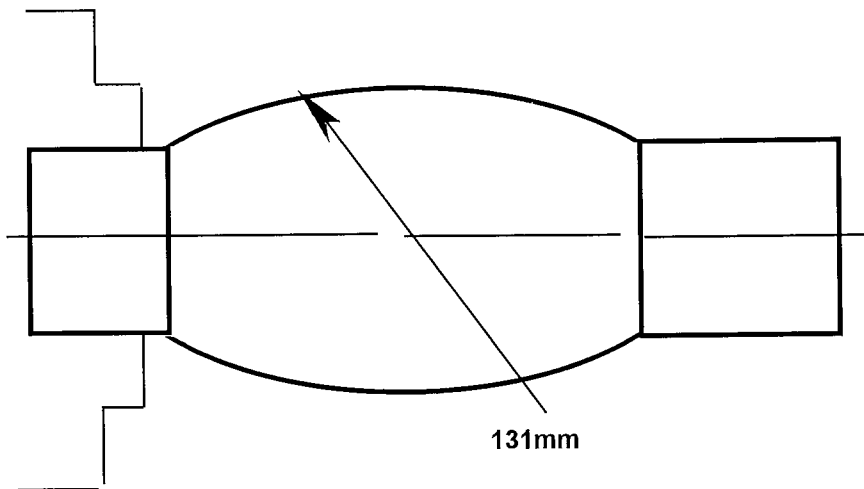
131mm

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

Right Hand

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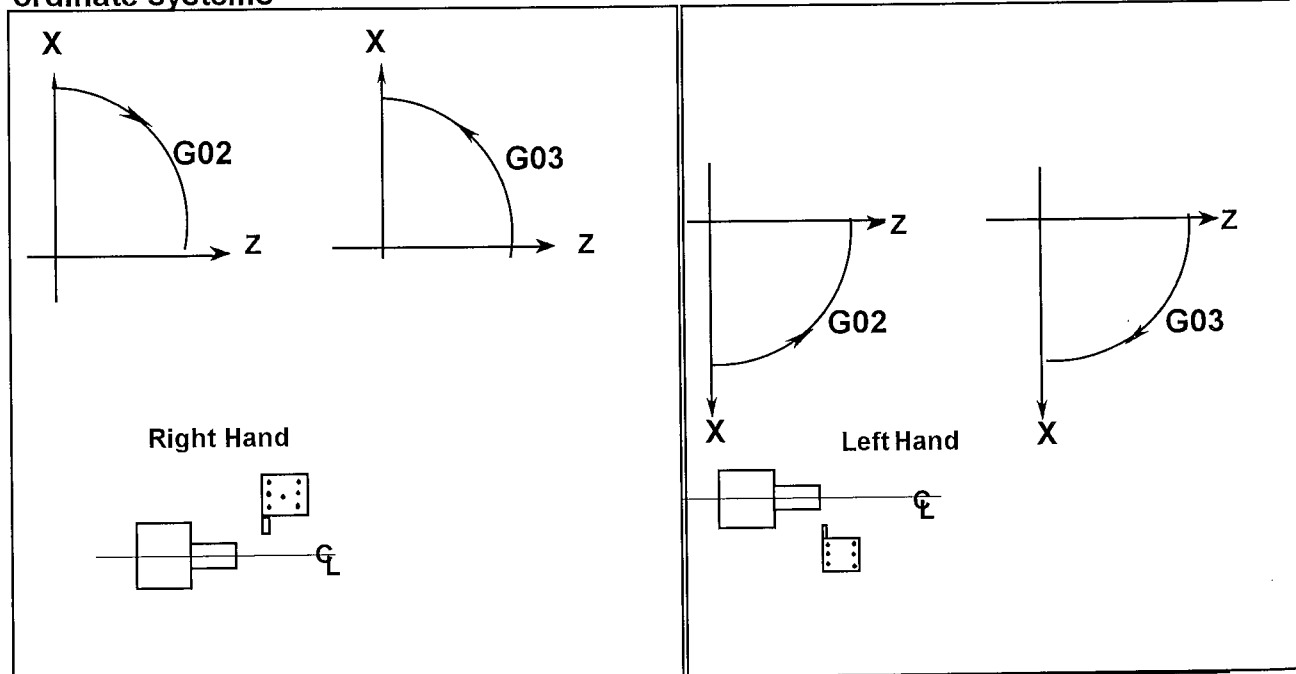
G02X ---- Z ---- I ---- K ---- F ----;
(absolute programming)

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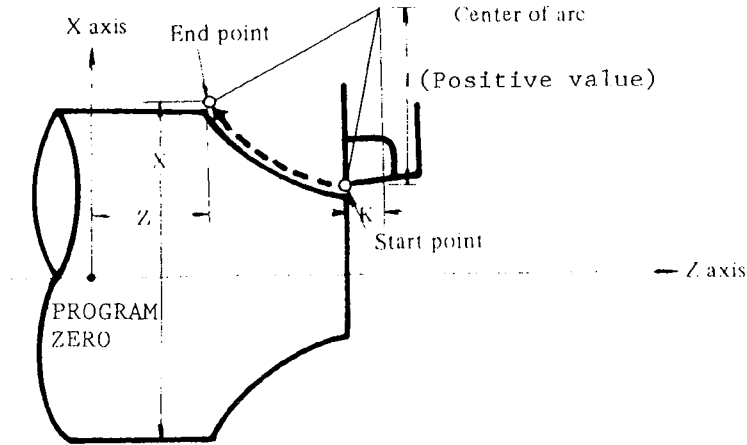
G03X ---- Z ---- I ---- K ---- F ----;
(absolute programming)

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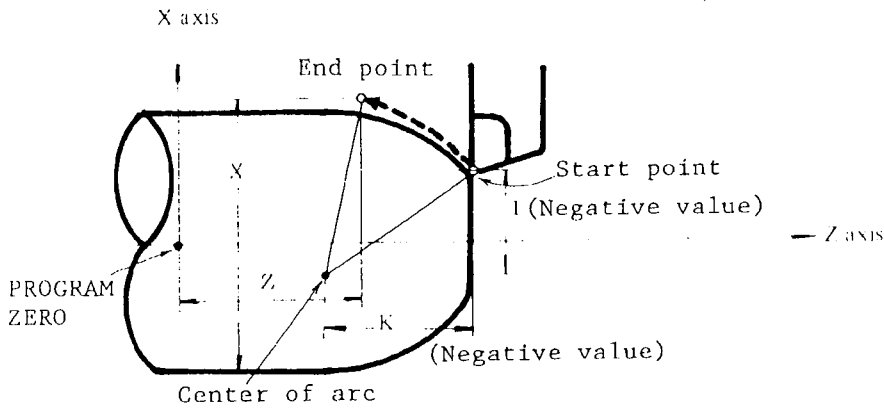
G02X ---- Z ---- R ---- F ----;
(absolute programming)

(Diameter programming)
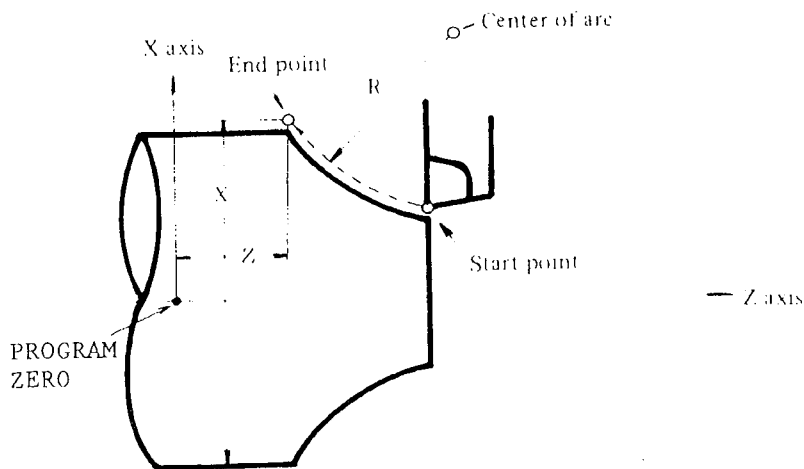
Program Example (Absolute Command)

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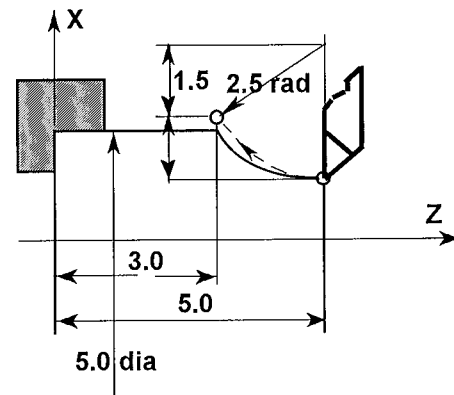
The feedrate for circular interpolation is specified by the F Code. The feedrate along an arc (tangent to an arc) is controlled to maintain the specified feedrate.

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.
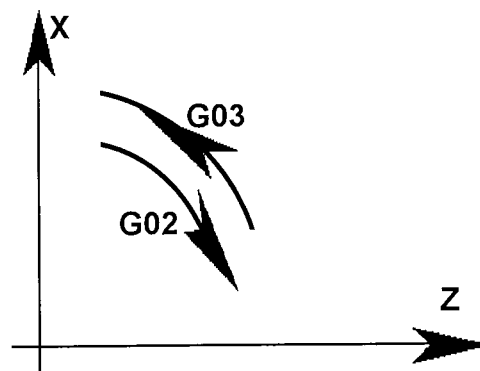
The end point of an arc is specified by address X, Z, or U, W, and is expressed as an absolute or incremental value. For the incremental value, the co-ordinate of the end point which is viewed from the start point of the arc is specified. The Arc Centre is specified by the address I, and K for the X 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.
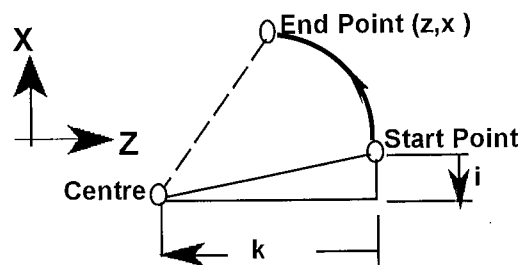
I and K must be aas signed according to the direction.

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;

X

②

End Point

Z

R = 50mm

①

Start Point

R = 50mm
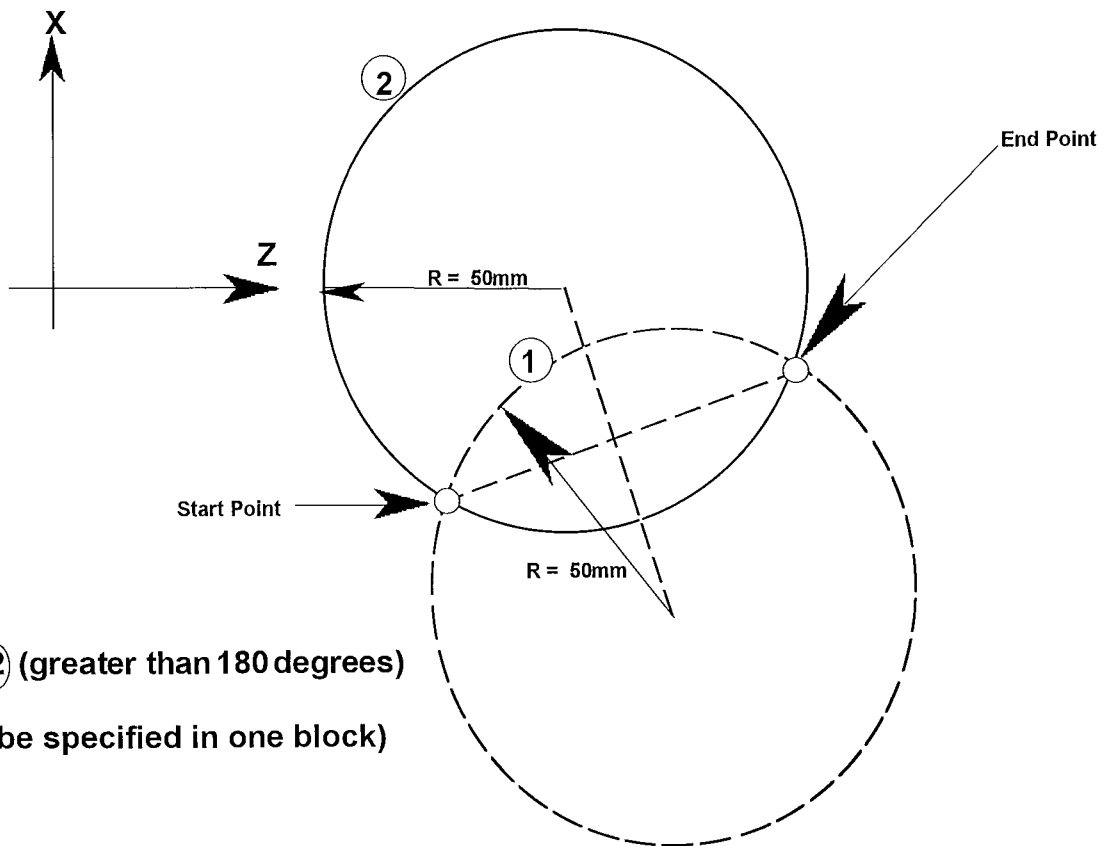
For arc ② (greater than 180 degrees)

(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.

```
┌─────────── CONTROL ───────────┐
│                               │
│   EXECUTE CNC                 │
│                               │
│   DRY RUN                     │
│                               │
│   TOOL OFFSETS                │
│                               │
│   LOAD TOOL OFFSETS           │
│                               │
│   SAVE TOOL OFFSETS           │
│                               │
│                               │
└───────────────────────────────┘
```
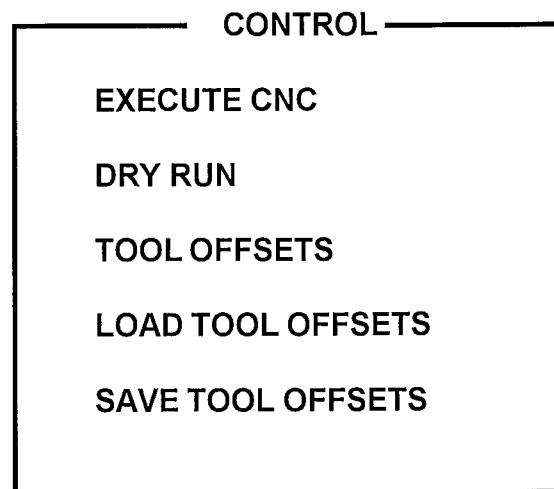
EXECUTE CNC.

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:-

MAIN PROGRAM   SUB-PROGRAMMS

01234	05678	08012
M98 P5678	M98 P8012	
M30	M99	M99

NESTED TO FOUR LEVELS

M98

SUB PPROGRAM CALL

P9999 ⟶ 5678

No

No. OF REPEATS
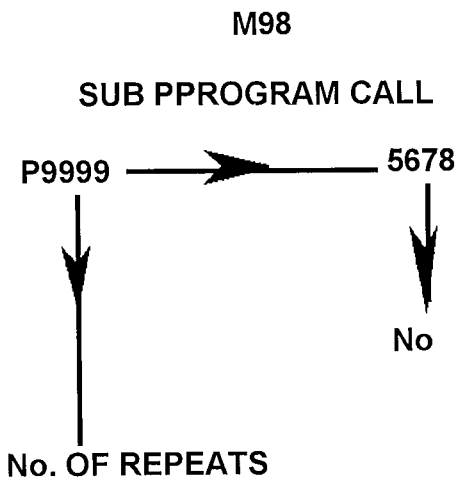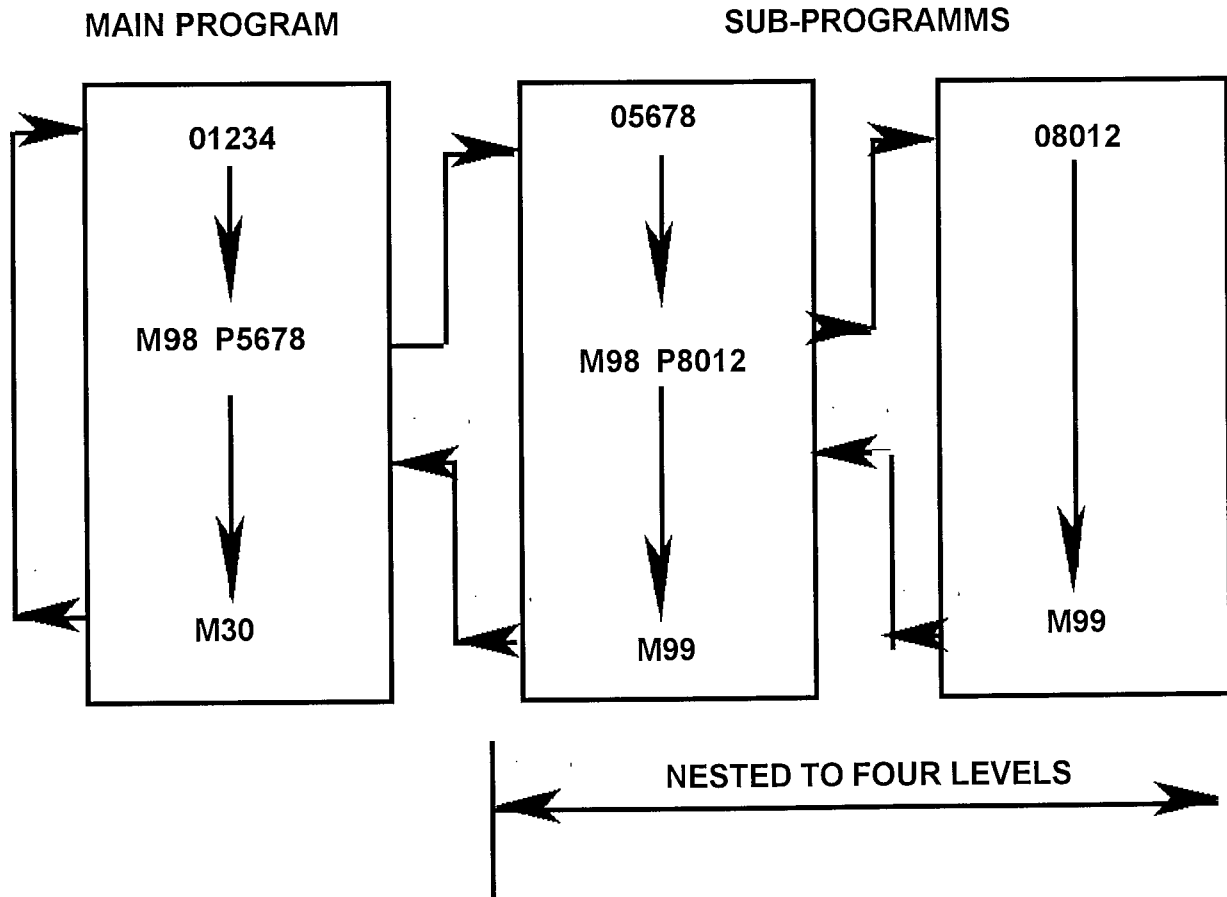
See page 18.1 for further details.

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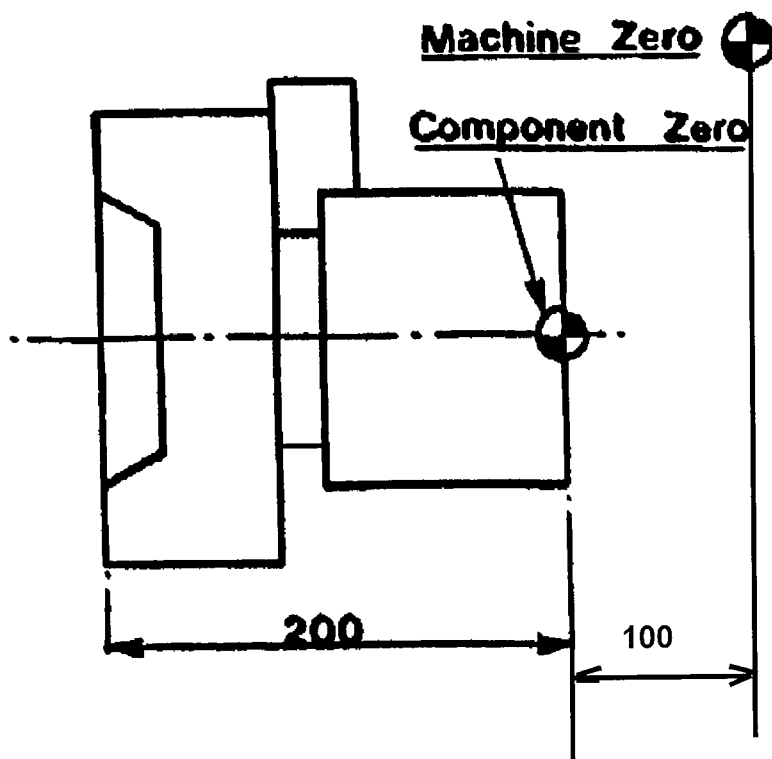
For the current operation, a zero position which has been selected for a first operation may not be the required position for the second operation.

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:-

```
T  0  3  0  3
```

TOOL SELECT CODE   TOOL NO.   OFFSET NO.
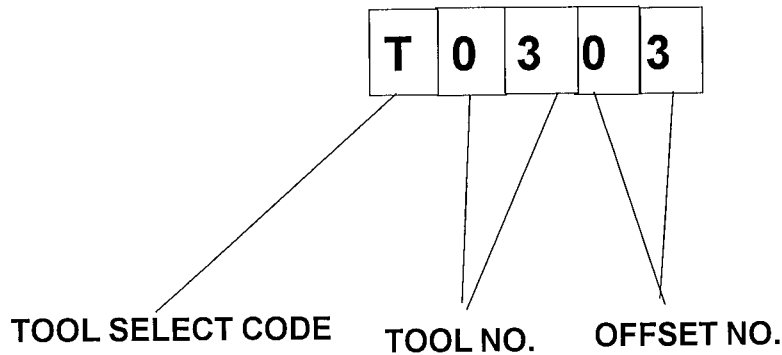
If the digits following the Tool Selecton Code (T) is less than 100 i.e. two digits, then the Tool Number and the Offset Number are the same.

Example:-

```
T  O  3
```

TOOL SELECT CODE   TOOL NO. & OFFSET NO.
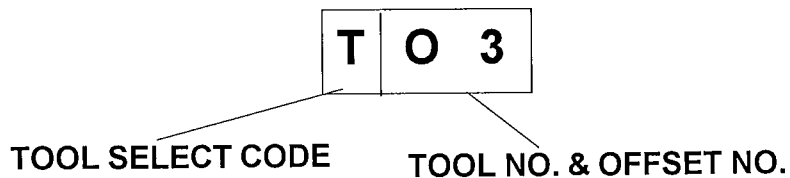
	Tool	Offset
T3	3	3
T03	3	3
T103	1	3
T0103	1	3
T0613	06	13
T0100	1	0 (no offset)

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.

DATUM

MACHINE

Z

X

X AND Z ZERO
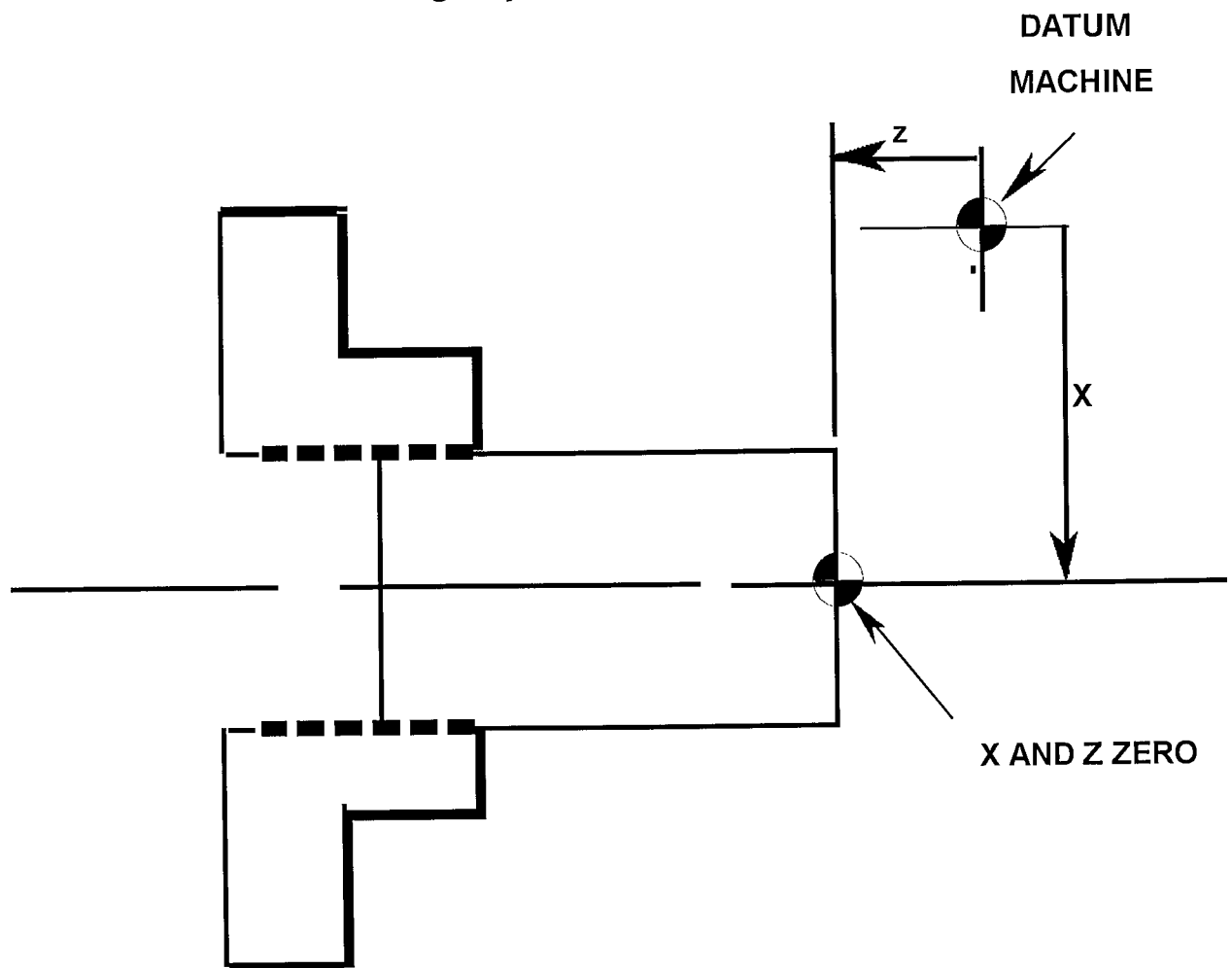
IMAGINARY TOOL NOSE
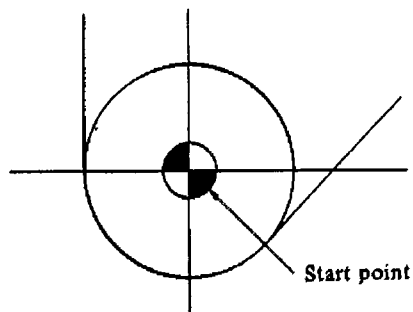
When programmed using the tool nose center
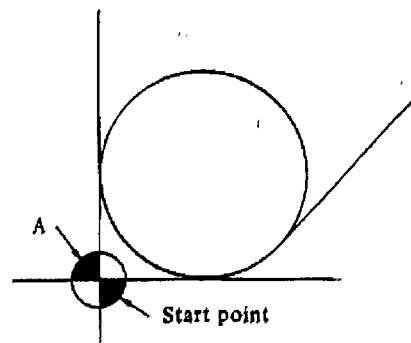
When programmed using the imaginary tool nose

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

POINT B

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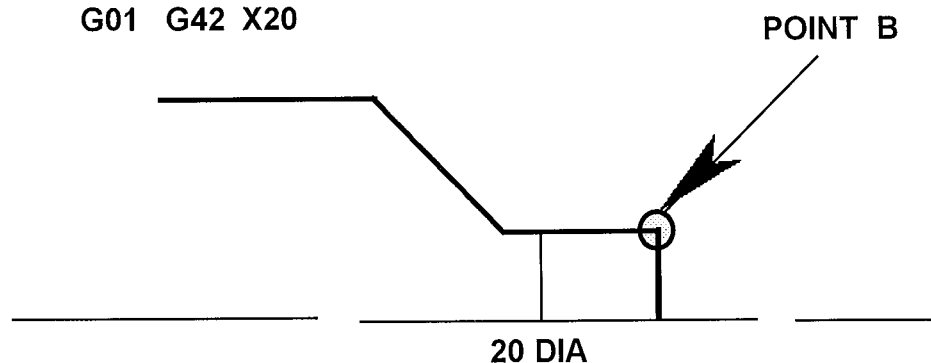
In the linear move before entering the TOOL NOSE COMPENSATION, always add on a value more than the radius of the tool.

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

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

DIRECTION OF IMAGINARY TOOL NOSE.

Imaginary tool nose
number 1

Imaginary tool nose
number 2

Imaginary tool nose
number 3

Imaginary tool nose
number 4

Imaginary tool nose
number 5

Imaginary too
nose number 6

Imaginary tool nose
number 7

Imaginary tool nose
number 8
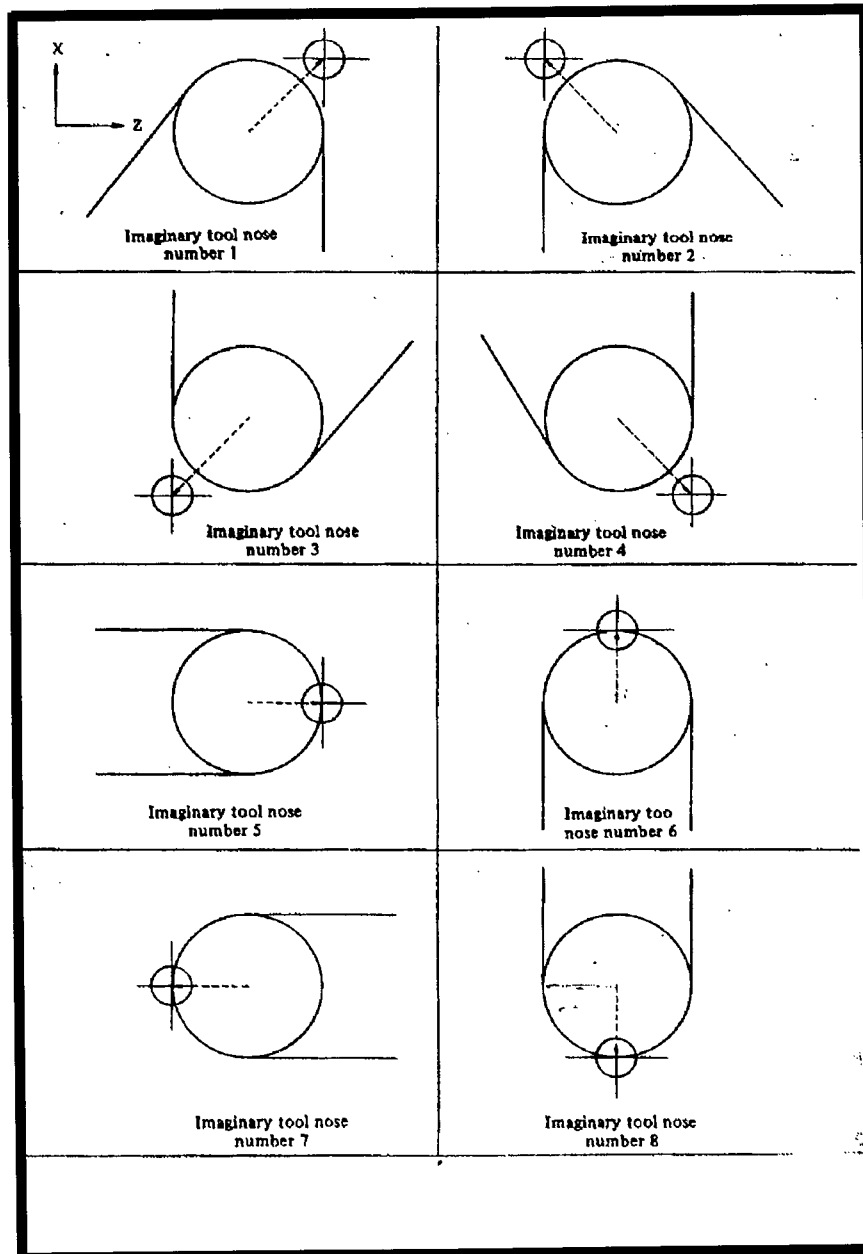
TYPICAL PART PROGRAM

The component shown below is to be faced and drilled with the tools shown.
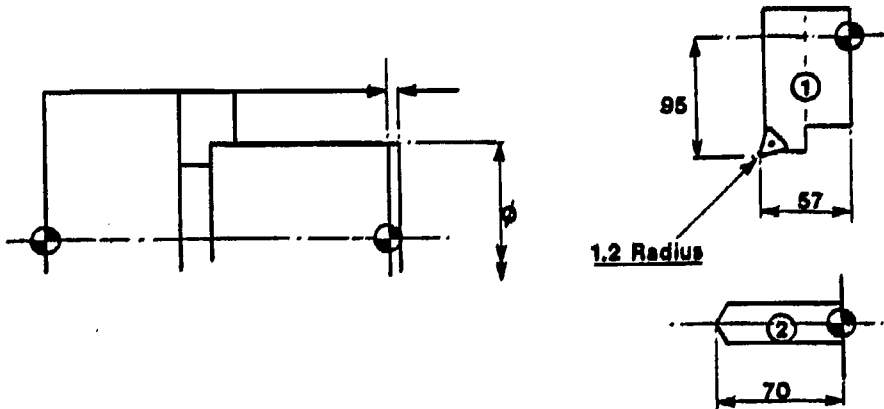
O0002 (TRAINING)

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.

Main Program	Sub-Program	Sub-Program
↓	↓	↓
O00005	0241	0340
↓	↓	↓
M98 P0241	M98 P20340	M30 M99
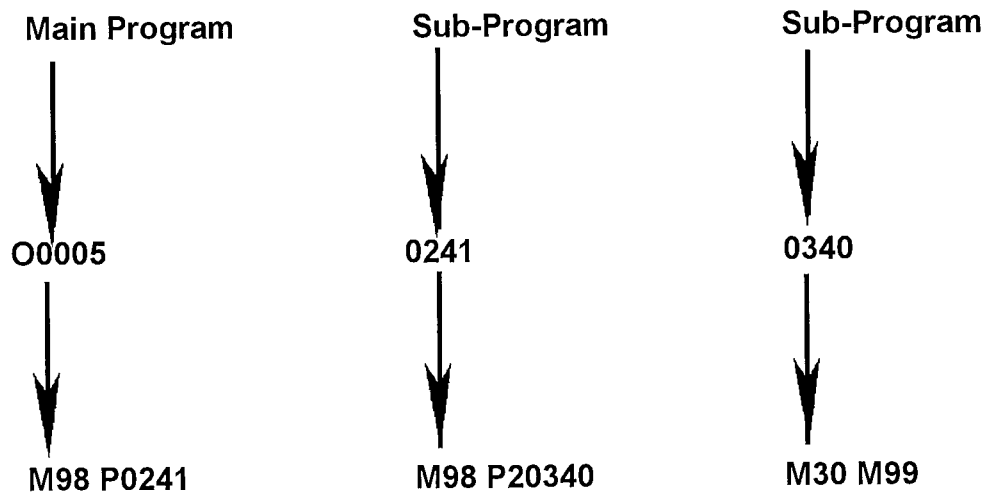
This repeats program 0340 twice.

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

```
            :0010
              |
              ↓

        N10 M00  ──────◄──────┐
              |               |
              |               ↑
              ↓               |
        N980 M99 P10 ────►────┘
```
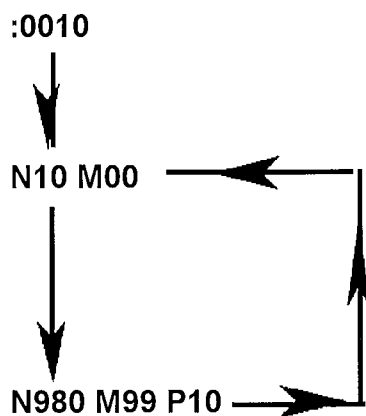
M99 generally indicates the end of a subroutine and allows the jump back to the main program. If it is used with "P" address, this indicates the "Jump To" block number.

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
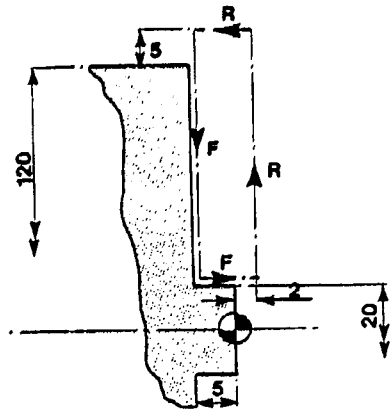
N500 G00 X130.00 Z2.0

N510 G94 X20.0 Z-5.0F0.3
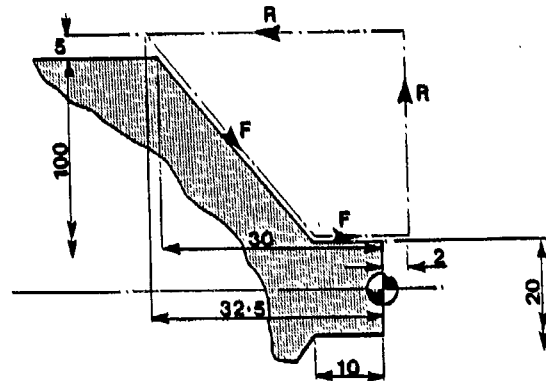
N400 G00 X130.00 Z2.0

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

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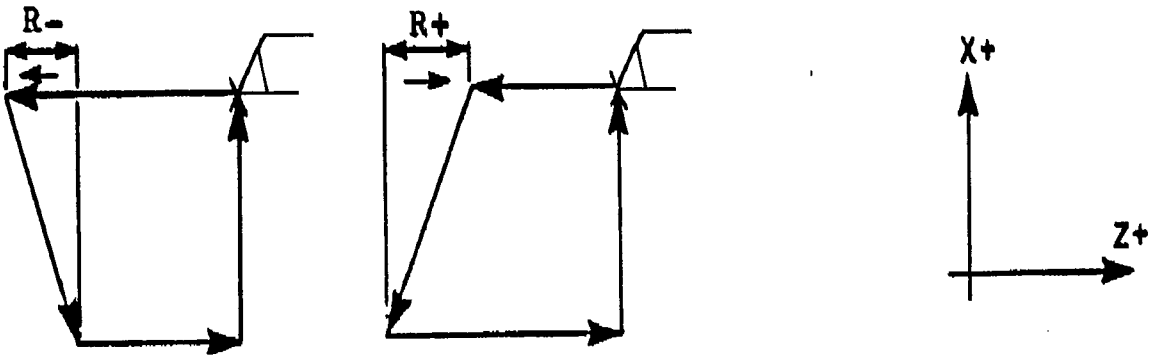
The G94 function in the above figure is a single "box type" cycle. One line of program information will enable the tool to perform the 4 distinct moves.

X - Final cutting position.

Z - Final cutting position.

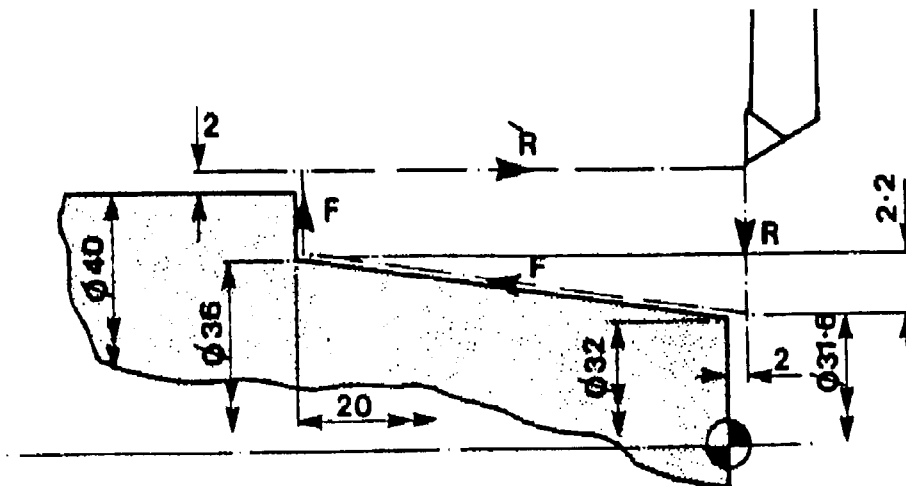R - Incremental distance to start of cut from final cutting position.

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

G90 TURNING CYCLE
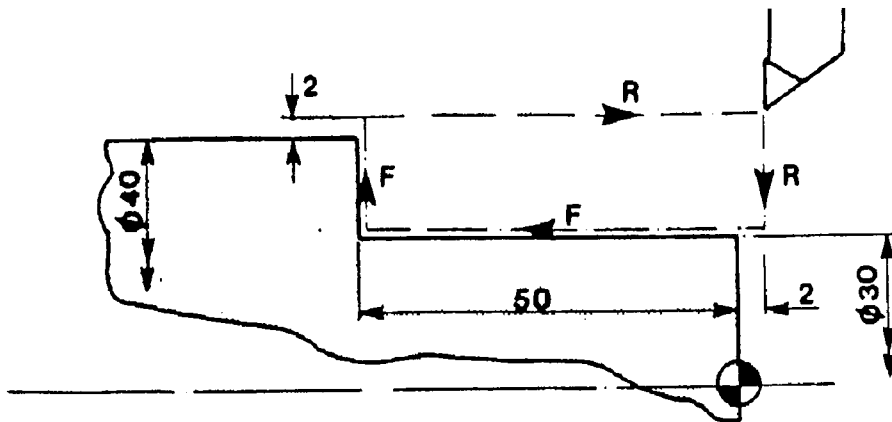
N600 G00 X44.0 Z2.0

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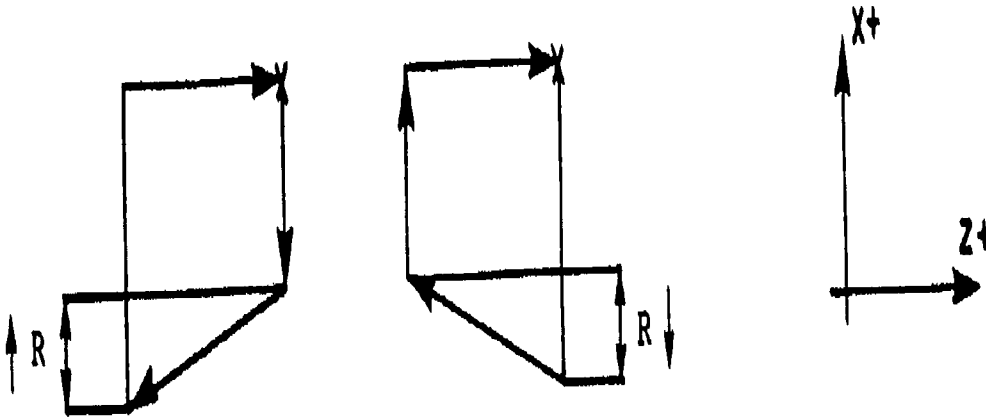
The G90 in the above figure is a single "box type" cycle. One line of program information will enable the tool to perform the four distinct moves.

    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.
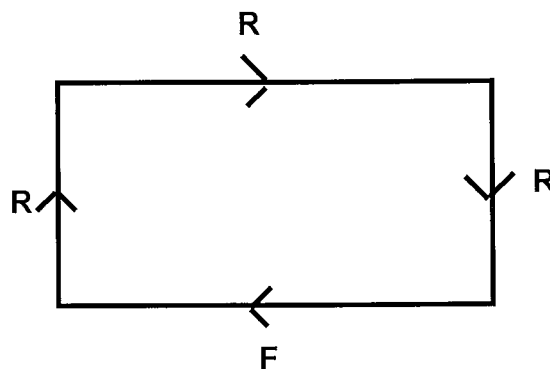
CANNED CYCLE (G92) - THREADING
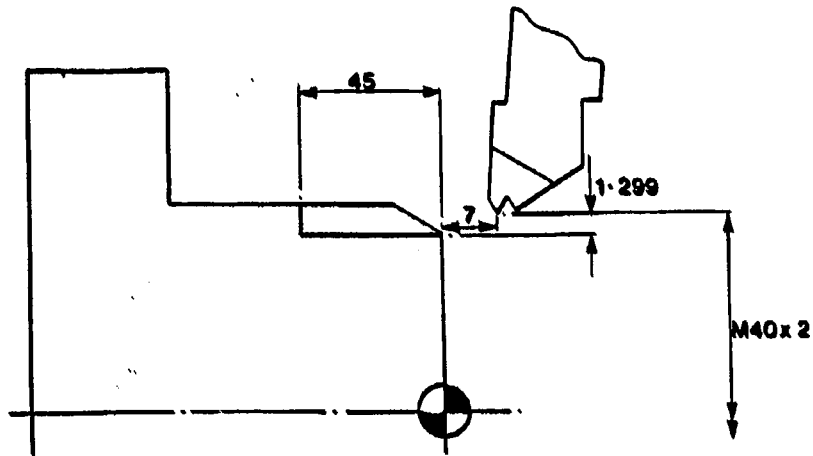
N690 M06 T0505

N700 M13 G00 X50.0 Z7.0

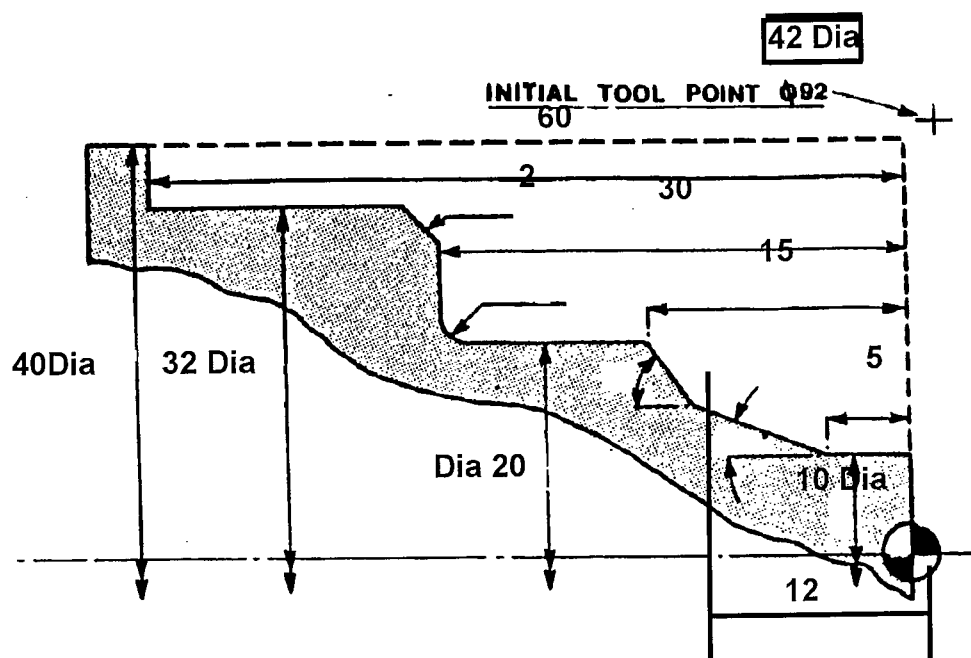**N710 G92 X39.35 Z-45.0 F2.0**

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