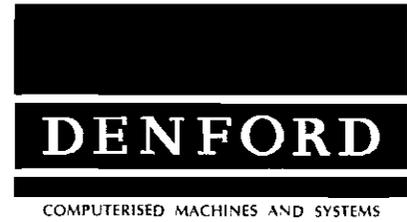


Denford Novaturn



Operating Guide for the Novaturn Series of CNC Lathes

- Installation
 - Specific
Features
 - Routine
Maintenance
-

This manual applies only to the machine having the serial number shown below.

Please note that this number will be required should Denford Limited be contacted regarding this machine.

Machine Serial Number : _ _ _ _ _

Year of Manufacture : _____



Manufactured by
Denford Limited,
Birds Royd, Brighouse, West Yorkshire, HD6 1NB, England.
Telephone: +44 (0)1484 712264.
Fax: +44 (0)1484 722160.
Email: service@denford.co.uk

CONTENTS.

Section	Page
Introduction / Warning	2
EC Declaration of Conformity	3
Unpacking & Lifting the Machine	4
Levelling & Positioning the Machine	5
Connecting the PC	6
Electrical Diagrams and Control Box Seal	6
Connecting the Mains Supply	7
Component Connection Schematic Diagrams ...	8
Removal of Protective Coatings	10
NovaTurn - General Layout	10
Install and Run the Machine Control Software ..	11
Running the Control Software	12
Automatic Search for Datum Point.....	12
Lubrication Chart	13
Maintenance Schedule.....	13
Lubrication Points on the Novaturn	14
Novaturn Specification	15
Denford Contacts, Products and Services	16

INTRODUCTION.

This guide will describe how to transport, site and setup your Denford Novaturn CNC lathe.

Any operational features, specific to the Novaturn range, are also covered in this guide. General operating functions are explained in the separate "Generic CNC Turning Manual" delivered with your machine.

A Routine Maintenance section is also included. Please note, the Electrical Diagrams for your machine are held in a folder fixed inside the electrical control box.

IF YOU HAVE ANY DOUBTS AND/OR QUESTIONS REGARDING THE SPECIFICATION, SERVICING OR FEATURES OF YOUR MACHINE, PLEASE CONTACT CUSTOMER SERVICES AT DENFORD.

WARNING.

The Warranty on this machine will be invalidated if any modifications, additional ancillary equipment is fitted, or any adjustments made to the controlling devices without prior notification from Denford Limited.

Do not carry out any portable appliance testing (PAT) on any of the supplied equipment.

EC DECLARATION OF CONFORMITY.

The responsible person :

Business Name :

Denford Limited.

Address :

Birds Royd,
Brighouse,
West Yorkshire,
HD6 1NB,
England.

Declares that the machinery described :

Manufacturer :

Denford Limited.

Model Name :

Novaturn

Serial Number :

conforms to the following directives :

EC Machinery directive 89/392/EEC as amended by directive 91/368 EEC and directive 93/44/EEC, CE marking directive 93/68/EEC and low voltage directive 73/23/EEC

and the following standards :

BS EN 60204 - 1 : 1993

and complies with the relevant health and safety requirements.

Signature :

Position within company :

DIRECTOR/GENERAL MANAGER

Signed at :

Denford Limited,
Birds Royd,
Brighouse,
West Yorkshire,
HD6 1NB,
England.

UNPACKING & LIFTING THE MACHINE.

Cut the top of the delivery box open and remove the styrofoam packaging carefully.

To obtain better access to the machine and the power supply box, remove all the sides from the delivery box, leaving the machine standing on its wooden delivery pallet.

Lift the power supply control box from the packaging. If possible, lift the power supply box using a porters trolley.

Lift the machine from the packaging. Denford does NOT recommend direct lifting of the machine !! Always use sensible lifting precautions in accordance with Health and Safety Regulations in your establishment.

Denford recommends using two 5 metre long slings and a professional hoist, arranged as shown in the diagram below.

Before fitting the slings, remove the guard. Do not lift the machine with the guard still in place - the pressure of the slings may cause the guard to crack. The guard is held in position using two nuts and bolts. Some machines will also have a side mounted safety interlock switch (this should be set "open" on delivery of the machine to allow the guard to be removed, but must be set "closed" when the machine operates). Ensure that the machine is both secure and balanced before lifting.

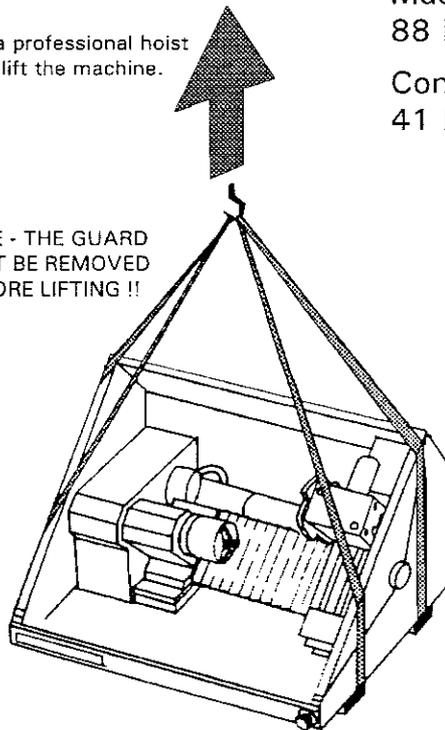
To transport the machine over longer distances, use a suitably sized trolley. Remember to refit the guard when the machine has been sited.

Use a professional hoist to lift the machine.

Machine Weight :
88 kgs (195 lbs)

Control Box Weight :
41 kgs (90 lbs)

NOTE - THE GUARD
MUST BE REMOVED
BEFORE LIFTING !!



Wrap two reinforced
straps round the
machine as shown.

LEVELLING & POSITIONING THE MACHINE.

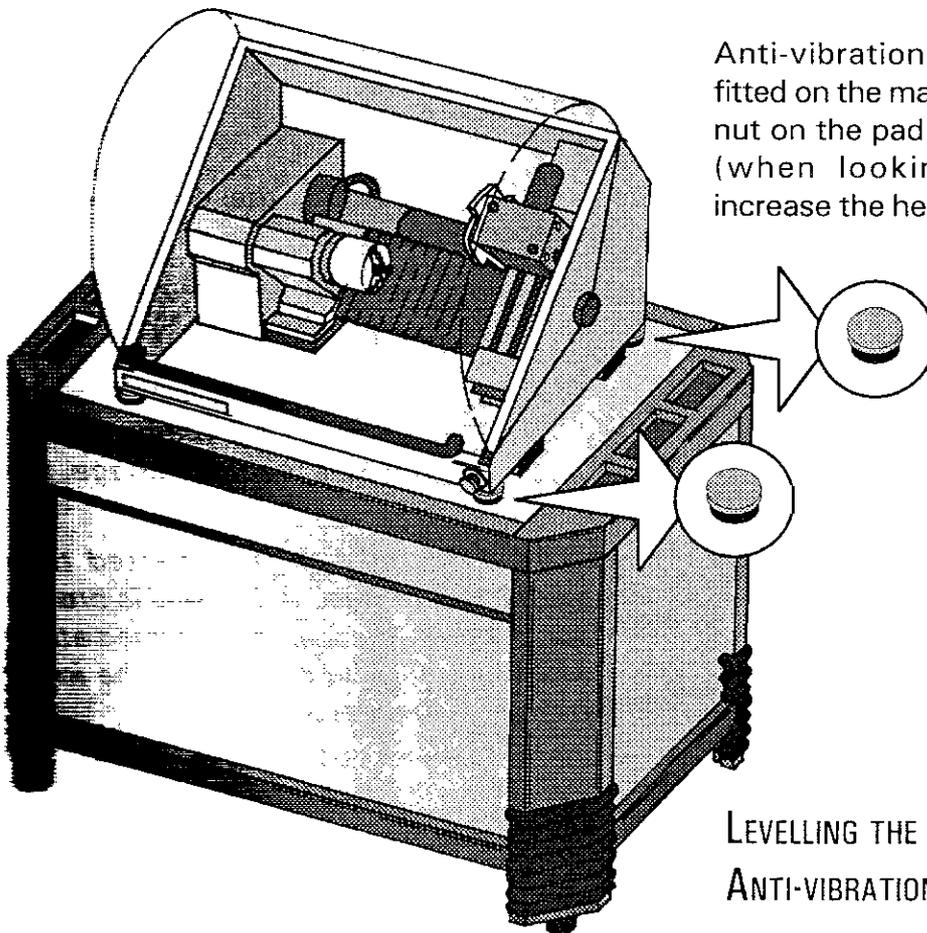
Remember when positioning the machine in the room, space will be required for the electrical control box. Sufficient room should also be provided for effective maintenance to be carried out.

The Novaturn is a bench mounted machine, so it should be sited on a bench of sturdy construction to take the weight of the machine, and of a height which enables comfortable operating and programming to take place.

The machine should rest level on the two hollow sections which run beneath the machine cabinet. The lathe itself has been levelled to the machine cabinet prior to dispatch, so it is only necessary to level the machine to the table on which it is to be situated.

If the machine is not stable resting on these two hollow sections, insert four anti-vibration pads under the sections at each corner of the machine - as illustrated in the diagram below. Adjust the pads until the machine is stable and level. The pads will also help to reduce the amount of noise and vibration generated when the machine is operating.

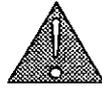
Note - the pads are *ONLY* used to *help* stabilise the machine, the main weight of the machine should still be taken by the hollow sections (ie, these sections should *ALWAYS* be in direct contact with the table surface).



Anti-vibration pads can be fitted on the machine. Turn the nut on the pad anti-clockwise (when looking down) to increase the height of the pad.

LEVELLING THE MACHINE USING
ANTI-VIBRATION PADS.

CONNECTING THE PC.



Warning! Do not connect cables between any electrical hardware with the mains power switched on, since this could damage the hardware.

Ideally, the pc (personal computer) should be placed next to the machine and its electrical control box, with access to a mains power supply.

Connect the elements of the pc together as described in the pc manufacturers operating manual.

The pc and the machines electrical control box are physically linked by the RS 232 cable (supplied). This cable is fitted with 25 pin connectors at both ends.

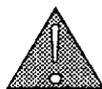
One end connects to the 25 pin port on the electrical control box labelled "RS 232".

The opposite end of this cable connects to the 25 pin port of the pc labelled "COM2". If this port cannot be identified on the back panel of the pc, please refer to the pc manufacturers operating manual.

Note. On some personal computers, this port may have a 9 pin connection. If this is the case, a 25 pin to 9 pin adapter, available from most good computer/electrical retailers must be fitted to allow the cable to be connected to the pc.

A schematic diagram illustrating these component connections is shown on page 8.

ELECTRICAL CONTROL BOX SEAL.

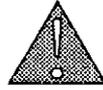


Warning! Do not connect cables between any electrical hardware with the mains power switched on, since this could damage the hardware.

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.

CONNECTING THE MAINS SUPPLY.

The large flexible machine power cable, leading from the side of the electrical control box is connected to the fixing bracket on the side of the lathe. Check that the connector is inserted into the bracket with the main cable from it leading towards the back of the machine (see the diagram on page 9).



Warning! Do not insert the connector into the bracket the wrong way round; this could damage the connector pins. Check that the letters and numbers on the two halves of the connector and bracket match before closing the junction.

Ensure the two roller clips holding the male connector against the female bracket are fully closed. There should be no free movement at the junction.

The mains power supply is fed to the electrical control box, which in turn, is connected to the lathe.

This electrical control box is delivered with the mains supply cable connected directly into the isolator with approximately 3 metres of cable. The cable should then be fitted with a standard 13 amp. plug suitable for the mains power supply.

The supply is 220/240volt Single Phase 50Hz.

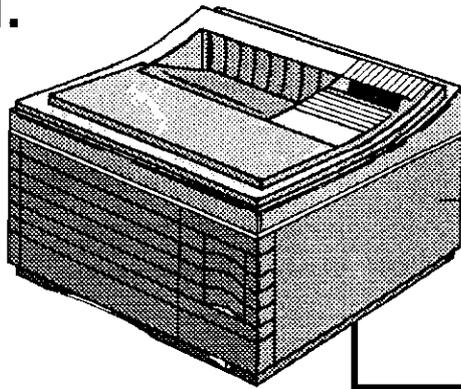
Cable required:- 2 Core + Earth, 1.5mm per core.

Current Taken 10 Amps.

All electrical connections should only be made by suitably qualified electrical engineers.

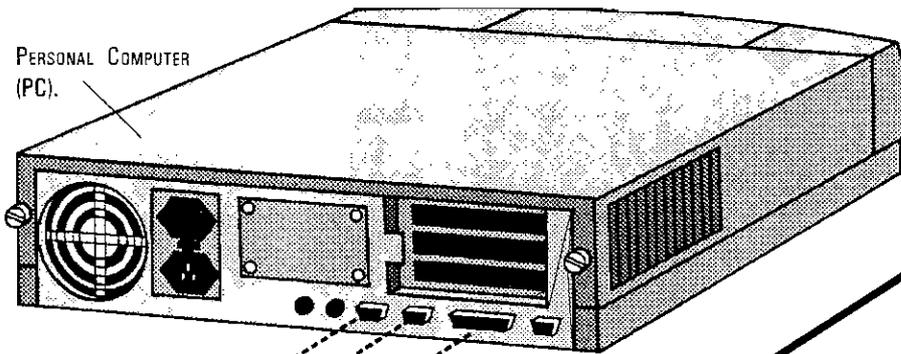
A schematic diagram illustrating these component connections is shown on page 8 and 9.

COMPONENT CONNECTION SCHEMATIC DIAGRAM.

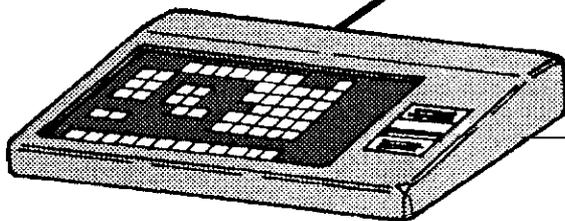
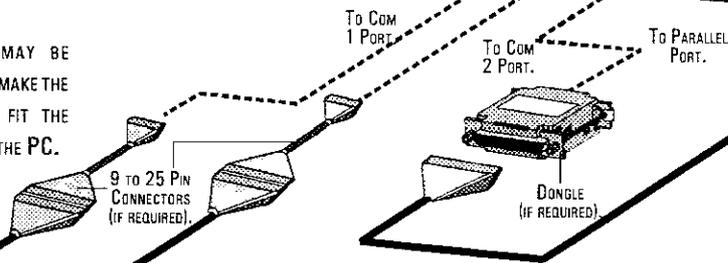


ANCILLARY EQUIPMENT, SUCH AS THIS
PRINTER, CONNECT TO THE PARALLEL
PORT ON THE PC.

PERSONAL COMPUTER
(PC).



ADAPTERS MAY BE
REQUIRED TO MAKE THE
CONNECTORS FIT THE
SOCKETS ON THE PC.



DESKTOP TUTOR
CONNECTS TO THE PC
PORT LABELLED COM 1.

THE RS232 CABLE CONNECTS
THE MACHINE ELECTRICAL
CONTROL BOX TO THE PC PORT
LABELLED COM 2.

COMPONENT CONNECTION SCHEMATIC DIAGRAM.

DENFORD NOVATURN
CNC LATHE.

ENSURE THIS CONNECTION IS
FITTED IN THE CORRECT
ORIENTATION (THE FLEXIBLE
BLACK CABLE SHOULD EXIT
TOWARDS THE BACK OF THE
MACHINE).

MACHINE ELECTRICAL
CONTROL BOX MAIN
CONNECTOR.

MAINS POWER
SUPPLY PLUG.

LARGE, FLEXIBLE
CORRUGATED CABLE.

THE ~~RS232~~ CABLE CONNECTS
THE MACHINE ELECTRICAL
CONTROL BOX TO THE PC PORT
LABELLED COM 2.

TO THE PORT LABELLED
"RS 232".

MACHINE ELECTRICAL
CONTROL BOX
(NOTE - THE PHYSICAL
LAYOUT ON INDIVIDUAL
BOXES MAY VARY).

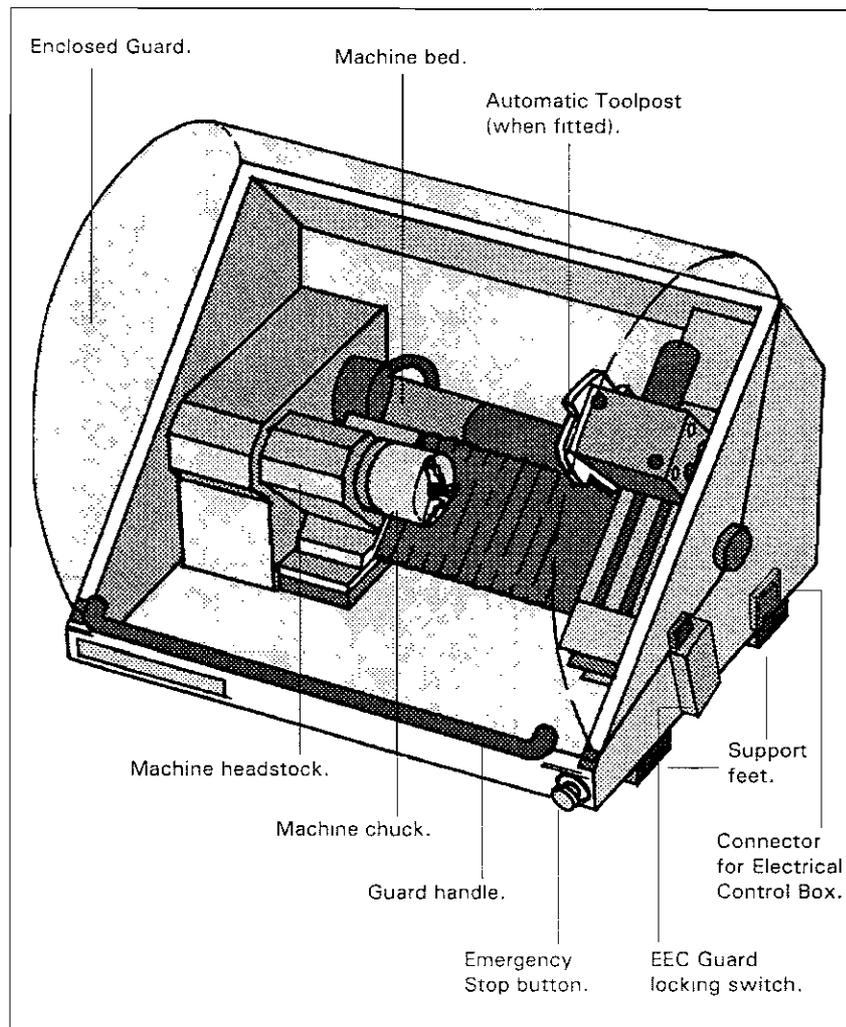
REMOVAL OF PROTECTIVE COATINGS.

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

The protective coatings applied to the slideways and bright surfaces can be removed using a kerosene based solvent. The coatings must be removed from the slideways before any attempt to move them is made. Once these protective coatings have been removed, all untreated surfaces should be coated with a light covering of machine oil. (e.g. BP: CS 68).

The protective plastic sheeting on the guard windows should be removed and the perspex cleaned with an anti-static cleaner.

NOVATURN - GENERAL LAYOUT.



INSTALLING & RUNNING THE MACHINE CONTROL SOFTWARE.



To install the machine control software onto your pc hard disk, switch on your pc and run/type the following:

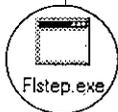
A:\Install.exe

....where **A:** is the label for the floppy (3.5 inch) disk drive.

Press the [ENTER] / [RETURN] key on the pc qwerty keyboard to confirm this command.

The install routine will ask you for a drive destination (the label assigned to your pc hard disk - the default is set as **C:**) and a directory destination (the name of the area/folder you would like the software to be stored - the default is set as **\NOVATURN**).

The machine control software is an MS-DOS application.



Each time you wish to run the machine control software from your pc hard disk, exit, if necessary, to the 'DOS' prompt and type the following:

C:

....where **C:** is the label for the drive where the software is installed, then press the [ENTER] / [RETURN] key on the pc qwerty keyboard. Next, type:

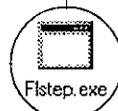
CD\NOVATURN

....where **NOVATURN** is the directory where the software is stored and press the [ENTER] / [RETURN] key on the pc qwerty keyboard. Then type:

FLSTEP

Press the [ENTER] / [RETURN] key on the pc qwerty keyboard and the software will start.

Alternatively, the software can be run directly off one floppy (3.5 inch) disk.



To load the machine controlling software from a floppy (3.5 inch) disk, switch on your pc and exit, if necessary, to the 'DOS' prompt.

Type the following line at the 'DOS' prompt:

A:\FLSTEP

....where **A:** is the label for the floppy (3.5 inch) disk drive.

Press the [ENTER] / [RETURN] key on the pc qwerty keyboard to confirm this command.

The machine control software will now start.



Note - The default configuration in the software is for the desk-top tutor to connect to COM 1 and the CNC machine to COM 2.

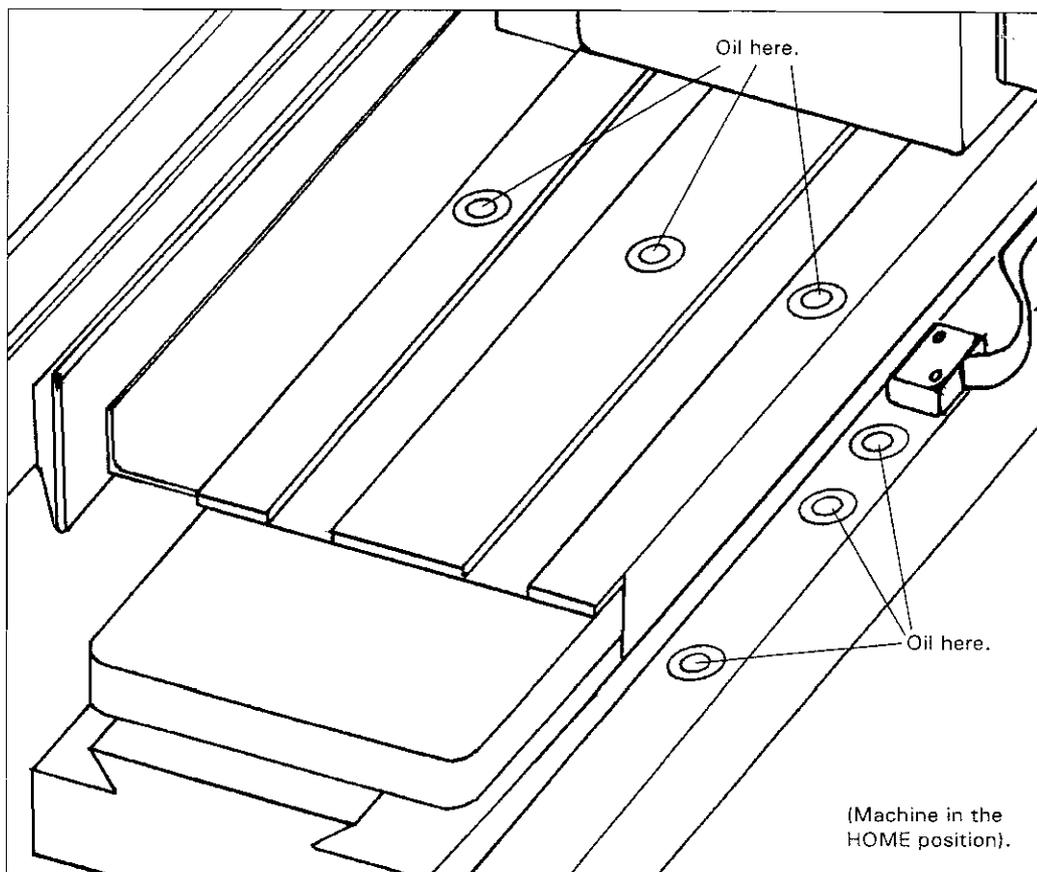
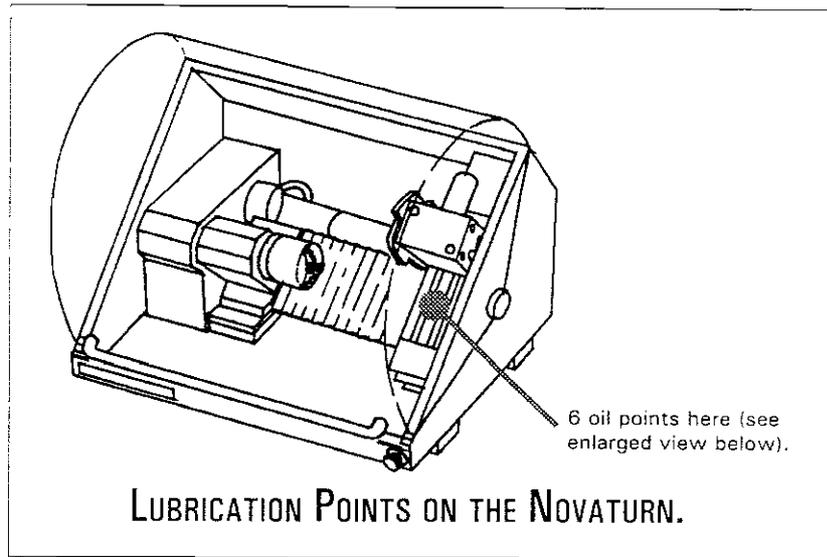
MAINTENANCE SCHEDULE.

<i>Daily</i>	<ul style="list-style-type: none"> - Clean and remove swarf. - Check that the slides are lubricated.
<i>Weekly</i>	<ul style="list-style-type: none"> - Clean machine. - Oil the slides and ballscrews. - Check exposed screws and nuts for tightness.
<i>Bi-annually</i>	<ul style="list-style-type: none"> - Check condition of electrical connections. - Check all cables for kinks and breaks. - Clean sensors and microswitches. - Remove chuck jaws, clean chuck.
<i>Annually</i>	<ul style="list-style-type: none"> - Check slides for wear. - Grease axis bearings

LUBRIACATION CHART.

Lubrication Point	Lubricating System	Frequency	Recommended Oil/Grease	Quantity
Slide ways and Ballscrews	Pump-action oil can	Weekly	BP : CS 68 Shell : Vitrea 68 Castrol : Perfecto NN	As required
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	2 cc/Bearing
Coolant			Cincinnati Millacron Simcool 60 Dilution 2.5%	

LUBRICATION POINTS ON THE NOVATURN.



CROSSLIDE SLIDEWAYS AND TABLE BALLSCREW.

The cross slideways can be lubricated by pumping oil into the oiling points on the top and side of the crossslide - see above.

Easy access to the oiling points is obtained by homing the machine axes. Oil can then be pumped directly into the oiling points.

Note - Always run the slides to distribute the oil along the slideways.

NOVATURN SPECIFICATION.

MECHANICAL.

Swing over Bed	160mm (6.25")
Swing over Cross Slide	80mm (3.125")
Distance between Centres	270mm (10.75")
Spindle Speed (Stepless)	0-3500 rpm
Spindle Bore	20.6mm (0.8125")
Spindle Taper	No. 3
45 degree slant bed design :	
X Axis Travel	140mm (5.5")
Z Axis Travel	225mm (8.75")
Z Axis with Tailstock	175mm (6.875")
X Axis Ballscrew	16mm dia. x 5mm pitch
Z Axis Ballscrew	16mm dia. x 5mm pitch
Machine Resolution	0.01mm (0.0004")
Machine Length	712mm (28")
Machine Width	690mm (27")
Machine Height	560mm (22")
Control Box Length	420mm (16.5")
Control Box Width	220mm (8.66")
Control Box Height	620mm (24.4")
Machine Weight	88 kgs (195 lbs)
Control Box Weight	41 kgs (90 lbs)

ELECTRICAL.

Mains Supply
50 60 Hz - Single phase - 220/240 Volts, 6 AMP

Spindle Motor:
0.746 Kw 1.6 H.P. DC

Axes Motor:
Stepper Motors - 200 steps/rev

DENFORD CONTACTS, PRODUCTS AND SERVICES

If you require specific help regarding the specification, operation or maintenance of this machine, contact Denford on the phone/fax number below. Please have the machine serial number and year of manufacture (printed on the front of this guide) to hand, when you call.

Telephone: +44 (0)1484 712264.

Fax: (01484) 722160.

Denford Limited,

Birds Royd, Brighouse, West Yorkshire, HD6 1NB, England.

Email: service@denford.co.uk

Stuck for projects and ideas?

Denford LatheCAM Designer is an easy to use CAD package specifically designed for use with Denford CNC lathes.

Components can be designed directly on-screen, or imported from other popular drawing packages. The G-code programs are then automatically generated by LatheCAM's post processor.

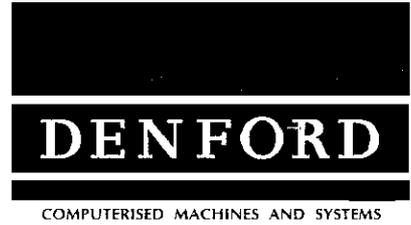
What simpler way is there of creating your own library of CNC files? !!

Need further training?

The Denford PTDC (Professional Training and Development Centre) is a purpose built centre specialising in project guidance, CNC machine training and software development skills for Denford customers. Training packages can be tailored to suit your needs, with the help of our experienced Education Support team. The centre can cater for training sessions from the very basics of CNC machine operation, upto the complexities of G-code programming, then further into 'new' Technology areas such as video conferencing.

Denford Limited is committed to the development of its training guides and manuals. If you have found certain sections in this setup guide useful, or feel that particular sections could be further developed, or new sections added in future, we would welcome your suggestions and comments.

Denford
Machining
Systems
Limited



Generic
Turning
Manual

Denford Limited
Birds Royd
Brighouse
West Yorkshire
HD6 1NB
Tel: + 44 (0)1484 712264
Fax: + 44 (0)1484 722160
EMail: service@denford.co.uk

CONTENTS.

SECTION	PAGE
Contents	0.0
Introduction	0.1
Start Machine and Software	1.1
Closing the Software	1.9
How to set-up LatheCAM Designer	1.11
Setting the Machine Datum	2.1
Clamping the Billet	3.1
Changing the Tool	3.3
Simulation Menu - Set Tooling	3.9
Setting the Tool Offsets	4.1
Setting the X and Z Offset values	4.3
Setting the Z co-ordinate value	4.5
Setting the X co-ordinate value	4.11
Setting the Offset values for other types of Tool ..	4.19
Control Menu	4.25
Control Menu - Load Tool Offsets	4.29
Control Menu - Save Tool Offsets	4.31
Changing the Drive Directory - Tool Offset Files ...	4.33
CNC Files Menu	5.1
CNC Files Menu - Load a CNC File	5.2
CNC Files Menu - Save a CNC File	5.6
Changing the Drive Directory - CNC Files	5.9
Simulate CNC File	6.1

CONTENTS.

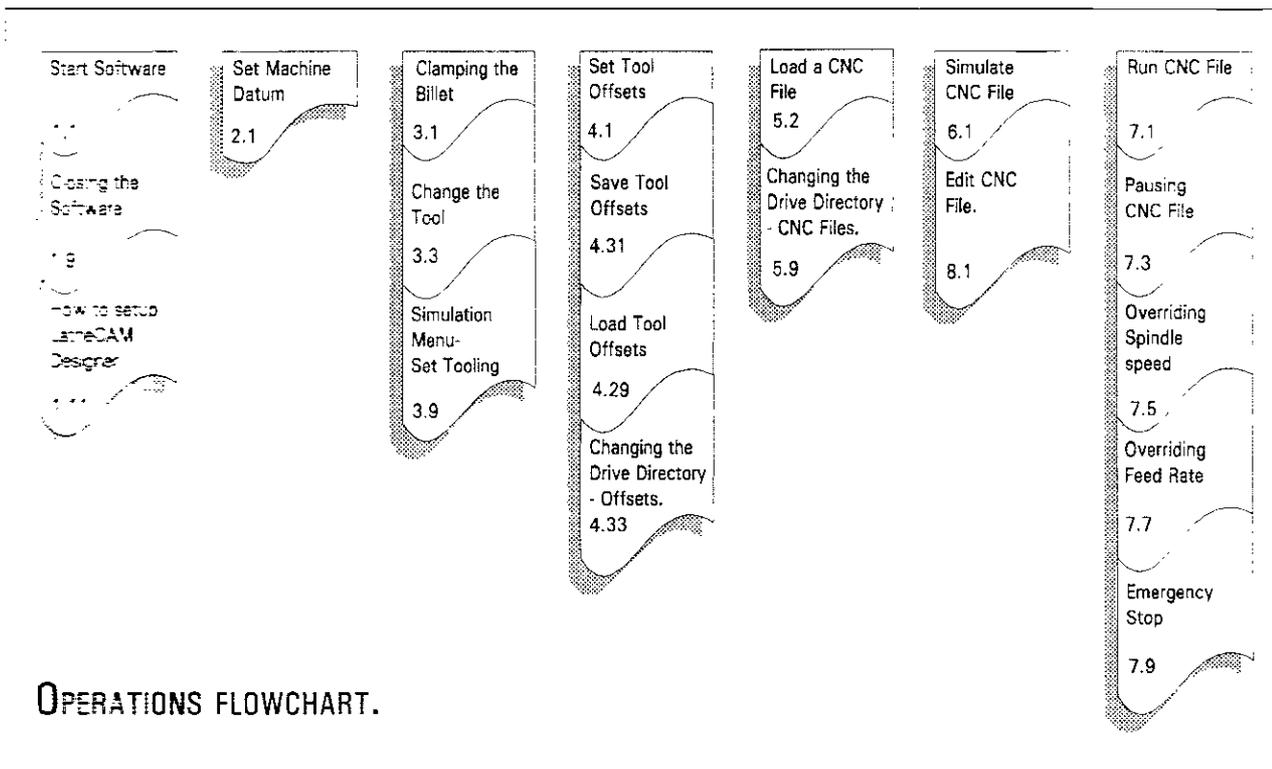
Run Program	7.1
Pausing Program	7.3
Overriding Spindle Speed	7.5
Overriding Feed Rate	7.7
Emergency Stop	7.9
Editing a CNC File	8.1
Main Menu - Machine Control	9.1
Main Menu - Print	9.2
Main Menu - Remote Link	9.4
Main Menu - Utilities	9.6
Change Settings	10.1
Change Settings - Editor	10.3
Change Settings - Simulation	10.8
Change Settings - Print Device	10.13
Change Settings - Print Page Layout	10.14
Change Settings - Remote Link	10.15
Change Settings - Miscellaneous	10.16
Change Settings - Load Settings	10.18
Change Settings - Save Settings	10.19
Change Settings - Delete Settings	10.20
What is a Part Program?	11.1
Listing of Address Characters	11.3
Listing of G Codes	11.4
Listing of M Codes	11.6
Denford Directives	11.8
Glossary	12.1

INTRODUCTION.

Using a new piece of hardware, such as your Denford CNC Lathe, can be quite daunting, especially if you have limited CNC experience or come from a non-engineering background! Unfortunately, many industrial machine manuals, though comprehensive, seem only to cater for those of us with either years of engineering experience, or time to thumb through acres of technical descriptions and confusing 'jargon'!

Although this manual is not aimed at the 'complete beginner' it does try to explain much of this 'jargon' so operations are easier to understand and follow.

All the operations and processes covered in this manual relate directly to the Denford series of CNC Lathes.



The Operations flowchart lists the progress of a typical operating procedure for a CNC Lathe. Sections in this manual correspond to each of the operations listed above.

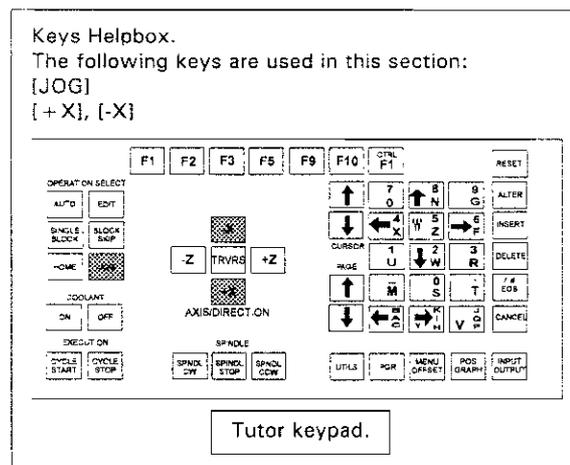
USING THE MANUAL.....

Conventions used in this manual follow this format :

[.....] - Square brackets with text show the individual keys to press, for example, [HOME] means press the Home key.

Italics. - Italics are used to show menu and text selections within the software.

Most sections of the manual contain the desktop tutor graphic, shown below. The keys required to complete each particular section are highlighted in grey. For example.....



The helpbox above would show that the [JOG], [+X] and [-X] keys are used at some point during that particular section.

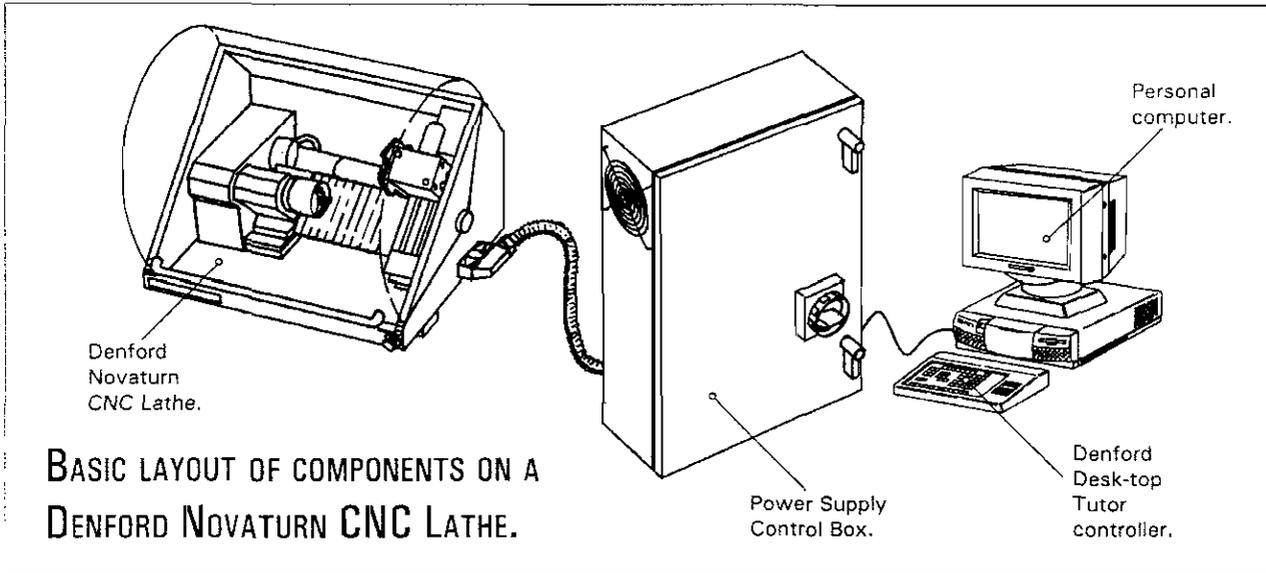
When relevant, screenshots and graphics are also provided to accompany the text. Specific screenshots and drawings of machines are only used for illustrative purposes. All the operations described in this manual relate to any Denford CNC Lathe.

Please note - the numerical figures depicted on certain screenshots (e.g. datum co-ordinates) may differ, according to the axis movement limits and settings of your machine.

Sections in the manual can be easily located using the indexing captions in the bottom corner of each page.

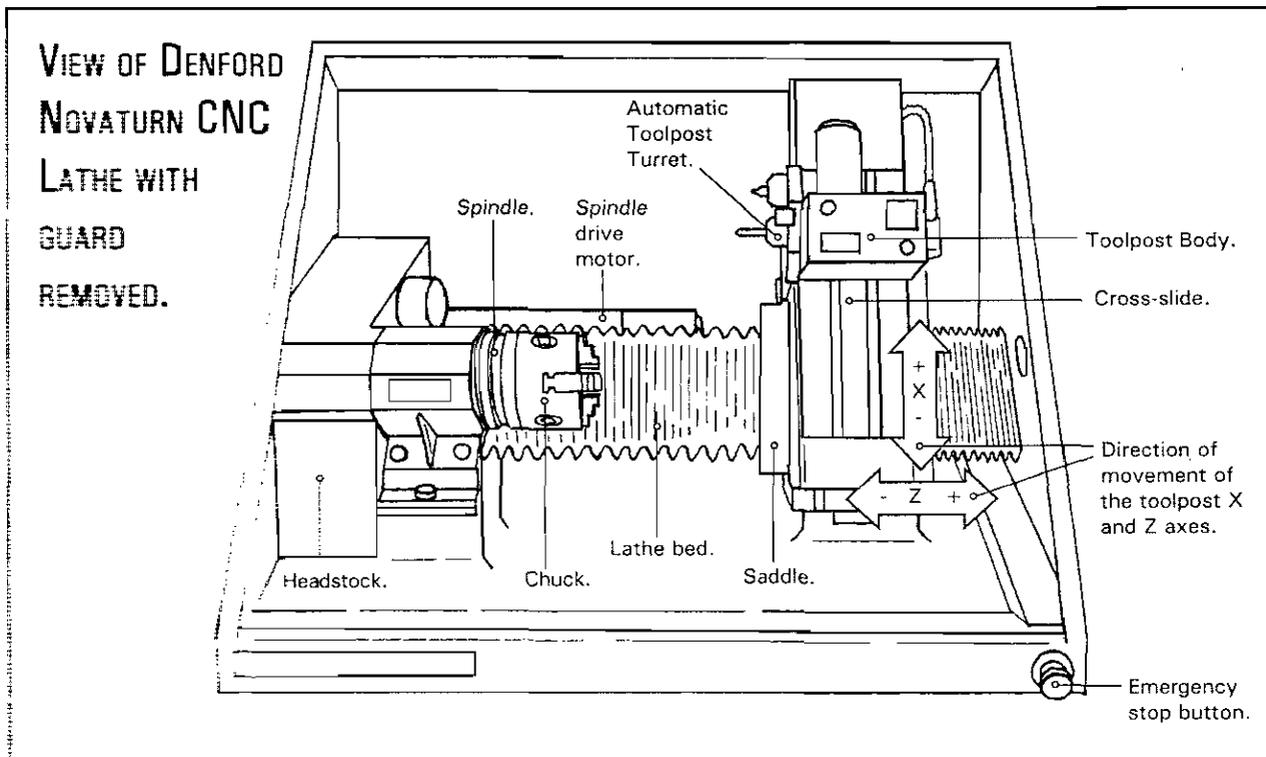
For Machine specific features and operations, please consult the separate Installation Manual supplied with your machine.

BASIC COMPONENTS....

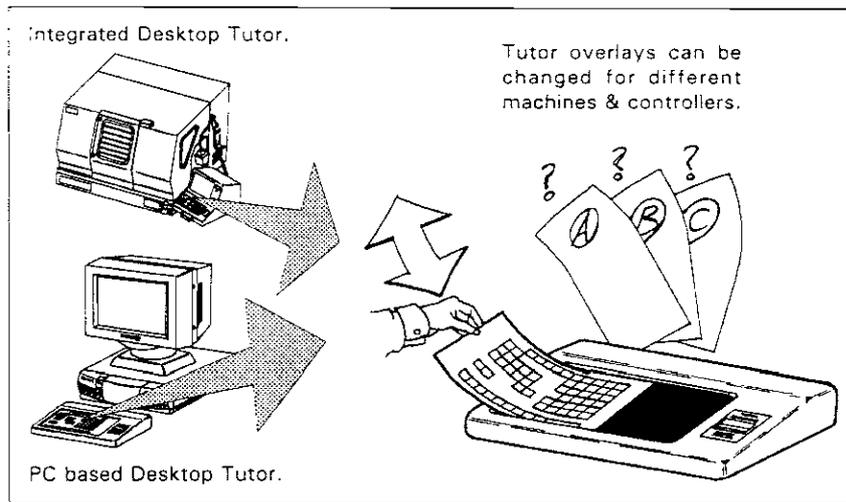


A lathe is used to change materials into cylindrical shapes, by traversing a toolpost parallel to the lathe bed, machining the rotating workpiece, called the billet. Traversing the tool across the lathe bed machines a flat surface. Lathes can also produce tapers, cut threads and bore holes.

On a CNC (Computer Numerical Control) lathe, the toolpost is moved by motors, controlled by the computer. Points on the workpiece are described by X and Z co-ordinates. The toolpost can be directed to these exact positions automatically, after programming the co-ordinate movements into the Desk-top Tutor controller.



OPERATING THE DESKTOP TUTOR CONTROL PANEL.



The Denford Desktop Tutor is the keypad input controller for the machine (i.e. the equivalent to a Qwerty keyboard on a pc), common to both pc driven and integrated machines. The overlays on Desktop Tutors are interchangeable, according to the type of machine and control method required.

		F1	F2	F3	F5	F9	F10	CTRL F1			RESET	
OPERATION SELECT							↑	7 0	8 N	9 G	ALTER	
AUTO	EDIT						↓	← 4 X	5 Z	→ 6 F	INSERT	
SINGLE BLOCK	BLOCK SKIP			-X			CURSOR	1 U	2 ↓ W	3 R	DELETE	
HOME	JOG	-Z	TRVRS	+Z			PAGE	↑	- M	0 S	/ # EOB	
COOLANT				+X				↓	← B A C	→ K I H	V J P	CANCEL
ON	OFF			AXIS/DIRECTION								
EXECUTION				SPINDLE								
CYCLE START	CYCLE STOP	SPNDL CW	SPINDL STOP	SPNDL CCW			UTILS	PGR	MENU OFFSET	POS GRAPH	INPUT OUTPUT	

This section will outline the use of each key on the Denford FANUC Turning keyboard controller overlay, shown above.

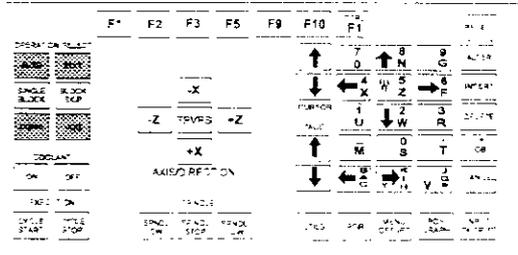
The Operation Select keys will set the mode in which the machine and controller will run:

The [AUTO] key is used to select *AUTO MODE* - pressing this key will allow the user to run any program loaded into the machine (ie, begin a machining operation).

The [EDIT] key is used to select *EDIT MODE* - program data can be simulated or changed when the machine is in this mode.

The [HOME] key is used to zero the machine around a set of reference points.

The [JOG] key is used to select a mode which allows the axes of the machine to be moved around. This can be either at a set feedrate (in *CONTINUOUS MODE*), or in stepped movements (in *INCREMENTAL MODE*).

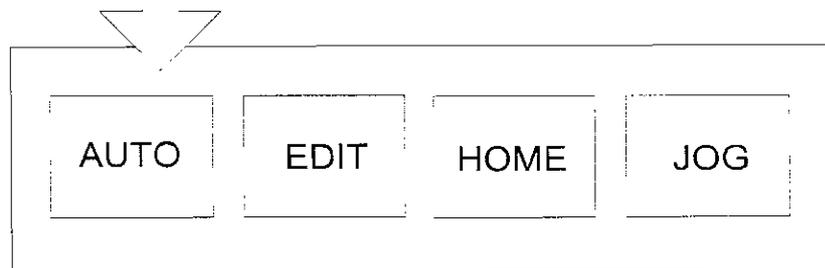


Tutor keypad.

Many of the other keys on the controller have multiple functions, according to the machine mode that is set.

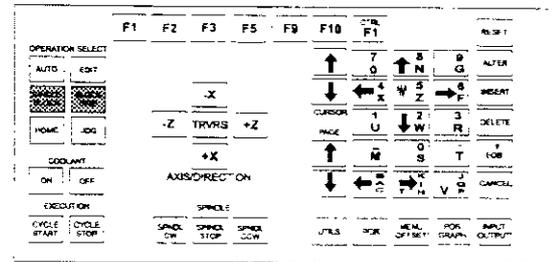
Look for the "*AUTO, EDIT, HOME* and *JOG*" symbols (shown below) - they will indicate the mode the machine must be running for the highlighted key to operate in the way described.

The highlighted keys will have no functions allocated to modes that are not shown by the boxes below.



The [SINGLE BLOCK] key is used to run a program in single blocks (ie, line by line).

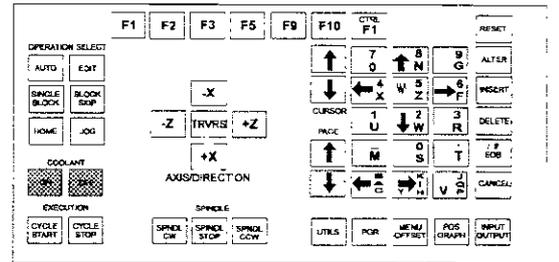
The [BLOCK SKIP] key is used to select the option to ignore, or include, specific program blocks (activated by a "/" character in front of the block).



Tutor keypad.

- The [ON] key is used to switch coolant liquid 'on'.

- The [OFF] key is used to switch coolant liquid 'off'.



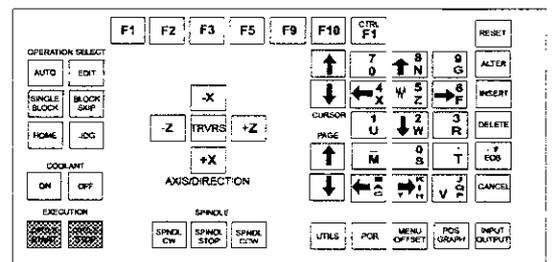
Tutor keypad.

The [CYCLE START] key is used to:

- 1) Start any program currently loaded into the machine.
- 2) Simulate any program currently loaded into the machine.
- 3) Resume a program which has been paused.

The [CYCLE STOP] key is used to:

- 1) Stop a program currently running on the machine.
- 2) Pause a program currently running on the machine.



Tutor keypad.

The [-X] key is used to control axis movement in the -X direction.

The [+X] key is used to home the machine (set the machine datum) in the X axis.

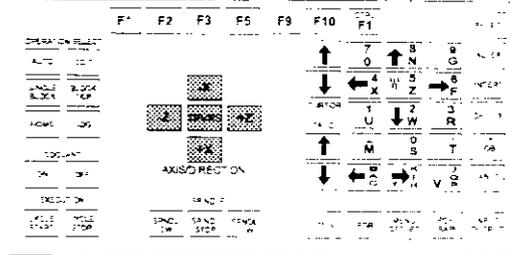
The [+X] key is used to control axis movement in the +X direction.

The [-Z] key is used to control axis movement in the -Z direction.

The [+Z] key is used to home the machine (set the machine datum) in the Z axis.

The [+Z] key is used to control axis movement in the +Z direction.

The [TRVRS.] key is used with the axis movement keys to achieve a rapid traverse.

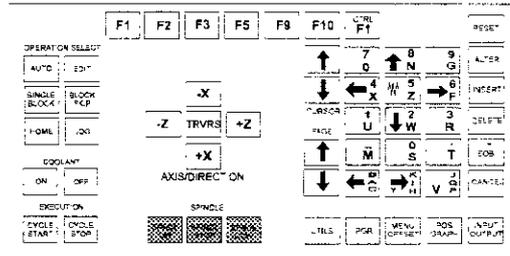


Tutor keypad.

The [SPNDL.CW] key is used to turn the spindle in a clockwise (forward) direction, viewed looking from the headstock end of the lathe.

The [SPNDL.STOP] key is used to stop the spindle turning.

The [SPNDL.CCW] key is used to turn the spindle in a counter-clockwise (backwards) direction, viewed looking from the headstock end of the lathe.



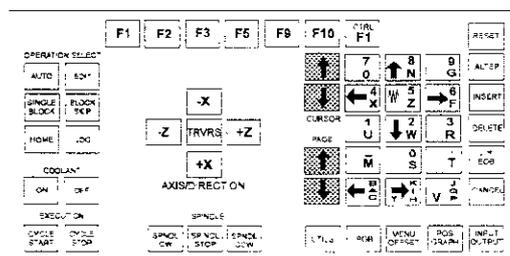
Tutor keypad.

The blue [CURSOR ARROWS] keys are used to:

- Move up or down lines of text on purple Menu Selection screens and when editing programs.
- Move up or down the program lines before machining is started.

The blue [PAGE ARROWS] cursor keys are used to:

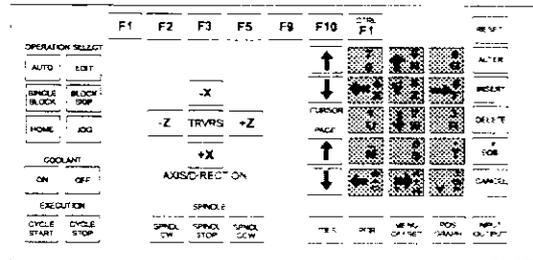
- Move between the top and bottom lines of purple Menu Selection screens and programs.
- Move up or down between the pages of the program before machining is started.



Tutor keypad.

   The [ALPHA/NUMERICAL] keys are used to enter characters and numbers used in program data.

Multi-character keys will toggle between the characters shown according to the number of times the key is pressed.



Tutor keypad.

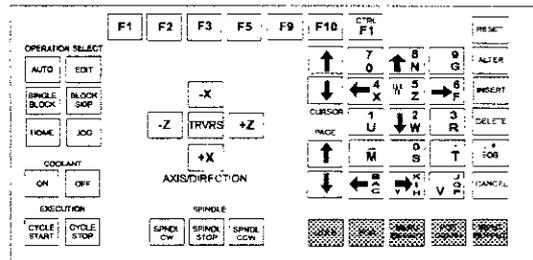
 The [UTILS] key is used to display any directives within a program, shown on screen as [YELLOW LINES]. Directives are Denford definitions for tool sizes, billet sizes etc....

    The [PGR.] key is used to toggle between these screen modes: *SIMULATE ONLY*, *EDIT ONLY* or *EDIT AND SIMULATE*.

   The [MENU OFFSET] key is used to display the *TOOL OFFSET TABLE*.

   The [POS. GRAPH] key is used to change the co-ordinates position read-out, on the VDU screen, between 'absolute' and 'distance to go' (useful when running in *AUTO MODE*).

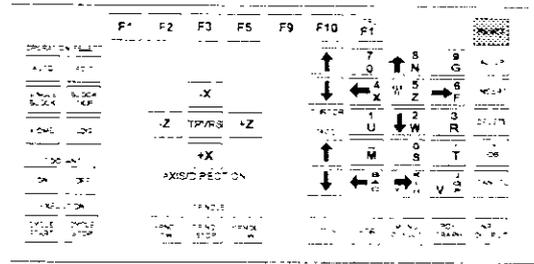
 The [INPUT OUTPUT] key is used to select the *REMOTE DEVICE LINK MENU* (this menu allows data to be sent or received from external peripherals).



Tutor keypad.

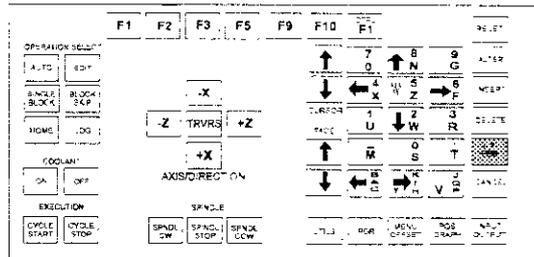
The [RESET] key is used to:

- 1) [AUTO] [EDIT] [HOME] [JOB] Move back through any menu screens accessed, one by one, until the highest (start) level is reached.
- 2) [AUTO] [EDIT] [CANCEL] [JOB] Clear any red warning or error messages from the screen.
- 3) [EDIT] Move to the top of a program.



Tutor keypad.

[EDIT] [JOB] The [EOB] key is the 'End of Block' command, used to signify the end of sequence of events or to confirm choices within the software. It is the equivalent of the 'return' key on a pc.



Tutor keypad.

The [ALTER] key is used to:

- 1) [EDIT] [HOME] [JOB] Change any words (made from an address letter and a number) in a program line.
- 2) [AUTO] [EDIT] [HOME] [JOB] Delete information in a text entry box (ie, load/save boxes).

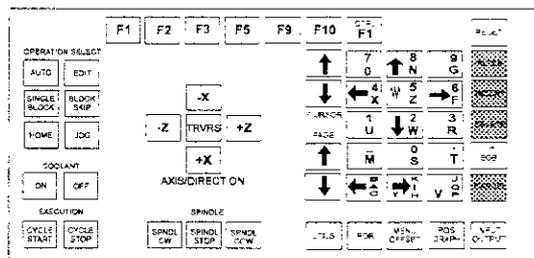
[EDIT] [HOME] [JOB] The [INSERT] key is used to place a word into a program line.

The [DELETE] key is used to:

- 1) [EDIT] [CANCEL] [JOB] Remove a word from a program line.
- 2) [AUTO] [EDIT] [HOME] [JOB] Remove unwanted characters that have been typed in.

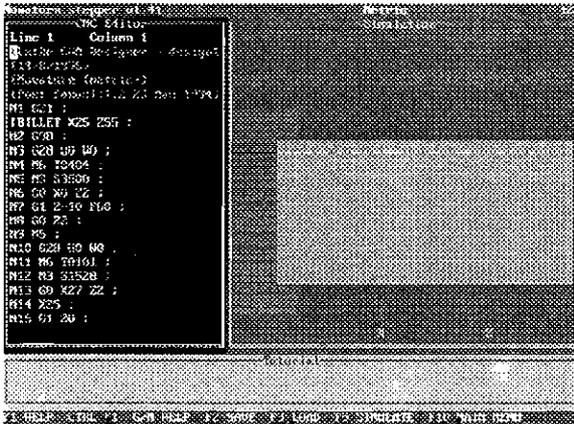
The [CANCEL] key is used to:

- 1) [EDIT] [HOME] [JOB] Remove a word from a program data entry line.
- 2) [AUTO] Abort a running program.

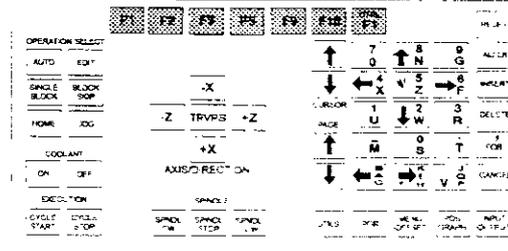


Tutor keypad.

The [FUNCTION NUMBERS] keys are used as short-cuts to other screens. An information bar is usually displayed at the base of the screen displaying the functions allocated to each particular key.



F1 HELP CTRL-F1 G/M HELP F2 SAVE F3 LOAD F5 SIMULATE F10 MAIN MENU



Tutor keypad.

The Fast key Strip displays the function allocated to each key..

The [F1] key is used to display context sensitive *General Help* screens.

The [F2] key is used to save a program to the default drive.

The [F3] key is used to load a program from the default drive.

The [F5] key is used to display the *Data Information Help* screen.

The [F9] key is used to:

- 1) Display the *CONTROL MENU*.
- 2) Display the *SIMULATION MENU*.

The [F10] key is used to display the *MAIN MENU*.

The [CTRL F1] key is used to display context sensitive *G and M code Help* screens.

OPERATING THE DESKTOP TUTOR CONTROL PANEL.

Both Feedrate and the Spindle Speed can be manually overridden, according to the controls fitted to the machine.

If the machine is fitted with override potentiometers (adjustable dial controls) on its front panel, these will be used to alter the feedrate and spindle speed values.

 This feature will operate in both *AUTO AND JOG MODES*.

On machines not fitted with override potentiometers, the following Tutor keys may be used:

Manual Feedrate Override - Use the key labelled [4X LEFT ARROW] to decrease the feedrate and the [6Z RIGHT ARROW] to increase the feedrate.

Manual Spindle Speed Override - Use the key labelled [2F DOWN ARROW] to decrease the spindle speed and the [8N UP ARROW] to increase the spindle speed.

 This feature will only operate in *AUTO MODE*.

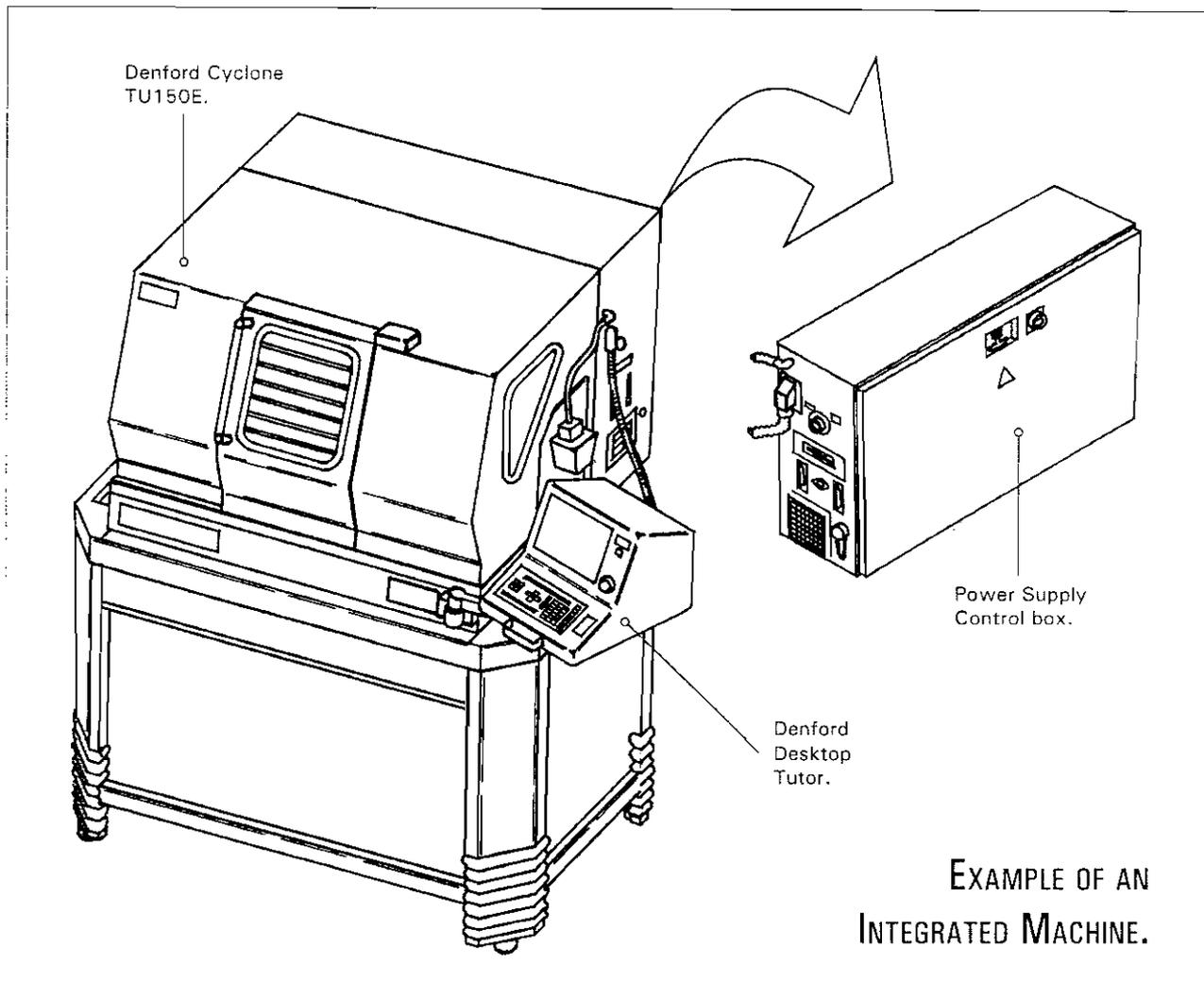
SWITCHING ON A MACHINE WITH AN INTEGRATED CONTROLLER.

An integrated machine has a permanently attached computer and controller, i.e. dedicated to operating that particular machine and nothing else!

The Power Supply controls for integrated machines are sited on either,

(i) free standing metal power supply cabinets
(as illustrated below).

(ii) power supply cabinets fixed to the back of the machine itself.



SWITCHING ON A MACHINE WITH AN INTEGRATED CONTROLLER.



To switch on your machine, locate the yellow rotary power supply switch on the power supply control box and turn it to the 'on' position.

All controlling software and necessary drivers will automatically load if installed on the computer hard disk.

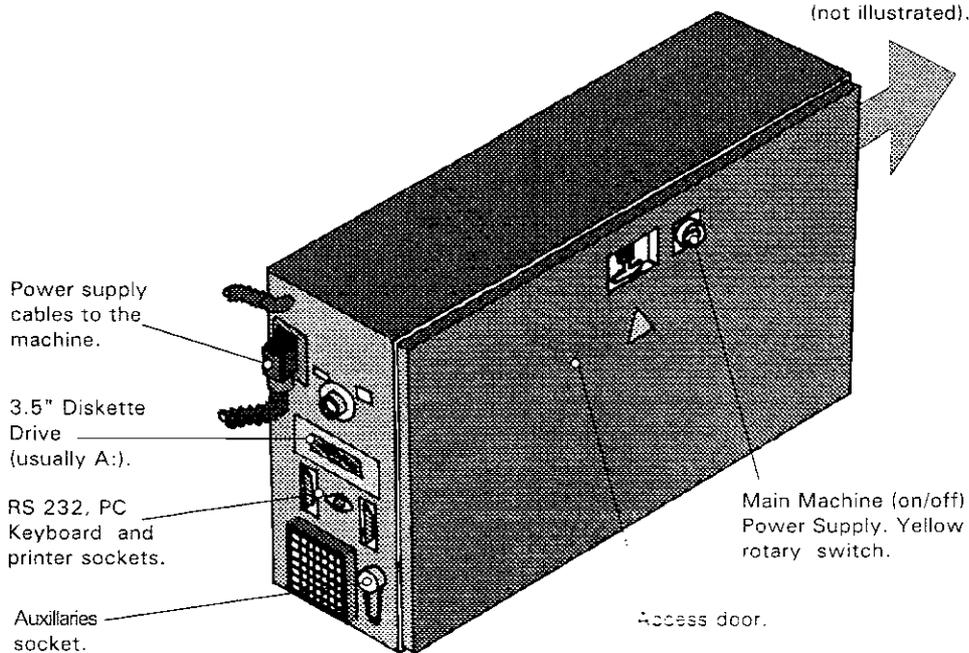
If the controller software is supplied on floppy (3.25 inch) disk, insert the disk into the floppy drive on the power supply control box before switching on the machine.



To switch off the machine, turn the yellow rotary power supply switch to the 'off' position. You do not have to fully exit the controlling software in order to switch off the machine, but note - the machine must not be turned off if a turning program is running, or the machine is cutting work.....

INTEGRATED MACHINE, POWER SUPPLY CONTROL BOX.

Mains plug exits from this side of the control box (not illustrated).



SWITCHING ON A MACHINE CONTROLLED BY A PC.

A pc operated machine has 3 main components:

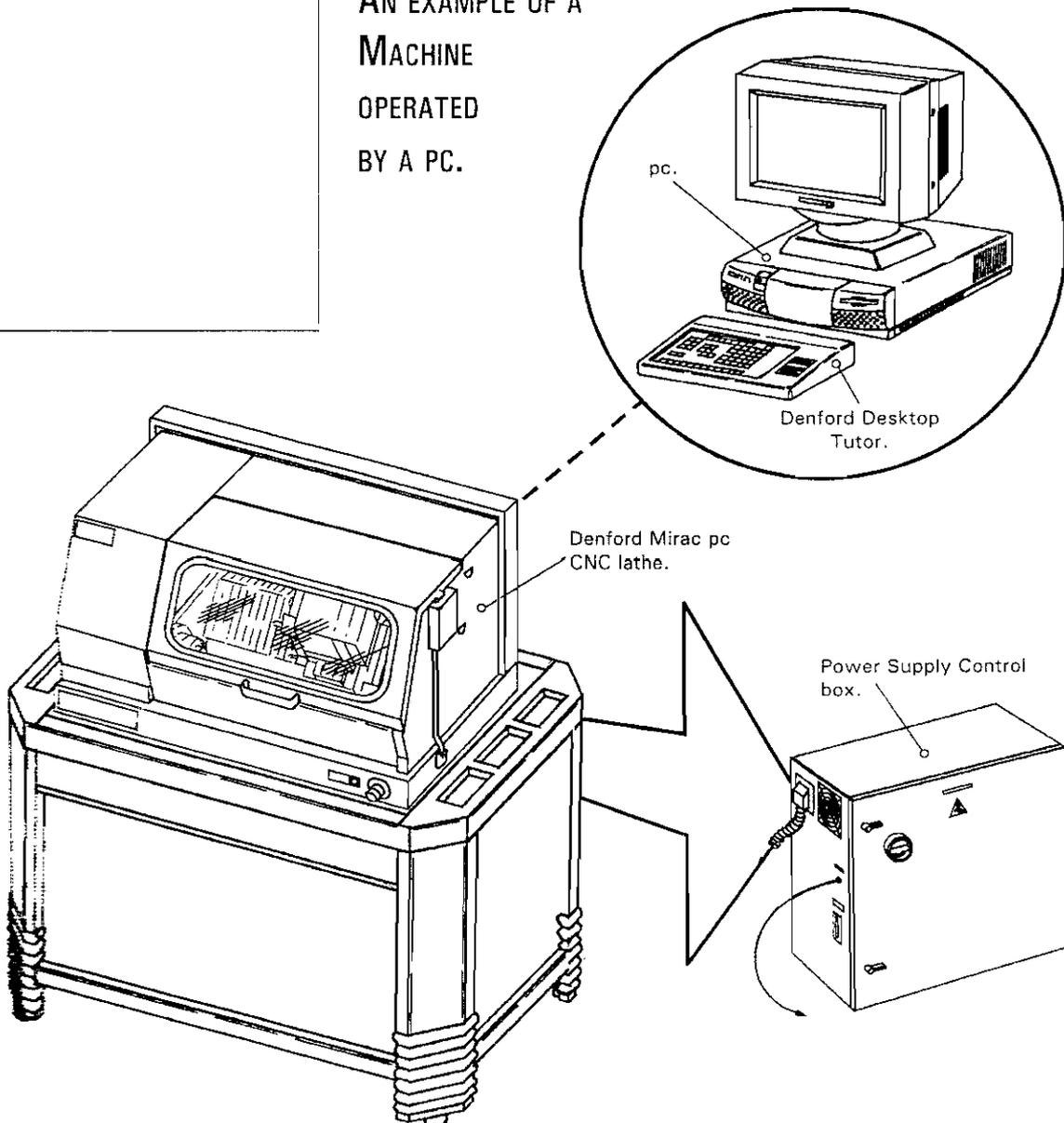
- 1) The pc (personal computer) itself, with an attached Denford Desktop Tutor.
- 2) The CNC lathe.
- 3) The machine power supply unit.

The pc and CNC lathe are linked together so the computer controls the operation of the machine via the Desktop Tutor. The Power Supply controls for pc operated machines are sited on either,

(i) free standing metal power supply cabinets (as illustrated below).

(ii) power supply cabinets fixed to the back of the machine itself.

AN EXAMPLE OF A MACHINE OPERATED BY A PC.



SWITCHING ON A MACHINE CONTROLLED BY A PC.

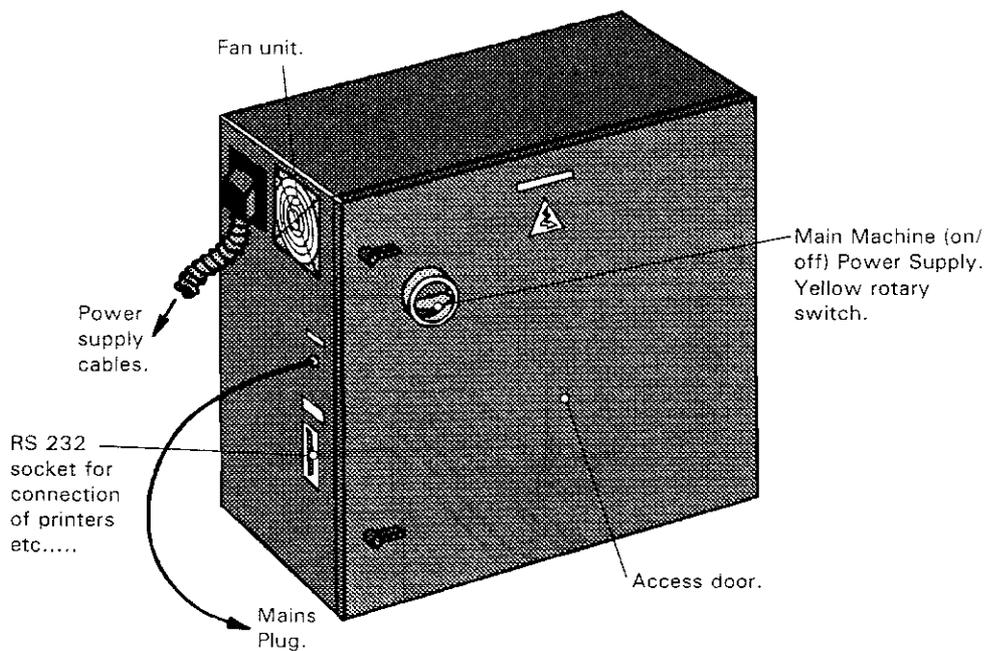


To switch on your machine, locate the yellow rotary power supply switch and turn it to the 'on' position. To load the machine controlling software, please refer to the next page.



To switch off the machine, turn the yellow rotary power supply switch to the 'off' position. You do not have to fully exit the controlling software in order to switch off the machine, but note - the machine must not be turned off if a turning program is running, or the machine is cutting work..... Exit the machine controlling software using the *MAIN MENU* (press the [F10] key) and highlight '*QUIT*' to shutdown (use the [CURSOR ARROWS] keys and [EOB]) - see page 1.9.

PC OPERATED MACHINE, POWER SUPPLY CONTROL BOX.



PC CONTROLLED MACHINES - LOADING THE CONTROL SOFTWARE.

To load the machine controlling software from your pc hard disk, switch on your pc and exit, if necessary, to the 'DOS' prompt.

The directory in which the software is held and the application start-up filename, will depend on the type of machine used. Please choose the correct directory and start-up filename from the list shown below....

Machine.	Directory.	Start-up filename.
Microturn	/DENFORD	FANUCML
Novaturn	/NOVATURN	FLSTEP
Mirac PC	/MIRACPC	FLSTEP

Please Note - the directories and filenames shown above are only applicable if the defaults are used when installing the software.

To start the machine controlling software, type the following at the 'DOS' prompt:

C:\Directory

(where 'C:' is the drive where the software has been installed and 'Directory' is the text chosen from the list shown above)

Press the [ENTER] / [RETURN] key on the pc.

Next, type in the '**Start-up filename**' chosen from the list above

Press the [ENTER] / [RETURN] key on the pc.

The machine controlling software will now load.

PC CONTROLLED MACHINES - LOADING THE CONTROL SOFTWARE.

To load the machine controlling software from a floppy (3.25 inch) disk, switch on your pc and exit, if necessary, to the 'DOS' prompt.

The application start-up filename will depend on the type of machine used. Please choose the start-up filename from the list shown below....

Machine.	Start-up filename.
Microturn	FANUCML
Novaturn	FLSTEP
Mirac PC	FLSTEP

To start the machine controlling software, type the following at the 'DOS' prompt:

A:\Start-up filename

(where 'A:' is the floppy (3.25 inch) disk drive and 'Start-up filename' is the text chosen from the list above).

Press the [ENTER] / [RETURN] key on the pc.

The machine controlling software will now load.

CONTROL SOFTWARE - MAIN MENU.

Keys Helobox.
The following keys are used in this section:
[F10]
[CURSOR ARROWS]
[EOB], [RESET]

Tutor keypad.

The *MAIN MENU* navigates around the most commonly used options of the control software.

Select the *MAIN MENU* by pressing the [F10] key. To select one of the ten options available, highlight the required option using the [CURSOR ARROWS] keys and press the [EOB] key to confirm this choice. Unwanted menus can be removed by pressing the [RESET] key.

MAIN MENU

- EDIT ONLY
- EDIT AND SIMULATE
- SIMULATE ONLY
- MACHINE CONTROL
- CNC FILES
- PRINT
- REMOTE LINK
- SETTINGS
- UTILITIES
- QUIT

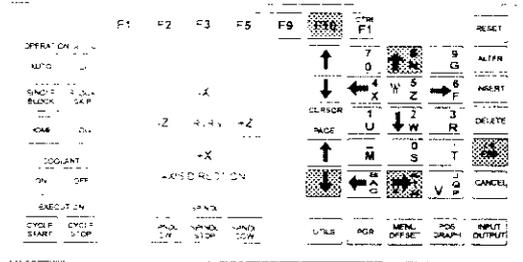
CONTROL SOFTWARE - MAIN MENU.

The *MAIN MENU* contains ten options:

- 1) *EDIT ONLY*. This option will display the full screen CNC File Editor with 241 characters sideways scrolling facility. The CNC File can be altered using this option. Simulation is not available from this section but pressing the [F9] key will run a syntax check on the CNC code.
- 2) *EDIT AND SIMULATE*. This option will display the CNC File Editor, Simulation graphics and Tutorial windows as a split screen. If the CNC line is longer than the Editor window, a sideways scrolling facility will be offered. During CNC File editing, a graphical Simulation can be started at any time. When this Simulation has been completed, the cursor will return to its last position in the CNC File Editor. The CNC File can be altered using this option.
- 3) *SIMULATE ONLY*. This option will display the CNC File in full screen graphical format only. The Tutorial window is still displayed at the bottom of the screen. If an error occurs during a CNC File execution, the *EDIT AND SIMULATE MODE* will be automatically selected and the error code highlighted. The CNC File cannot be altered using this option.
- 4) *MACHINE CONTROL*. This option will change an *EDIT MODE* screen display to the last *MACHINE CONTROL MODE (JOG, AUTO or HOME MODE)* used.
- 5) *CNC FILES*. This option gives access to a sub-menu allowing CNC Files to be loaded, saved, deleted and the drive directories changed.
- 6) *PRINT*. This option will print the currently loaded CNC File in various formats.
- 7) *REMOTE LINK*. This option gives access to a sub-menu allowing the machine controller to be linked to an external device (such as a paper tape punch etc) for CNC File transfer.
- 8) *SETTINGS*. This option gives access to a sub-menu allowing many of the options listed above to be customised and configured.
- 9) *UTILITIES*. This option will allow access to other software products running through DOS.
- 10) *QUIT*. This option will exit the machine controlling software and returns to DOS.

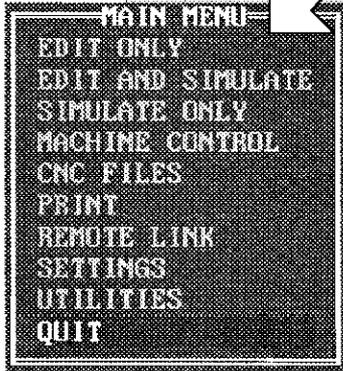
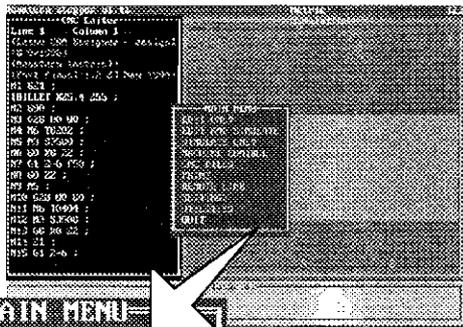
CLOSING THE CONTROL SOFTWARE.

Keys Helpbox
 The following keys are used in this section:
 [F10]
 [PAGE DOWN]
 [N], [Y], [EOB],
 [NUMBERS]: - not highlighted.



Tutor keypad.

Select the *MAIN MENU* by pressing the [F10] key.
 Press the [PAGE DOWN] key to highlight the *QUIT* option and press the [EOB] key to close the software.



When the *QUIT* option is selected a message may be displayed, asking whether the current program being used needs to be saved.

If this program does not need to be saved, press the [N] key.

If this program does need to be saved, press the [Y] key. Enter the filename (only number characters can be used with the Desktop Tutor) and press [EOB] to save the program on the currently selected drive.

Question
 Current program not saved - Save it first?

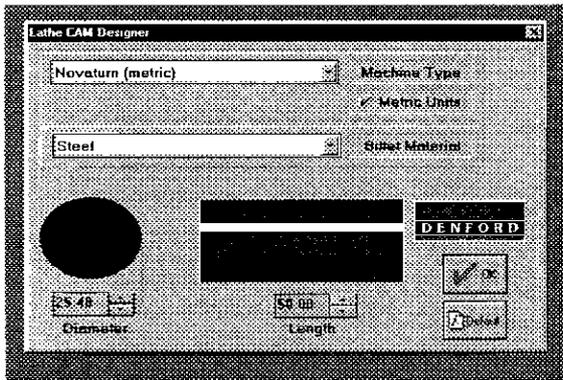
HOW TO SET-UP LATHECAM DESIGNER TO WORK WITH YOUR CNC LATHE.



Denford's LatheCAM Designer is a user friendly, "Windows" based software package, especially developed as an introduction to the world of CAM (Computer Aided Manufacture).

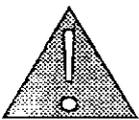
Essentially, it allows a part to be designed "on-screen" using a series of shapes selected from a software menu. Once designed, the G-code program, used by the machine to manufacture the part, can be generated automatically.

This section is not intended as a guide on how to use LatheCAM Designer - please refer to the separate LatheCAM Designer manual for explanations about its specific functions and commands. This section should be used to help configure LatheCAM Designer so it works efficiently with your CNC lathe.



When LatheCAM Designer is run, it will first display a start-up screen allowing the following items to be defined:

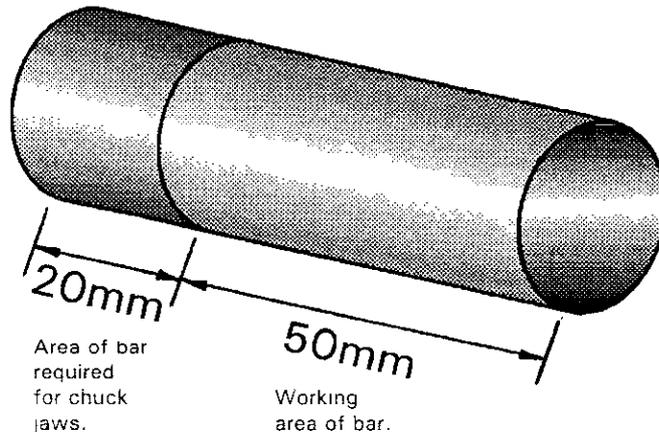
- the type of machine that will be used.
- the type of material used as the billet.
- the length and diameter of the billet (this defines the working area).
- the units of measurement.



The definable size of the billet is its working area, ie, the area of the material that can be safely reached by the cutting tools.

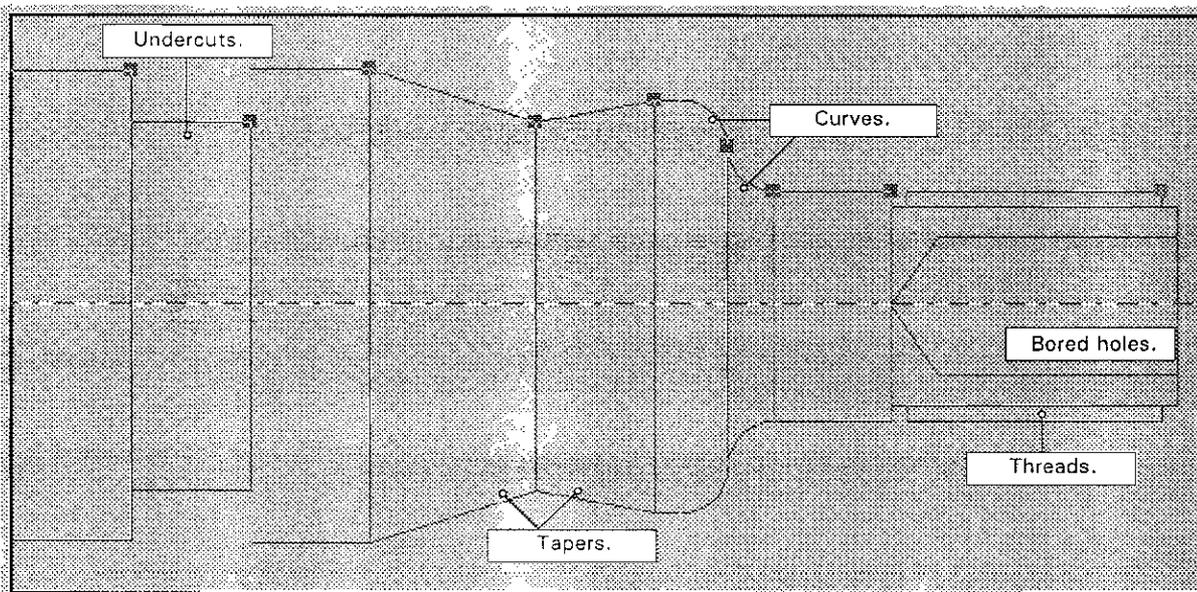
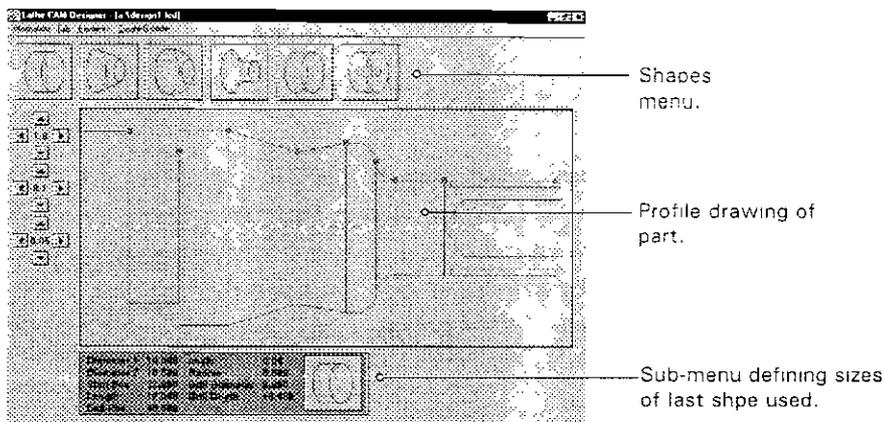
Extra material needed to actually hold the billet in the lathe chuck jaws (ie, material that cannot be machined) is not included in this working area figure.

Example - The 25.4mm diameter steel bar used in the screenshot above needs to be longer than 50mm when used on the lathe, since the 50mm figure does not include the section of the bar required for the chuck jaws. In this case, an extra 20mm is needed.



The main LatheCAM Designer screen displays the shapes menu, some cursor keys for sizing each shape and a large rectangular window representing the working area (ie, the billet with no material removed).

The design of the part is built up in stages using the shapes menu. The range of shapes available cover most profiles that would be required in a design.

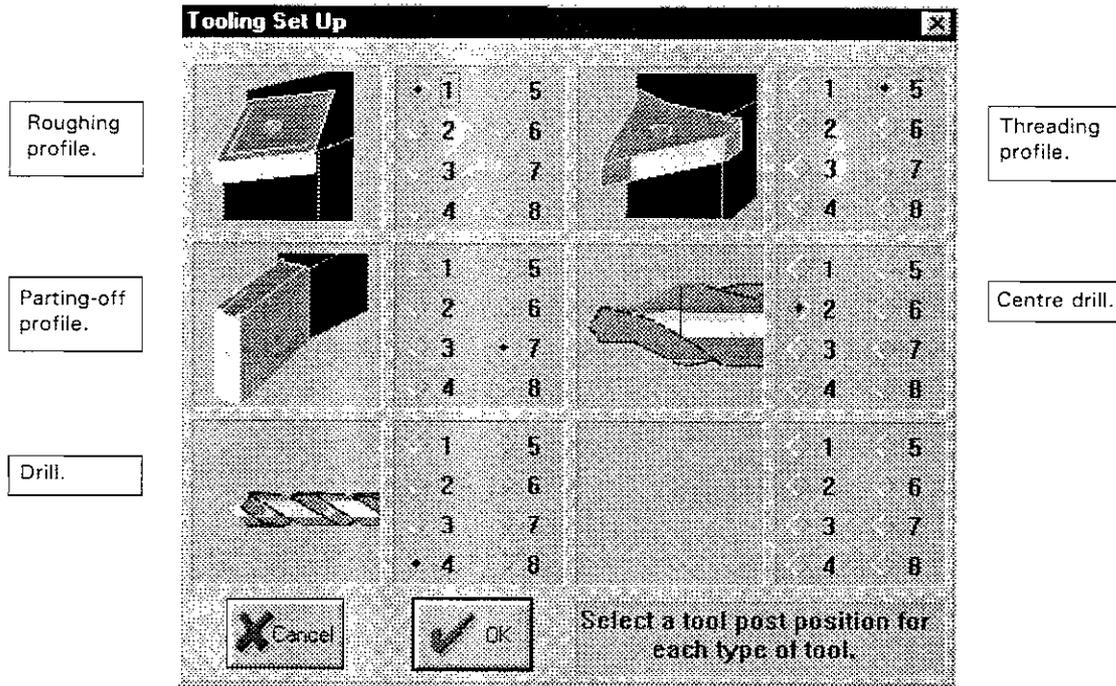


The design shown above was produced using each shape from the menu at least once.

When the design is complete, the tool profile numbers need to be set. The "Tooling Set Up" screen can be accessed in the "Create G-code" menu.

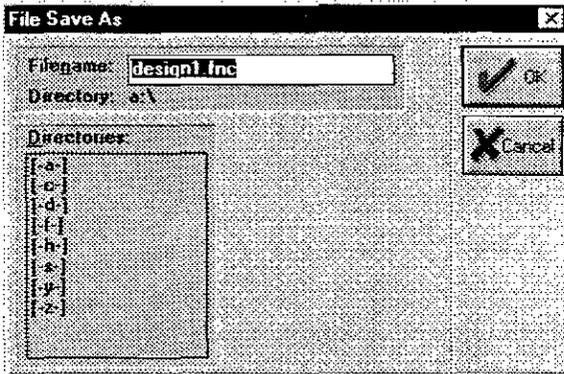
It is very important that the tool number profiles on both LatheCAM Designer and the actual toolpost on the CNC lathe match each other, ie, if tool 1 on the CNC lathe is a roughing profile, then tool 1 on LatheCAM needs to be a roughing profile, etc.....

If the tool profiles do not match, the part may not be cut successfully and the CNC lathe could be damaged. Note that although most Automatic Toolposts have spaces for upto 8 tools, LatheCAM Designer only uses 5 main tools, shown below.



The part drawn in this example (shown opposite) is designed to be produced on a Denford Novaturn CNC lathe. The two tables below show an example of how the LatheCAM software and CNC lathe toolpost would need to be set, in order for the program to work.

<p>The following tools were present on the Novaturn toolpost:</p> <ul style="list-style-type: none"> Tool 1 - Roughing tool. Tool 2 - Centre drill. Tool 3 - Threading tool. Tool 4 - 6mm drill. Tool 5 - Threading tool. Tool 6 - No Tool present. Tool 7 - Parting tool. Tool 8 - No Tool present. 	<p>The tool numbers in the LatheCAM Designer software (see above) were set to match this:</p> <ul style="list-style-type: none"> Roughing tool - 1. Parting tool - 7. Drill - 4. Threading tool - 5. Centre drill - 2.
------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------	---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------



Once the tooling has been set, the G-code program can be generated, by selecting "Make file..." from the "Create G-code" menu.

Ensure that the G-code parameters (speeds and feeds) are correct before making the G-code program file.

This program can then be transferred to the CNC lathe.



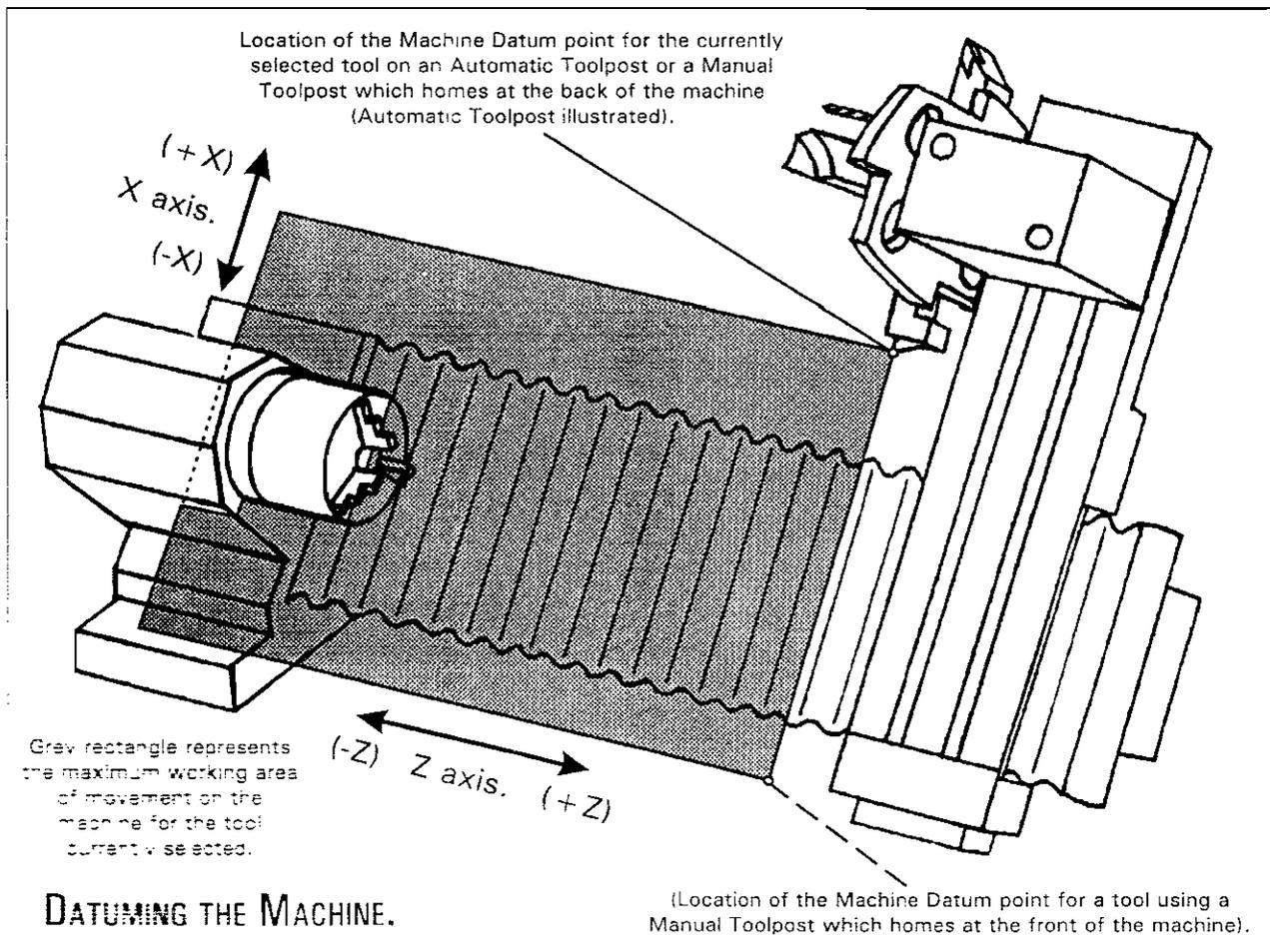
SETTING THE MACHINE DATUM POINT.

It is necessary to home (datum) the machine whenever it is switched on. Essentially, this operation finds the limits of movement for both machine axes - X and Z. The toolpost will move to its maximum limits of travel on the X and Z axes. Automatic toolposts will home in the back right-hand corner of the machine (see diagram below). Manual toolposts home either in the front or back right-hand corner of the machine, according to the particular machine model and the type of control software being used.

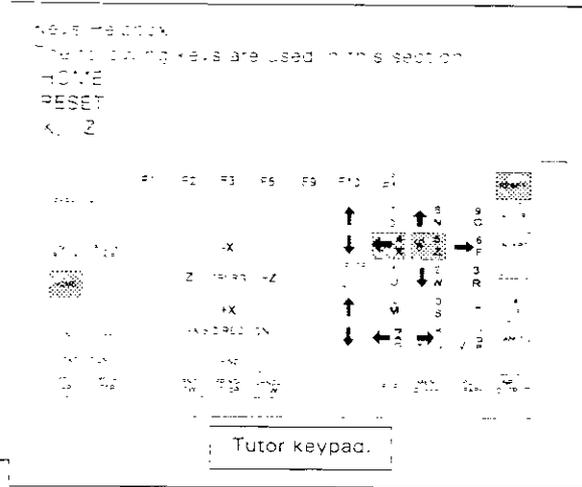
Why are X and Z axes used, instead of X and Y? The Industrial standard is Z always runs along the axis which has rotating, or moving parts (ie, the chuck).

When the machine has just been datumed, the tip of the currently selected tool will always be the machine datum point with the co-ordinates $X=0$, $Z=0$. If the tool is changed for another, with a different length, the machine datum point will then be the tip of this "new" tool, again with the co-ordinates $X=0$, $Z=0$.

The machine datum point is used as a reference for describing other co-ordinates within the working area of the selected tool, shown by the shaded area on the diagram below....

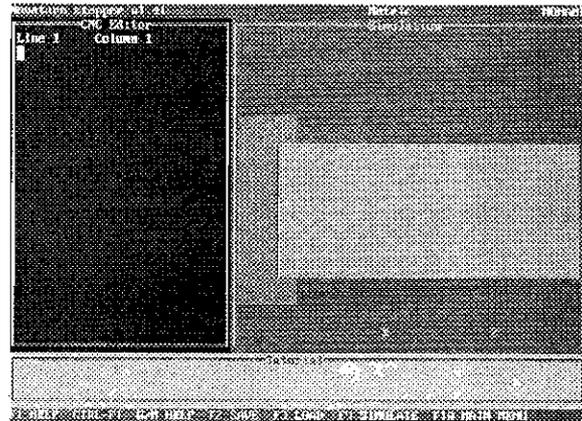


SETTING THE MACHINE DATUM POINT.

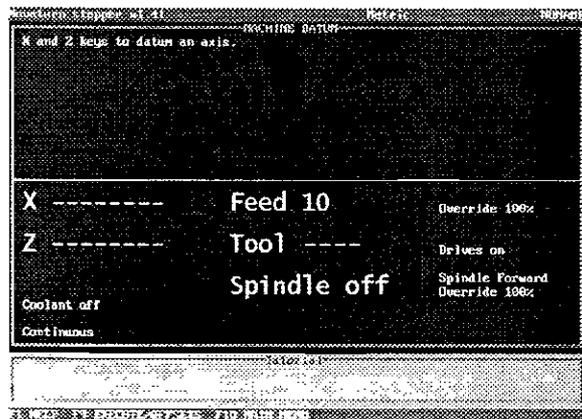


The following procedure shows how a Denford Novaturn is datumed - most Denford CNC lathes will datum using this procedure.....

On loading up the DENFORD TURNING software, the start up screen will be displayed.....



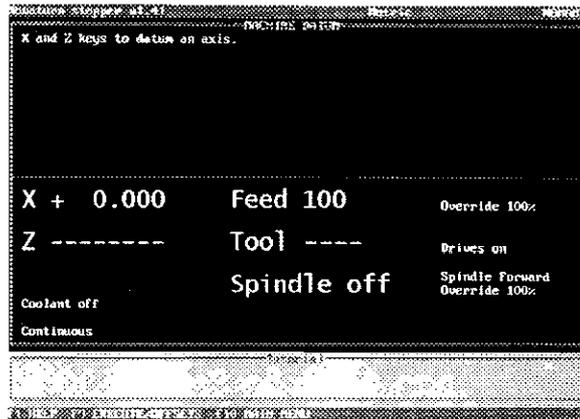
To set the machine datum point, first press the [HOME] key.....



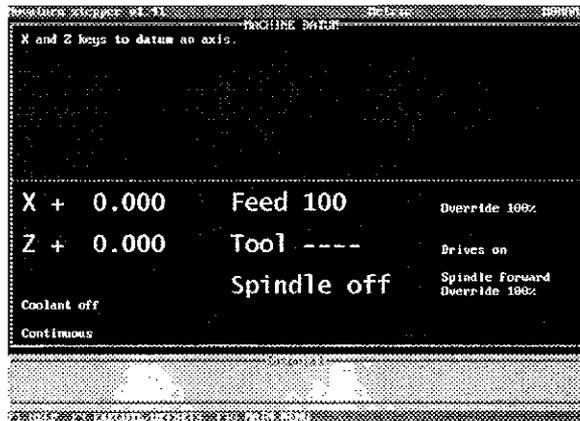
Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

SETTING THE MACHINE DATUM POINT.

Next, press the [X] key. The toolpost will move until it finds the X axis reference point.....



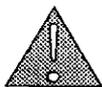
Then, press the [Z] key. The toolpost will move until it finds the Z axis reference point.



continued....

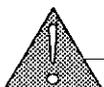
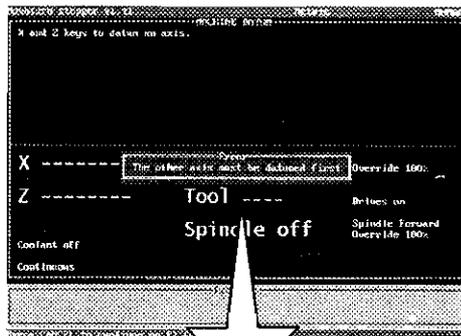
SETTING THE MACHINE DATUM POINT.

continued....



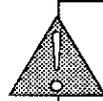
Note - some machines may be configured so that the Z axis is datumed before the X axis (Denfords Microturn, for example), so if the X axis is datumed first an error message will be displayed. If this occurs, press [RESET] and datum the Z axis first.

The machine cannot be damaged if you try to datum with the wrong axis first.

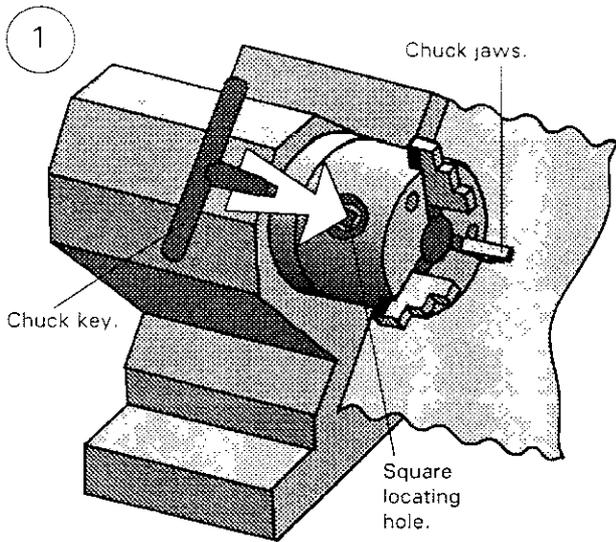


Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

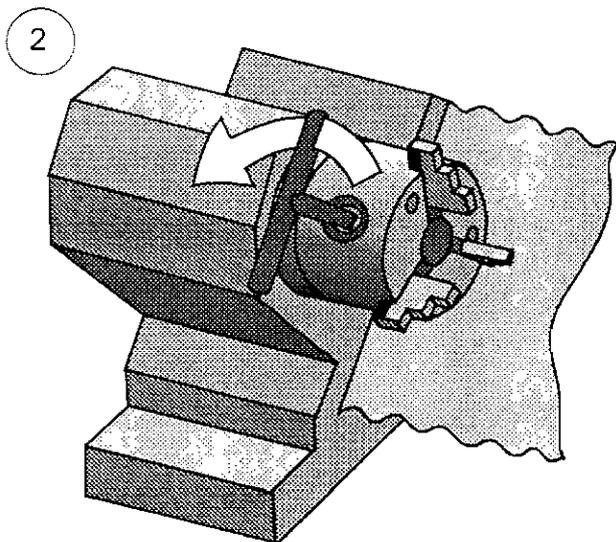
CLAMPING THE BILLET.



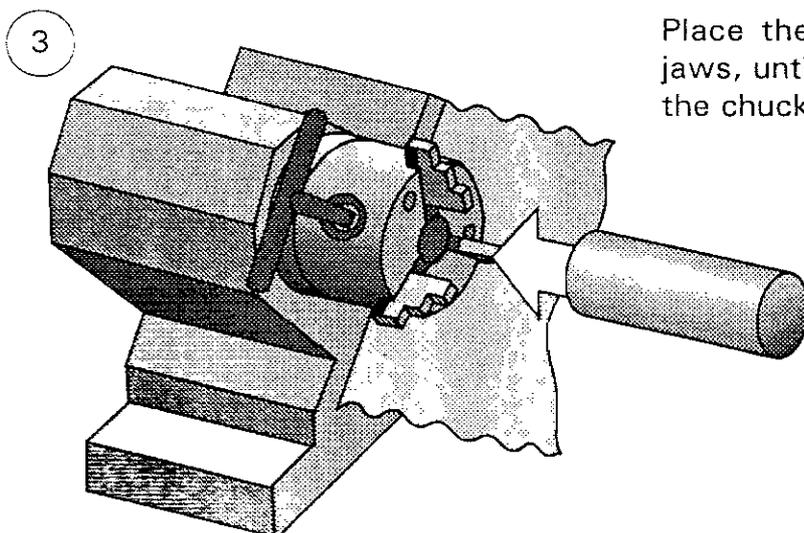
Never leave the chuck key fitted in one of the locating holes. If the spindle is switched on it could hit the operator or machine and cause serious damage.



The billet is held onto the spindle by a 3 jaw chuck. The chuck jaws are adjusted using a chuck key - a "T" shaped bar with a square locating peg. This fits into one of the three square holes equally spaced around the perimeter of the chuck.

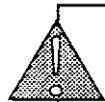


Open the chuck jaws by rotating the chuck key anticlockwise until there is sufficient space to fit the billet.



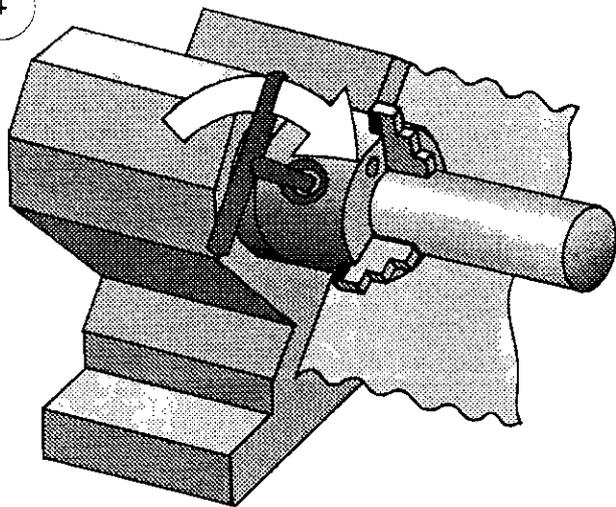
Place the billet between the chuck jaws, until it touches the front face of the chuck.

CLAMPING THE BILLET.



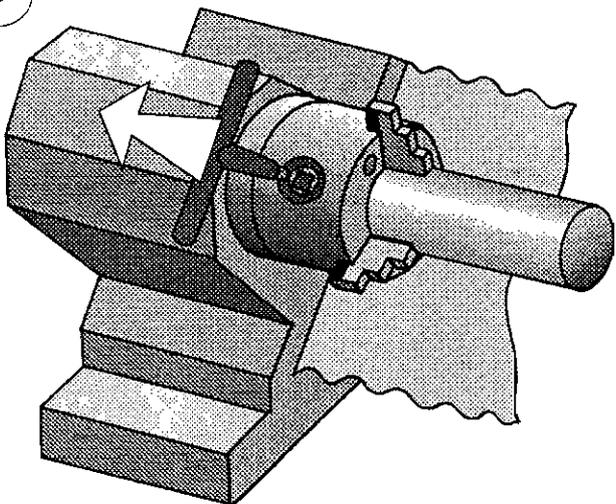
Never leave the chuck key fitted in one of the locating holes. If the spindle is switched on it could hit the operator or machine and cause serious damage.

4



Whilst holding the billet against the front face of the chuck, close the chuck jaws by rotating the chuck key clockwise.

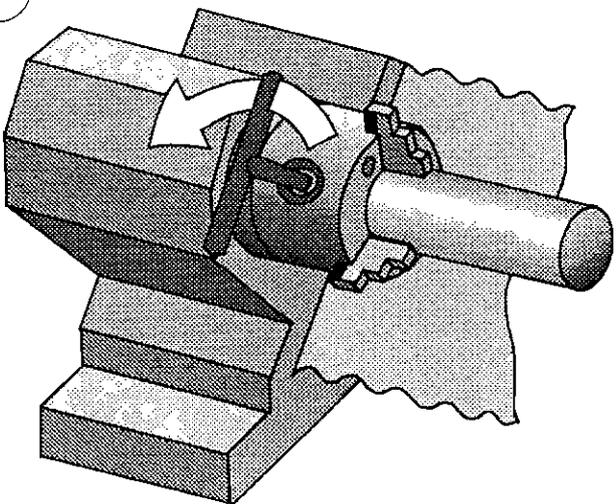
5



Check the billet is held securely in the chuck jaws.

Remove the chuck key from the square locating hole.

6



To remove the billet, insert the chuck key into one of the square locating holes. Rotate the chuck key anti-clockwise to open the chuck jaws and remove the billet. Remove the chuck key.

CHANGING THE TOOL.

Each of the lathe tools is held in a toolholder which, in turn, is mounted upon a toolpost.

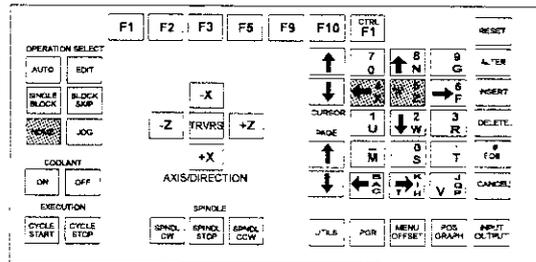
A CNC lathe equipped with an Automatic Toolpost will perform all the necessary functions required for changing a tool automatically - the operator only needs to enter the number of the tool required into the desktop tutor keypad.

A CNC lathe equipped with a manual toolpost will rely on the operator to physically change the tool to the correct number, although the tool changing software screens remain the same as changing a tool using an Automatic Toolpost. When running a multiple tool program with a manual toolpost, the software will prompt the operator to change tools when required.

Keys Helpbox.

The following keys are used in this section:

- [HOME]
- [X]
- [Z]



Tutor keypad.

In most cases, the safest position to change tools is when the toolpost is parked in its "home" position. This position usually leaves the greatest distance possible between the toolpost and the billet.

To "Home" the machine, press the [HOME] key, followed by the [X] key and [Z] key. Note, on some machines, the order of pressing X, then Z may need to be reversed.

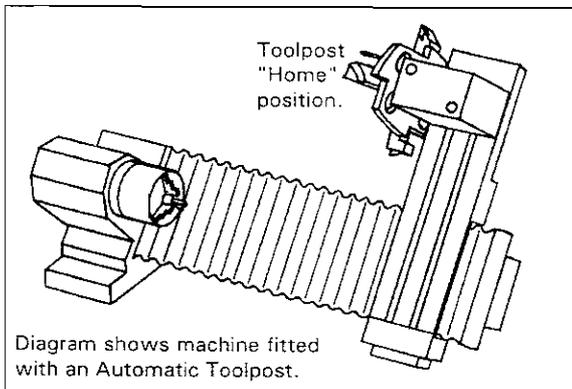


Diagram shows machine fitted with an Automatic Toolpost.

continued....

CHANGING THE TOOL.

continued....

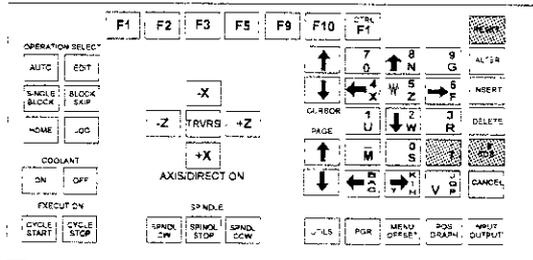
The toolpost will move to its "Home" position. On machines with an Automatic Toolpost this is the back right-hand corner of the machine. On machines fitted with a Manual Toolpost this is either the front or back right-hand corner of the machine, according to the machine model and control software used.

When the tool carousel is rotated on the Automatic Toolpost, there is less chance of damage being caused by the tools hitting part of the machine or billet during the operation. Similarly, on a manual toolpost, there is more space to remove and refit tools.

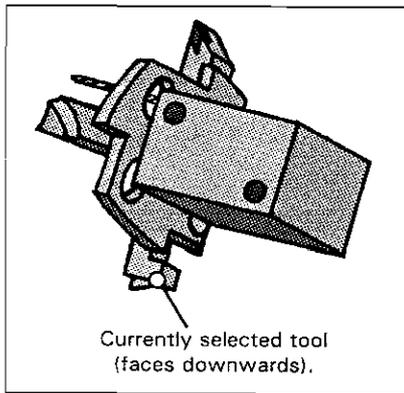
CHANGING THE TOOL USING AN AUTOMATIC TOOLPOST.

Keys Helpbox.

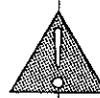
The following keys are used in this section:
 [JOG], [T]
 [NUMBERS] - not highlighted
 [EOB]



Tutor keypad.



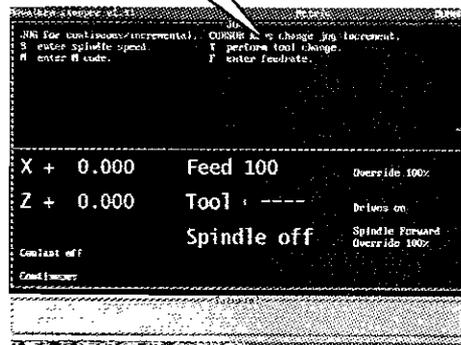
Currently selected tool (faces downwards).



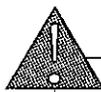
Move the toolpost to a safe position before starting any tool changing operation.

To change a tool, the lathe needs to be set in "JOG MODE". This is selected by pressing the [JOG] key. A list of commands which operate in "JOG MODE" are displayed, one of which is tool change.

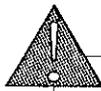
T perform tool change.



continued....



Tool Orientation - Fit the tool facing down on Automatic Toolposts.



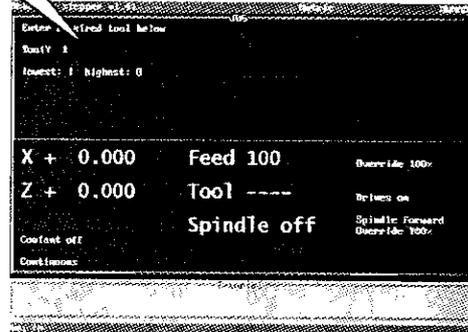
Please note - The co-ordinate values depicted on screenshots are used for illustrative purposes only.

CHANGING THE TOOL USING AN AUTOMATIC TOOLPOST.

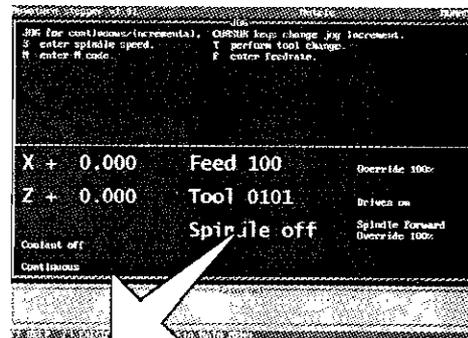
continued....

Next, enter the "new" number of the tool required, using the [NUMBERS] keys.

Enter required tool below
Tool? 1
lowest: 1 highest: 8



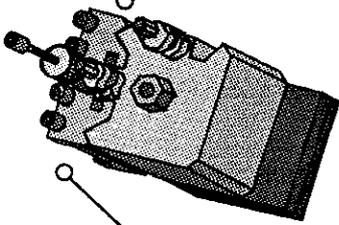
When the [EOB] key is pressed, the tool changer will automatically change from the currently selected tool number to the "new" tool number. This "new" tool number will be displayed "on-screen" when the operation has been completed.



Tool 0101

CHANGING THE TOOL USING A MANUAL TOOLPOST.

Manual toolposts which home in front right-hand corner of machine - cutting tool tips on this side of toolpost facing upwards.



Manual toolposts which home in back right-hand corner of machine - cutting tool tips on this side of toolpost facing downwards.



Tool Orientation -

Fit the tool facing up on manual toolposts that home in the front right-hand corner of the machine.

Fit the tool facing down on manual toolposts that home in the back right-hand corner of the machine.



Please note -

The co-ordinate values depicted on screenshots are used for illustrative purposes only.

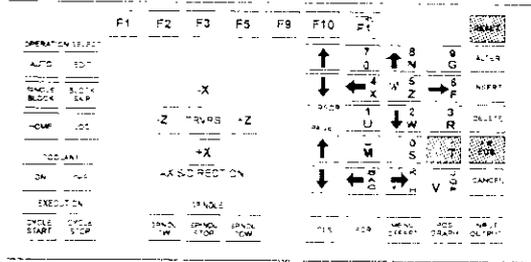
Keys Highlighted.

The following keys are used in this section:

[JOG], [T]

[NUMBERS] - not highlighted

[EOB]



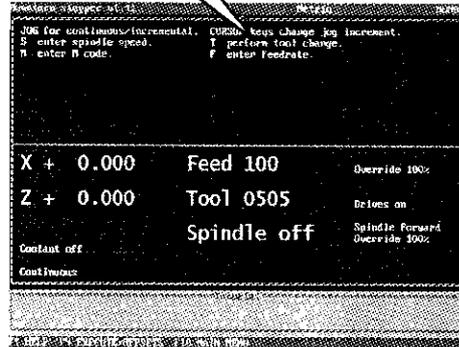
Tutor keypad.



Move the toolpost to a safe position before starting any tool changing operation.

To change a tool, the lathe needs to be set in "JOG MODE". This is selected by pressing the [JOG] key. A list of commands which operate in "JOG MODE" are displayed, one of which is tool change.

T perform tool change.

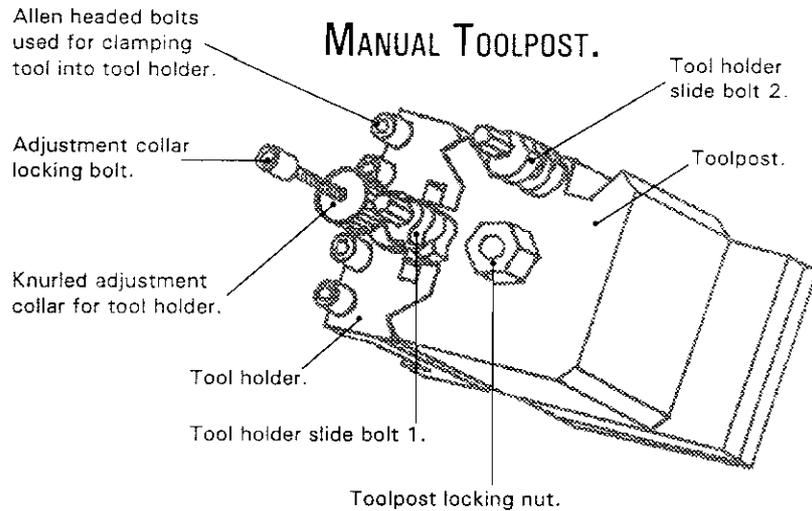


continued....

CHANGING THE TOOL USING A MANUAL TOOLPOST.

continued....

Next, open the machine door to gain access to the toolpost. Loosen each of the four allen headed bolts on the tool holder (turning anticlockwise) and remove the tool. Place the "new" tool into the tool holder and tighten each of the four allen headed bolts (turning clockwise) until the tool is fixed securely.



Adjust the tool tip, if necessary, so it is level with the spindle centreline. One method of achieving this is to use a drill or previously turned spike as a visual guide to the spindle centreline position. Move the tool close to this reference point, then adjust the tool holder height until the tool tip coincides exactly with the reference point.

The tool holder is released by turning the allen headed locking bolt anticlockwise. Move the tool holder to the correct position by rotating the knurled adjustment collar, then lock the tool holder in position by turning the allen headed locking bolt clockwise.

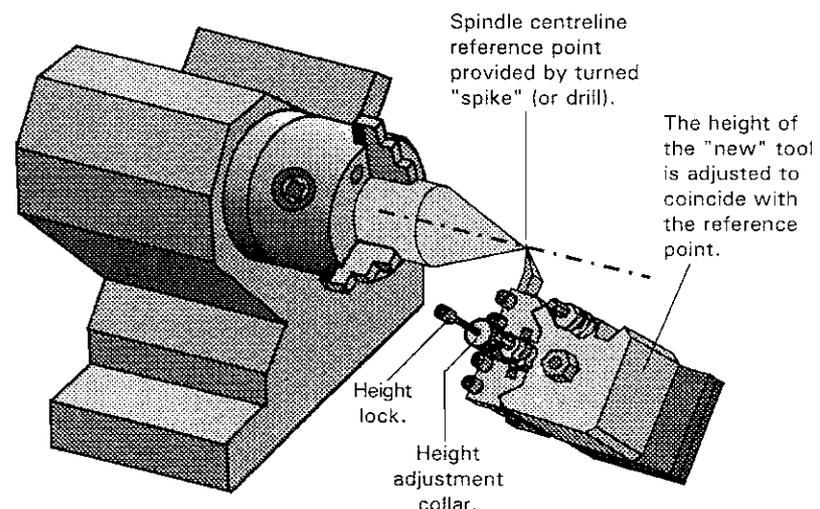


Diagram shows Manual Toolpost that homes in the front right-hand corner of the machine.

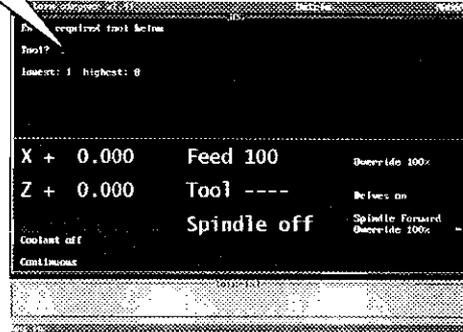
continued....

CHANGING THE TOOL USING A MANUAL TOOLPOST.

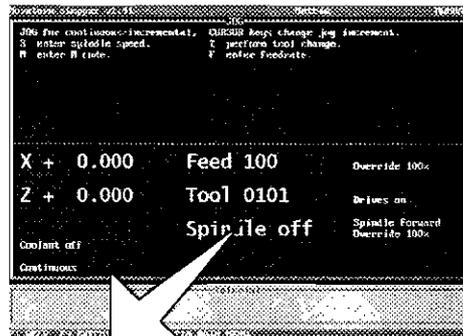
continued....

Next, enter the "new" number of the tool which has just been fitted to the machine, using the [NUMBERS] keys.

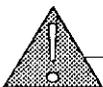
Enter required tool below
Tool? 1
lowest: 1 highest: 8



When the [EOB] key is pressed, the "new" tool number will be displayed "on-screen" confirming that the operation has been completed.



Tool 0101



Please note -
The co-ordinate values depicted
on screenshots are used for
illustrative purposes only.

SIMULATION MENU - SET TOOLING.

The machine controlling software contains a "Simulation" option, which allows any G-code file loaded to be tested without cutting the billet. Before using this option the on-screen graphics need to be set to match the tools numbers used by the G-code file and the machine toolpost.

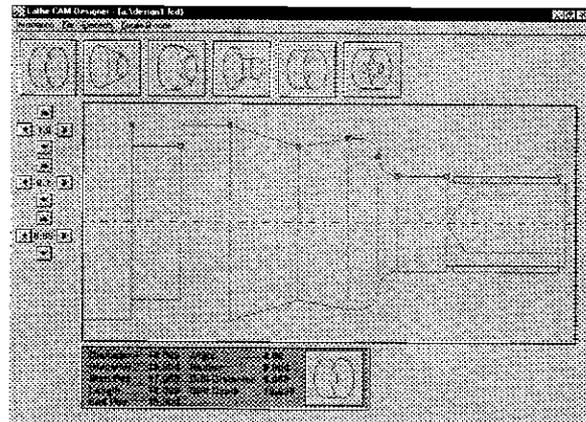
The *SET TOOLING* option is used to configure these graphics in the machine controlling software. This will ensure that the correct tools are drawn and used on the display screen at the right time.

The *SET TOOLING* option must be completed, if either the *EDIT AND SIMULATE* or *SIMULATE ONLY* options are to be used, ie, "on-screen" graphics will be used in the machine controlling software.

The following example shows how the tooling is set for the program produced in section 1.11 - "How to set-up LatheCAM Designer".

The design shown right was drawn using Denford's LatheCAM Designer software package.

After drawing the part, the tooling is set within LatheCAM so it matches the tool profiles present on the actual toolpost of the CNC lathe. This is shown by the table below....

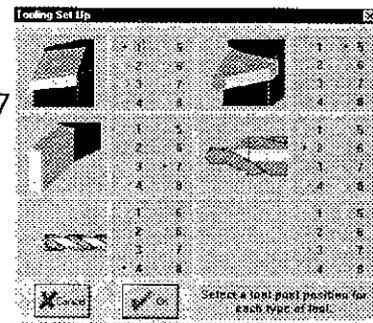


The following tools were present on the Novaturn toolpost:

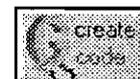
- Tool 1 - Roughing tool.
- Tool 2 - Centre drill.
- Tool 3 - Threading tool.
- Tool 4 - 6mm drill.
- Tool 5 - Threading tool.
- Tool 6 - No tool present.
- Tool 7 - Undercutting tool.
- Tool 8 - No tool present.

The tool numbers in the LatheCAM Designer software (ie, the actual CNC program) were set to match this:

- Roughing tool - 1.
- Undercutting tool - 7
- Drill - 4.
- Threading tool - 5.
- Centre drill - 2.



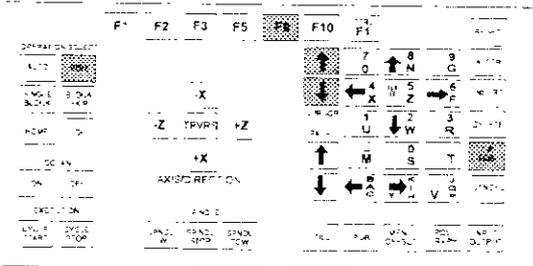
Next, the CNC File is generated using LatheCAM's post processors.



SIMULATION MENU - SET TOOLING.

Keys Helpbox.

The following keys are used in this section:
[EDIT], [F9]
[CURSOR ARROWS]
[EOB]

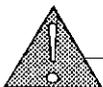
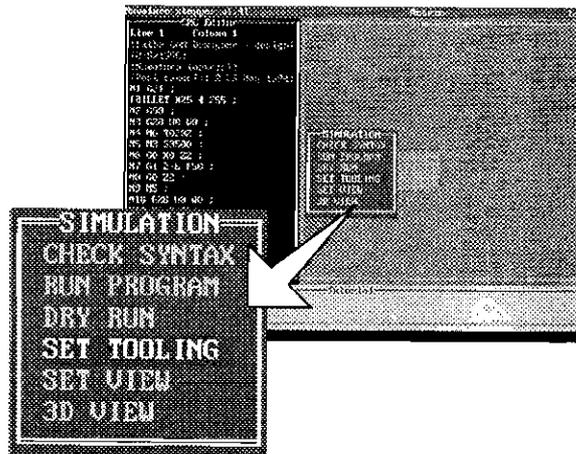


Tutor keypad.

The CNC File generated by LatheCAM's post-processor is transferred to the CNC lathe and loaded for "simulation" on the machine software.

To access the *SET TOOLING* option, first set the machine to operate in *EDIT MODE* by pressing the [EDIT] key. Next, select the *SIMULATION MENU* by pressing the [F9] key.

From the list of options, highlight 'SET TOOLING' using the [CURSOR ARROWS] keys and press [EOB].

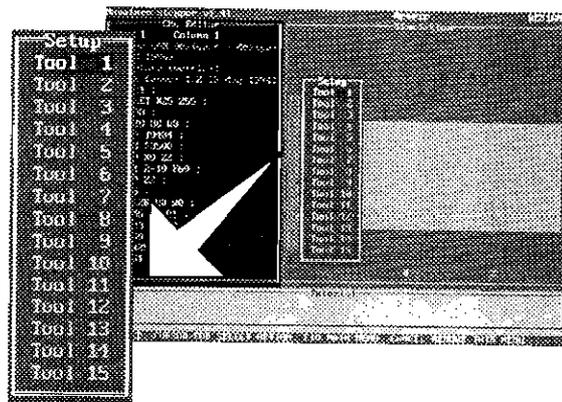


Please note -

The co-ordinate values depicted on screenshots are used for illustrative purposes only.

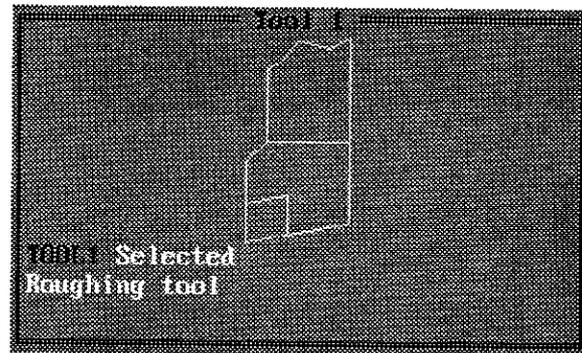
SIMULATION MENU - SET TOOLING.

Next, select the number of one of the tools from the list, using the [CURSOR ARROW] keys and press [EOB]. In this example, tool number 1 is highlighted.



Cycle through the green coloured tool screens available, using the [CURSOR ARROWS] keys. This tool screen needs to match the type of tool present in the selected tool number from your machine. In this example, tool number 1 on the machine toolpost was a "roughing" tool, so the tool screen was set to match this.

When the correct tool profile is found press the [EOB] key.



Tool 1 - Roughing tool.

SIMULATION

MENU -

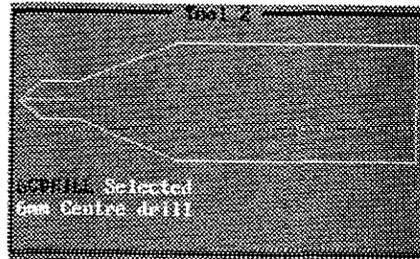
SET TOOLING.

Using the same method, set the tool screens for all the other tool numbers on your machine.

(These screenshots show how the graphics would be set to match the tool numbers in the LatheCAM software/CNC File and the toolpost, shown at the beginning of this section).

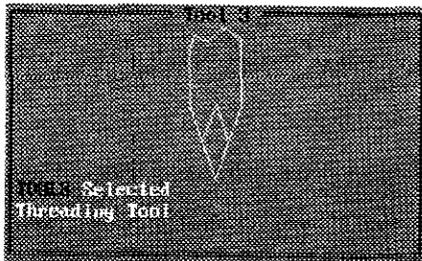
Setup
Tool 1
Tool 2
Tool 3
Tool 4
Tool 5
Tool 6
Tool 7
Tool 8
Tool 9
Tool 10
Tool 11
Tool 12
Tool 13
Tool 14
Tool 15

Tool 2 - Centre drill.



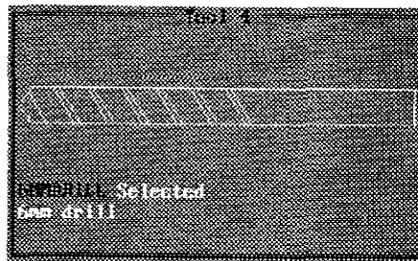
Setup
Tool 1
Tool 2
Tool 3
Tool 4
Tool 5
Tool 6
Tool 7
Tool 8
Tool 9
Tool 10
Tool 11
Tool 12
Tool 13
Tool 14
Tool 15

Tool 3 - Threading tool.



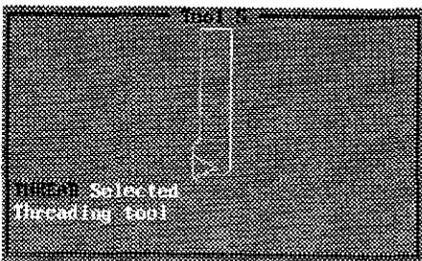
Setup
Tool 1
Tool 2
Tool 3
Tool 4
Tool 5
Tool 6
Tool 7
Tool 8
Tool 9
Tool 10
Tool 11
Tool 12
Tool 13
Tool 14
Tool 15

Tool 4 - 6mm drill.



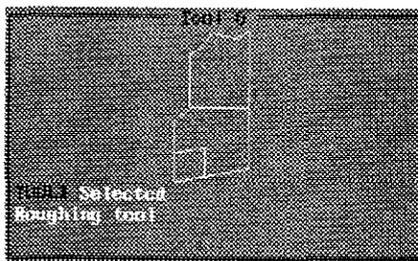
Setup
Tool 1
Tool 2
Tool 3
Tool 4
Tool 5
Tool 6
Tool 7
Tool 8
Tool 9
Tool 10
Tool 11
Tool 12
Tool 13
Tool 14
Tool 15

Tool 5 - Threading tool.



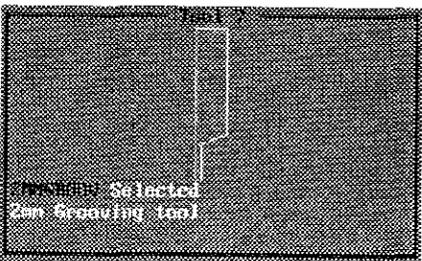
Setup
Tool 1
Tool 2
Tool 3
Tool 4
Tool 5
Tool 6
Tool 7
Tool 8
Tool 9
Tool 10
Tool 11
Tool 12
Tool 13
Tool 14
Tool 15

Tool 6 - No tool present.



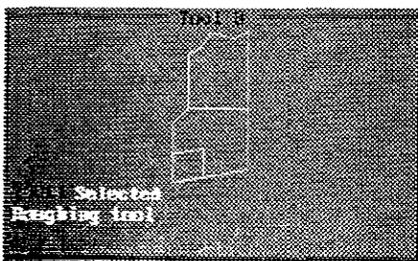
Setup
Tool 1
Tool 2
Tool 3
Tool 4
Tool 5
Tool 6
Tool 7
Tool 8
Tool 9
Tool 10
Tool 11
Tool 12
Tool 13
Tool 14
Tool 15

Tool 7 - Undercutting tool.



Setup
Tool 1
Tool 2
Tool 3
Tool 4
Tool 5
Tool 6
Tool 7
Tool 8
Tool 9
Tool 10
Tool 11
Tool 12
Tool 13
Tool 14
Tool 15

Tool 8 - No tool present.

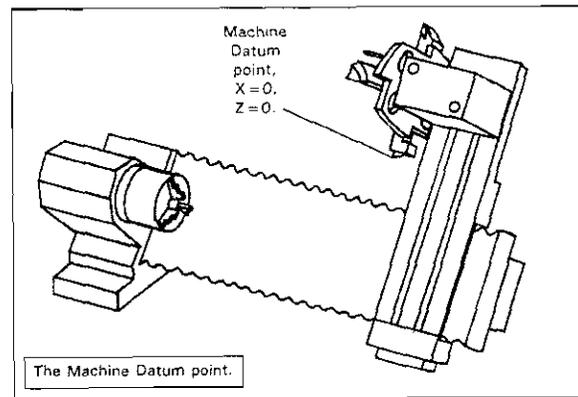


SETTING THE TOOL OFFSETS.

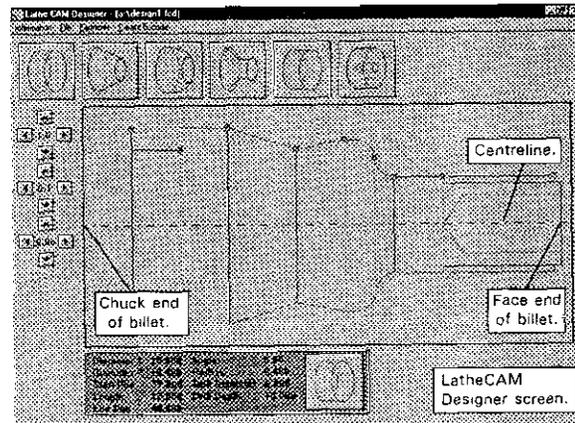
Setting the offsets tells each tool where the *actual* billet is positioned, so it can start cutting in the correct place.

Please Note - All diagrams show tools fitted to toolposts which home in the back right-hand corner of the machine (all Automatic Toolpost and most manual toolpost machines).

When the CNC lathe is first switched on, the machine datum point is set, by homing the machine. The machine datum point is the tip of the currently selected tool, with the co-ordinates $X=0$, $Z=0$ - see the diagram below....

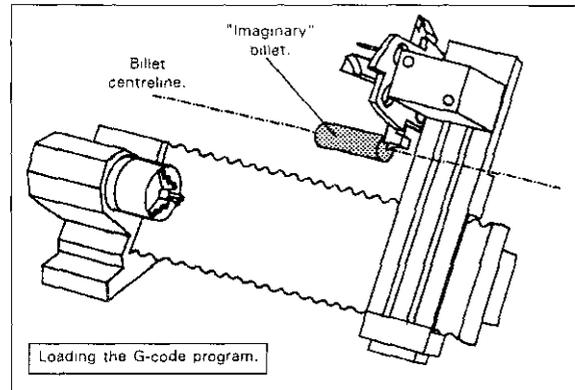


The G-code program from LatheCAM Designers post-processor describes the path taken by the tools when cutting the part. All these movement commands are relative to the position $X=0$, $Z=0$. This position is usually the face end of the billet along its centreline - see the screenshot below....



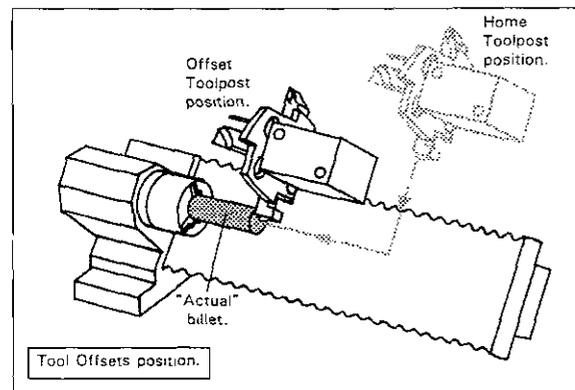
SETTING THE TOOL OFFSETS.

When this G-code program is loaded into the machine, the machine datum point (ie, the tip of the currently selected tool) is also the starting position for the G-code program. The program will presume that the billet that we want to cut is also positioned here - see the diagram below....

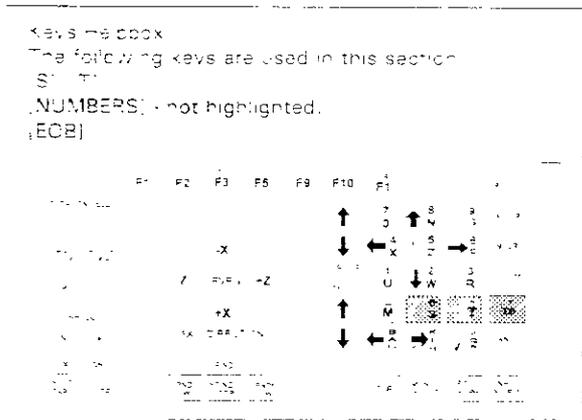


However, the *actual* billet, held in the chuck, is positioned in a different area of the machine. Therefore, the starting position for the G-code program must be moved, or "offset", so it coincides with the face end of the *actual* billet along its centreline - see the diagram below....

This amount of offset between the machine datum point for the currently selected tool and the fixed position of the actual billet will differ according to the type of tool, since all the tools differ in size. The offsets must be set individually for every tool that will be used to make the part.



SETTING THE X AND Z OFFSET VALUES.



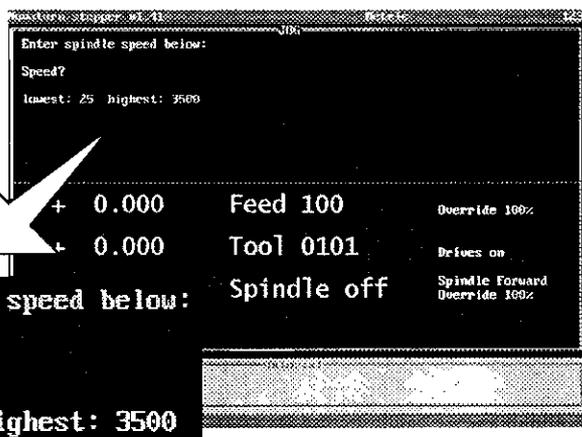
Tutor keypad.

2

Although the tips of lathe tools are very hard, they are also quite brittle. It is much safer to touch the tip of the tool on the billet when it is rotating to avoid damage on any of the tools.

A global value of 1000rpm, suitable for use with all the tools, can be set for the billet speed. To change the spindle speed press the [S] key.

Enter a value of "1000" using the [NUMBERS] keys and press [EOB] to set this value as the "new" spindle speed.



Enter spindle speed below:
Speed? 1000
lowest: 25 highest: 3500

Check that the tool displayed on screen is the correct tool number you want to use for setting the offsets. If not, the tool must be changed - check that the toolpost is at its home position, this avoids damage when the tools are changed (follow section 2.1 "Setting the Machine Datum Point" to home the machine). Press [T] to change tool numbers, enter the correct number of the tool you want to use and press [EOB]. Changing tools is covered in greater depth in section 3.3 "Changing Tools".

Enter required tool below
Tool?
lowest: 1 highest: 8

 Please note - The co-ordinate values depicted on screenshots are used for illustrative purposes only.

SETTING THE Z CO-ORDINATE VALUE.

Movement keys.....

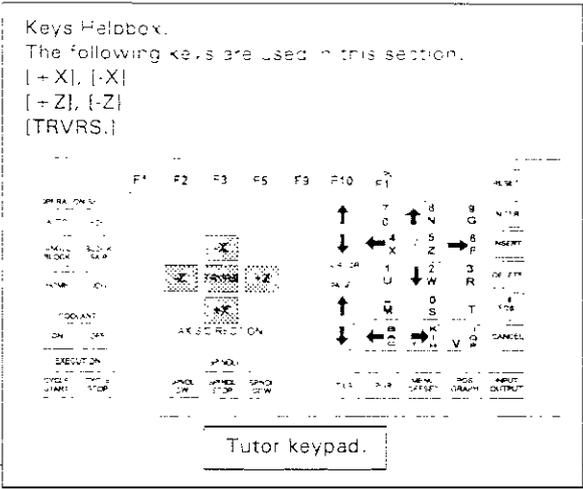
[+X] - moves the toolpost up/away from the front of the machine.

[-X] - moves the toolpost down/towards the front of the machine.

[+Z] - moves the toolpost right/away from the chuck.

[-Z] - moves the toolpost left/towards the chuck.

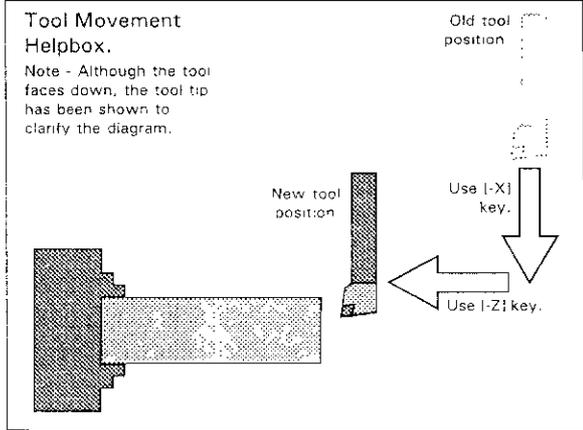
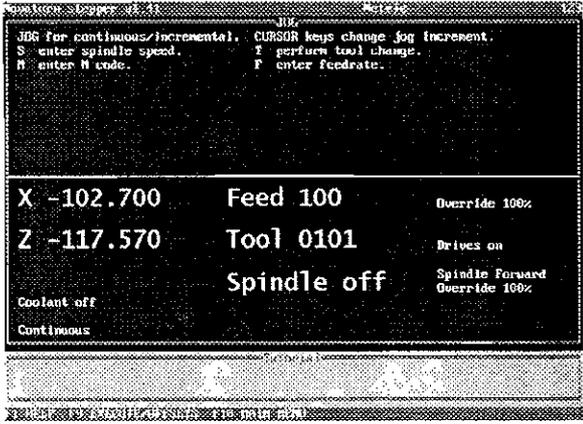
3



The Z co-ordinate value can be found by touching the tip of the tool onto the end surface of the billet.

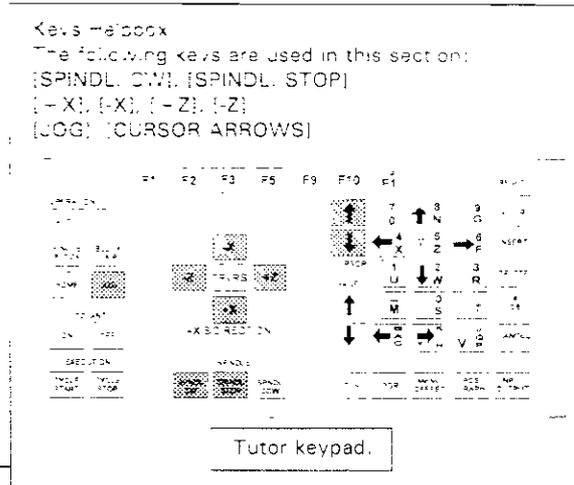
Check that *JOG MODE* is set to *CONTINUOUS*, then move the tool until it is close to the end of the billet, but not touching. Move the tool using the [+X], [-X], [+Z] and [-Z] keys. Pressing the [TRVRS.] key at the same time will increase this rate of movement - see the "Tool Movement Helpbox", below.

Notice that as the tool is moved, the X and Z co-ordinate values displayed on the screen will change. These numbers are the current co-ordinates of the cutting tool tip, relative to the last datum point set (if the machine has just been switched on, this will be the machine datum point).

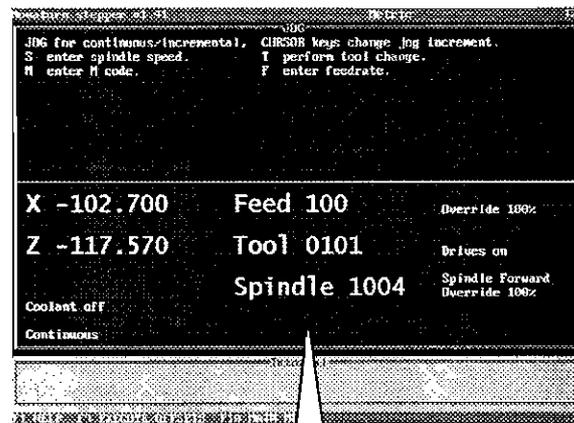


SETTING THE Z CO-ORDINATE VALUE.

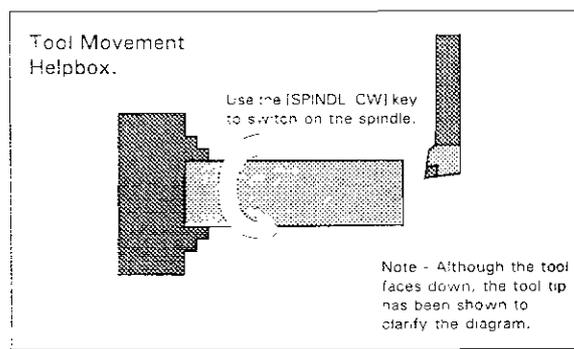
4



It is safer to touch the tip of the tool on the end of the billet when it is rotating. Switch on the spindle so it rotates in a clockwise direction by pressing the [SPINDL, CW] key. The at speed which the spindle rotates is displayed on screen - it should read approximately 1000rpm (this was set in the earlier part of this section).



Spindle 1004



When the tool has been moved fairly close to the end of the billet (within approx. 5mm) the [JOG] key can be used to switch from *CONTINUOUS* to *JOG STEP* - this is a more accurate degree of adjustment using stepped movements. continued....

Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

SETTING THE Z CO-ORDINATE VALUE.

4

continued... In Jog Step, the tool will move a set amount every time one of the movement keys is pressed. The size of this movement will be displayed, for example *JOG STEP 0.012* is a movement of 0.012mm every time a movement key is pressed.

Select the Jog Step value required using the blue [CURSOR ARROWS] keys. Jog Steps can be selected between a movement of 0.006mm minimum to 1mm maximum. The values given for the smaller sizes of Jog Steps may seem irregular - this is caused by the limitations imposed by the size of the stepped movements of the control motors.

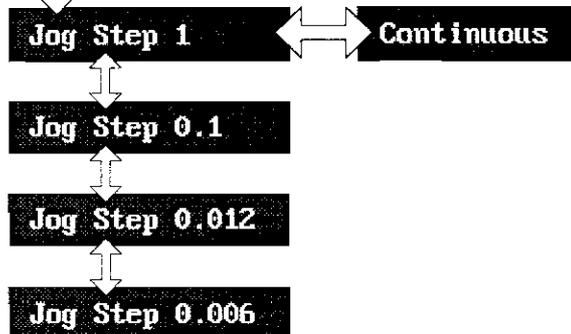
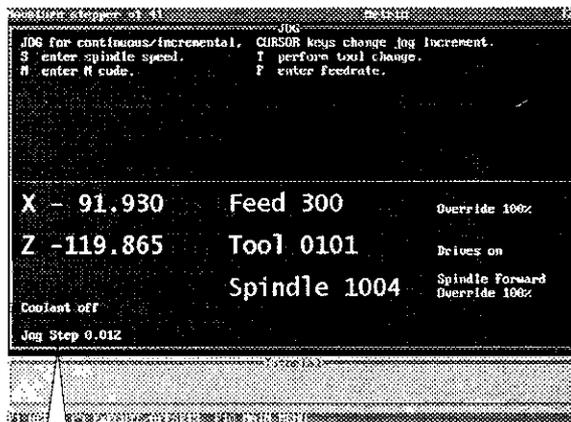
Movement keys.....

[+X] - moves the toolpost up/away from the front of the machine.

[-X] - moves the toolpost down/towards the front of the machine.

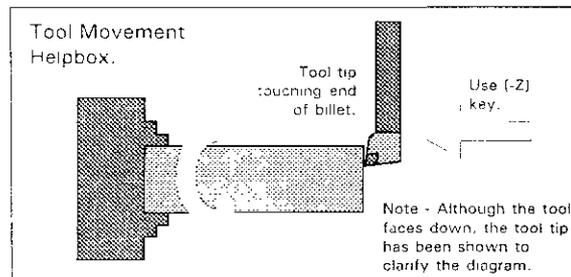
[+Z] - moves the toolpost right/away from the chuck.

[-Z] - moves the toolpost left/towards the chuck.

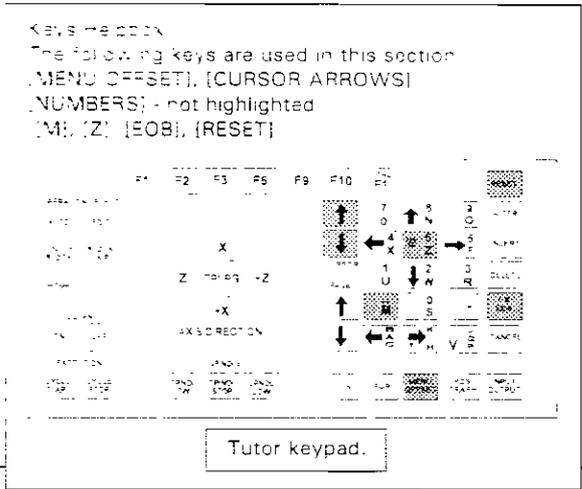


Using *JOG STEP* and the [+X], [-X], [+Z], [-Z] movement keys touch the tip of the tool on the end of the billet. You should hear when the tip starts to make contact - at this point stop advancing the tool into the billet - see the "Tool Movement Helpbox", below.

At this stage the spindle can be switched off, if required, by pressing the [SPINDL. STOP] key.



SETTING THE Z CO-ORDINATE VALUE.

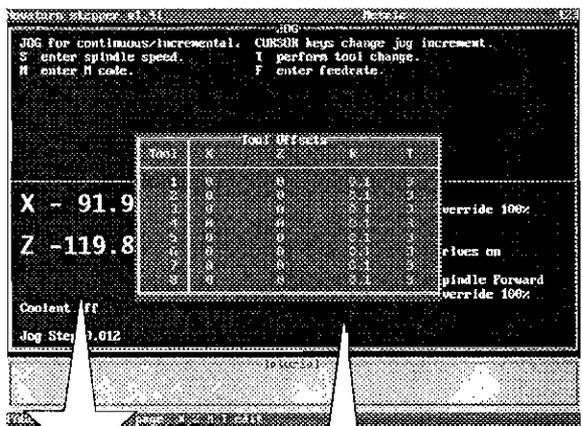


5

The Z Tool Offset value for the currently selected tool number must now be transferred into the *TOOL OFFSET TABLE*.

Make a note of the Z co-ordinate offset value from the main screen, then select the *TOOL OFFSET TABLE* by pressing the [MENU OFFSET] key.

Highlight the number of the tool currently being used with the blue [CURSOR ARROWS] keys.



Z -119.865
Main screen Z co-ordinate offset value.

Tool	X	Z	R	T
1	0	0	0.1	3
2	0	0	0.1	3
3	0	0	0.1	3
4	0	0	0.1	3
5	0	0	0.1	3
6	0	0	0.1	3
7	0	0	0.1	3
8	0	0	0.1	3

continued....

Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

SETTING THE Z CO-ORDINATE VALUE.

5

continued.... Next press the [M] key. This tells the machine that you want to "make" the current position of the tool read a particular value. An 'M' cursor should appear next to the tool number that has been highlighted.

Tool Offsets				
Tool	X	Z	R	T
M 1	0	0	0.1	3
2	0	0	0.1	3
3	0	0	0.1	3
4	0	0	0.1	3
5	0	0	0.1	3
6	0	0	0.1	3
7	0	0	0.1	3
8	0	0	0.1	3

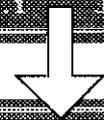
Press the [Z] key to move the cursor into the Z Tool Offset values column.

Use the [NUMBERS] keys to enter the Z Tool offset value as "zero".

(If the Z value is not being entered with this "M" prefix, enter the Z Tool Offset using the value noted earlier from the main screen. In this example, a value of -119.865 would be entered. This value has a "-" sign, since Z values to the left the current datum are negative).

Press the [EOB] key to confirm this value. The numbers which then appear in the Z Tool Offset column should be identical to the values noted from the main screen at the beginning of this section.

Tool Offsets				
Tool	X	Z	R	T
M 1	0	0	0.1	3
2	0	0	0.1	3
3	0	0	0.1	3
4	0	0	0.1	3
5	0	0	0.1	3
6	0	0	0.1	3
7	0	0	0.1	3
8	0	0	0.1	3



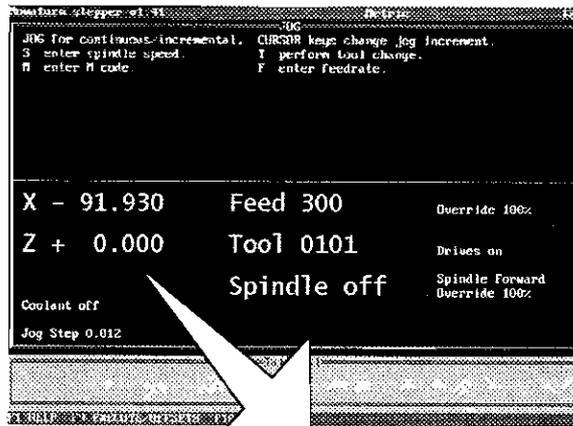
Tool Offsets				
Tool	X	Z	R	T
1	0	119.86	0.1	3
2	0	0	0.1	3
3	0	0	0.1	3
4	0	0	0.1	3
5	0	0	0.1	3
6	0	0	0.1	3
7	0	0	0.1	3
8	0	0	0.1	3

continued....

SETTING THE Z CO-ORDINATE VALUE.

5

continued.... To register this value as the Z Tool Offset, press the [RESET] key. The main screen Z co-ordinate should return to "zero", indicating that this point is now recognised by the machine as the "new" Z datum.



Z + 0.000

Main screen Z co-ordinate value.

Recap....

The [M] key allows you to "make" the current position of the tool read a particular value. For example, [M], [Z], [0] would mean "make Z offset read zero at the current tool position".



Please note -

The co-ordinate values depicted on screenshots are used for illustrative purposes only.

SETTING THE X CO-ORDINATE VALUE.

Movement keys.....

[+X] - moves the toolpost up/away from the front of the machine.

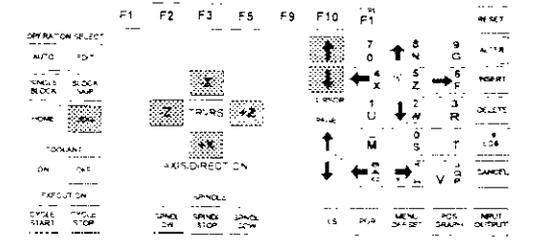
[-X] - moves the toolpost down/towards the front of the machine.

[+Z] - moves the toolpost right/away from the chuck.

[-Z] - moves the toolpost left/towards the chuck.

6

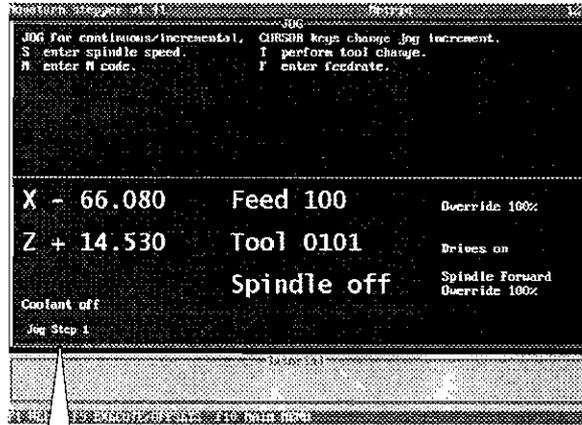
Keys Helpbox.
The following keys are used in this section:
[JOG], [CURSOR ARROWS],
[+X], [-X], [+Z], [-Z]



Tutor keypad.

The X co-ordinate value can be found by touching the tip of the tool onto the side surface of the billet.

Currently, the tool is touching the end of the billet. Select *JOG MODE* and set it on *JOG STEP* movements, by pressing the [JOG] key until the words '*JOG STEP....*' are displayed on the screen (select the step size using the [CURSOR ARROWS] keys). Move the tool clear from the billet in stepped movements, using the [+X], [-X], [+Z] and [-Z] keys.



Jog Step 1

continued....

SETTING THE X CO-ORDINATE VALUE.

6

continued.... When the tool is sufficiently clear from the billet, set the *JOG MODE* to *CONTINUOUS* by pressing the [JOG] key until *CONTINUOUS* is displayed on the screen. Keep moving the tool until you are sure it will be able to pass along the side of the billet safely (with a gap of at least 5mm) - see the "Tool Movement Helpbox", below.

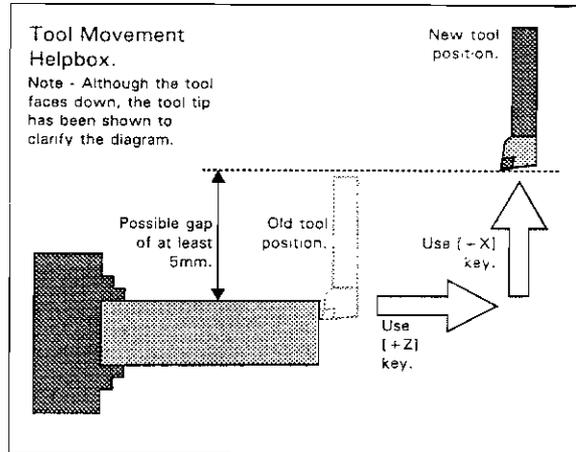
Movement keys.....

[+X] - moves the toolpost up/away from the front of the machine.

[-X] - moves the toolpost down/towards the front of the machine.

[+Z] - moves the toolpost right/away from the chuck.

[-Z] - moves the toolpost left/towards the chuck.



Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

SETTING THE X CO-ORDINATE VALUE.

Movement keys.....

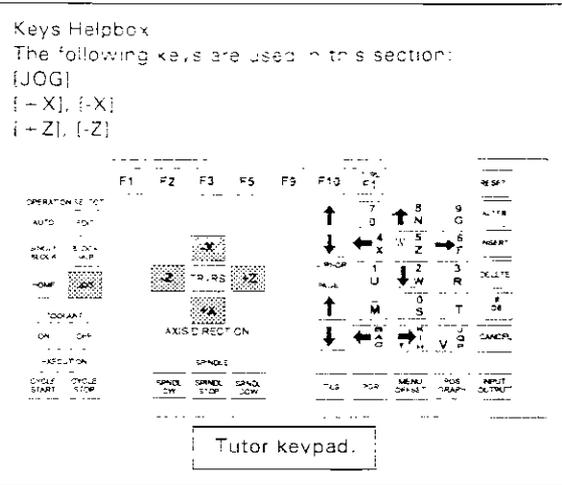
[+X] - moves the toolpost up/away from the front of the machine.

[-X] - moves the toolpost down/towards the front of the machine.

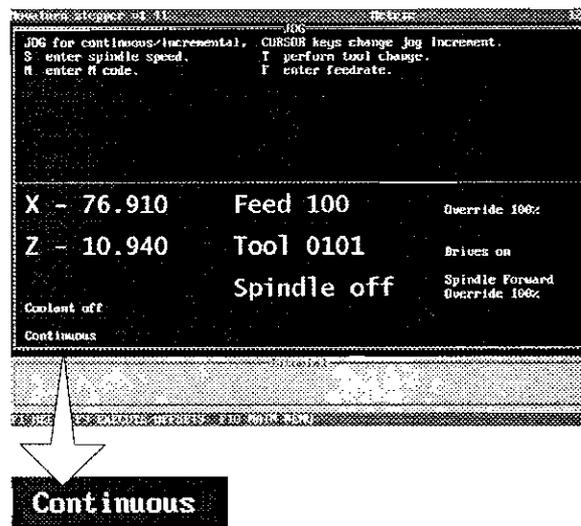
[+Z] - moves the toolpost right/away from the chuck.

[-Z] - moves the toolpost left/towards the chuck.

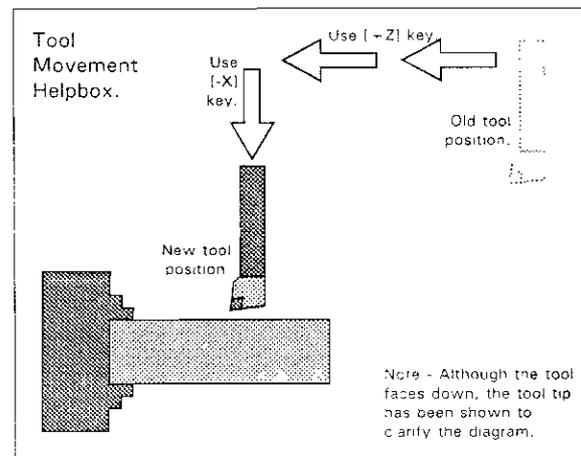
7



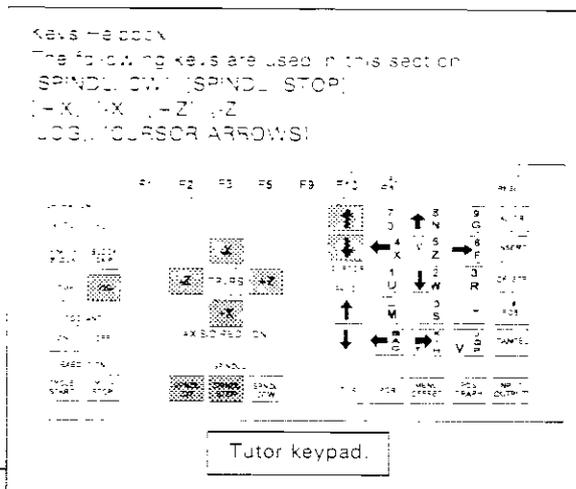
Check that *JOG MODE* is set to *CONTINUOUS* by pressing the [JOG] key until *CONTINUOUS* is displayed on the screen.



Move the tool until it is positioned along the side of the billet, but not touching the billet surface, using the [+X], [-X], [+Z] and [-Z] keys - see the "Tool Movement Helpbox", below.



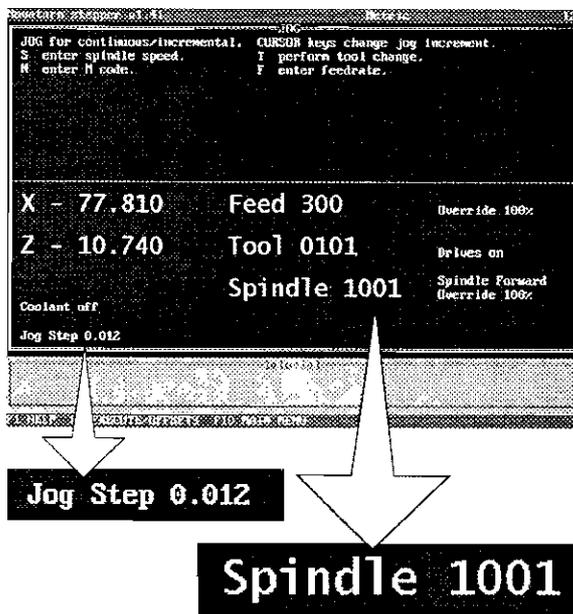
SETTING THE X CO-ORDINATE VALUE.



8

It is safer to touch the tip of the tool on the side of the billet when it is rotating. Switch on the spindle so it rotates in a clockwise direction by pressing the [SPINDL. CW] key. The at speed which the spindle rotates is displayed on screen - it should read approximately 1000rpm (this was set in the earlier part of this section).

Select *JOG MODE* and set it on *JOG STEP* movements, by pressing the [JOG] key until the words 'JOG STEP....' are displayed on the screen (select the step size using the [CURSOR ARROWS] keys).



continued...

 Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

SETTING THE X CO-ORDINATE VALUE.

8

continued.... Using *JOG STEP* and the [+X], [-X], [+Z], [-Z] movement keys touch the tip of the tool on the side of the billet. You should hear when the tip starts to make contact - at this point stop advancing the tool into the side of the billet - see the "Tool Movement Helpbox", below.

At this stage the spindle can be switched off, if required, by pressing the [SPINDL. STOP] key.

Movement keys.....

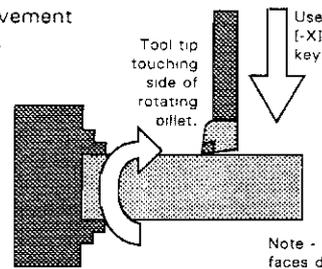
[+X] - moves the toolpost up/away from the front of the machine.

[-X] - moves the toolpost down/towards the front of the machine.

[+Z] - moves the toolpost right/away from the chuck.

[-Z] - moves the toolpost left/towards the chuck.

Tool Movement Helpbox.



Note - Although the tool faces down, the tool tip has been shown to clarify the diagram.

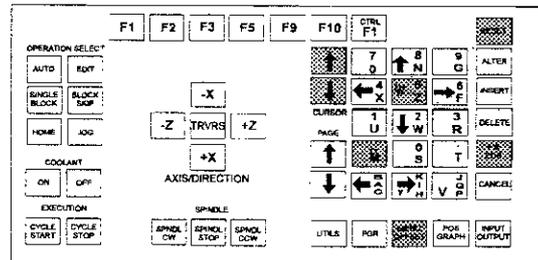
Keys Helpbox.

The following keys are used in this section:

[MENU OFFSET], [CURSOR ARROWS]

[NUMBERS] - not highlighted.

[M], [Z], [EOB], [RESET]



Tutor keypad.

9

The X Tool Offset value for the currently selected tool number must now be transferred into the *TOOL OFFSET TABLE*.

Make a note of the X co-ordinate offset value from the main screen, then select the *TOOL OFFSET TABLE* by pressing the [MENU OFFSET] key.

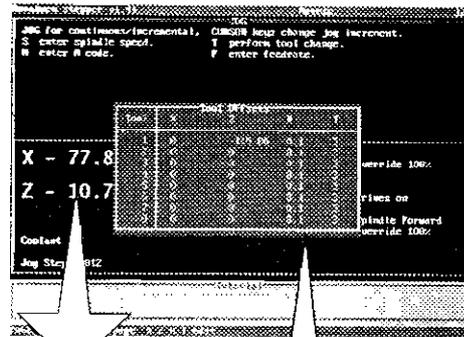
Highlight the number of the tool currently being used with the blue [CURSOR ARROWS] keys.

continued....

SETTING THE X CO-ORDINATE VALUE.

9

continued....



X - 77.810

Main screen X co-ordinate offset value.

Tool Offsets				
Tool	X	Z	R	T
1	0	-119.86	0.1	3
2	0	0	0.1	3
3	0	0	0.1	3
4	0	0	0.1	3
5	0	0	0.1	3
6	0	0	0.1	3
7	0	0	0.1	3
8	0	0	0.1	3

Next press the [M] key. This tells the machine that you want to "make" the current position of the tool read a particular value. An 'M' cursor should appear next to the tool number that has been highlighted.

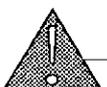
Tool Offsets				
Tool	X	Z	R	T
M 1	0	-119.86	0.1	3
2	0	0	0.1	3
3	0	0	0.1	3
4	0	0	0.1	3
5	0	0	0.1	3
6	0	0	0.1	3
7	0	0	0.1	3
8	0	0	0.1	3

Press the [X] key to move the cursor into the X Tool Offset values column.

Use the [NUMBERS] keys to enter the X Tool offset value - the value that must be entered is the diameter of the billet (measured in millimetres) - in this example, a 25.4mm diameter piece of steel bar was used, so the value entered was 25.4.

(If the X value is not being entered with this "M" prefix, the value which must be entered is the X Tool Offset value noted earlier from the main screen added to the diameter of the billet - for this example the values would be $77.81 - 25.4 = 103.21$. This value must be typed in as -103.21, since X values below the current datum are negative).

Press the [EOB] key to confirm this value. continued....



Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

SETTING THE X CO-ORDINATE VALUE.

9

continued....

Tool Offsets				
Tool	X	Z	R	T
M 1	25.40	-119.86	0.1	3
2	0	0	0.1	3
3	0	0	0.1	3
4	0	0	0.1	3
5	0	0	0.1	3
6	0	0	0.1	3
7	0	0	0.1	3
8	0	0	0.1	3



Tool Offsets				
Tool	X	Z	R	T
1	-103.21	-119.86	0.1	3
2	0	0	0.1	3
3	0	0	0.1	3
4	0	0	0.1	3
5	0	0	0.1	3
6	0	0	0.1	3
7	0	0	0.1	3
8	0	0	0.1	3

To register this value as the X Tool Offset, press the [RESET] key. The main screen X co-ordinate should change to show the value of the billet diameter, indicating that the X datum is now recognised as the centreline running through the billet.

Recap....

The [M] key allows you to "make" the current position of the tool read a particular value. For example, [M], [X], [0] would mean "make X offset read zero at the current tool position".

300 For continuous/incremental, CURSOR keys change jog increment.
 S enter spindle speed. T perform tool change.
 R enter R code. F enter feedrate.

X + 25.400 Feed 300 Override 100%
 Z - 10.74 Tool 0101 Drives on
 Spindle off Spindle Forward
 Coolant off Override 100%
 Jog Step 0.012

X + 25.400

Main screen X co-ordinate value

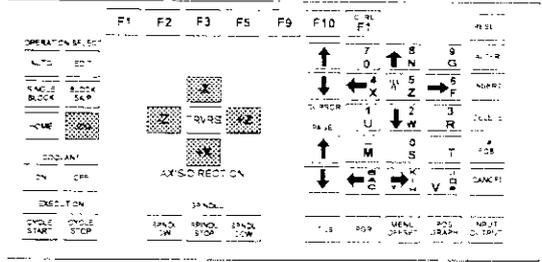
SETTING THE TOOL OFFSET OF THE NEXT TOOL.

Please note -
Tool Offsets need to be set for each Tool profile/number used on the billet.



10

Keys Helpbox.
The following keys are used in this section:
[JOG]
[+X], [-X]
[+Z], [-Z]



Tutor keypad.

The offset values for the currently selected tool have now been set and registered.

Due to the different length, size and shapes of the different tool profiles, the offset values must be set individually for every tool that will be used to machine the billet.

The current tool must be removed and replaced by a different tool that will be used to cut the billet. This "new" tool number must then have its own offset values set.

Before performing a tool change, check that the toolpost is at its home position. This avoids damage when the tools are changed (follow section 2.1 - "Setting the Machine Datum Point" to home the machine).

Press [T] to change tool numbers, enter the correct number of the tool you want to use and press [EOB]. Changing tools is covered in greater depth in section 3.3 - "Changing the Tool".

Enter required tool below
Tool?
lowest: 1 highest: 8

SETTING THE TOOL OFFSETS - ROUGHING OR FINISHING TOOL.

This section shows the touching points used when setting the offsets, for the most common tools used on a CNC lathe....

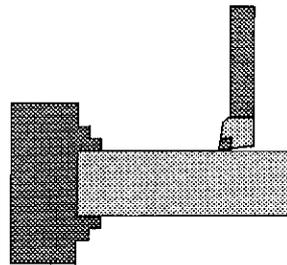
ROUGHING OR FINISHING TOOL.

The roughing tool is used for removing large amounts of material and general shaping down to within around 1mm of the required shape.

The finishing tool is used for light cuts on the workpiece to give a high quality finish to its surfaces.

X TOUCHING-ON POINT.

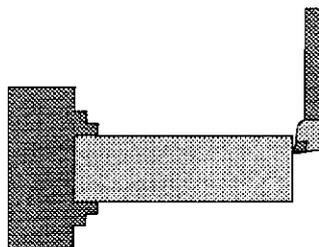
Touch the pointed tip of the tool onto the side of the rotating billet. Enter the X value following the procedures outlined in section ⑤ of setting the offsets.



Note - Although the tool faces down, the tool tip has been shown to clarify the diagram.

Z TOUCHING-ON POINT.

Touch the pointed tip of the tool onto the face end of the rotating billet. Enter the Z value following the procedures outlined in section ⑨ of setting the offsets.



Note - Although the tool faces down, the tool tip has been shown to clarify the diagram.

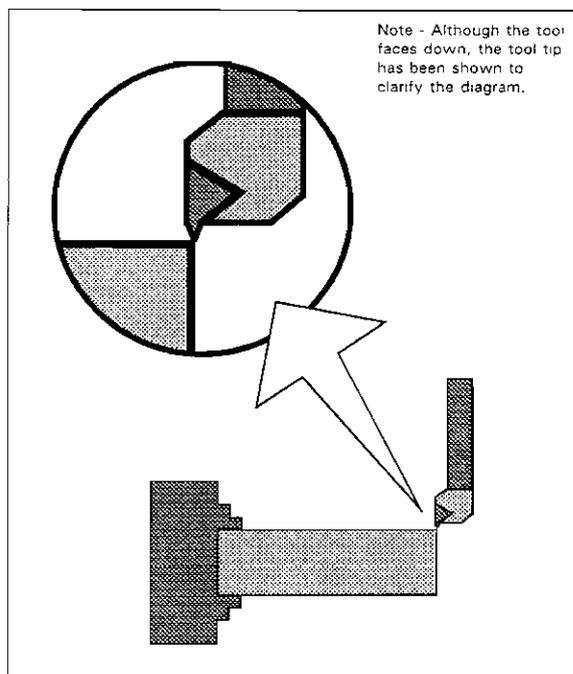
SETTING THE TOOL OFFSETS - THREADING TOOL.

THREADING TOOL.

The threading tool has a profile ground to the angle of the external thread required.

Z AND X TOUCHING-ON POINT.

The Z and X co-ordinates are found when the tip of the tool is touching the outer rim of the billet. Enter these values following the procedures outlined in sections ⑤ (Z) and ⑨ (X) of setting the offsets. Note that the billet does not need to be rotating. Do not let the tip of the tool hit the billet, since it may be damaged.



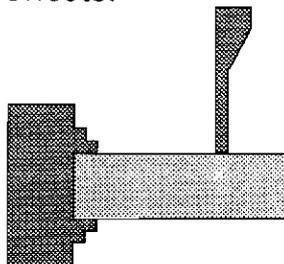
SETTING THE TOOL OFFSETS - PARTING TOOL.

PARTING TOOL.

The parting-off tool is used to separate the finished workpiece from the stock billet.

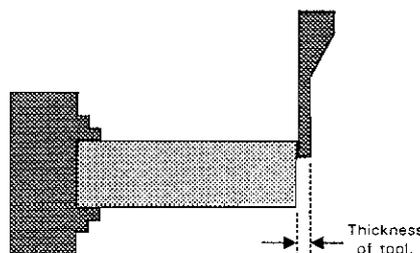
X TOUCHING-ON POINT.

Touch the tip of the tool onto the side of the rotating billet. Enter the X value following the procedures outlined in section ⑤ of setting the offsets.



Z TOUCHING-ON POINT.

Touch the tip of the tool onto the face end of the rotating billet. Enter the Z value following the procedures outlined in section ⑨ of setting the offsets.



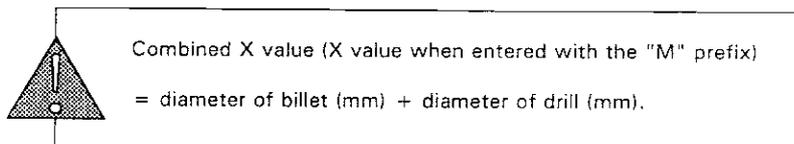
Note - Although the left side of the tool is used for touching the face end of the billet, it is the right side of the tool that will be touching the end of the workpiece when it is parted-off. The thickness of the parting-off tool is automatically compensated for by LatheCAM Designer software.

SETTING THE TOOL OFFSETS - DRILLS.

DRILLS - ENTERING THE X VALUE INTO THE TOOL OFFSETS TABLE.

The X value, when entered with the "M" prefix, relies on the tip of the tool being touched on the side of the billet. For the X datum to be the centreline of the billet, the value entered must be the billet diameter.

However, a drill uses its side, not its tip, to touch on the side of the billet. The diameter of the drill must be included in this X value....



Note - once this value has been set and registered in the offsets table for one drill, the offset value displayed will be identical for any other type of drill used on the same program. On some machines, this offset value is displayed on the machine door. For example, a Denford Mirac lathe displays '*X Datum to centreline X-149*', so the value entered into the X offsets column would be -149.

If the X centreline value is known, or has been previously set, only the Z value needs to be found. This X offset value can be copied into the X offsets column without using the 'M' prefix.

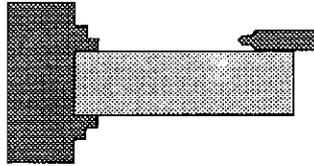
SETTING THE TOOL OFFSETS - CENTRE DRILL.

CENTRE DRILL.

The centre drill is used to mark the position of a hole before the twist drill is used.

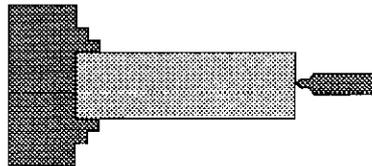
X TOUCHING-ON POINT.

Touch the side of the centre drill (its widest part) onto the side of the rotating billet. Enter the X value following the procedures outlined in section ⑤ of setting the offsets.



Z TOUCHING-ON POINT.

Touch the tip of the centre drill onto the face end of the rotating billet. Enter the Z value following the procedures outlined in section ⑨ of setting the offsets.



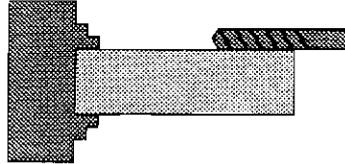
SETTING THE TOOL OFFSETS - TWIST DRILL.

TWIST DRILL.

The twist drill is used to bore holes along the centreline of the chuck.

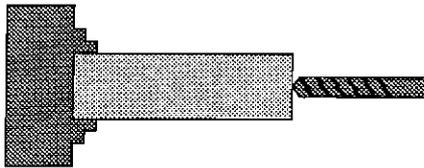
X TOUCHING-ON POINT.

Touch the side of the twist drill onto the side of the rotating billet. Enter the X value following the procedures outlined in section ⑤ of setting the offsets.



Z TOUCHING-ON POINT.

Touch the tip of the twist drill onto the face end of the rotating billet. Enter the Z value following the procedures outlined in section ⑨ of setting the offsets.



CONTROL MENU.

Keys Helpbox
The following keys are used in this section:
[F9]
[RESET]

Tutor keypad.

! Please note -
The function of the [F9] key will depend on the machine mode:
AUTO MODE = Control Menu.
EDIT MODE = Simulation Menu.
HOME MODE = Control Menu.
JOG MODE = Control Menu.

Select the *OFFSET CONTROL MENU* by pressing the [F9] key.

CONTROL

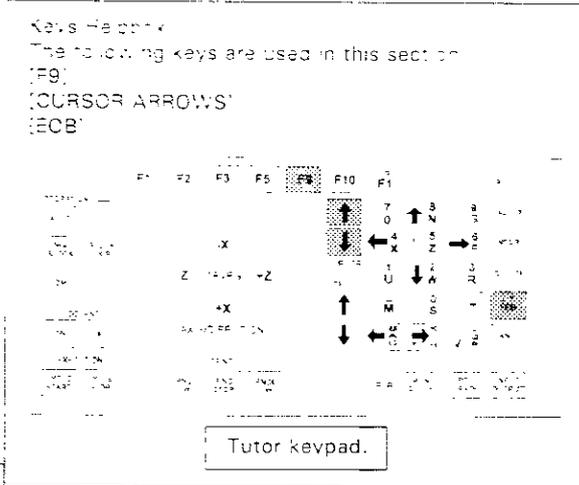
- EXECUTE CNC
- DRY RUN
- TOOL OFFSETS
- LOAD TOOL OFFSETS
- SAVE TOOL OFFSETS
- DELETE TOOL OFFSETS

- The *OFFSET CONTROL MENU* contains 6 options :
- i) Execute CNC - run the currently loaded CNC File on the machine.
 - ii) Dry Run - check the syntax and machining requirements for the currently loaded CNC File.
 - iii) Tool Offsets - display the Tool Offset Table, from which individual offset values can be altered.
 - iv) Load Tool Offsets - load a collection of Tool Offsets.
 - v) Save Tool Offsets - save a collection of Tool Offsets.
 - vi) Delete Tool Offsets - delete a selected collection of Tool Offsets.

Press the [RESET] key to clear any menus.

! Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

CONTROL MENU - EXECUTE CNC.

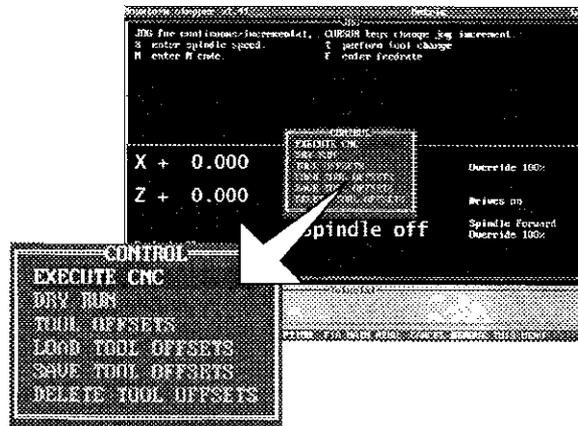


! Please note -
The function of the [F9] key will depend on the machine mode:
AUTO MODE = Control Menu.
EDIT MODE = Simulation Menu.
HOME MODE = Control Menu.
JOG MODE = Control Menu.

The '*EXECUTE CNC*' command will run any CNC File currently loaded.

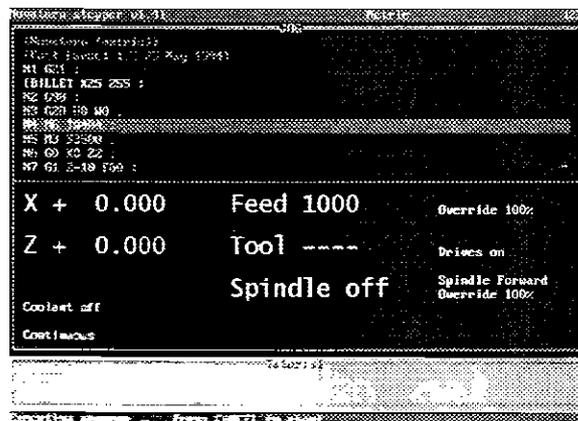
To run the currently loaded CNC File, select the *OFFSET CONTROL MENU* by pressing the [F9] key (not available in *EDIT MODE*).

Highlight the '*EXECUTE CNC*' option using the blue [CURSOR ARROWS] keys and press the [EOB] key.



The machine will switch to *AUTO MODE* and run the currently loaded CNC File.

If no CNC File is present, the *OFFSET CONTROL MENU* will be removed from the current display.



! Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

CONTROL MENU - DRY RUN.

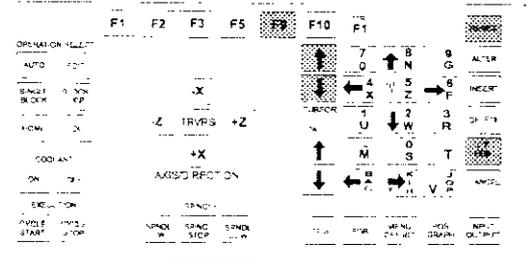
Keys Helpbox

The following keys are used in this section:

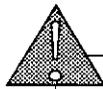
[F9]

[CURSOR ARROWS]

[EOB], [RESET]



Tutor keypad.



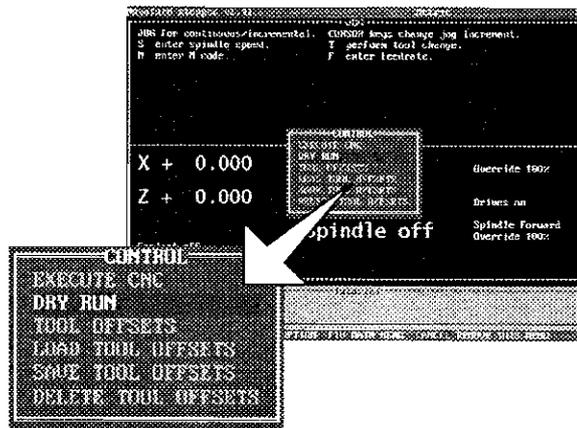
Please note -

The function of the [F9] key will depend on the machine mode:
AUTO MODE = Control Menu.
EDIT MODE = Simulation Menu.
HOME MODE = Control Menu.
JOG MODE = Control Menu.

The '*DRY RUN*' command will run the currently loaded CNC File in the computer memory. '*DRY RUN*' will check the CNC File for validity of both the G-code syntax and machining requirements (eg, axes limits, speeds and feeds suitable for the machine model used).

To check the currently loaded CNC File, select the *OFFSET CONTROL MENU* by pressing the [F9] key (not available in *EDIT MODE*).

Highlight the '*DRY RUN*' option using the blue [CURSOR ARROWS] keys and press the [EOB] key.

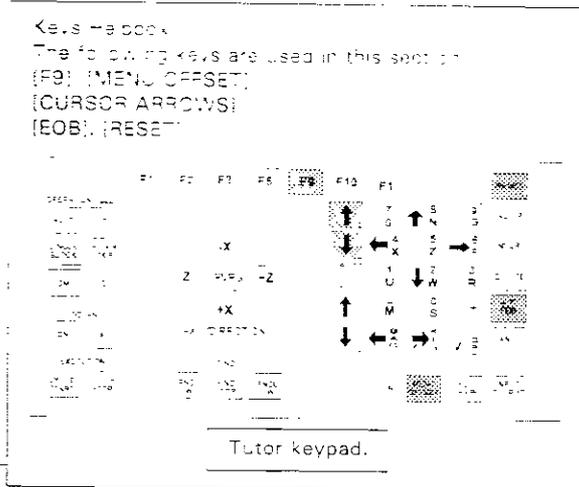


The *OFFSET CONTROL MENU* will be automatically cleared as the '*DRY RUN*' operation is started. If the '*DRY RUN*' operation is successful, no message windows will be displayed.

Note - The '*DRY RUN*' of a CNC File is performed very quickly. On short CNC Files it may appear that nothing has happened due to the speed at which the operation is performed.

A message window relating to any errors encountered will be displayed if part of the CNC File is incorrect (note - in *EDIT MODE*, the line in the CNC File where the error occurred is also highlighted). Remove the error message window by pressing the [RESET] key and correct the highlighted error.

CONTROL MENU - DISPLAY TOOL OFFSETS TABLE.



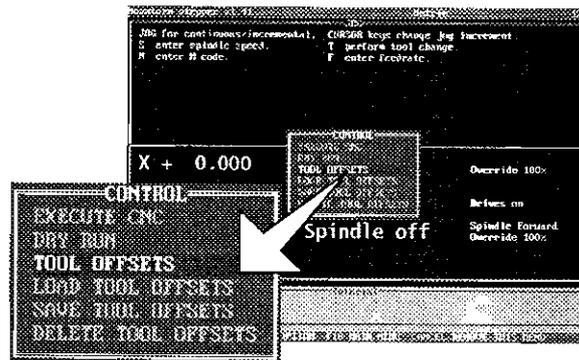
! Please note -
The function of the [F9] key will depend on the machine mode:
AUTO MODE = Control Menu.
EDIT MODE = Simulation Menu.
HOME MODE = Control Menu.
JOG MODE = Control Menu.

The '*TOOL OFFSETS*' command will display a table showing the offset values assigned to the current collection of tool (numbers).

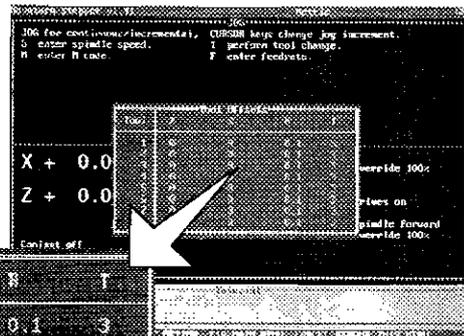
To display the Tool Offsets table, press the [F9] key (not available in *EDIT MODE*).

Highlight the '*TOOL OFFSETS*' option using the blue [CURSOR ARROWS] keys and press the [EOB] key.

Note that the *TOOL OFFSET TABLE* can be directly accessed by pressing the [MENU OFFSET] key (not available in *EDIT MODE*).



See section 4.1 - "Setting the Tool Offsets" for details about entering and editing the individual values in the Tool Offset table.



Tool	X	Z	R	T
1	0	0	0.1	3
2	0	0	0.1	3
3	0	0	0.1	3
4	0	0	0.1	3
5	0	0	0.1	3
6	0	0	0.1	3
7	0	0	0.1	3
8	0	0	0.1	3

Press the [RESET] key to clear any tables or menus.

! Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

CONTROL MENU - LOAD TOOL OFFSETS.

Keys Helpbox.
The following keys are used in this section
[F9], [CURSOR ARROWS],
[NUMBERS] - not highlighted
[DELETE], [EOB], [RESET]

Tutor keypad.

! Please note -
The function of the [F9] key will depend on the machine mode:
AUTO MODE = Control Menu.
EDIT MODE = Simulation Menu.
HOME MODE = Control Menu.
JOG MODE = Control Menu.

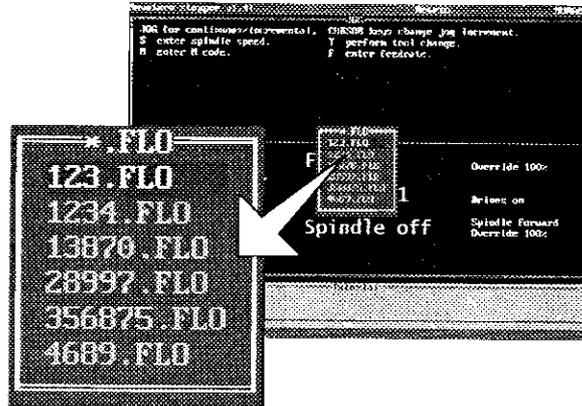
! Please Note - your software will be set to read either the computers hard drive (usually C:) or the floppy disk drive (usually A:) by default. If you do not want to load the Tool Offset Files from the default drive, then the drive destination must be changed. See page 4.33 "Changing the Drive Directory - Tool Offset Files".

To load a collection of Tool Offsets from the currently selected drive, select the *OFFSET CONTROL MENU* by pressing the [F9] key (not available in *EDIT MODE*). Highlight the '*LOAD TOOL OFFSETS*' option using the blue [CURSOR ARROWS] keys and press the [EOB] key.

Enter the file name (number) for the collection of Tool Offsets using the [NUMBER] keys. Use the [DELETE] key to remove any incorrectly typed characters and confirm the filename by pressing the [EOB] key.

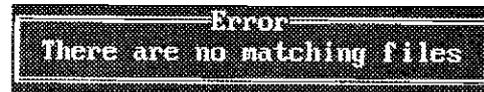
CONTROL MENU - LOAD TOOL OFFSETS.

If the file name is unknown, press the [EOB] key to list all the offset files saved on the currently selected drive. Files within these lists can be loaded by highlighting them using the [CURSOR ARROWS] and pressing the [EOB] key.



If no files can be found on the selected drive, an error message will be displayed. To clear this message, press the [RESET] key.

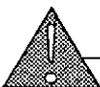
In most cases, this message (shown below) is displayed when the computer is trying to read the files from the wrong drive or directory - see page 4.33 "Changing the Drive Directory - Tool Offset Files".



The error message shown below may be displayed if the file you want to load has the wrong extension for the machine being used, or the filename has been typed and entered incorrectly on the tutor keypad.



Press the [RESET] key to clear any error messages or menus.



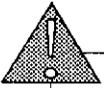
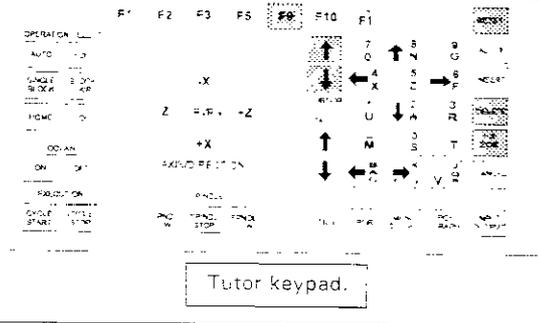
Please note -

The co-ordinate values depicted on screenshots are used for illustrative purposes only.

CONTROL MENU - SAVE TOOL OFFSETS.

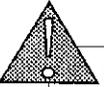
Keys Helpbox.

The following keys are used in this section:
[F9], [CURSOR ARROWS];
[NUMBERS] - not highlighted;
[DELETE], [EOB], [RESET]



Please note -

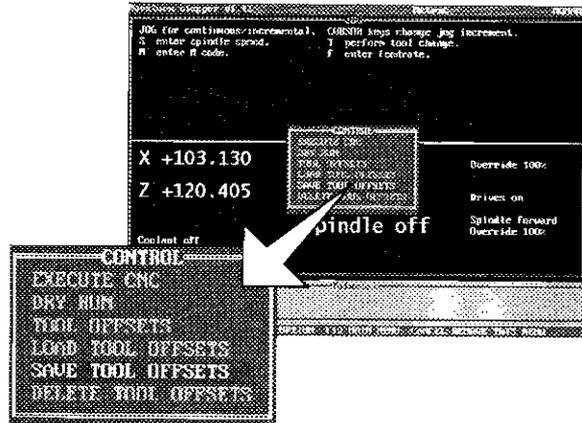
The function of the [F9] key will depend on the machine mode:
AUTO MODE = Control Menu.
EDIT MODE = Simulation Menu.
HOME MODE = Control Menu.
JOG MODE = Control Menu.



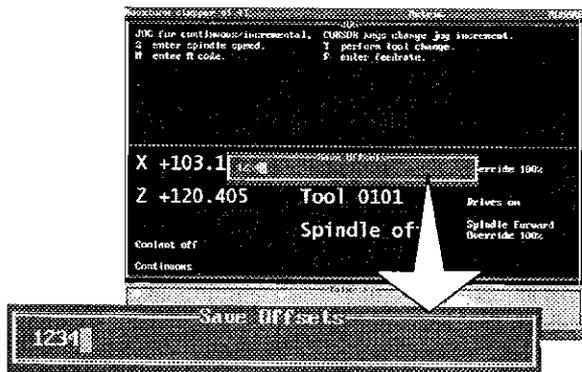
Please Note - your software will be set to read either the computers hard drive (usually C:) or the floppy disk drive (usually A:) by default. If you do not want to save the Tool Offset Files to the default drive, then the drive destination must be changed. See page 4.33 "Changing the Drive Directory - Tool Offset Files".

To save a collection of Tool Offsets to the currently selected drive, select the *OFFSET CONTROL MENU* by pressing the [F9] key (not available in *EDIT MODE*).

Highlight the '*SAVE TOOL OFFSETS*' option using the blue [CURSOR ARROWS] keys and press the [EOB] key.

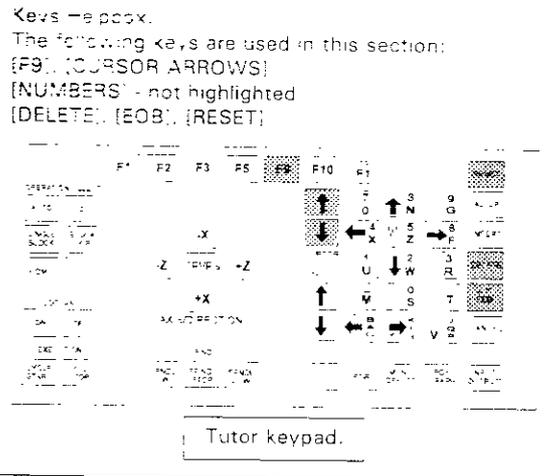


Enter a number (representing the offsets filename) using the [NUMBER] keys. Use the [DELETE] key to remove any incorrectly typed characters and confirm the filename by pressing the [EOB] key. Note that the file will be saved on the currently selected drive. Make a note of the name of your new offset file for future reference.



Press the [RESET] key to clear any menus.

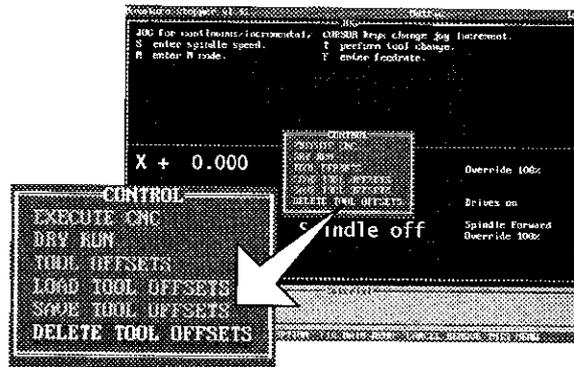
CONTROL MENU - DELETE TOOL OFFSETS.



Please note -
The function of the [F9] key will depend on the machine mode:
AUTO MODE = Control Menu.
EDIT MODE = Simulation Menu.
HOME MODE = Control Menu.
JOG MODE = Control Menu.

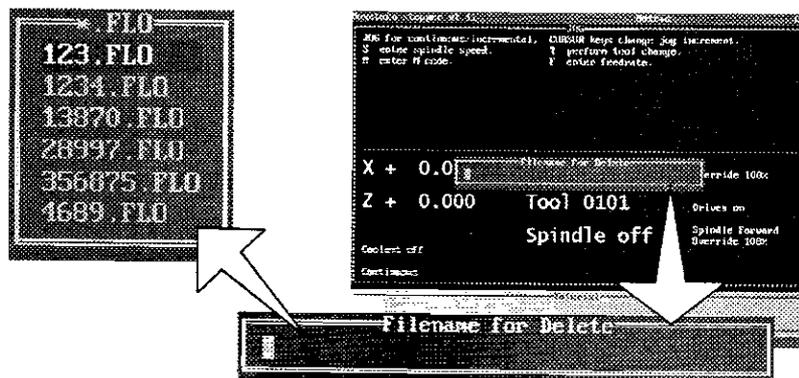
The '**DELETE**' command will permanently remove a selected offsets file from the currently selected drive.
To remove a selected offsets file, select the **OFFSET CONTROL MENU** by pressing the [F9] key (not available in *EDIT MODE*).

Highlight the '**DELETE**' option using the blue [CURSOR ARROWS] keys and press the [EOB] key.



Enter the filename (number) for the offsets file using the [NUMBER] keys. Use the [DELETE] key to remove any incorrectly typed characters and confirm the delete file command by pressing the [EOB] key.

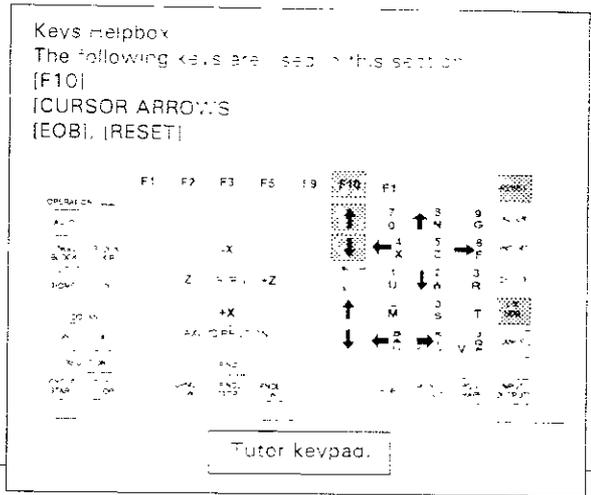
If the file name is unknown, press the [EOB] key to list all the offset files saved on the currently selected drive. Files within these lists can be deleted by highlighting them using the [CURSOR ARROWS] and pressing the [EOB] key.



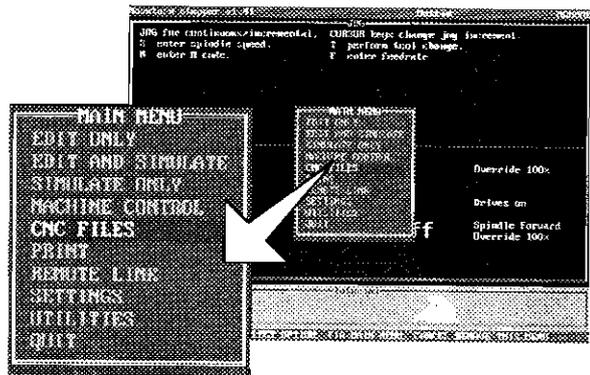
Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

Press the [RESET] key to clear any menus.

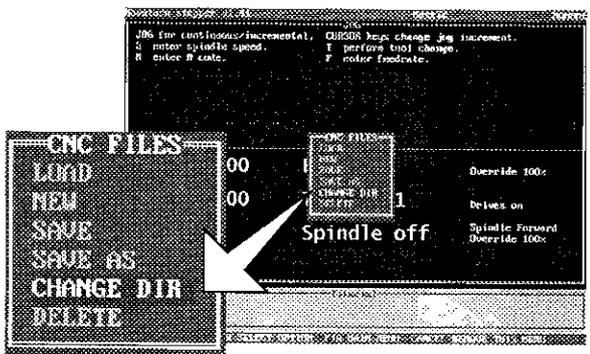
CHANGING THE DRIVE DIRECTORY - TOOL OFFSET FILES.



Select the *MAIN MENU* by pressing the [F10] key. Highlight '*CNC FILES*' using the [CURSOR ARROWS] keys and press the [EOB] key to confirm this choice.



Highlight '*CHANGE DIR*' with the [CURSOR ARROWS] keys and press the [EOB] key.

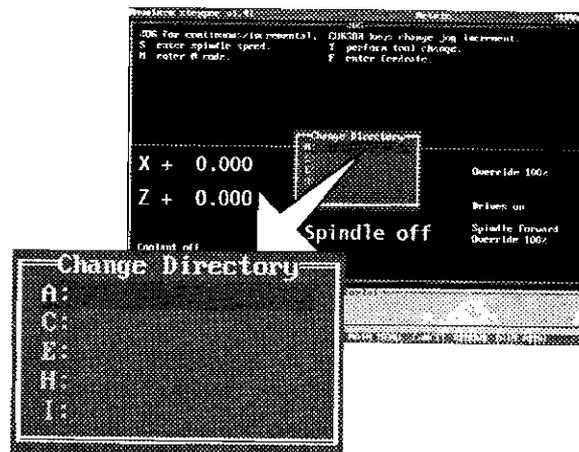


continued....

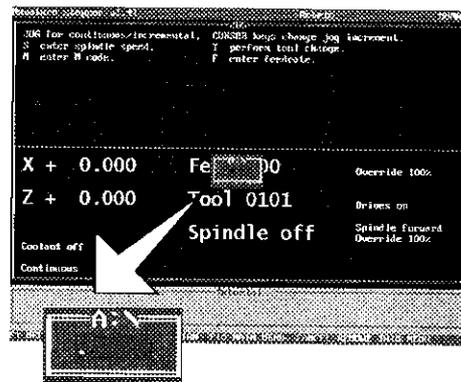
CHANGING THE DRIVE DIRECTORY - TOOL OFFSET FILES.

continued....

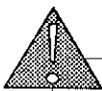
Highlight the drive required (in this example 'A:' is selected) using the [CURSOR ARROWS] keys.



Upon pressing the [EOB] key the selected drive will be displayed.



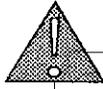
Press the [RESET] key to clear the screen of any unwanted menus or information.



Please note -

The co-ordinate values depicted on screenshots are used for illustrative purposes only.

SAVING TOOL OFFSET FILES ON A CHANGED DRIVE.

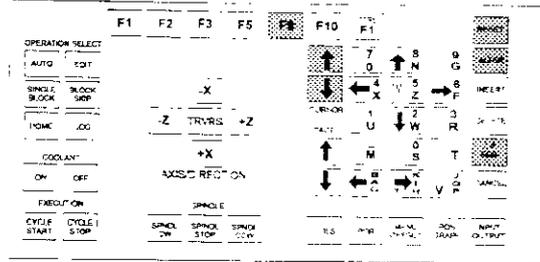


Please note -

The function of the [F9] key will depend on the machine mode:
AUTO MODE = Control Menu.
EDIT MODE = Simulation Menu.
HOME MODE = Control Menu.
JOG MODE = Control Menu.

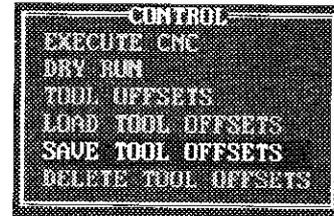
Keys Helpbox.

The following keys are used in this section:
 [F9], [CURSOR ARROWS],
 [NUMBERS] - not highlighted
 [EOB], [ALTER], [RESET].



Tutor keypad.

Select the **OFFSET CONTROL MENU** by pressing the [F9] key, highlight 'SAVE TOOL OFFSETS' using the [CURSOR ARROWS] keys and press the [EOB] key.



Note - the screen may display the previous setting for the drive. In the example below, the screen displays the drive as 'C:', even though it has just been changed to save on 'A:', as shown in the last section. If this occurs, press the [ALTER] key to reset to the new drive.



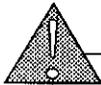
Enter the filename using the [NUMBERS] keys and press [EOB] to confirm this. The offset files will now be saved on the new drive.



Please Note -

CNC Files are stored in ".fnc"
 or ".fnc" format.
 Offset Files are stored in ".flo"
 format.

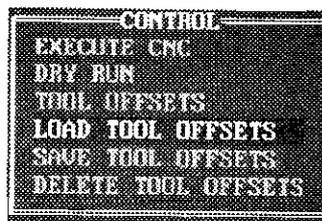
LOADING TOOL OFFSET FILES ON A CHANGED DRIVE.



Please note -

The function of the [F9] key will depend on the machine mode:
AUTO MODE = Control Menu.
EDIT MODE = Simulation Menu.
HOME MODE = Control Menu.
JOG MODE = Control Menu.

Select the *OFFSET CONTROL MENU* by pressing the [F9] key, highlight 'LOAD TOOL OFFSETS' using the [CURSOR ARROWS] keys and press the [EOB] key.



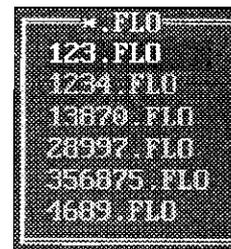
Note - the screen may display the previous setting for the drive. In the example above, the screen displays the drive as 'C:', even though it has just been changed to load from 'A:', as shown in the last section. If this occurs, press the [ALTER] key to reset to the new drive.



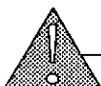
Enter the filename using the [NUMBERS] keys and press [EOB] to confirm this. The offset file will now be loaded from the new drive.



If the filename is unknown, the list of offset files stored on the drive can be accessed by pressing the [EOB] key.



The screen above will be shown, if there are no matching offset files on the drive that is being read. To clear this message, press the [RESET] key.



Please Note -

CNC Files are stored in ".fnc" or ".fnl" format.
 Offset Files are stored in ".flo" format.