



**DENFORD**



---

**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](mailto:service@denford.co.uk)



# CONTENTS.

SECTION	PAGE
Notes .....	0-2
Introduction .....	0-5
Operations flowchart .....	0-5
Using the Manual .....	0-6
Basic Components .....	0-7
Operating the Desktop Tutor Control Panel .....	0-9
Switching on a Machine with an Integrated Controller .....	1-1
Switching on a Machine controlled by a PC .....	1-3
PC Controlled Machines - Loading the Control Software .....	1-5
Control Software - Main Menu .....	1-7
Closing the Control Software .....	1-9
Setting up LatheCAM Designer for your CNC Lathe .....	1-11
Setting the Machine Datum Point .....	2-1
Clamping the Billet .....	3-1
Changing the Tool .....	3-3
Changing the Tool using an Automatic Toolpost .....	3-4
Changing the Tool using a Manual Toolpost .....	3-6
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 Tool Offset of the Next Tool .....	4-18
Setting the Tool Offsets - Roughing or Finishing Tool .....	4-19
Setting the Tool Offsets - Threading Tool .....	4-20
Setting the Tool Offsets - Parting Tool .....	4-21
Setting the Tool Offsets - Drills .....	4-22
Setting the Tool Offsets - Centre Drill .....	4-23
Setting the Tool Offsets - Twist Drill .....	4-24
Control Menu .....	4-25
Control Menu - Execute CNC .....	4-26
Control Menu - Dry Run .....	4-27
Control Menu - Display Tool Offsets Table .....	4-28
Control Menu - Load Tool Offsets .....	4-29
Control Menu - Save Tool Offsets .....	4-31
Control Menu - Delete Tool Offsets .....	4-32
Changing the Drive Directory - Tool Offset Files .....	4-33
Saving Tool Offset Files on a Changed Drive .....	4-35
Loading Tool Offset Files on a Changed Drive .....	4-36
CNC Files Menu .....	5-1
CNC Files Menu - Load .....	5-2
CNC Files Menu - AdvancedLoad Setup .....	5-4
CNC Files Menu - New .....	5-5
CNC Files Menu - Save .....	5-6
CNC Files Menu - Save as .....	5-7
CNC Files Menu - Delete .....	5-8
Changing the Drive Directory - CNC Files .....	5-9
Saving CNC Files on a Changed Drive .....	5-11
Loading CNC Files on a Changed Drive .....	5-12

# CONTENTS.

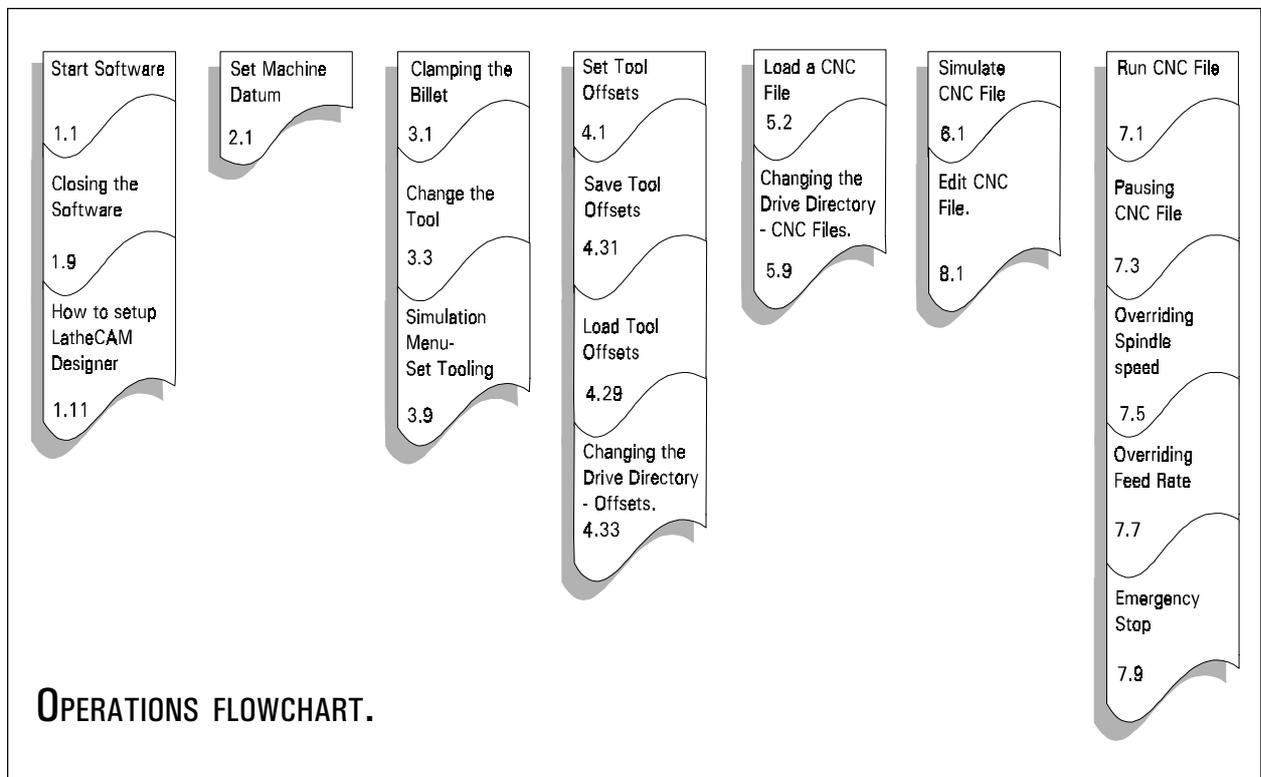
SECTION	PAGE
Simulate CNC File .....	6-1
Simulate - Screen View Options. ....	6-1
Simulate - Select Screen View .....	6-2
Simulation Menu .....	6-3
Run Program .....	7-1
Pausing Program .....	7-3
Overriding Spindle Speed .....	7-5
Overriding Spindle Speed - Potentiometer Controls .....	7-5
Overriding Spindle Speed - Tutor Controls .....	7-6
Overriding Feed Rate .....	7-7
Overriding Feed Rate - Potentiometer Controls .....	7-7
Overriding Feed Rate -Tutor Controls .....	7-8
Emergency Stop .....	7-9
Editing a CNC File - Screen View Options .....	8-1
Editing a CNC File - Key Functions .....	8-3
Editing a CNC File - Editor Window Layout. ....	8-4
Editing a CNC File .....	8-5
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 - Menu .....	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
Programming Terms and Conventions .....	11-1
What is a Part Program? .....	11-1
Composition of a Part Program .....	11-2
Listing of Address Characters .....	11-3
G Codes Listing for Denford CNC FANUC Lathes.....	11-4
M Codes Listing for Denford CNC FANUC Lathes .....	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.

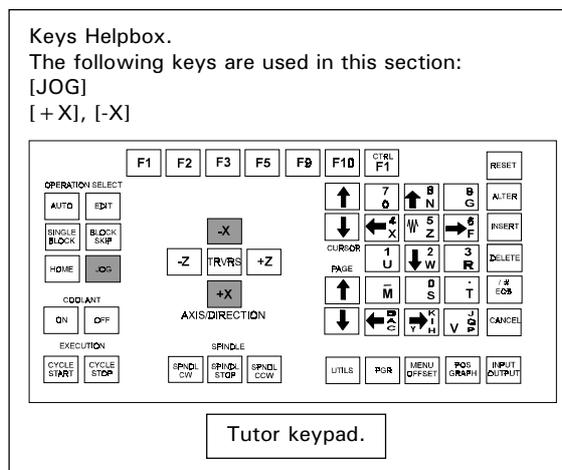
# USING THE MANUAL.....

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.

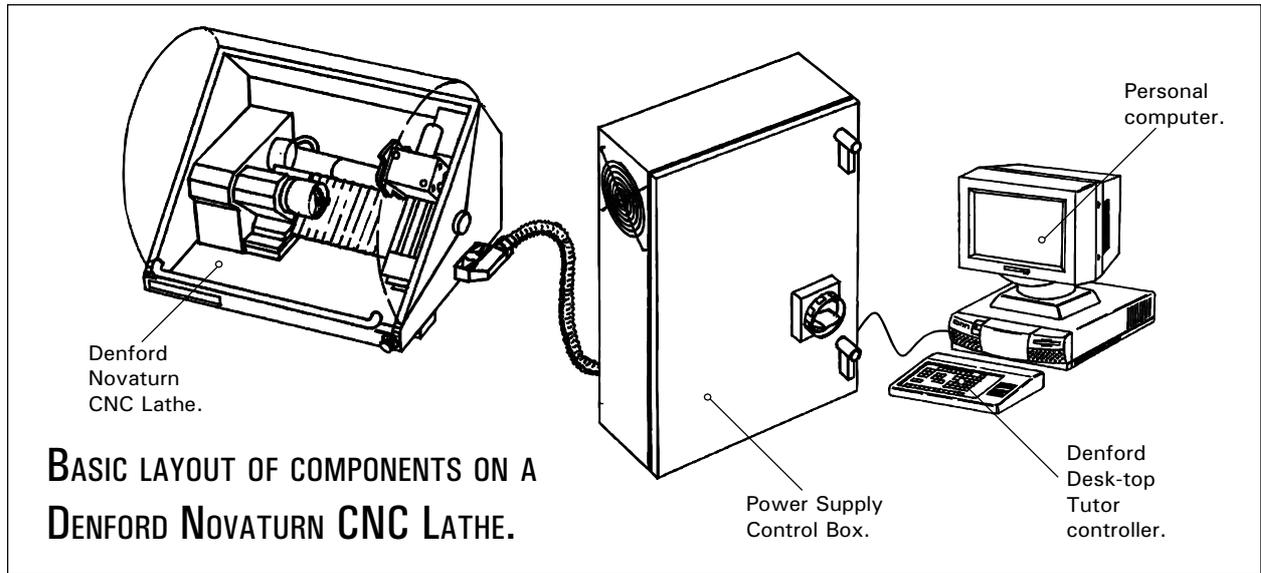
When relevant, screenshots and graphics are also provided to accompany the text. Specific screenshots and drawings of machines are only used for illustrative purposes. All the operations described in this manual relate to any Denford CNC Lathe.

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.

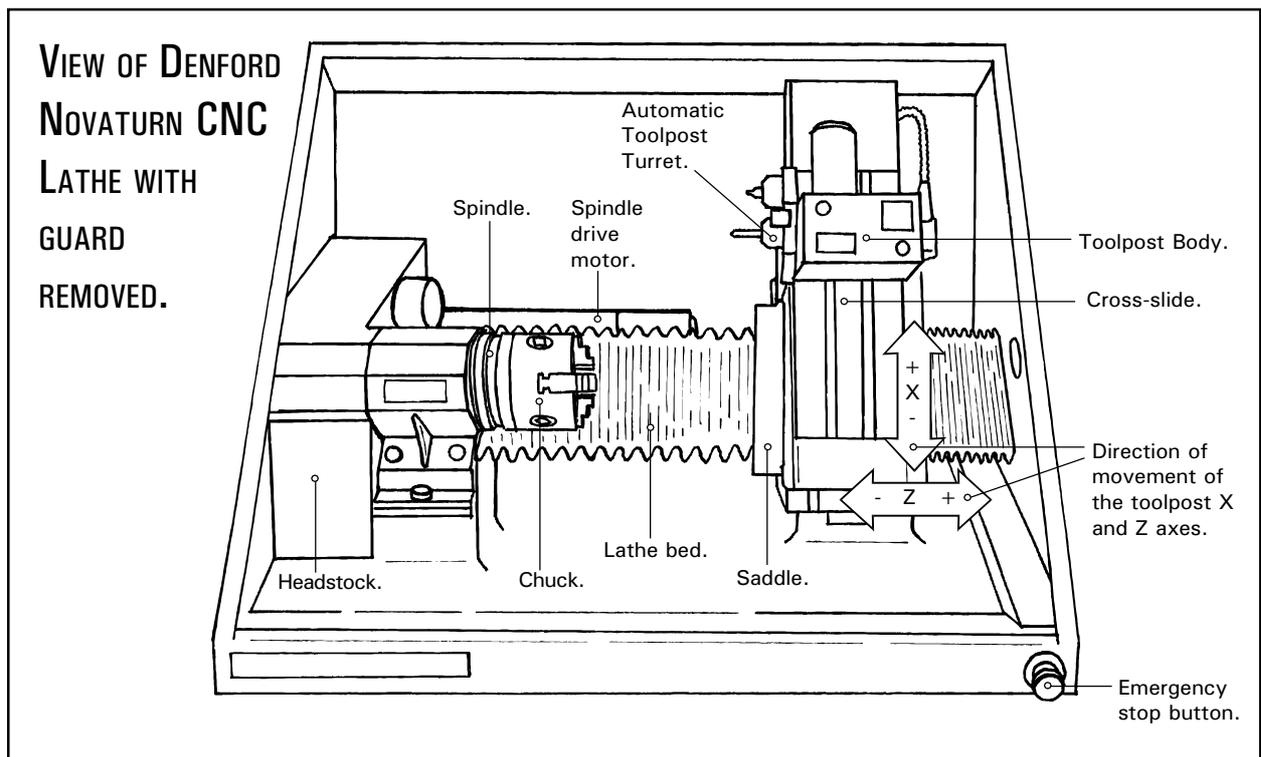
For Machine specific features and operations, please consult the separate Installation Manual supplied with your machine.

# 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.







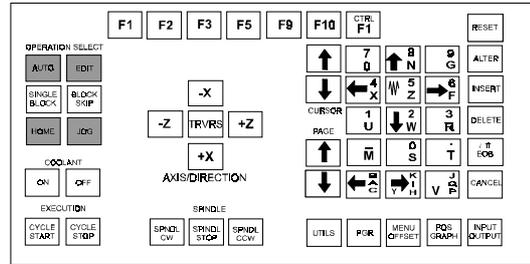
The Operation Select keys will set the mode in which the machine and controller will run:

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*).

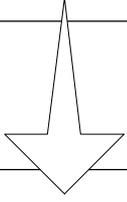


Tutor keypad.

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

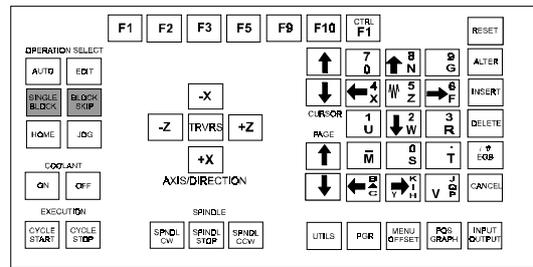
Look for the "*AUTO, EDIT, HOME and JOG*" symbols (shown below) - they will indicate the mode the machine must be running for the highlighted key to operate in the way described.

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).

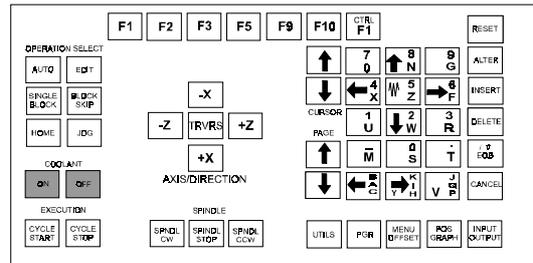
The [BLOCK SKIP] key is used to select the option to ignore, or include, specific program blocks (activated by a "/" character in front of the block).



Tutor keypad.

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

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



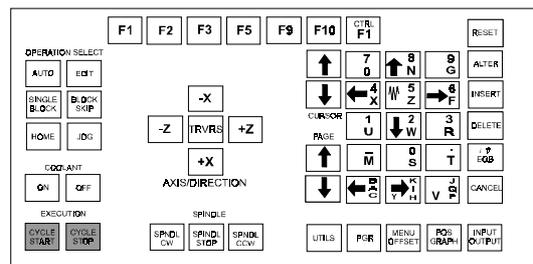
Tutor keypad.

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.

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

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

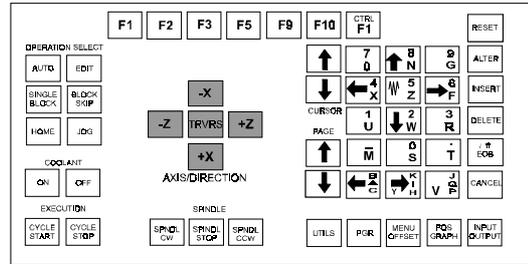
**JOG** The [+X] key is used to control axis movement in the +X direction.

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

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

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

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

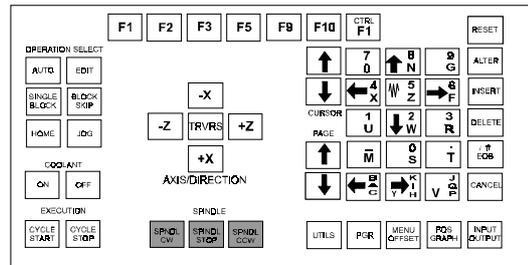


Tutor keypad.

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

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

**HOME** **JOG** 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.



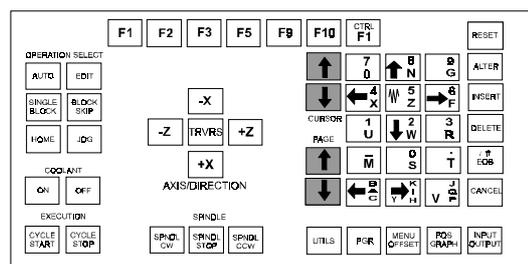
Tutor keypad.

The blue [CURSOR ARROWS] keys are used to:

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

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

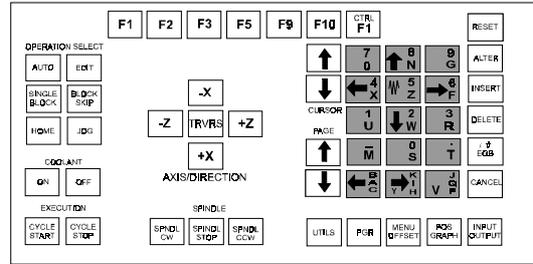
- EDIT** **HOME** **JOG** Move between the top and bottom lines of purple Menu Selection screens and programs.
- AUTO** Move up or down between the pages of the program before machining is started.



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.

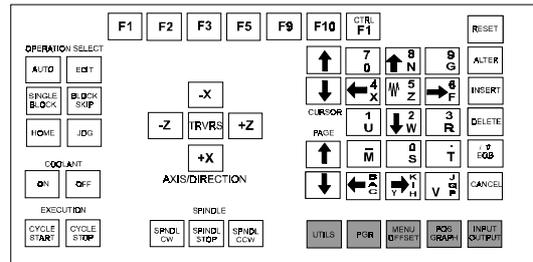
 The [UTILS] key is used to display any directives within a program, shown on screen as [YELLOW LINES]. Directives are Denford definitions for tool sizes, billet sizes etc....

    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*).

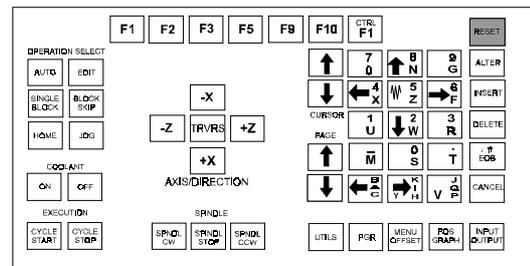
 The [INPUT OUTPUT] key is used to select the *REMOTE DEVICE LINK MENU* (this menu allows data to be sent or received from external peripherals).



Tutor keypad.

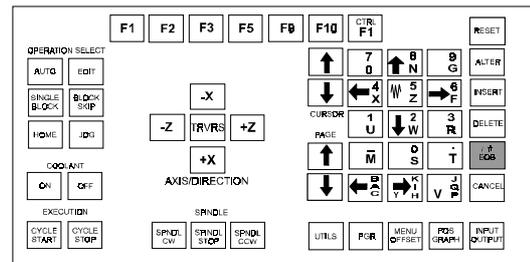
The [RESET] key is used to:

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



Tutor keypad.

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

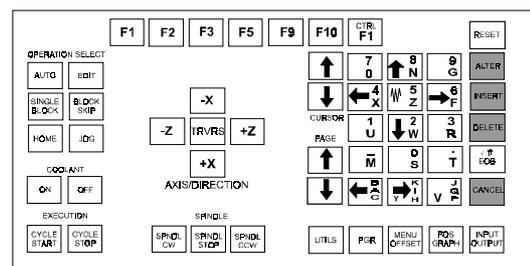
The [INSERT] key is used to place a word into a program line.

The [DELETE] key is used to:

- 1)    Remove a word from a program line.
- 2)     Remove unwanted characters that have been typed in.

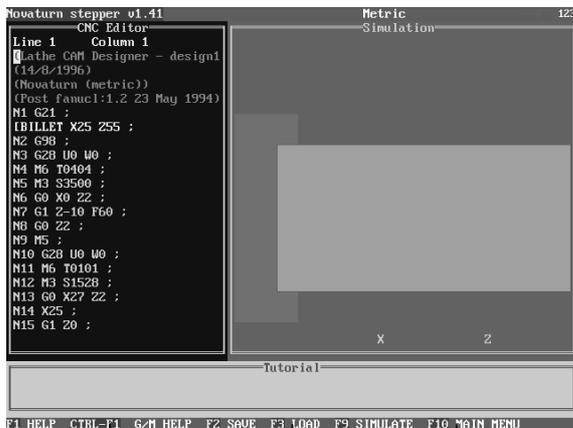
The [CANCEL] key is used to:

- 1)    Remove a word from a program data entry line.
- 2)  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 F9 SIMULATE F10 MAIN MENU**

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.

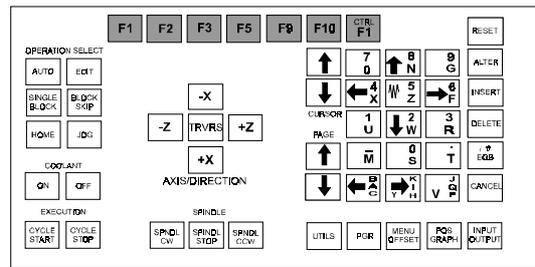
The [F5] key is used to display the *Data Information Help* screen.

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.



Tutor keypad.

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

# 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.

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

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

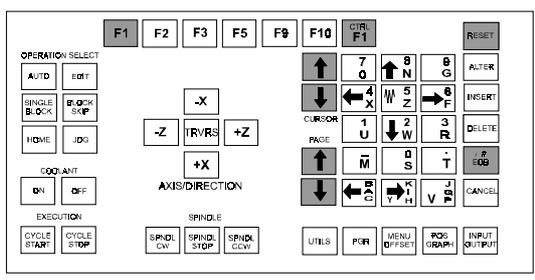
Manual Feedrate Override - Use the key labelled [4X LEFT ARROW] to decrease the feedrate and the [6Z RIGHT ARROW] to increase the feedrate.

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.

AUTO This feature will only operate in *AUTO MODE*.

# CONTEXT SENSITIVE HELP.

Keys Helpbox.  
 The following keys are used in this section:  
 [F1], [CTRL F1]  
 [CURSOR ARROWS], [PAGE UP], [PAGE DOWN]  
 [EOB], [RESET]

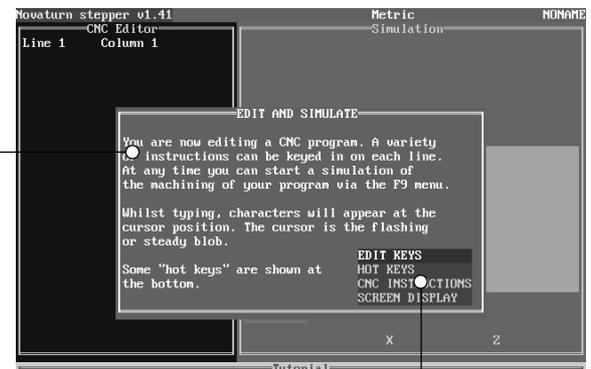


Tutor keypad.

Context sensitive help is available at all times, with sub-related topics. To select the *GENERAL HELP* screen press the [F1] key. The information on the help screen will depend on the position of the cursor when the [F1] key was pressed (eg, if the cursor was highlighting an option in the Main Menu, the help screen will display information about options available within the Main Menu). Pressing the [F1] key again will display a sub-related topic.

Pressing the [CTRL F1] key will display *G AND M CODE HELP*. Again, this help will be context sensitive, if the cursor is highlighting a particular code when editing a program.

Main Help screen.



Help Fast Keys Strip.



Help Options Menu.

Navigate around the help screens using the following keys :

[CURSOR ARROWS] keys -  
 Used to select categories from the pink/red help options menus.

[PAGE UP] and [PAGE DOWN] keys -  
 Used to cycle through different help pages, when more than one page is available.

[RESET] key -  
 Used to exit the help screens.



# SWITCHING ON A MACHINE WITH AN INTEGRATED CONTROLLER.

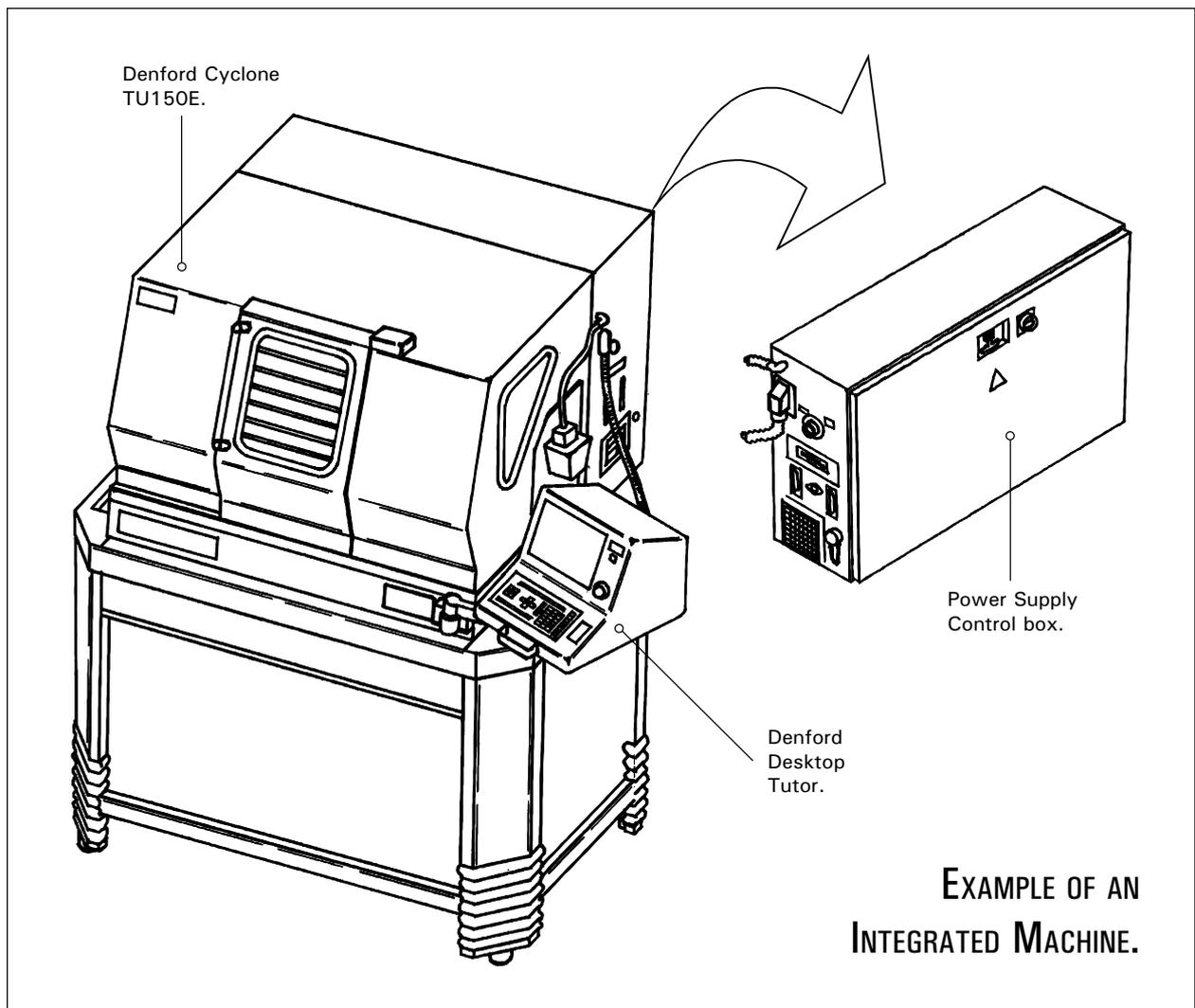
An integrated machine has a permanently attached computer and controller, i.e. dedicated to operating that particular machine and nothing else!

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.

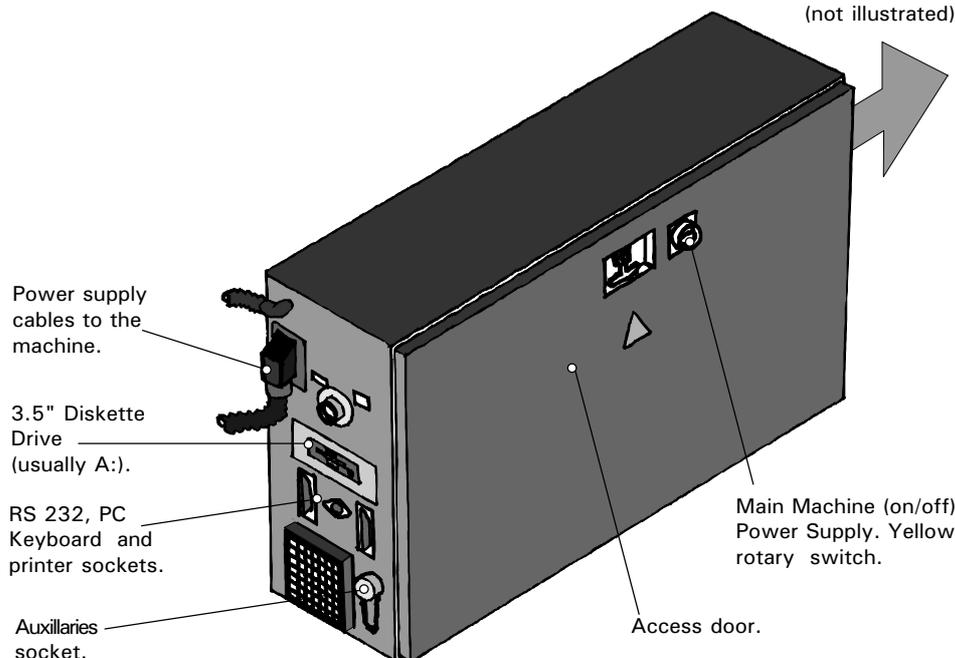
If the controller software is supplied on floppy (3.25 inch) disk, insert the disk into the floppy drive on the power supply control box before switching on the machine.



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.

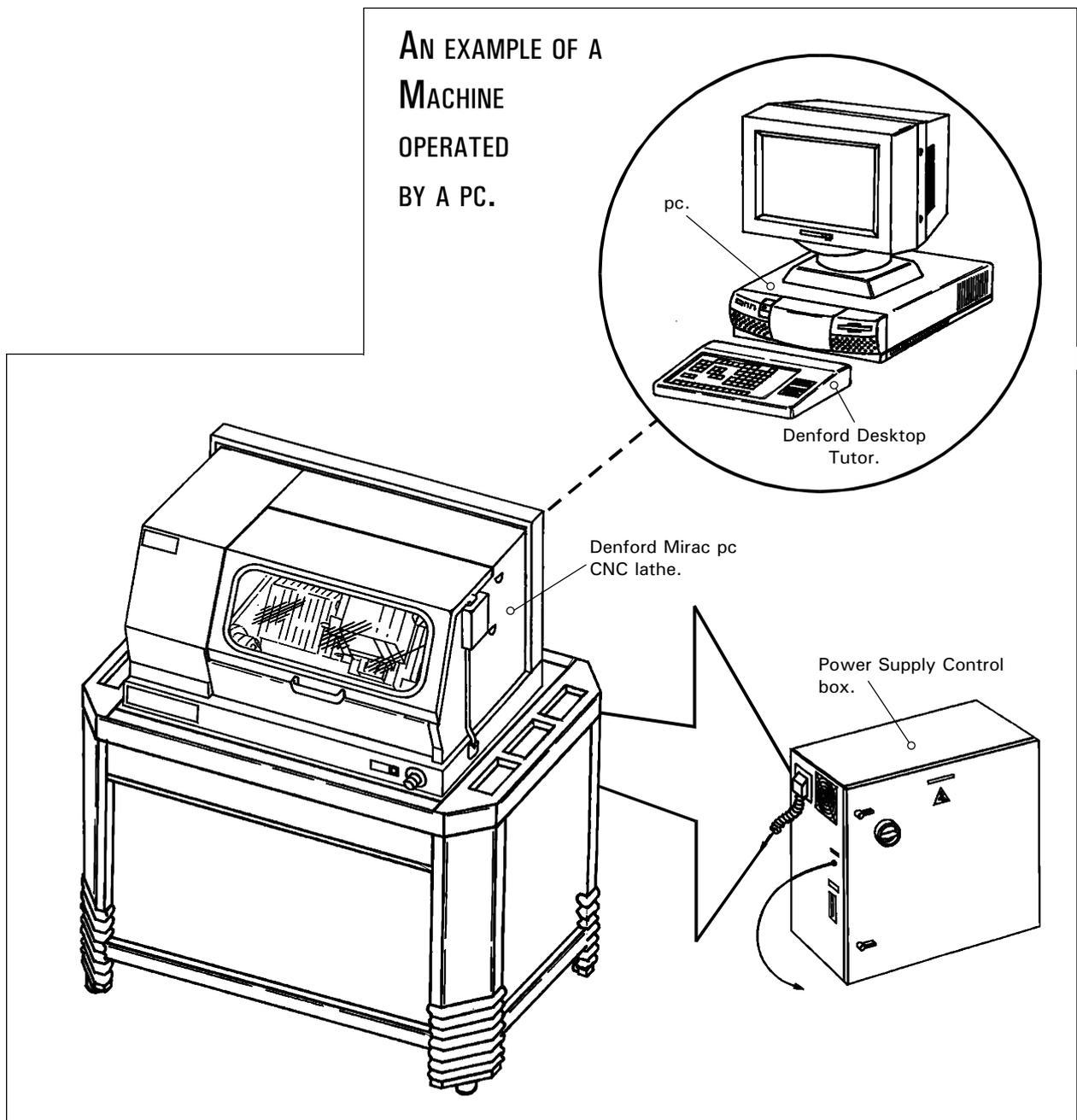
A pc operated machine has 3 main components:

- 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.

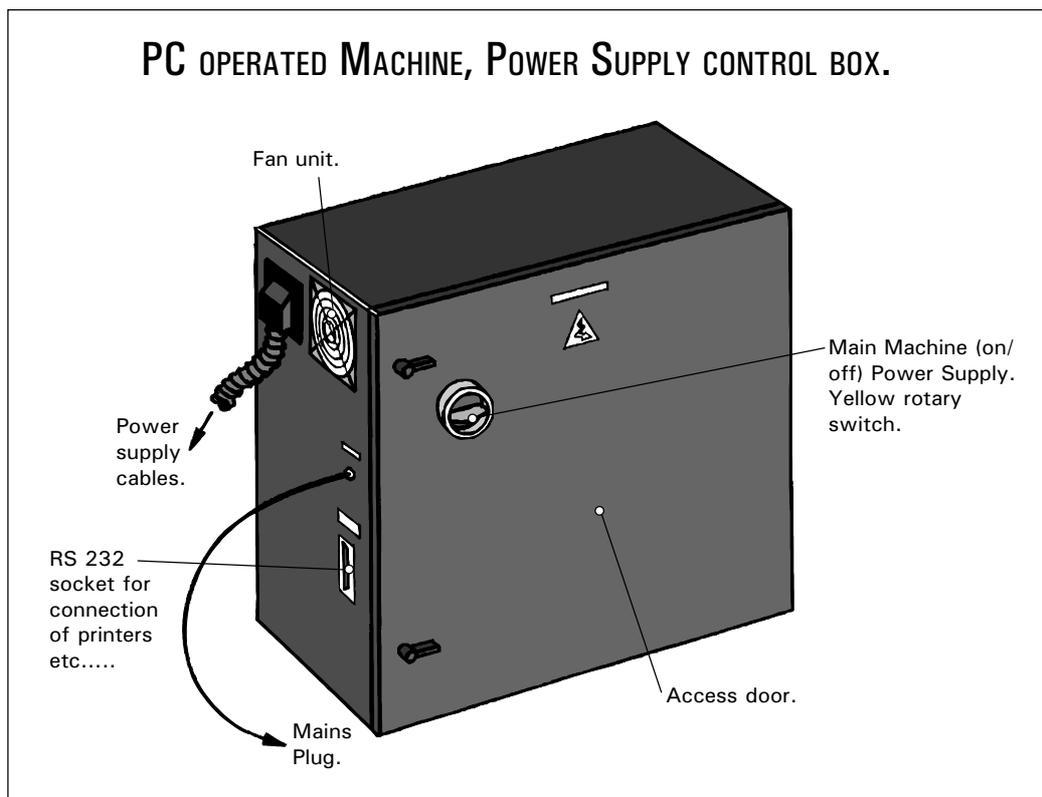


To switch on your machine, locate the yellow rotary power supply switch and turn it to the 'on' position. To load the machine controlling software, please refer to the next page.



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**

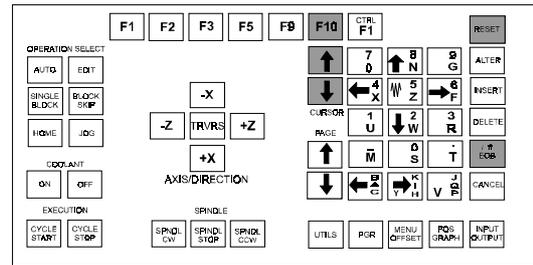
(where '**A:**' is the floppy (3.25 inch) disk drive and '**Start-up filename**' is the text chosen from the list above).

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

The machine controlling software will now load.

# CONTROL SOFTWARE - MAIN MENU.

Keys Helpbox.  
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.



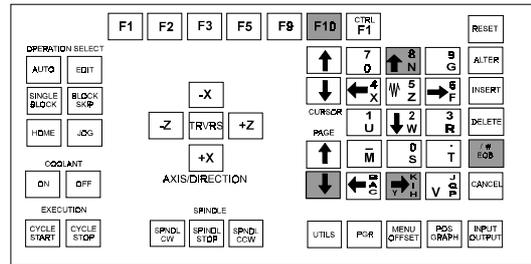
# CONTROL SOFTWARE - MAIN MENU.

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.

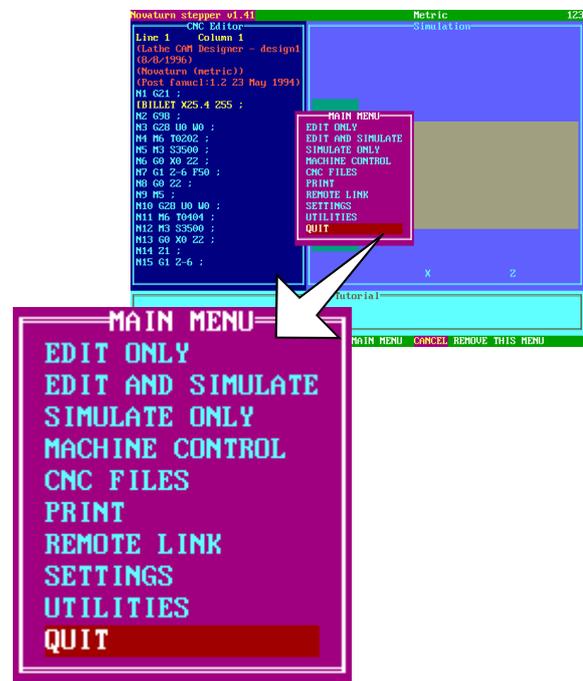
Keys Helpbox.  
The following keys are used in this section:  
[F10]  
[PAGE DOWN]  
[N], [Y], [EOB]  
[NUMBERS] - not highlighted.



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.

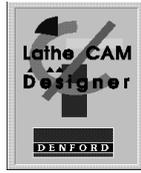
If this program does not need to be saved, press the [N] key.

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?



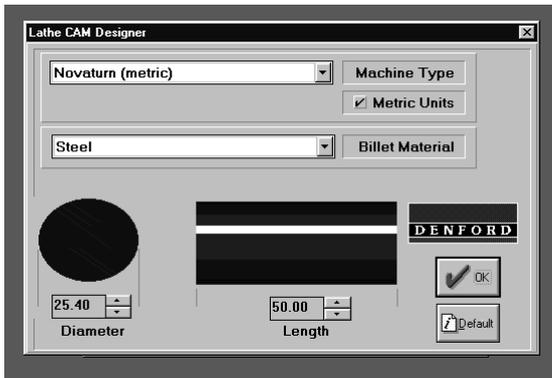
# HOW TO SET-UP LATHECAM DESIGNER TO WORK WITH YOUR CNC LATHE.



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).

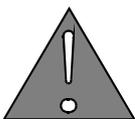
Essentially, it allows a part to be designed "on-screen" using a series of shapes selected from a software menu. Once designed, the G-code program, used by the machine to manufacture the part, can be generated automatically.

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:

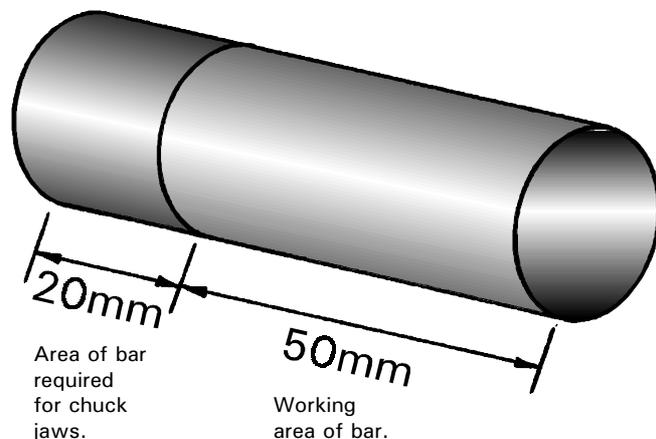
- the type of machine that will be used.
- the type of material used as the billet.
- the length and diameter of the billet (this defines the working area).
- the units of measurement.



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.

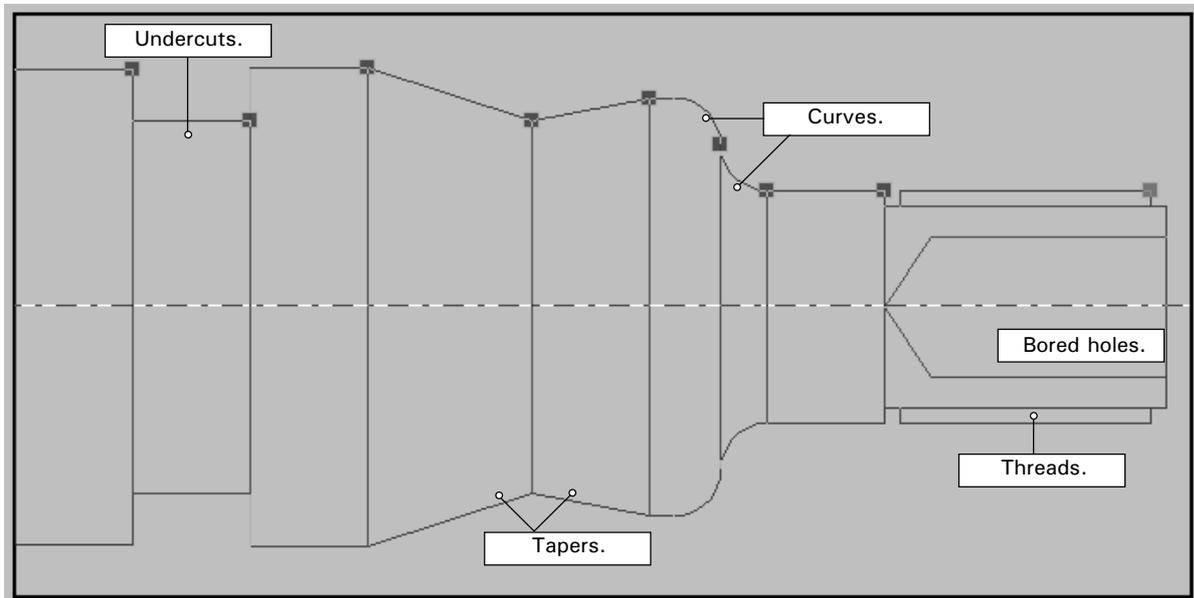
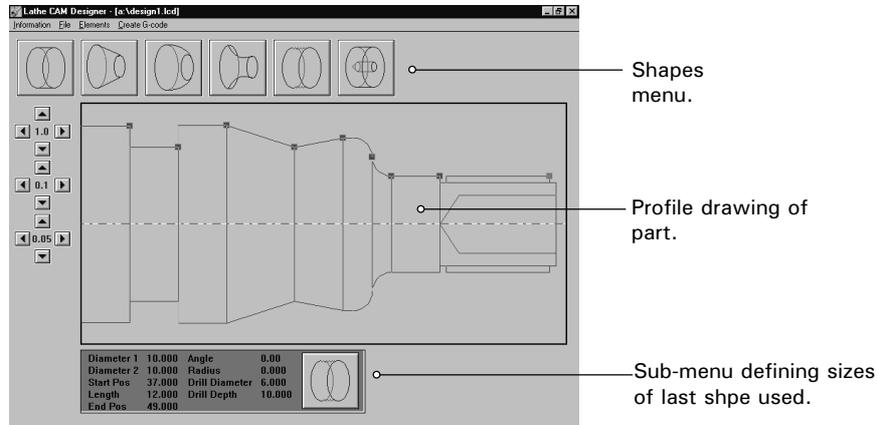
Extra material needed to actually hold the billet in the lathe chuck jaws (ie, material that cannot be machined) is not included in this working area figure.

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.

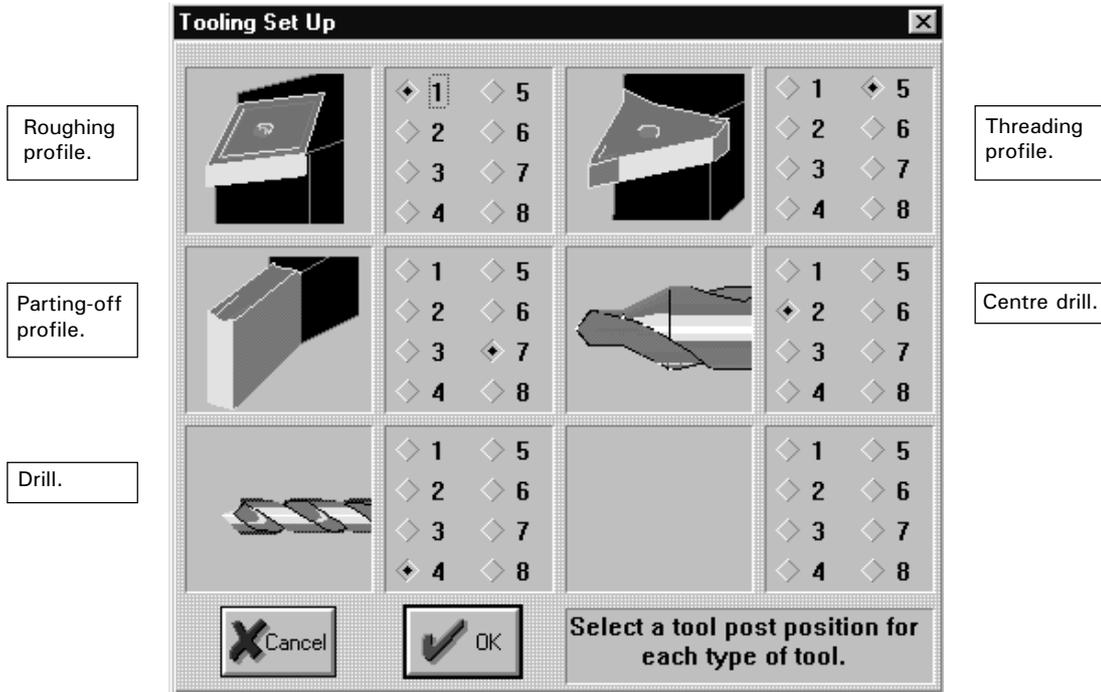


The design shown above was produced using each shape from the menu at least once.

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.

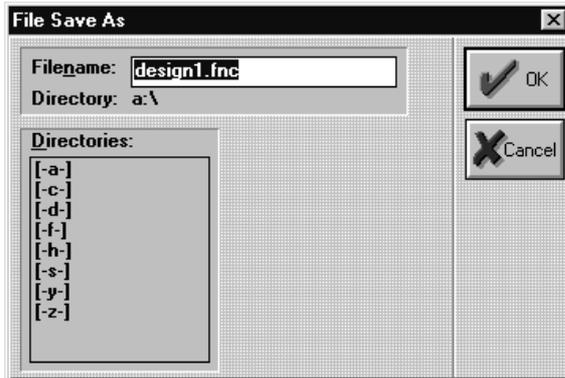
It is very important that the tool number profiles on both LatheCAM Designer and the actual toolpost on the CNC lathe match each other, ie, if tool 1 on the CNC lathe is a roughing profile, then tool 1 on LatheCAM needs to be a roughing profile, etc.....

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



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.

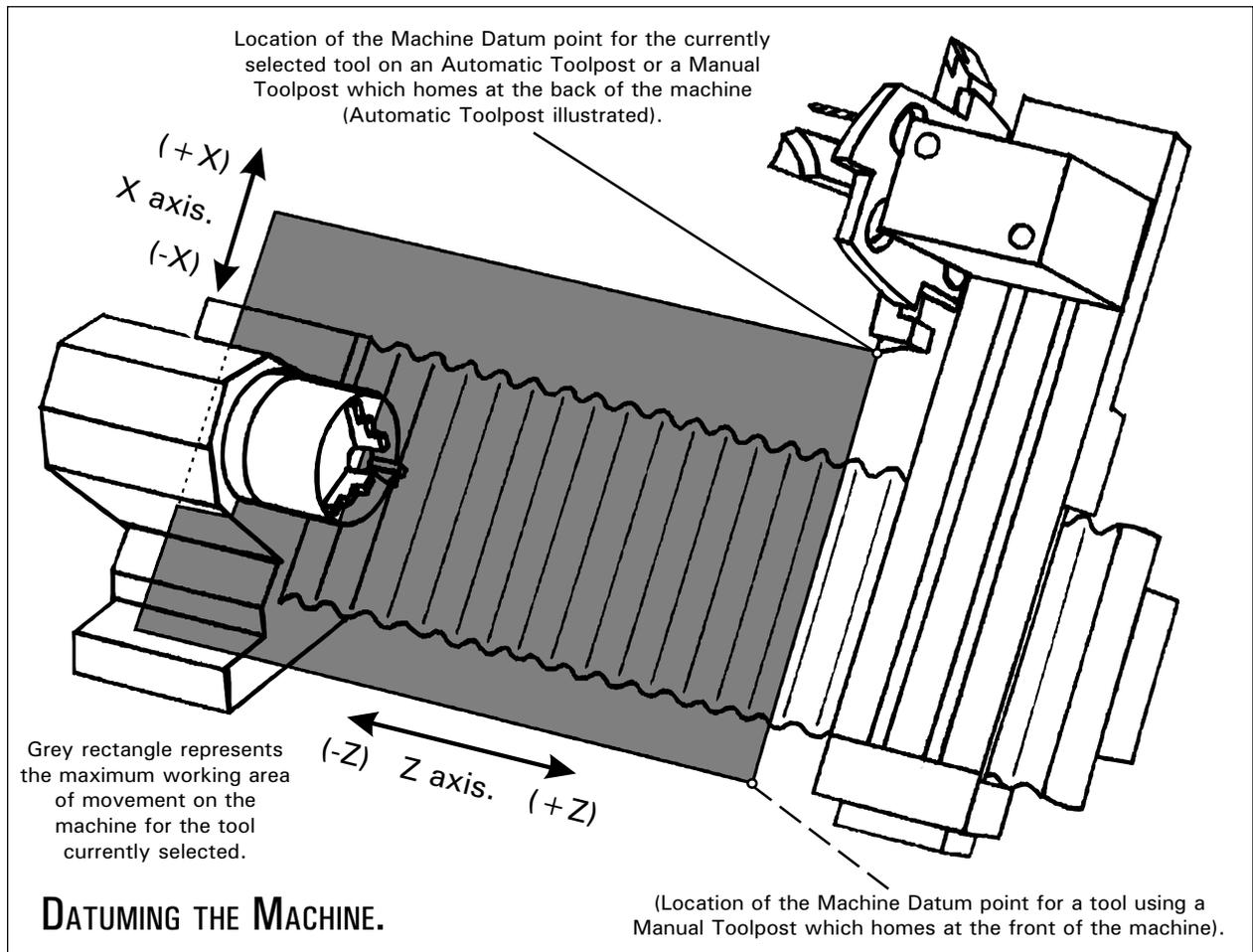
# SETTING THE MACHINE DATUM POINT.

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).

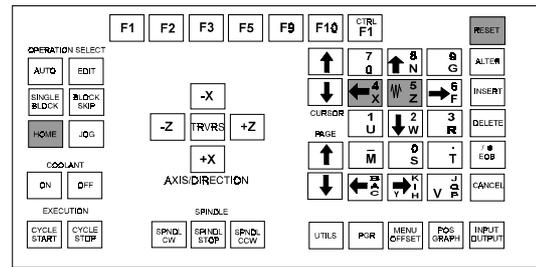
When the machine has just been datumed, the tip of the currently selected tool will always be the machine datum point with the co-ordinates  $X=0, Z=0$ . If the tool is changed for another, with a different length, the machine datum point will then be the tip of this "new" tool, again with the co-ordinates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

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.

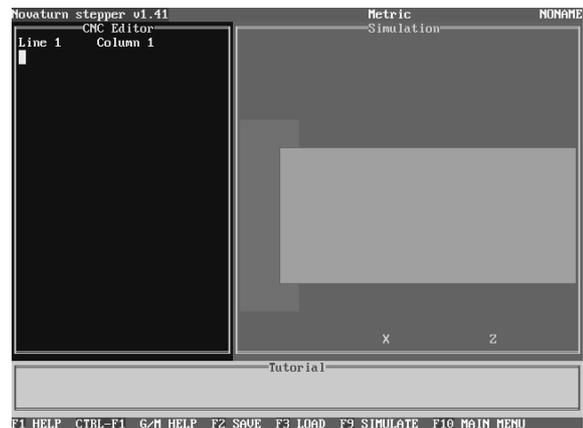
Keys Helpbox.  
 The following keys are used in this section:  
 [HOME]  
 [RESET]  
 [X], [Z]



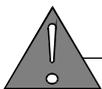
Tutor keypad.

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.....



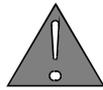
Then, press the [Z] key. The toolpost will move until it finds the Z axis reference point.



continued....

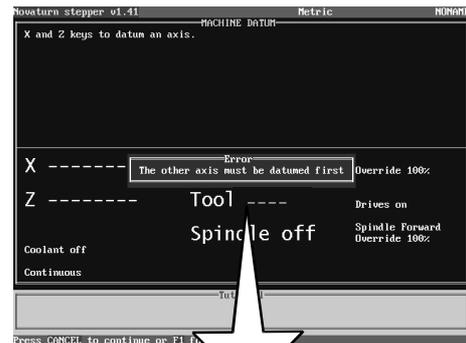
# SETTING THE MACHINE DATUM POINT.

continued....

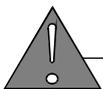


Note - some machines may be configured so that the Z axis is datumed before the X axis (Denfords Microturn, for example), so if the X axis is datumed first an error message will be displayed. If this occurs, press [RESET] and datum the Z axis first.

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



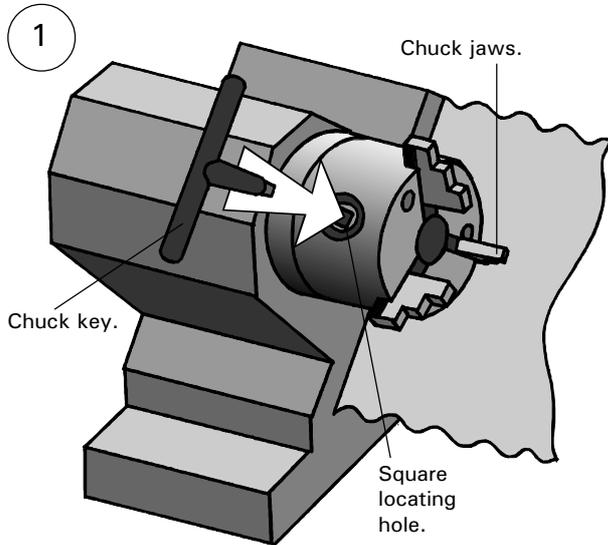
**Error**  
The other axis must be datumed first



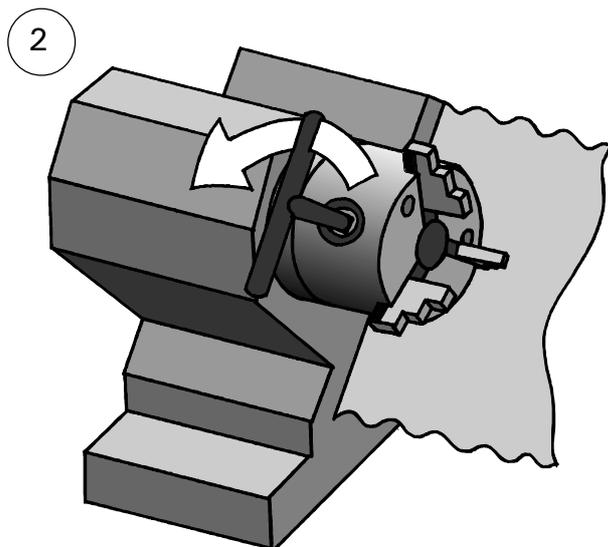
Please note -  
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

# CLAMPING THE BILLET.

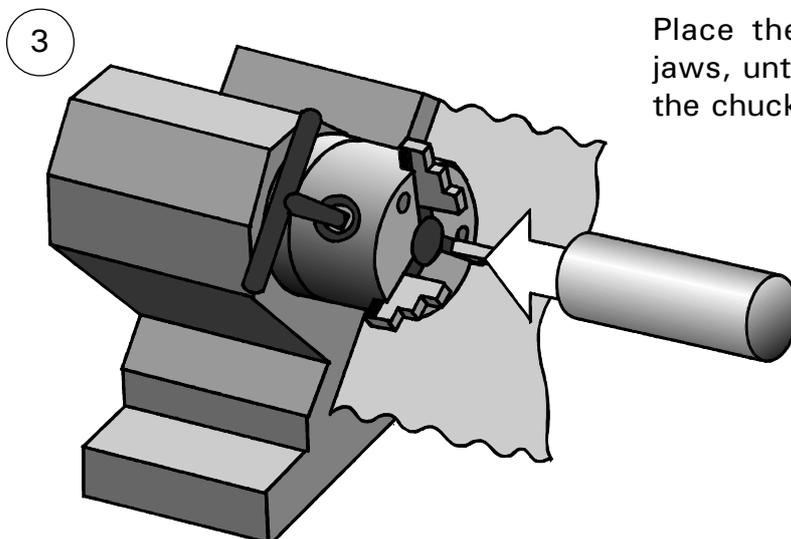
 Never leave the chuck key fitted in one of the locating holes. If the spindle is switched on it could hit the operator or machine and cause serious damage.



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.

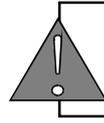


Open the chuck jaws by rotating the chuck key anticlockwise until there is sufficient space to fit the billet.

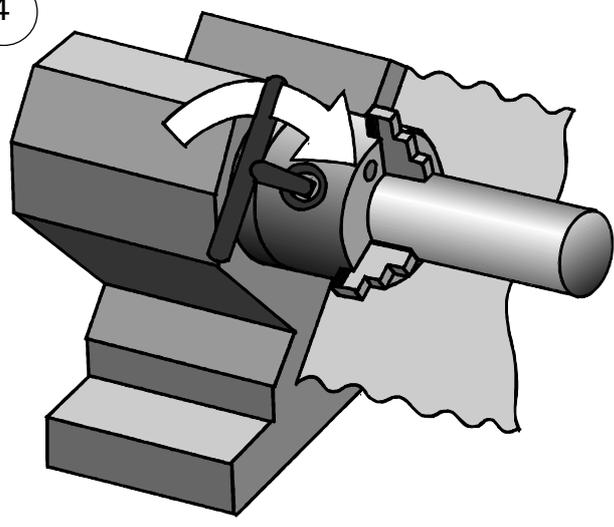


Place the billet between the chuck jaws, until it touches the front face of the chuck.

# CLAMPING THE BILLET.

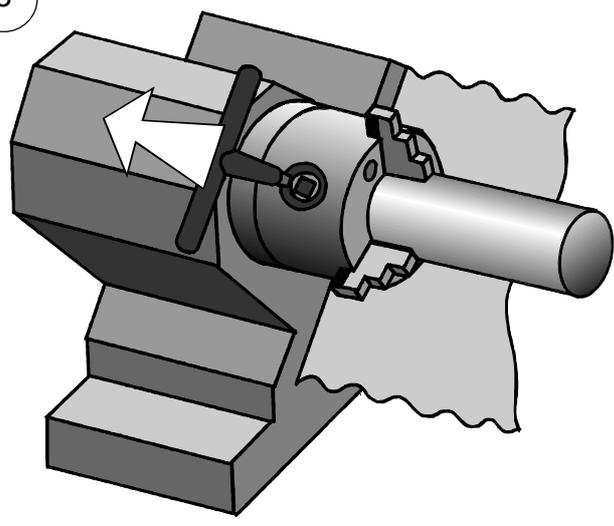
 Never leave the chuck key fitted in one of the locating holes. If the spindle is switched on it could hit the operator or machine and cause serious damage.

4



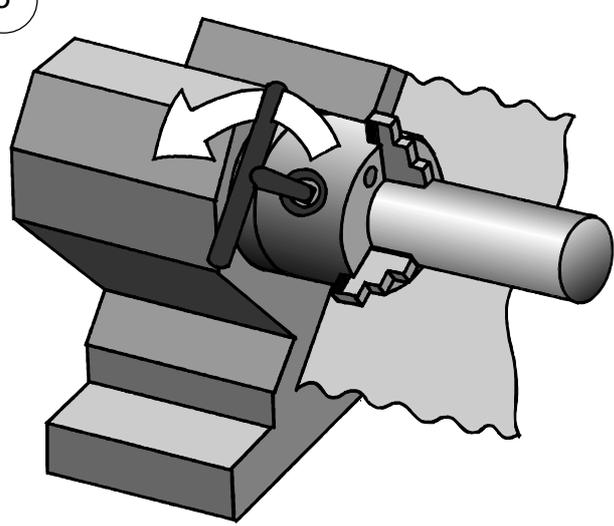
Whilst holding the billet against the front face of the chuck, close the chuck jaws by rotating the chuck key clockwise.

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.

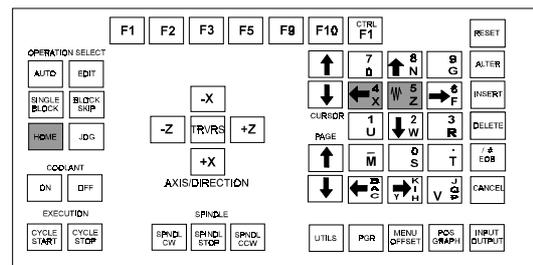
## Keys Helpbox.

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.

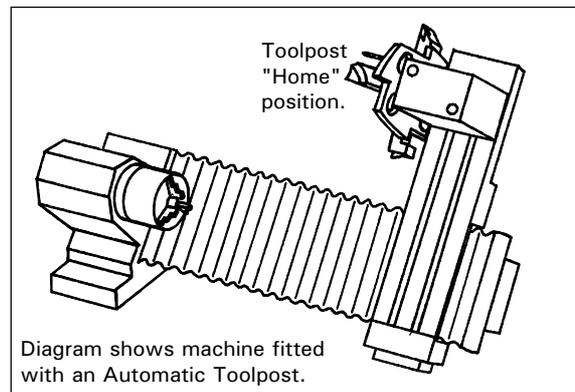


Diagram shows machine fitted with an Automatic Toolpost.

continued....

# CHANGING THE TOOL.

continued....

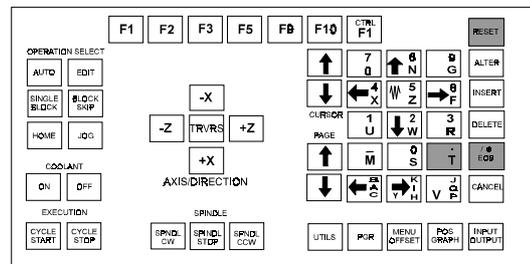
The toolpost will move to its "Home" position. On machines with an Automatic Toolpost this is the back right-hand corner of the machine. On machines fitted with a Manual Toolpost this is either the front or back right-hand corner of the machine, according to the machine model and control software used.

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.

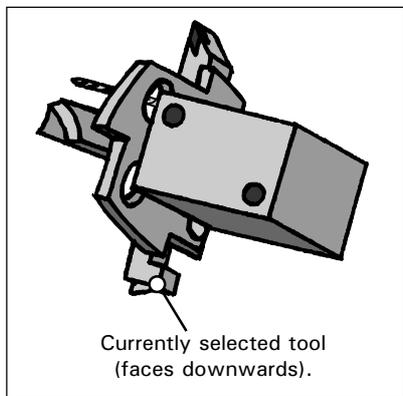
# CHANGING THE TOOL USING AN AUTOMATIC TOOLPOST.

Keys Helpbox.

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



Tutor keypad.



Currently selected tool (faces downwards).

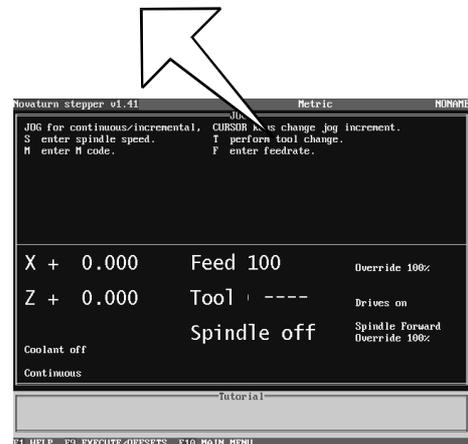
**!** Move the toolpost to a safe position before starting any tool changing operation.

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.

**!** 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.



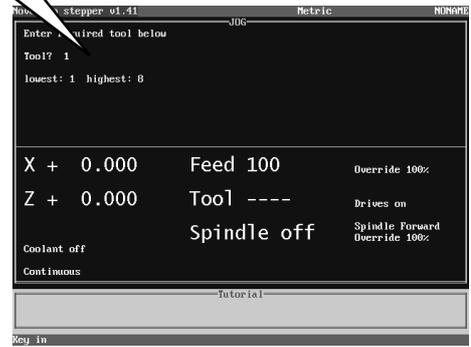
continued....

# CHANGING THE TOOL USING AN AUTOMATIC TOOLPOST.

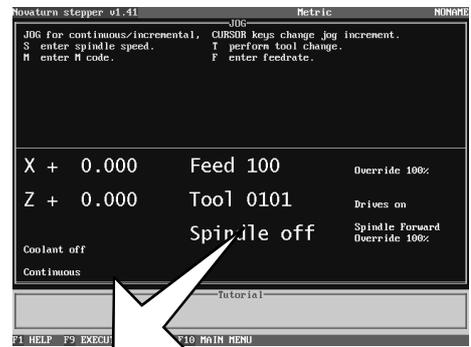
continued....

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

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



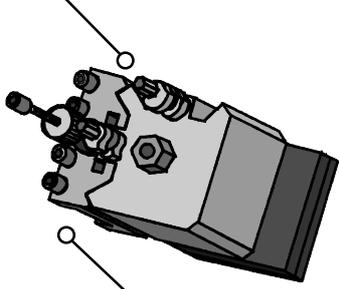
When the [EOB] key is pressed, the tool changer will automatically change from the currently selected tool number to the "new" tool number. This "new" tool number will be displayed "on-screen" when the operation has been completed.



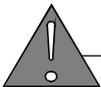
```
Tool 0101
```

# 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.

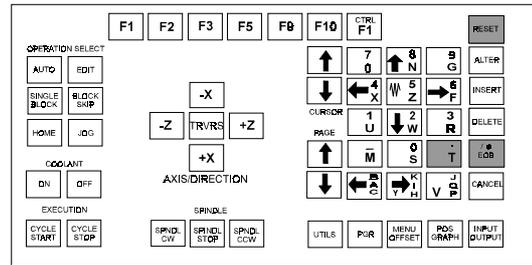


### Tool Orientation -

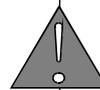
Fit the tool facing up on manual toolposts that home in the front right-hand corner of the machine.  
Fit the tool facing down on manual toolposts that home in the back right-hand corner of the machine.

### Keys Helpbox.

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



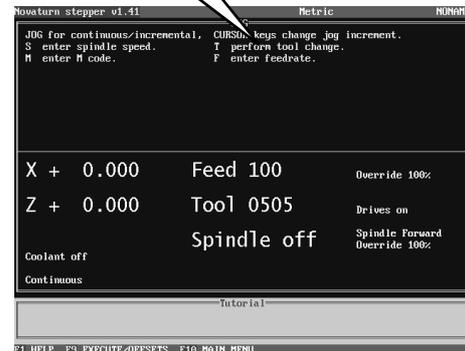
Tutor keypad.



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....



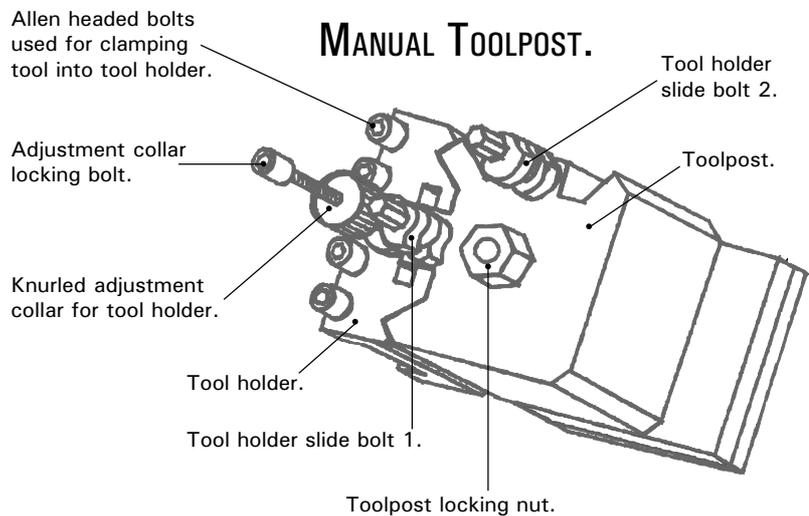
### Please note -

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

# 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.

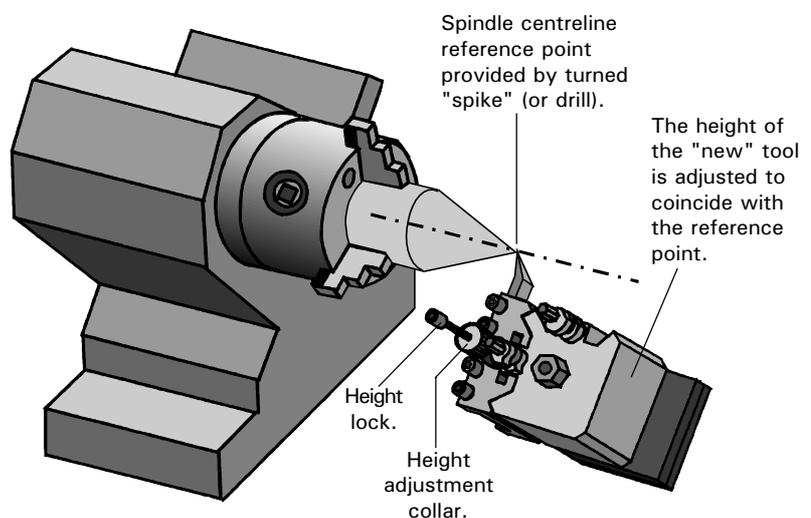


Diagram shows Manual Toolpost that homes in the front right-hand corner of the machine.

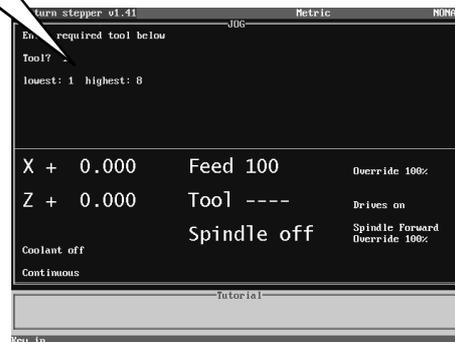
continued....

# CHANGING THE TOOL USING A MANUAL TOOLPOST.

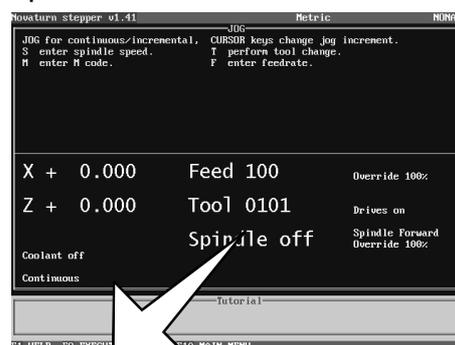
continued....

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

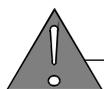
```
Enter required tool below
Tool? 1
lowest: 1 highest: 8
```



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.

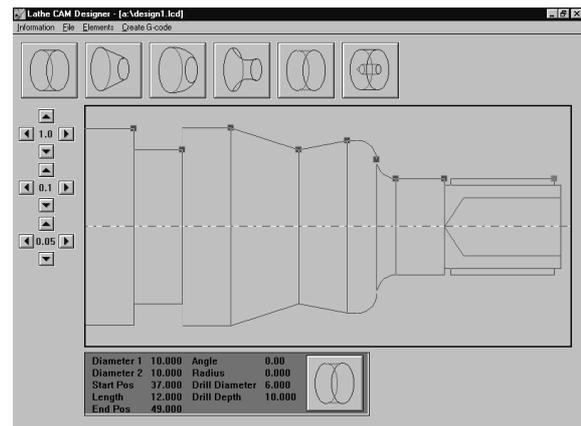
The *SET TOOLING* option is used to configure these graphics in the machine controlling software. This will ensure that the correct tools are drawn and used on the display screen at the right time.

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....

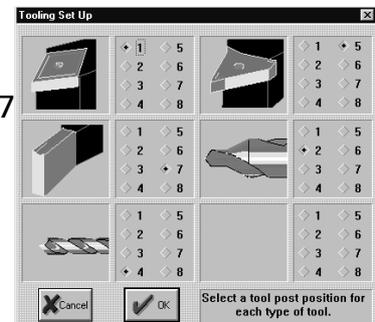


The following tools were present on the Novaturn toolpost:

- 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.

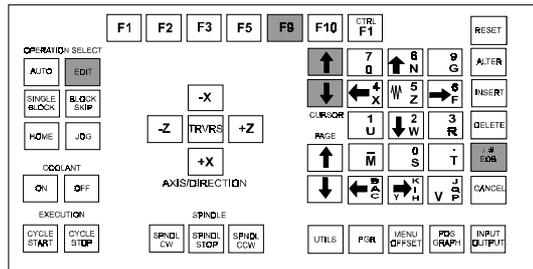
Keys Helpbox.

The following keys are used in this section:

[EDIT], [F9]

[CURSOR ARROWS]

[EOB]

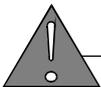
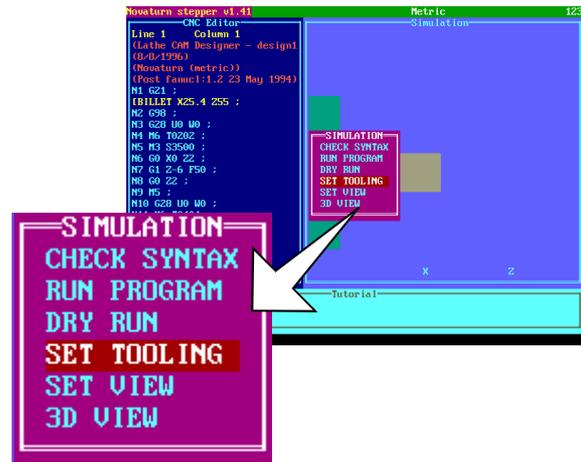


Tutor keypad.

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].



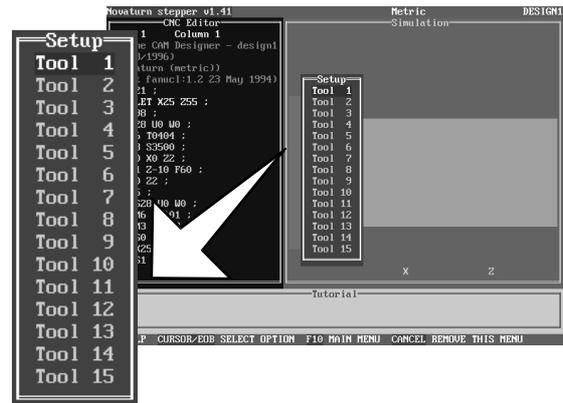
Please note -

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

# SIMULATION MENU

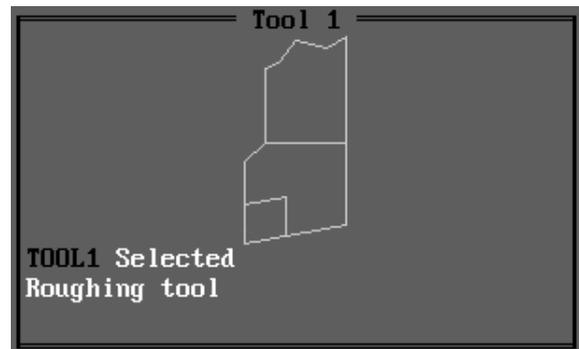
## - SET TOOLING.

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.

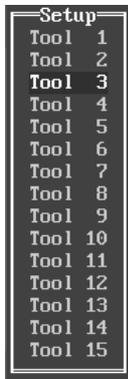
# SIMULATION

## MENU

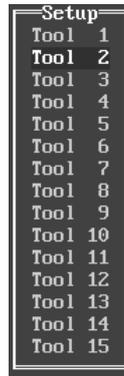
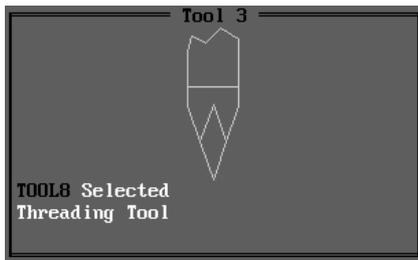
### - SET TOOLING.

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

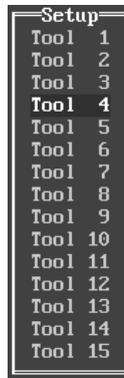
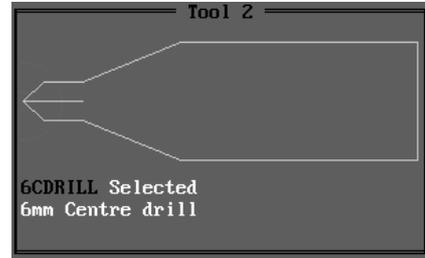
(These screenshots show how the graphics would be set to match the tool numbers in the LatheCAM software/ CNC File and the toolpost, shown at the beginning of this section).



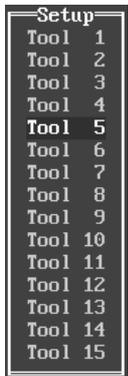
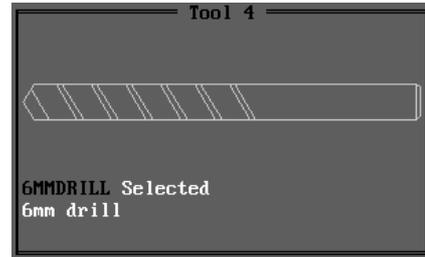
Tool 3 - Threading tool.



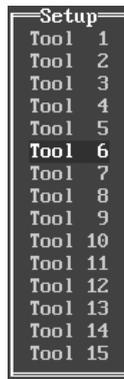
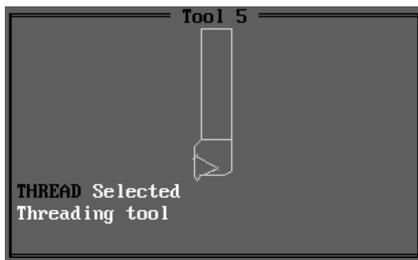
Tool 2 - Centre drill.



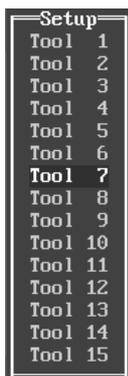
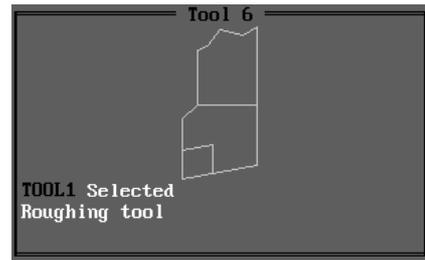
Tool 4 - 6mm drill.



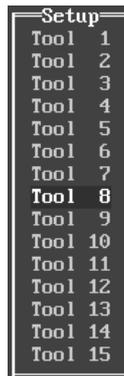
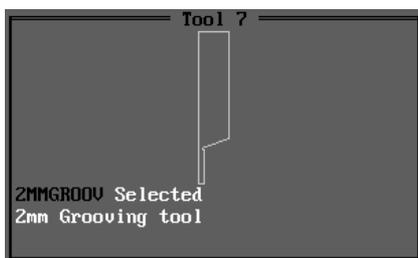
Tool 5 - Threading tool.



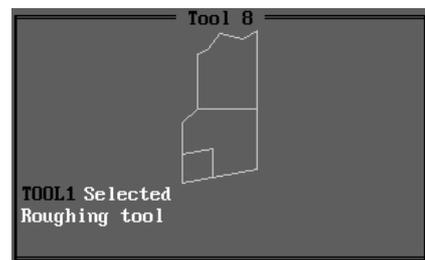
Tool 6 - No tool present.



Tool 7 - Undercutting tool.



Tool 8 - No tool present.

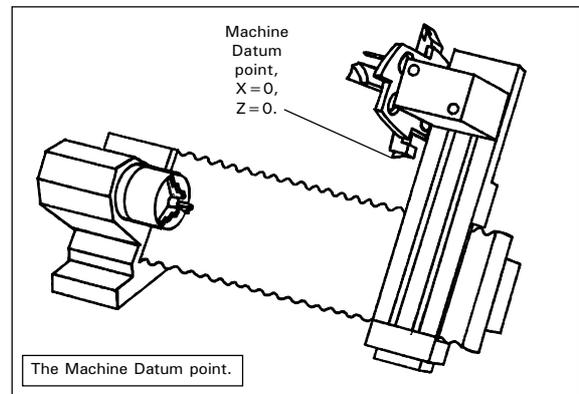


# SETTING THE TOOL OFFSETS.

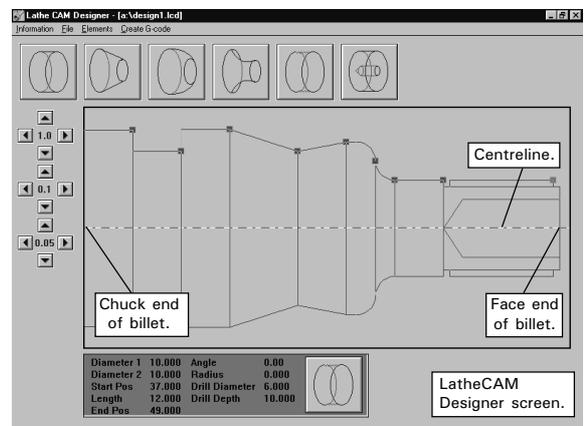
Setting the offsets tells each tool where the *actual* billet is positioned, so it can start cutting in the correct place.

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....

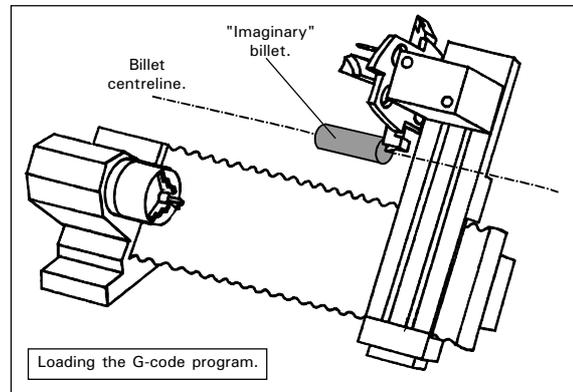


The G-code program from LatheCAM Designers post-processor describes the path taken by the tools when cutting the part. All these movement commands are relative to the position  $X=0$ ,  $Z=0$ . This position is usually the face end of the billet along its centreline - see the screenshot below....



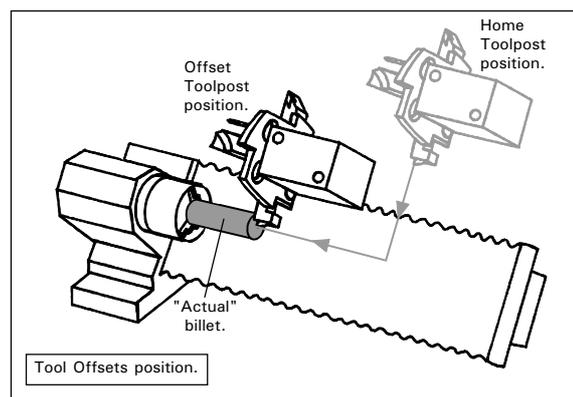
# SETTING THE TOOL OFFSETS.

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.

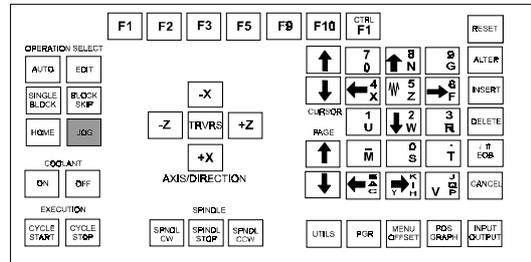


# SETTING THE X AND Z OFFSET VALUES.



This section uses the roughing tool as an example for setting the offsets - page 4.19 shows how to set the offsets using other tool profiles.

Keys Helpbox.  
The following keys are used in this section:  
[JOG]



Tutor keypad.

### 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.

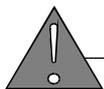
[-Z] - moves the toolpost left/towards the chuck.

1

To set the X and Z co-ordinates, the tip of the tool needs to be moved so it is just touching the side of the billet (X) and the end of the billet (Z).

Select *JOG MODE* by pressing the [JOG] key. When in *JOG MODE*, the lathe can be controlled manually. The screen will display the X and Z co-ordinates (measured in millimetres), the tool selected and the spindle action (this should be "off").

Before moving the tool in any direction, you need to set the movement to *CONTINUOUS* (ie, the tool will move continuously, so long as one of the movement keys are being pressed). To do this, press the [JOG] key again until the screen shows the word *CONTINUOUS* in the lower area of the display. Continued pressing of the [JOG] key will toggle the *JOG MODE* between its two settings - *CONTINUOUS* and *JOG STEP*.



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



# SETTING THE X AND Z OFFSET VALUES.

Keys Helpbox.  
The following keys are used in this section:  
[S], [T]  
[NUMBERS] - not highlighted.  
[EOB]

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?  
lowest: 25 highest: 3500

+ 0.000 Feed 100 Override 100%  
+ 0.000 Tool 0101 Drives on  
Spindle off Spindle Forward Override 100%

Speed? 1000  
lowest: 25 highest: 3500

Tutorial

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.

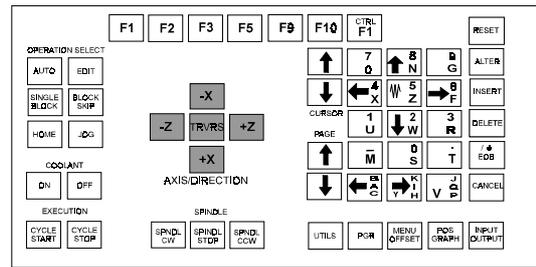
[-X] - moves the toolpost down/towards the front of the machine.

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

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

3

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



Tutor keypad.

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

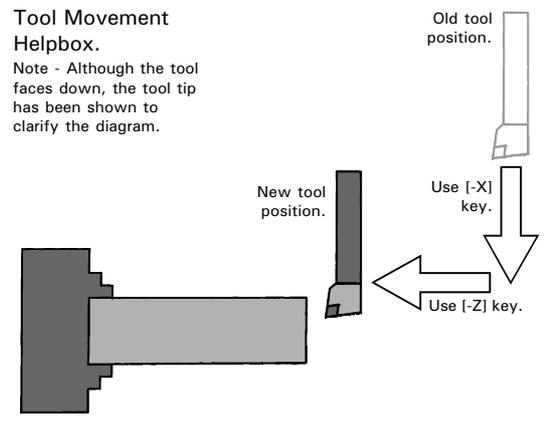
Check that *JOG MODE* is set to *CONTINUOUS*, then move the tool until it is close to the end of the billet, but not touching. Move the tool using the [+X], [-X], [+Z] and [-Z] keys. Pressing the [TRVRS.] key at the same time will increase this rate of movement - see the "Tool Movement Helpbox", below.

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).



## Tool Movement Helpbox.

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



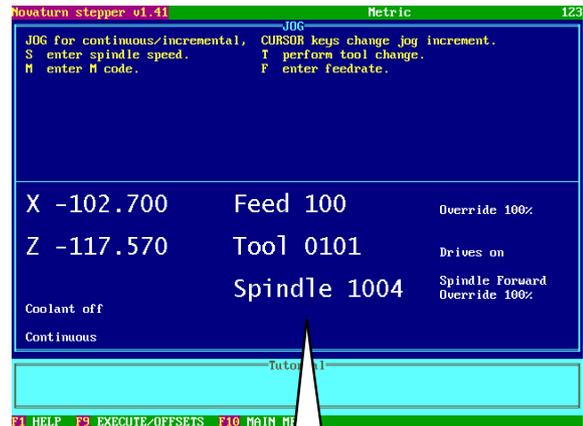
# SETTING THE Z CO-ORDINATE VALUE.

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

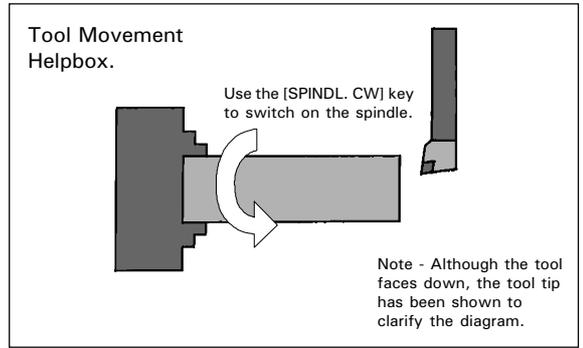
Tutor keypad.

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

### Movement keys.....

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

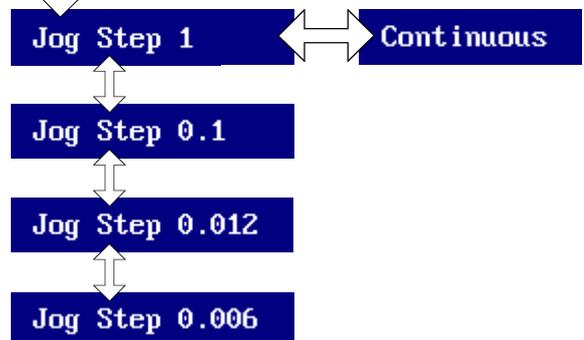
[-X] - moves the toolpost down/towards the front of the machine.

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

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

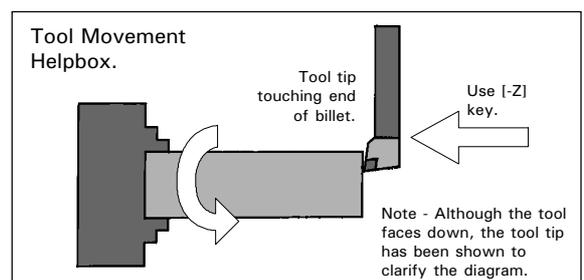
continued.... In Jog Step, the tool will move a set amount every time one of the movement keys is pressed. The size of this movement will be displayed, for example *JOG STEP 0.012* is a movement of 0.012mm every time a movement key is pressed.

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.



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.

Keys Helpbox.  
The following keys are used in this section:  
[MENU OFFSET], [CURSOR ARROWS]  
[NUMBERS] - not highlighted.  
[M], [Z], [EOB], [RESET]

Tutor keypad.

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.

Novaturn stepper v1.41 Metric 123

JOG for continuous/incremental, CURSOR keys change jog increment.  
S enter spindle speed. T perform tool change.  
M enter M code. F enter feedrate.

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

X - 91.9  
Z - 119.8

Coolant Off  
Jog Step 0.012

override 100%  
drives on  
spindle Forward  
override 100%

Tutorial

page X Z R T edit

**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

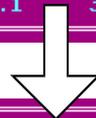
Press the [Z] key to move the cursor into the Z Tool Offset values column.

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



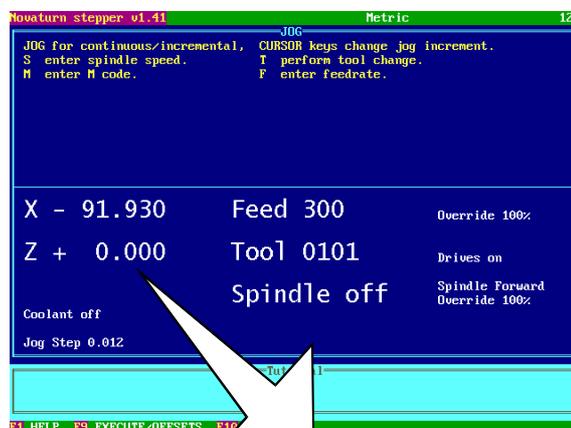
Tool Offsets				
Tool	X	Z	R	T
1	0	-119.86	0.1	3
2	0	0	0.1	3
3	0	0	0.1	3
4	0	0	0.1	3
5	0	0	0.1	3
6	0	0	0.1	3
7	0	0	0.1	3
8	0	0	0.1	3

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.

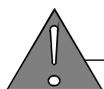


Z + 0.000

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.

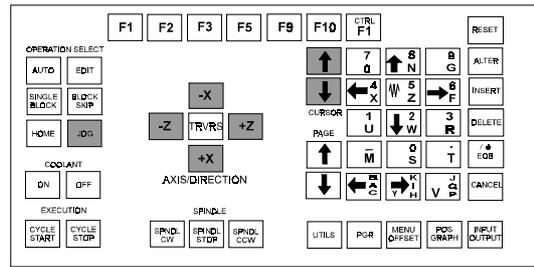
[+Z] - moves the toolpost right/away from the chuck.

[-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.

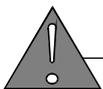
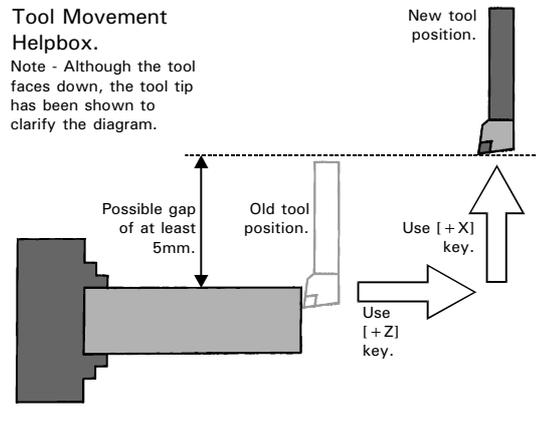
[-X] - moves the toolpost down/towards the front of the machine.

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

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

## Tool Movement Helpbox.

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



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.

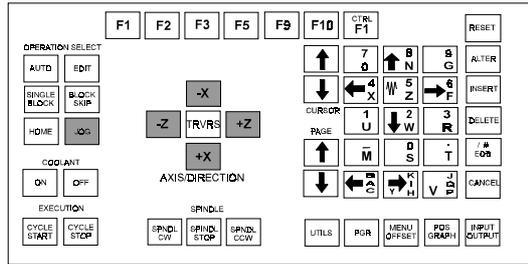
[-X] - moves the toolpost down/towards the front of the machine.

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

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

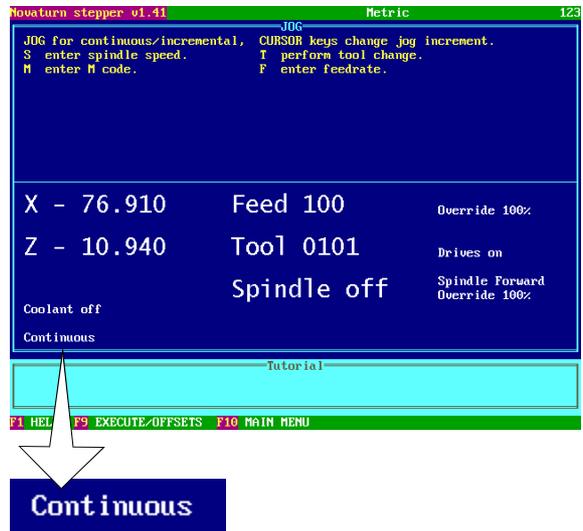
7

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

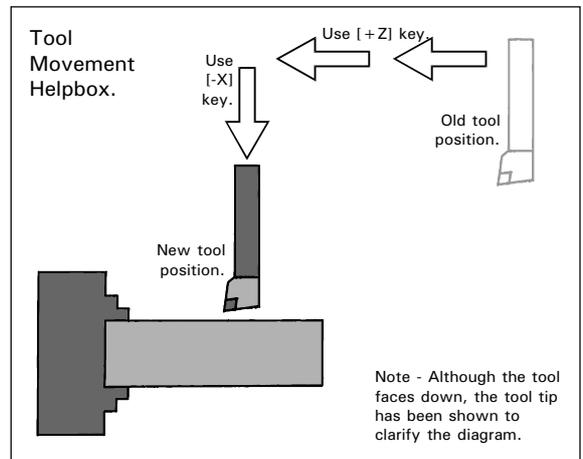


Tutor keypad.

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

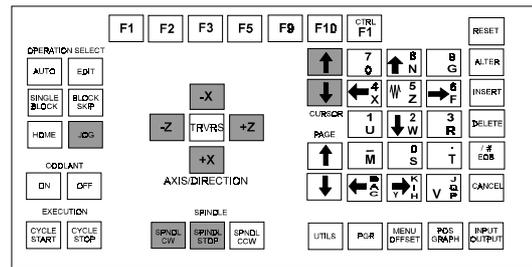


Move the tool until it is positioned along the side of the billet, but not touching the billet surface, using the [+X], [-X], [+Z] and [-Z] keys - see the "Tool Movement Helpbox", below.



# SETTING THE X CO-ORDINATE VALUE.

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

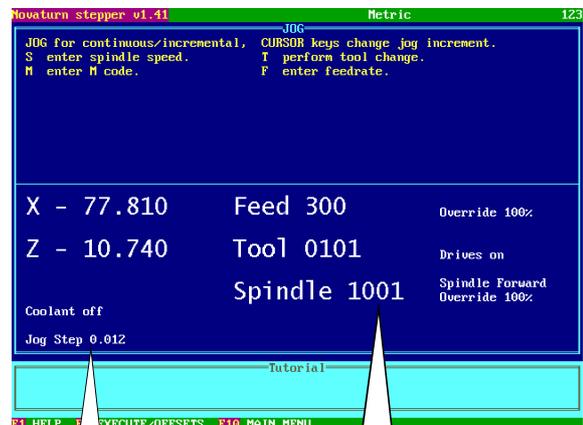


Tutor keypad.

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).



Jog Step 0.012

Spindle 1001

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.....

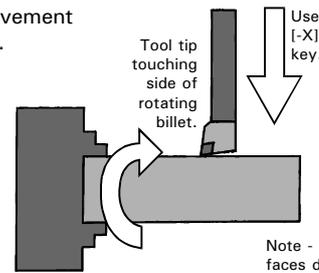
[+X] - moves the toolpost up/away from the front of the machine.

[-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.

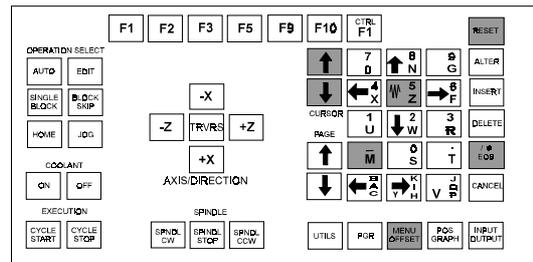
### Keys Helpbox.

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*.

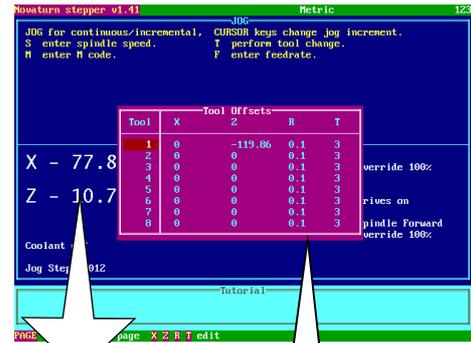
Make a note of the X co-ordinate offset value from the main screen, then select the *TOOL OFFSET TABLE* by pressing the [MENU OFFSET] key.

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	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.

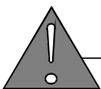
Tool	X	Z	R	T
<b>M 1</b>	0	-119.86	0.1	3
2	0	0	0.1	3
3	0	0	0.1	3
4	0	0	0.1	3
5	0	0	0.1	3
6	0	0	0.1	3
7	0	0	0.1	3
8	0	0	0.1	3

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	25.4	-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".

```

Mouaturn stepper v1.41 Metric 123
JOG
JOG for continuous/incremental, CURSOR keys change jog increment.
S enter spindle speed. T perform tool change.
M enter M 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

F1 HELP F9 EXECUTE/OFFSETS F10
  
```

**X + 25.400**

Main screen X co-ordinate value.



# 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....

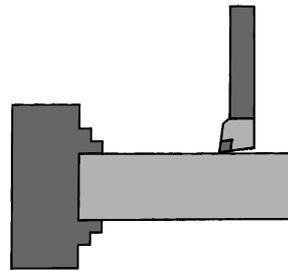
## ROUGHING OR FINISHING TOOL.

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.

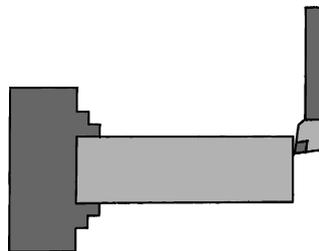
Touch the pointed tip of the tool onto the side of the rotating billet. Enter the X value following the procedures outlined in section ⑤ of setting the offsets.



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.

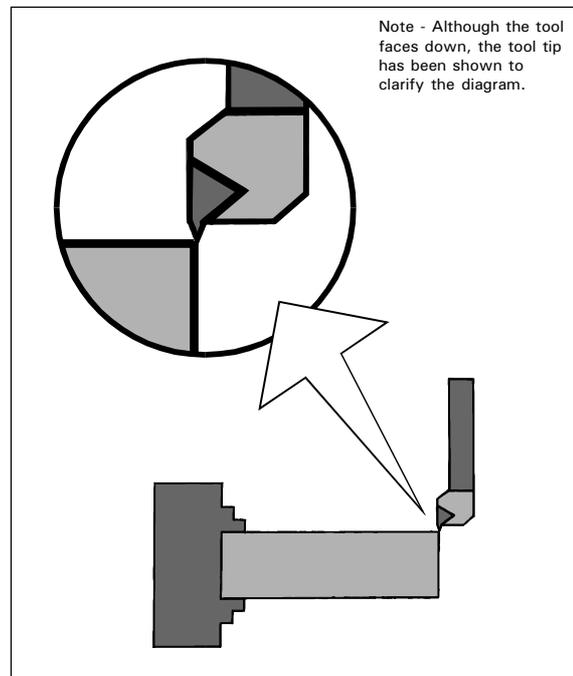
# SETTING THE TOOL OFFSETS - THREADING TOOL.

## 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.



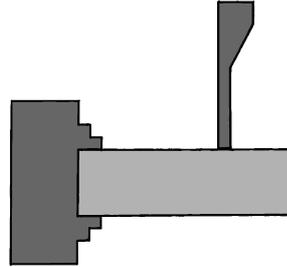
# SETTING THE TOOL OFFSETS - PARTING TOOL.

## PARTING TOOL.

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

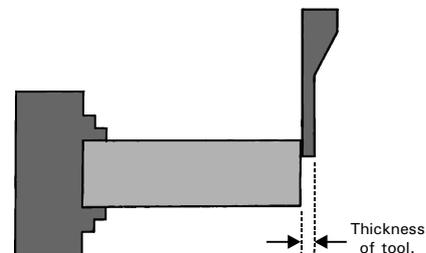
### X TOUCHING-ON POINT.

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.



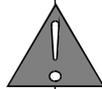
Note - Although the left side of the tool is used for touching the face end of the billet, it is the right side of the tool that will be touching the end of the workpiece when it is parted-off. The thickness of the parting-off tool is automatically compensated for by LatheCAM Designer software.

# 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....



Combined X value (X value when entered with the "M" prefix)  
= diameter of billet (mm) + diameter of drill (mm).

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.

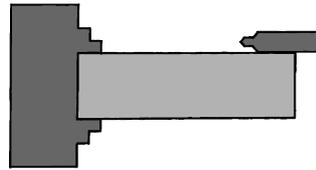
# SETTING THE TOOL OFFSETS - CENTRE DRILL.

## CENTRE DRILL.

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

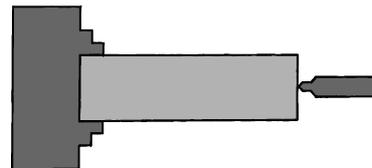
### X TOUCHING-ON POINT.

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

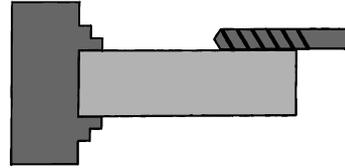
## - TWIST DRILL.

### TWIST DRILL.

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

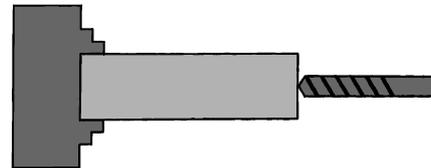
#### X TOUCHING-ON POINT.

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.



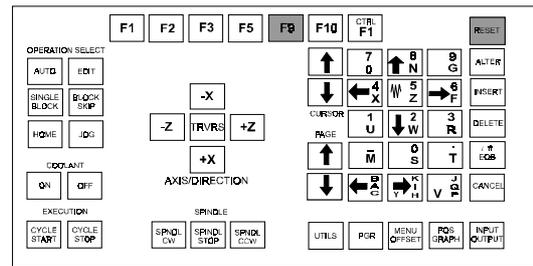
#### Z TOUCHING-ON POINT.

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.

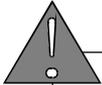


# CONTROL MENU.

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



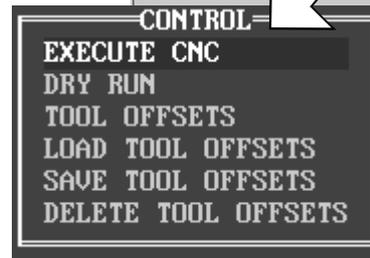
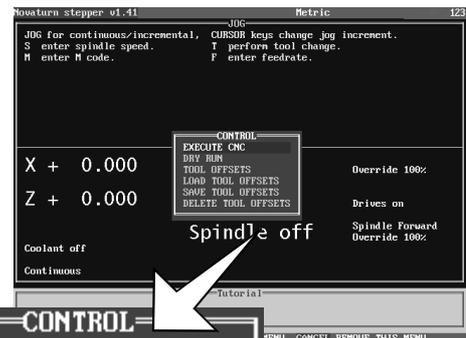
Tutor keypad.



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.

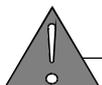
Select the *OFFSET CONTROL MENU* by pressing the [F9] key.



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.

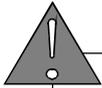


Please note -

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.

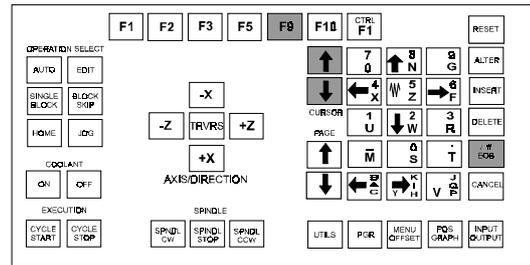
Keys Helpbox.

The following keys are used in this section:

[F9]

[CURSOR ARROWS]

[EOB]

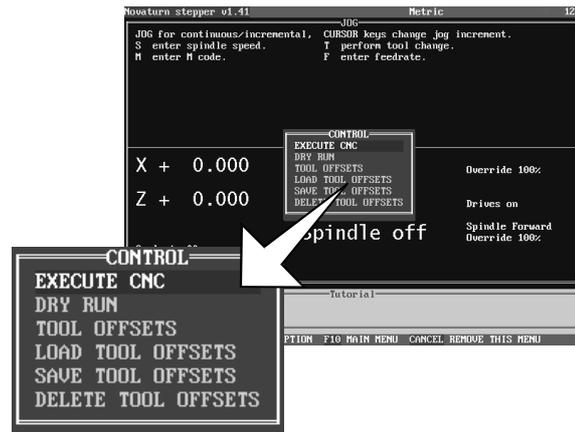


Tutor keypad.

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.



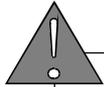
Please note -

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



# CONTROL MENU

## - DRY RUN.



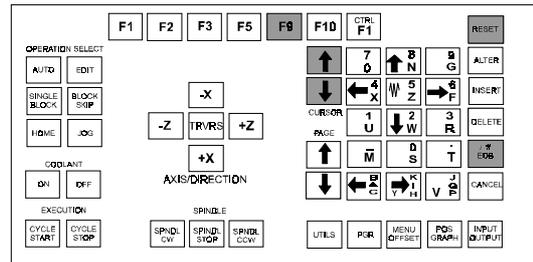
Please note -

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]  
 [EOB], [RESET]

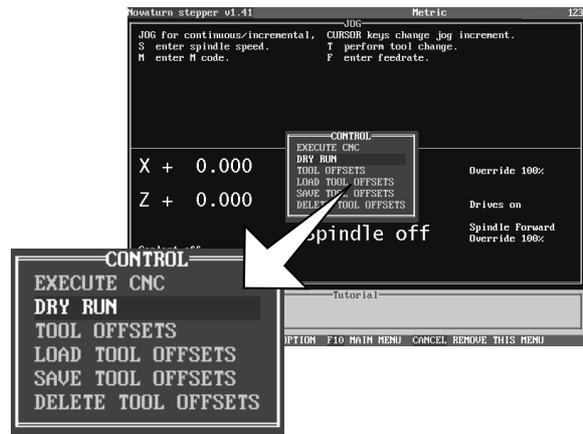


Tutor keypad.

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.

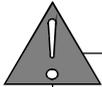


The *OFFSET CONTROL MENU* will be automatically cleared as the '*DRY RUN*' operation is started. If the '*DRY RUN*' operation is successful, no message windows will be displayed.

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.

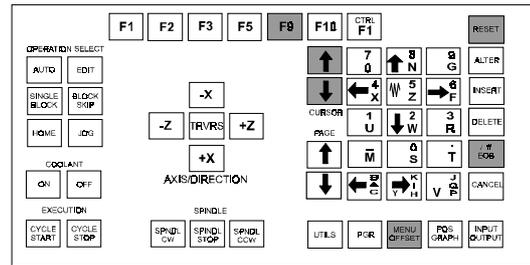


Please note -

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.

Keys Helpbox.

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



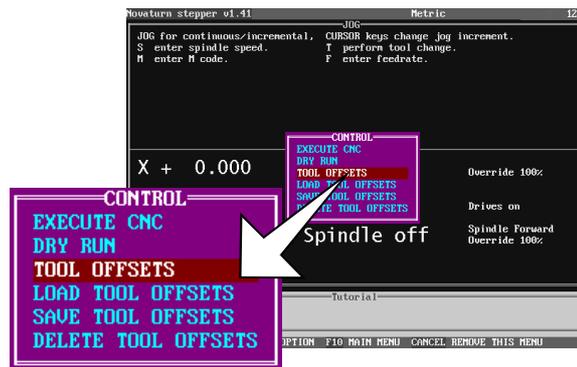
Tutor keypad.

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

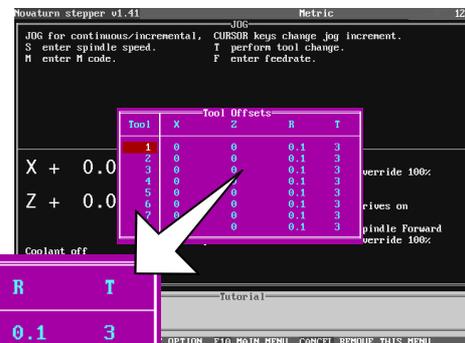
To display the Tool Offsets table, press the [F9] key (not available in *EDIT MODE*).

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



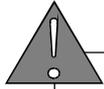
Please note -

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

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

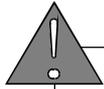
# CONTROL MENU

## - LOAD TOOL OFFSETS.



Please note -

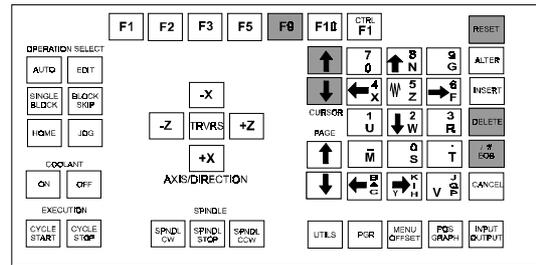
The function of the [F9] key will depend on the machine mode:  
*AUTO MODE* = Control Menu.  
*EDIT MODE* = Simulation Menu.  
*HOME MODE* = Control Menu.  
*JOG MODE* = Control Menu.



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".

Keys Helpbox.

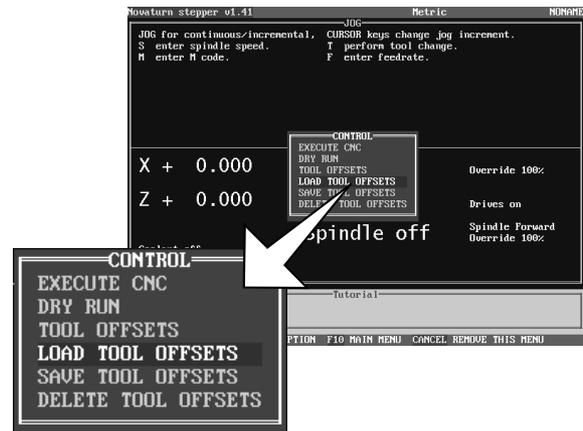
The following keys are used in this section:  
 [F9], [CURSOR ARROWS]  
 [NUMBERS] - not highlighted  
 [DELETE], [EOB], [RESET]



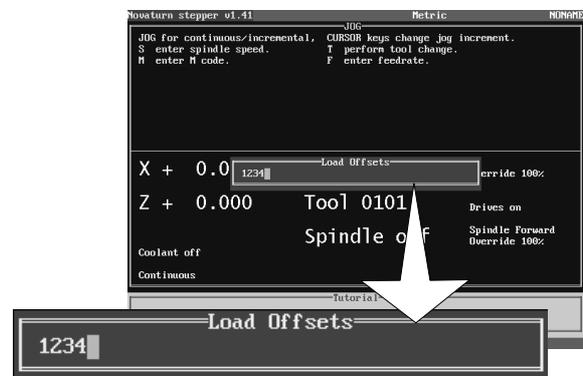
Tutor keypad.

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.



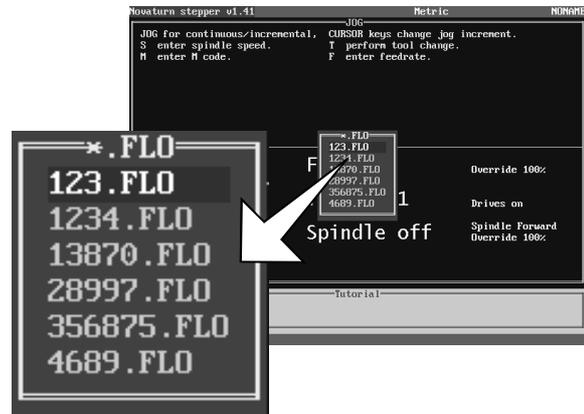
Enter the file name (number) for the collection of Tool Offsets using the [NUMBER] keys. Use the [DELETE] key to remove any incorrectly typed characters and confirm the filename by pressing the [EOB] key.



# 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.

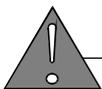
In most cases, this message (shown below) is displayed when the computer is trying to read the files from the wrong drive or directory - see page 4.33 "Changing the Drive Directory - Tool Offset Files".



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.



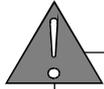
Press the [RESET] key to clear any error messages or menus.



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

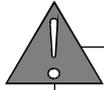
# CONTROL MENU

## - SAVE TOOL OFFSETS.



Please note -

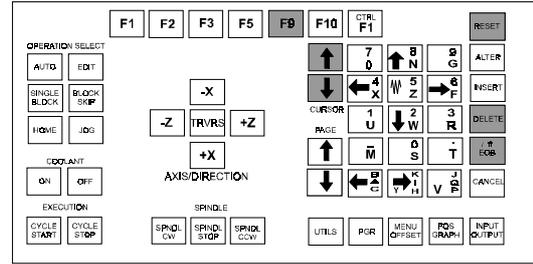
The function of the [F9] key will depend on the machine mode:  
*AUTO MODE* = Control Menu.  
*EDIT MODE* = Simulation Menu.  
*HOME MODE* = Control Menu.  
*JOG MODE* = Control Menu.



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".

Keys Helpbox.

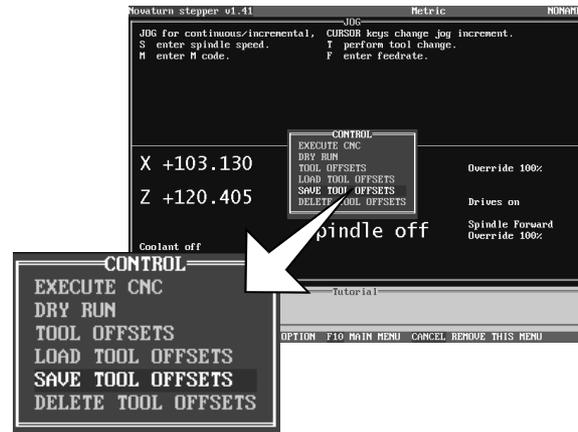
The following keys are used in this section:  
 [F9], [CURSOR ARROWS]  
 [NUMBERS] - not highlighted  
 [DELETE], [EOB], [RESET]



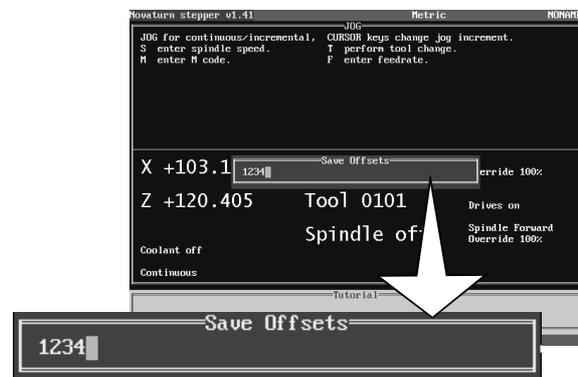
Tutor keypad.

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.



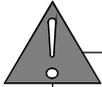
Enter a number (representing the offsets filename) using the [NUMBER] keys. Use the [DELETE] key to remove any incorrectly typed characters and confirm the filename by pressing the [EOB] key. Note that the file will be saved on the currently selected drive. Make a note of the name of your new offset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



Press the [RESET] key to clear any menus.

# CONTROL MENU

## - DELETE TOOL OFFSETS.

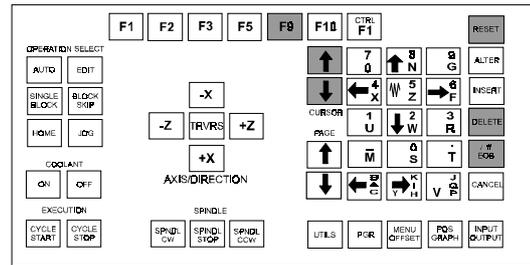


Please note -

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.

Keys Helpbox.

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

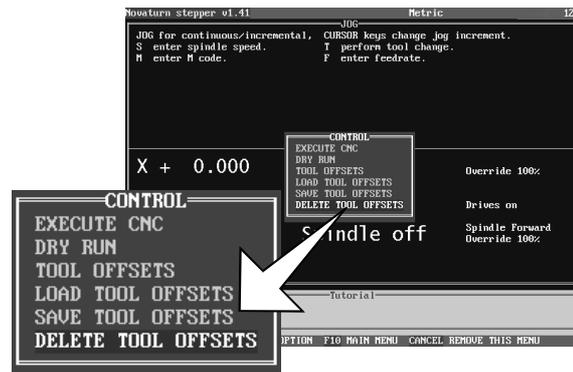


Tutor keypad.

The '*DELETE*' command will permanently remove a selected offsets file from the currently selected drive.

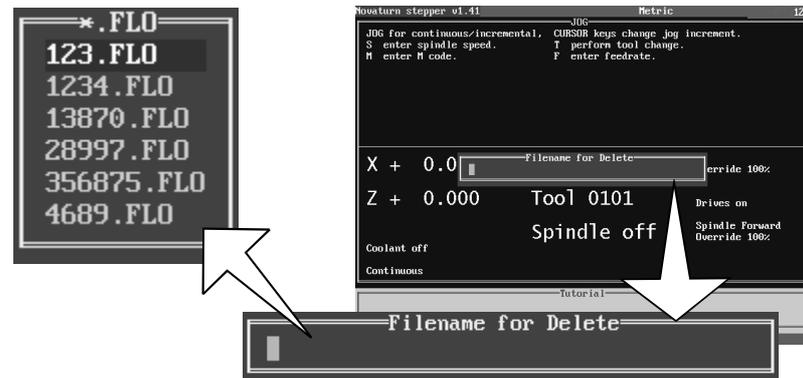
To remove a selected offsets file, select the *OFFSET CONTROL MENU* by pressing the [F9] key (not available in *EDIT MODE*).

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.

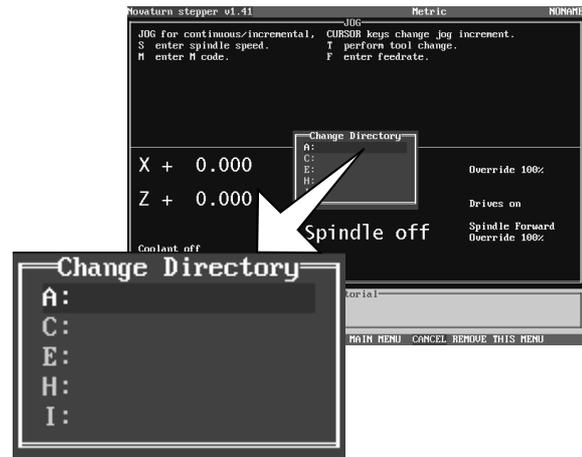


# CHANGING THE DRIVE DIRECTORY

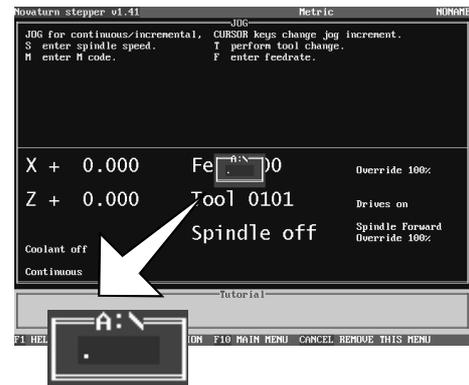
## - TOOL OFFSET FILES.

continued....

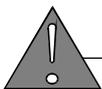
Highlight the drive required (in this example 'A:') using the [CURSOR ARROWS] keys.



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

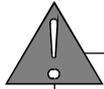


Press the [RESET] key to clear the screen of any unwanted menus or information.



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

# SAVING TOOL OFFSET FILES ON A CHANGED DRIVE.

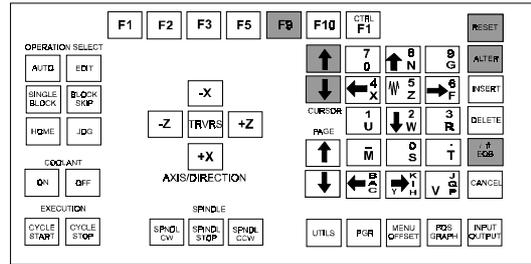


Please note -

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.

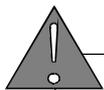
Select the *OFFSET CONTROL MENU* by pressing the [F9] key, highlight 'SAVE TOOL OFFSETS' using the [CURSOR ARROWS] keys and press the [EOB] key.



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.



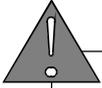
Enter the filename using the [NUMBERS] keys and press [EOB] to confirm this. The offset files will now be saved on the new drive.



Please Note -

CNC Files are stored in ".fnc" or ".fnc1" 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.



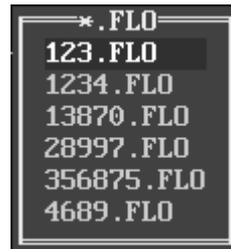
Note - the screen may display the previous setting for the drive. In the example above, the screen displays the drive as 'C:', even though it has just been changed to load from 'A:', as shown in the last section. If this occurs, press the [ALTER] key to reset to the new drive.



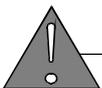
Enter the filename using the [NUMBERS] keys and press [EOB] to confirm this. The offset file will now be loaded from the new drive.



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.

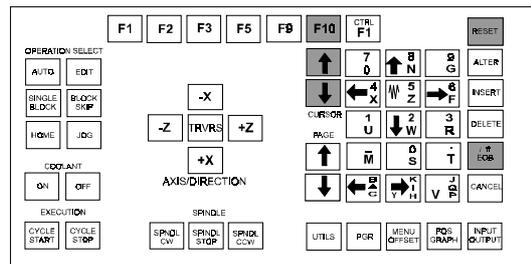


Please Note -

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

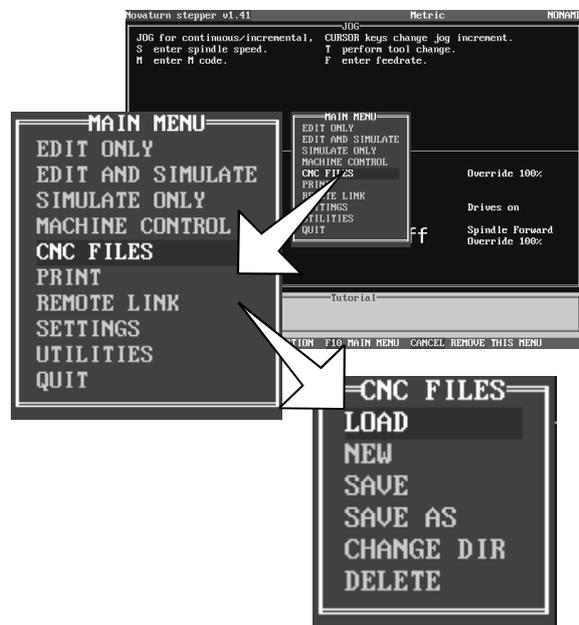
# CNC FILES MENU.

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



Tutor keypad.

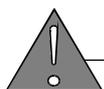
Select the *MAIN MENU* by pressing the [F10] key. Highlight '*CNC FILES*' using the [CURSOR ARROWS] keys and press the [EOB] key to confirm this choice.



The *CNC FILES MENU* contains 5 or 6 options (depending on your machine set-up) :

- i) *LOAD* - load a selected CNC File.
- ii) *NEW* - clear a currently loaded CNC File.
- iii) *SAVE* - overwrite a CNC File with the same name or save a newly created CNC File.
- iv) *SAVE AS* - save a CNC File with a specific name.
- v) *CHANGE DIR* - change the drive used to load and save CNC Files.
- vi) *DELETE* - delete a selected CNC File (this option is not be available on some versions of the control software).

Press the [RESET] key to clear any menus.



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

# CNC FILES MENU - LOAD.

**!** 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 CNC Files from the default drive, then the drive destination must be changed. See page 5.9 "Changing the Drive Directory - CNC Files".

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

Tutor keypad.

Highlight 'LOAD' in the *CNC FILES MENU* using the [CURSOR ARROWS] keys and press the [EOB] key.

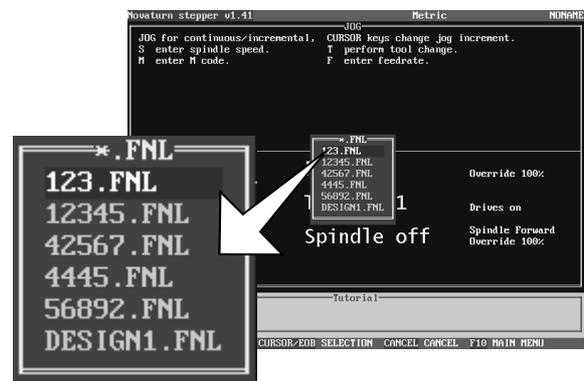
Type in the name of the CNC File you wish to load, using the [NUMBERS] keys. Incorrect characters can be removed using the [DELETE] key. Press the [EOB] key to load the CNC File.

If the filename contains any alphabet characters, it can only be loaded from a directory listing (see the next page).

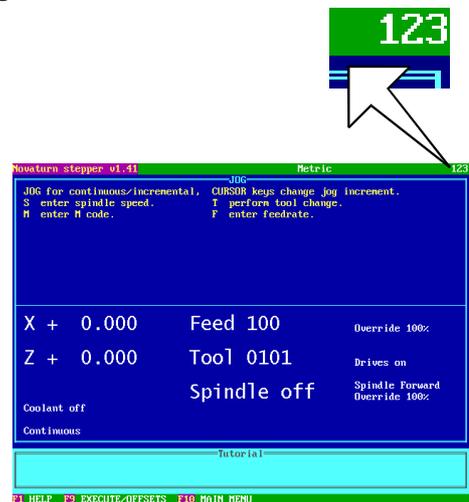
**!** Please note - The co-ordinate values depicted on screenshots are used for illustrative purposes only.

# CNC FILES MENU - LOAD.

If the filename is unknown, press the [EOB] key to list all the CNC Files stored on the currently selected drive. CNC Files within these lists can be loaded by highlighting them using the [CURSOR ARROWS] keys and pressing the [EOB] key.



The name of the CNC File you have just loaded is displayed in the top right hand corner of the screen.



Note - If there is a CNC File currently in the editor when loading a new CNC File, the screen will display '*Do you want to merge?*' (ie, do you want to combine the program you wish to load with the program already loaded). To merge CNC Files press the [EOB] key, otherwise, press the [N] key to clear the current CNC File and load the selected CNC File into the editor.



# CNC FILES MENU - LOAD.

If no CNC Files can be found on the selected drive, an error message will be displayed (see example below). To clear this message, press the [RESET] key.

In most cases, this message is displayed when the computer is trying to read the CNC Files from the wrong drive or directory - refer to section 5.9 "Changing the Drive Directory - CNC Files".

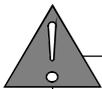
Error  
There are no matching files

The error message below may be displayed if the CNC 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.

Error  
Invalid filename given

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

# CNC FILES MENU - ADVANCEDLOAD SETUP.



Please note - Any changes made to these settings should be carried out by either your IT Manager or computer technician.

Please Note - Denford CNC Lathes will recognise both ".fnc" and ".fnl" extension filenames. Denford LatheCAM software saves G-code programs with the file extension ".fnc".

If errors are encountered when loading CNC Files, check that the correct drive and directory (pathnames) are being used by the control software and correct if necessary (see page 5.9).

To change the type of CNC File recognised by the control software, open a DOS box and change the pathname to read the directory where the machine options file is stored, for example type....

## CD \Machine

.... where **Machine** is the name of your lathe and press the [RETURN] key on the QWERTY keyboard.

Type the following to change to recognise files with the extension ".fnc"....

## OPTION FLSTEP FILETYPE FNC

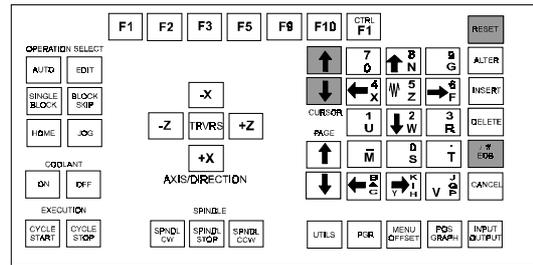
Type the following to change to recognise files with the extension ".fnl"....

## OPTION FLSTEP FILETYPE FNL

Note - For the Denford Microturn replace the word **FLSTEP** with **FANUCML**.

# CNC FILES MENU - NEW.

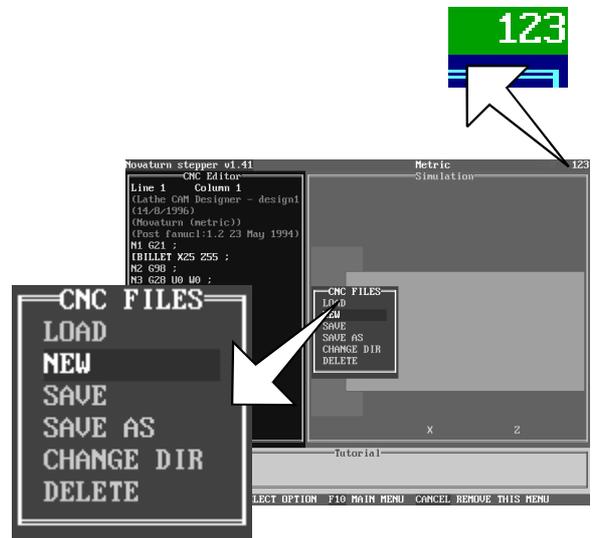
Keys Helpbox.  
The following keys are used in this section:  
[CURSOR ARROWS]  
[EOB]  
[RESET]



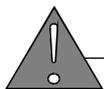
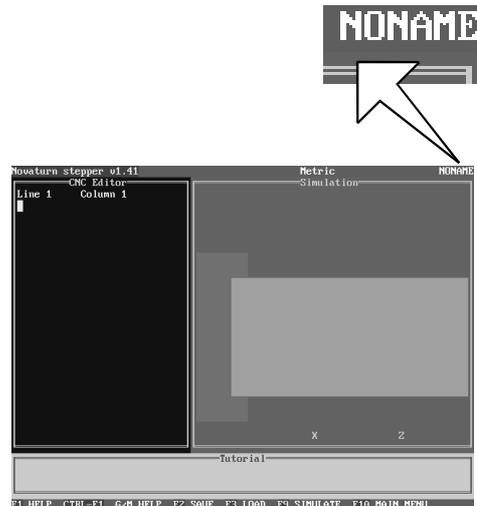
Tutor keypad.

The 'NEW' command will clear any CNC File currently loaded in the editor.

Highlight 'NEW' in the *CNC FILES MENU* with the [CURSOR ARROWS] keys and press the [EOB] key.



The current CNC File is cleared leaving the editor with no CNC File loaded.



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

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

# CNC FILES MENU - SAVE.

**!** 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 CNC Files on the default drive, then the drive destination must be changed. See page 5.9 "Changing the Drive Directory - CNC Files".

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

Tutor keypad.

Highlight 'SAVE' in the *CNC FILES MENU* using the [CURSOR ARROWS] keys and press the [EOB] key.

If the CNC File has never been saved, ie, it has just been manually entered, the software will prompt for a filename. Use the [NUMBERS] keys to write a filename into the dialog box. Incorrect characters can be removed using the [DELETE] key. Press the [EOB] key to save the CNC File.

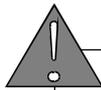
Note that CNC Files can only be saved with alphabet characters when using a Qwerty keyboard.

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

If a CNC File has been previously loaded, then edited, the 'SAVE' command will overwrite the original CNC File with the new edited version - no filename dialog box will be displayed. Press the [RESET] key to clear the screen of any unwanted menus or information.

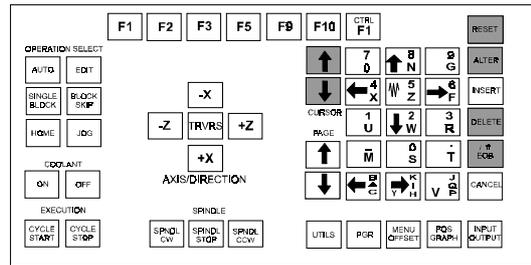
# CNC FILES

## MENU - SAVE AS.



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 CNC Files on the default drive, then the drive destination must be changed. See page 5.9 "Changing the Drive Directory - CNC Files".

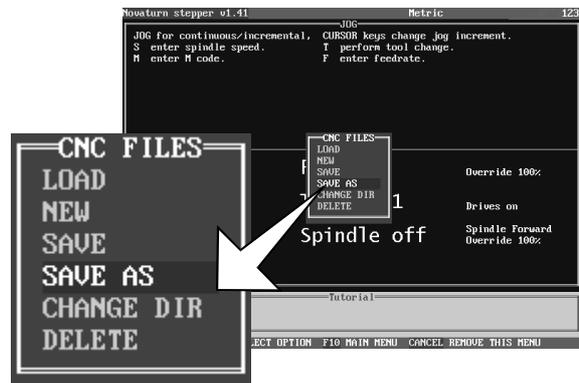
Keys Helpbox.  
The following keys are used in this section:  
[CURSOR ARROWS], [EOB]  
[NUMBERS] - not highlighted  
[DELETE], [RESET]



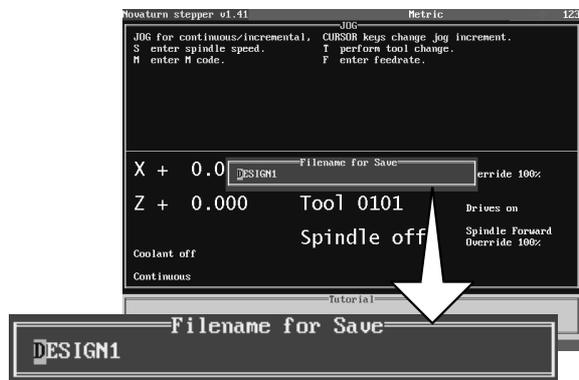
Tutor keypad.

To save a newly edited CNC File with a different name from the original version, the 'SAVE AS' command must be used.

Highlight 'SAVE AS' in the *CNC FILES MENU* using the [CURSOR ARROWS] keys and press the [EOB] key.



To remove the old filename, press the [ALTER] key, then enter the new filename using the [NUMBERS] keys. Any characters typed in by mistake can be corrected using the [DELETE] key.

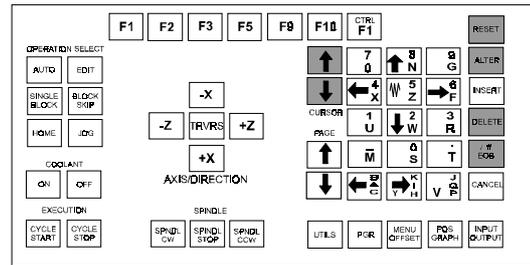


Press the [EOB] key to save this new CNC File to the currently selected drive. Press the [RESET] key to clear the screen of any unwanted menus or information.

# CNC FILES MENU - DELETE.

Keys Helpbox.

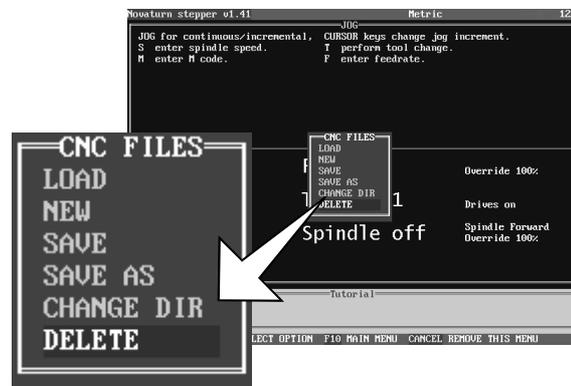
The following keys are used in this section:  
[CURSOR ARROWS], [EOB]  
[NUMBERS] - not highlighted  
[DELETE], [RESET]



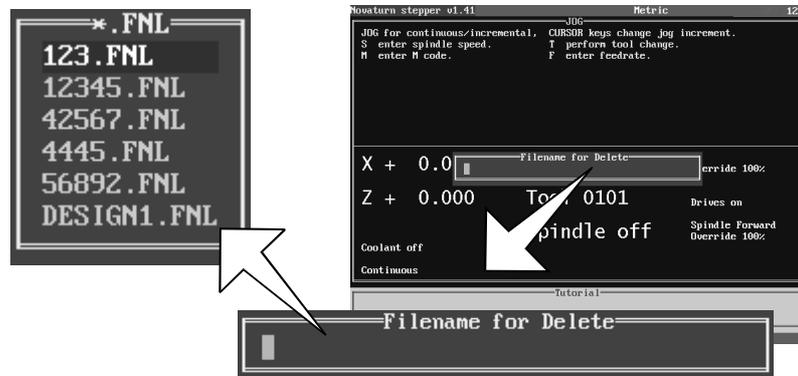
Tutor keypad.

To permanently remove a CNC File, use the '*DELETE*' command.

Highlight '*DELETE*' in the *CNC FILES MENU* using the [CURSOR ARROWS] keys and press the [EOB] key.



To remove a CNC File, enter the filename using the [NUMBERS] keys. Any characters typed in by mistake can be corrected using the [DELETE] key. If the filename is unknown, press the [EOB] key to list all the files on the currently selected drive, then highlight the required file using the [CURSOR ARROWS] keys.



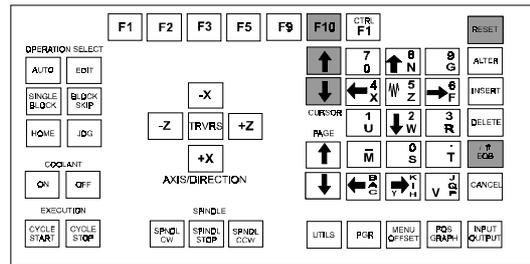
Please note -

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

Press the [EOB] key to delete the selected CNC File. Press the [RESET] key to clear the screen of any unwanted menus or information.

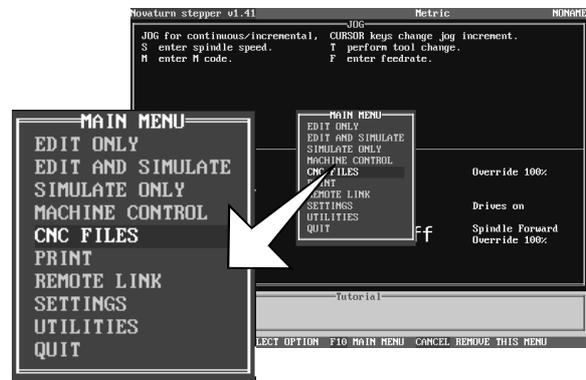
# CHANGING THE DRIVE DIRECTORY - CNC FILES.

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

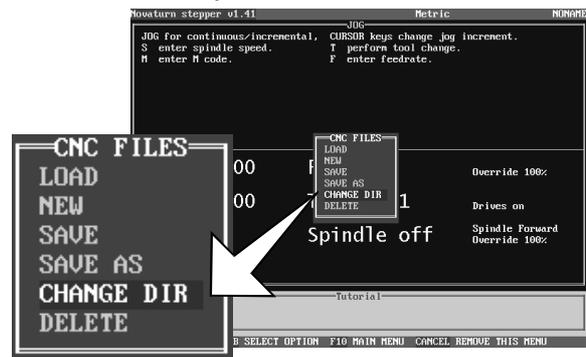


Tutor keypad.

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.

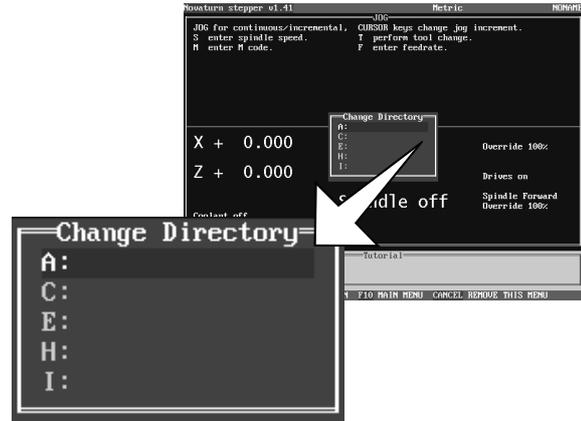


continued....

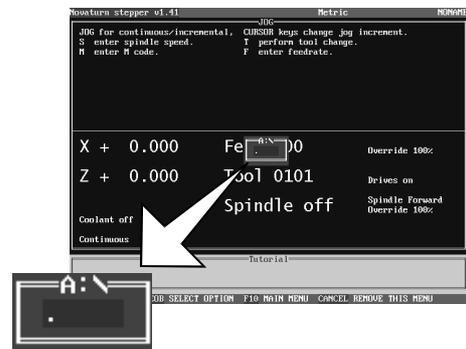
# CHANGING THE DRIVE DIRECTORY - CNC FILES.

continued....

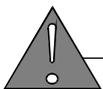
Highlight the drive required (in this example 'A:') using the [CURSOR ARROWS] keys.



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



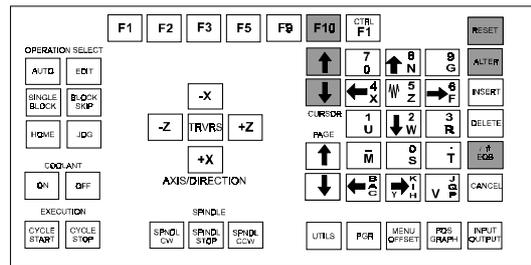
Press the [RESET] key to clear the screen of any unwanted menus or information.



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

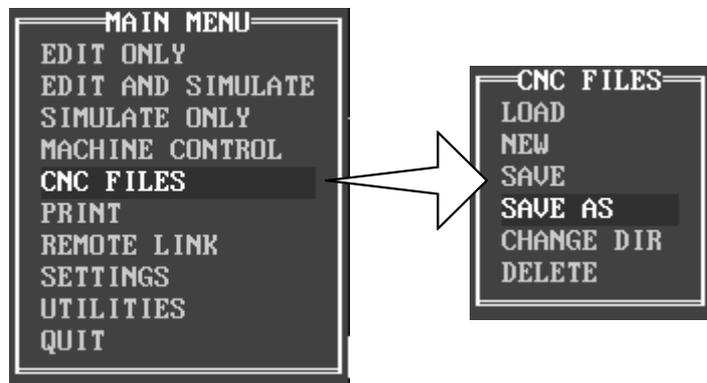
# SAVING CNC FILES ON A CHANGED DRIVE.

Keys Helpbox.  
The following keys are used in this section:  
[F10], [CURSOR ARROWS]  
[NUMBERS] - not highlighted  
[EOB], [ALTER], [RESET]



Tutor keypad.

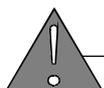
Select the *MAIN MENU* by pressing the [F10] key, highlight 'CNC FILES' and press [EOB]. Highlight the 'SAVE AS' option and press the [EOB] key.



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 save on 'A:', as shown in the last section. If this occurs, press the [ALTER] key to reset to the new drive.



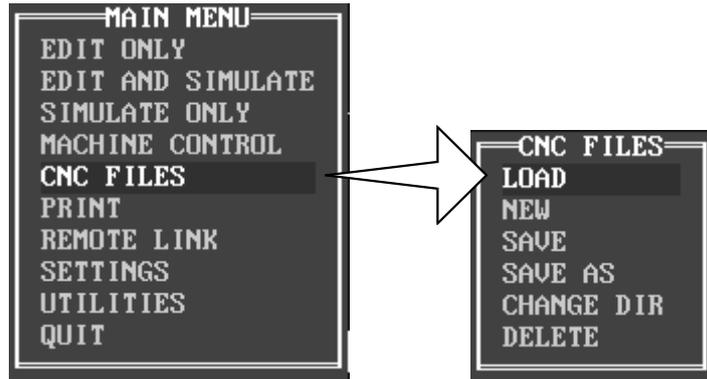
Enter the filename using the [NUMBERS] keys and press [EOB] to confirm this. The CNC File will now be saved on the new drive.



Please Note -  
CNC Files are stored in ".fnc"  
or ".fml" format (see page 5.4).  
Offset Files are stored in ".flo"  
format.

# LOADING CNC FILES ON A CHANGED DRIVE.

Select the *MAIN MENU* by pressing the [F10] key, highlight 'CNC FILES' and press the [EOB] key. Highlight the 'LOAD' option 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.



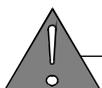
Enter the filename using the [NUMBERS] keys and press [EOB] to confirm this. The CNC File will now be loaded from the new drive.



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



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



Please Note - CNC Files are stored in ".fnc" or ".fnl" format (see page 5.4). Offset Files are stored in ".flo" format.

# SIMULATE CNC FILE.

Once a CNC file has been loaded, its action can be simulated on screen. Remember that the name of the CNC file is displayed in the top right-hand corner of the display. In the example screenshots, a program called "123" has been loaded.

Simulating a program can be useful for checking, the order of cutting commands, the appearance of the end result and whether the program contains any mistakes etc ..... Work materials do not have to be clamped in the chuck at this stage, since the machine cannot cut during a simulation exercise.

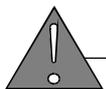
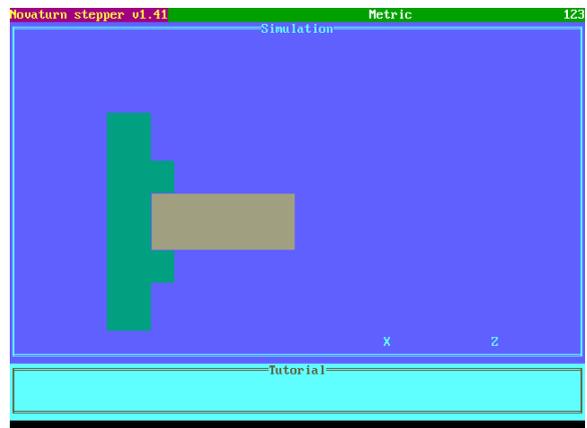
# SIMULATE - SCREEN VIEW OPTIONS.

CNC Files can be simulated in two main screen view options:

*EDIT AND SIMULATE.* The CNC File can be altered using the editor side of the screen and then simulated using the graphical display(plan or 3d).



*SIMULATE ONLY.* The CNC File can be simulated using a full screen graphical display(plan or 3d).



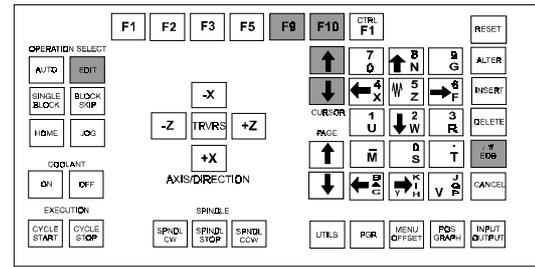
Please note -  
The co-ordinate values depicted  
on screenshots are used for  
illustrative purposes only.

# SIMULATE

## - SELECT

# SCREEN VIEW.

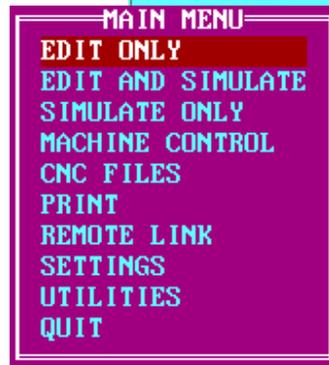
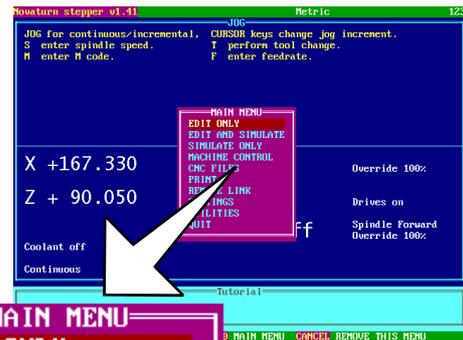
Keys Helpbox.  
 The following keys are used in this section:  
 [F10], [F9]  
 [CURSOR ARROWS], [EOB]  
 [EDIT]



Tutor keypad.

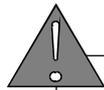
The required screen view can be accessed by selecting the *MAIN MENU* by pressing the [F10] key. Highlight either the '*EDIT AND SIMULATE*' or '*SIMULATE ONLY*' options, using the [CURSOR ARROWS] keys and press the [EOB] key to select the highlighted option.

Alternatively, pressing the [EDIT] key will directly access the '*EDIT AND SIMULATE*' option.



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

# SIMULATION MENU.

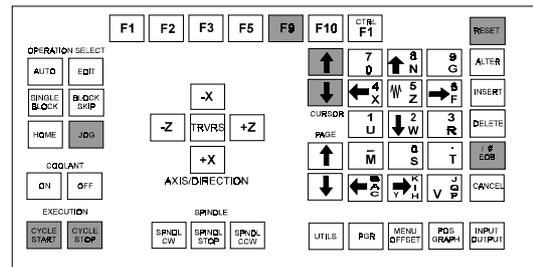


Please note -

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.

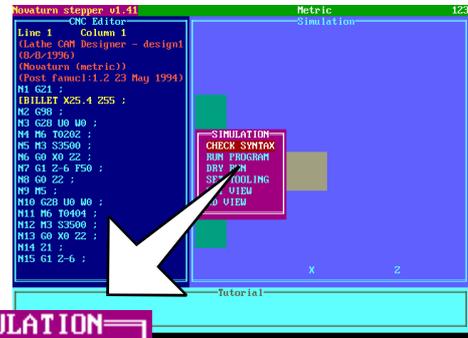
The following keys are used in this section:  
 [F9], [CURSOR ARROWS]  
 [EOB], [JOG]  
 [CYCLE START], [CYCLE STOP], [RESET]



Tutor keypad.

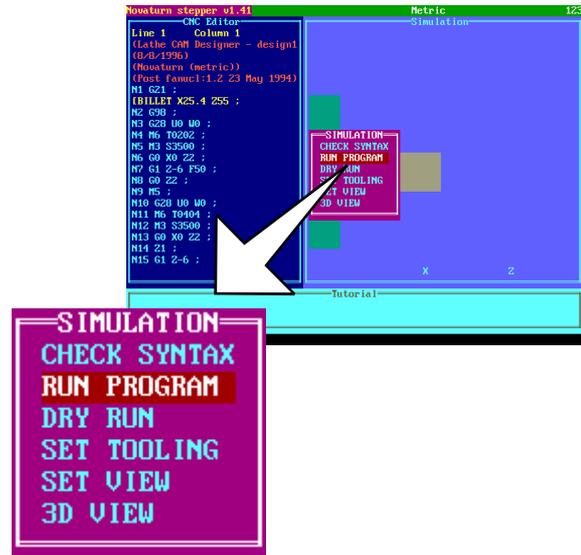
To start simulating the CNC File (program) in the chosen screen view, select the *SIMULATION MENU* by pressing the [F9] key. All example screenshots are shown using the '*EDIT AND SIMULATE*' option.

1) *CHECK SYNTAX*. This checks for illegal G-codes without running the program. To run this option, highlight '*CHECK SYNTAX*' using the [CURSOR ARROWS] keys and press [EOB]. The message indicating the results of this exercise is cleared by pressing the [RESET] key.



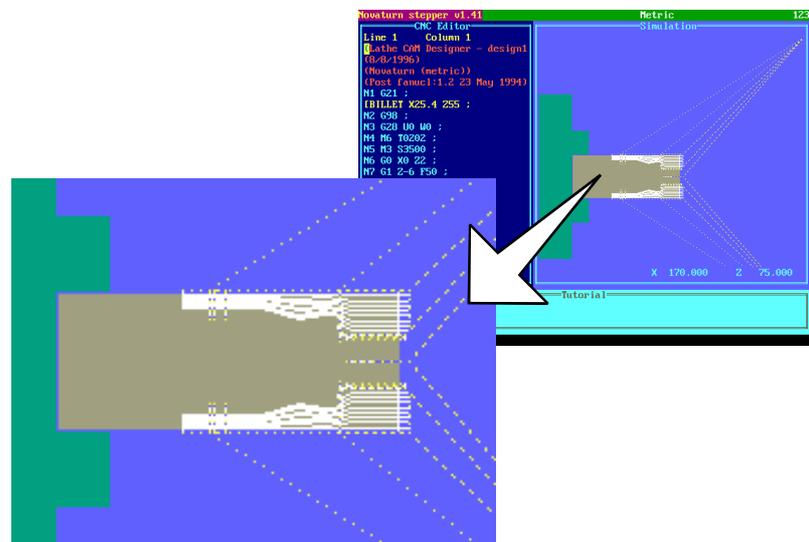
# SIMULATION MENU.

2) *RUN PROGRAM*. This instructs the computer to run through the program. To start this option, highlight *RUN PROGRAM* using the [CURSOR ARROWS] keys and press [EOB].



The *RUN PROGRAM* option is set to run through the entire program cycle, from start to finish. Whilst the program is running, the written text will scroll down the screen and the pictorial view will be simultaneously updated.

Note - the *RUN PROGRAM* option does not use your tool offsets - it tests the function of the program only!



! The [CYCLE STOP] key can also be used to pause a run. The remainder of the program will be run as a seperate cycle by pressing the [CYCLE START] key. Once this smaller cycle has finished press the [RESET] key to return to the start of the program.

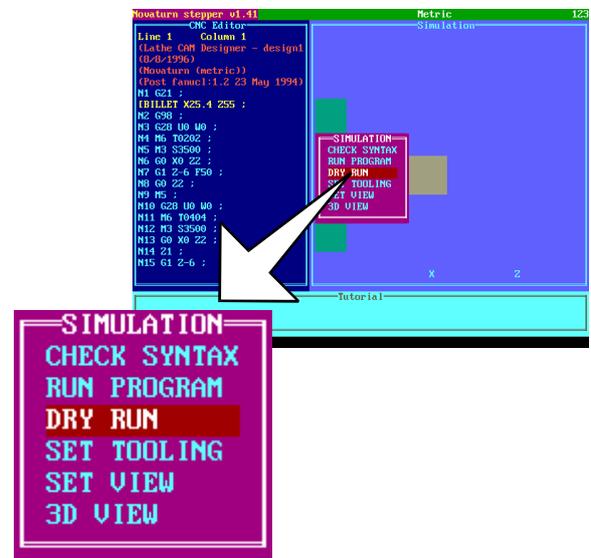
! Please note - The co-ordinate values depicted on screenshots are used for illustrative purposes only.

Pressing the [CYCLE STOP] key at any time will abort the run. To reset back to the start of the program, press the [RESET] key twice.



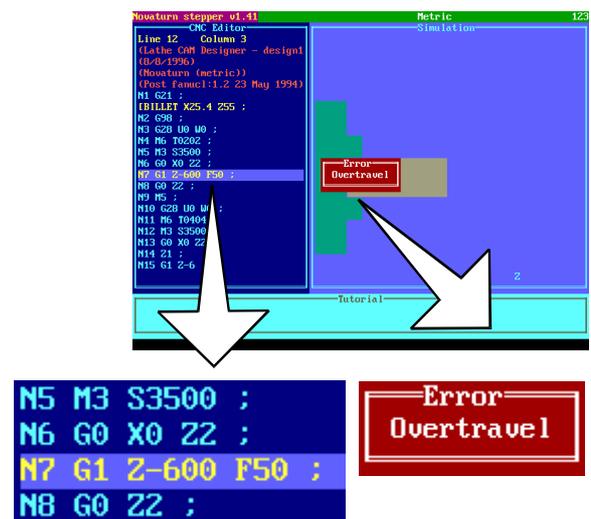
# SIMULATION MENU.

3) *DRY RUN*. This option performs a *CHECK SYNTAX* and also checks the validity of the machining requirements (for example, can the design be manufactured within the axes movement of the machine) by running the program in the computer memory. To run this option, highlight *DRY RUN* using the [CURSOR ARROWS] keys and press [EOB].



On short programs it may appear that nothing has happened, since the *DRY RUN* operation may take less than a second to complete.

*DRY RUN* will display any errors in your program, so if none are shown after pressing [EOB], your CNC program has run correctly. Error messages are displayed with the appropriate incorrect line in your program highlighted. Any error messages which are displayed can be cleared by pressing the [RESET] key.

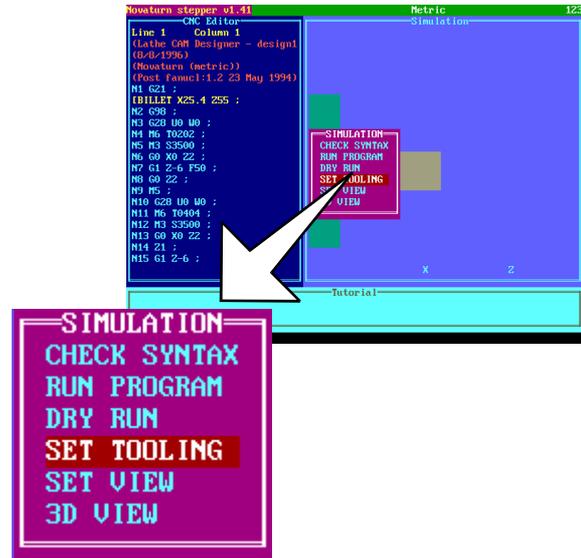


# SIMULATION MENU.

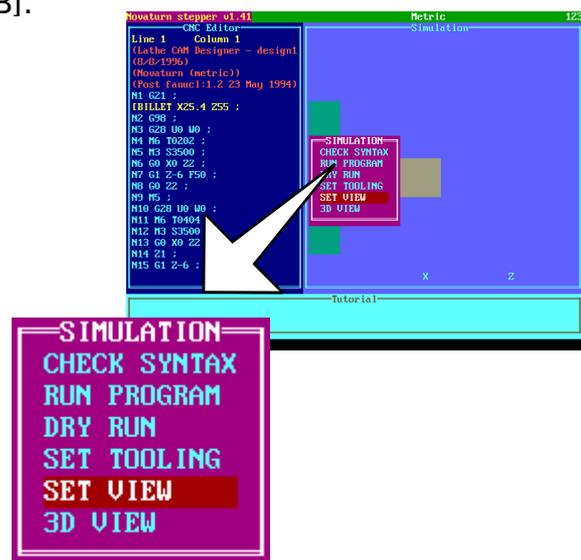
4) *SET TOOLING*. This allows the graphics for the *EDIT AND SIMULATE* and the *SIMULATE ONLY* options to be configured to match the type of tool used by both the machine toolpost and the program.

To select this option, highlight *SET TOOLING* using the [CURSOR ARROWS] keys and press [EOB].

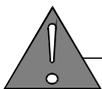
Please refer to the separate section 3.9 "Simulation Menu - Set Tooling", for details on how to set this option.



5) *SET VIEW* (not applicable to the *EDIT ONLY* option). This allows the detail level of the plan pictorial view to be selected. To select this option, highlight *SET VIEW* using the [CURSOR ARROWS] keys and press [EOB].



continued.....

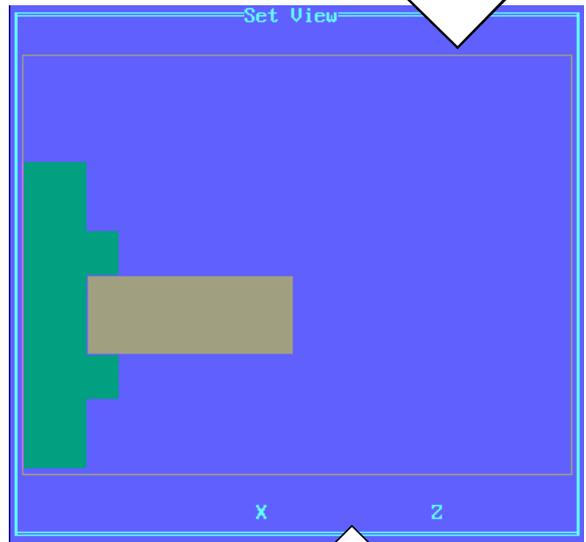
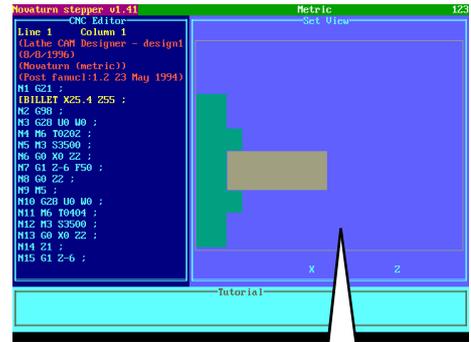


Please note -  
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

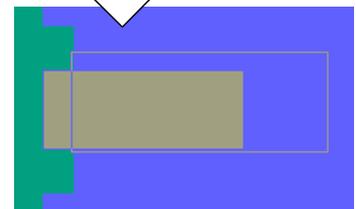
# SIMULATION MENU.

continued.....

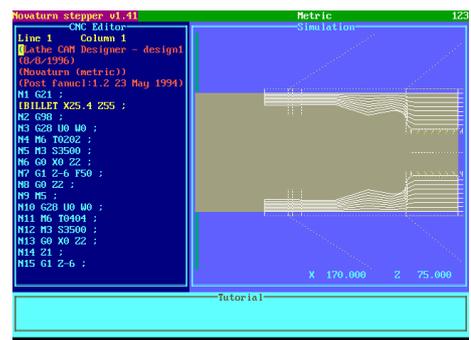
Press the [JOG] key to cycle through the different views. To select the view highlighted press the [EOB] key.



[JOG] key toggles around the different sized views.

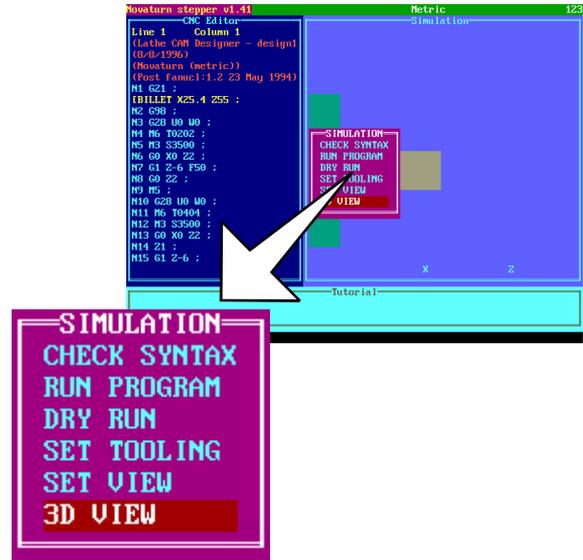


Now, when *RUN PROGRAM* is selected, the plan pictorial view shown will be the one previously chosen in the *SET VIEW* option.



# SIMULATION MENU.

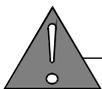
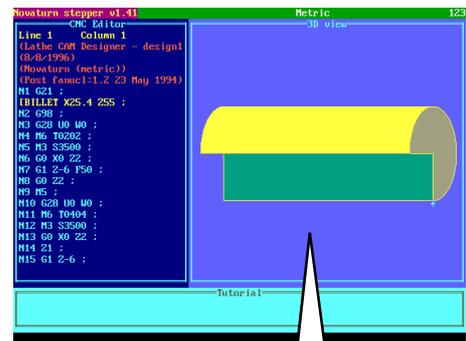
6) *3D VIEW*. This allows a three dimensional model of the work to be displayed in place of the side elevational view. To select this option, highlight *3D VIEW* using the [CURSOR ARROWS] and press [EOB].



The *3D VIEW* option will only display a static 3d view of the current stage of the work.

To return to the 2d side view, select the *RUN PROGRAM* option.

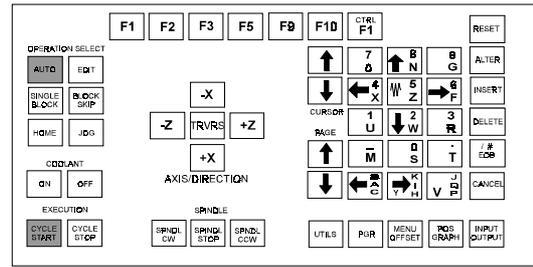
If a 3d view is permanently required during the *RUN PROGRAM* operation, the 3d view must be set from the *CHANGE SETTINGS (SIMULATION) MENU*.



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

# RUN PROGRAM.

Keys Helpbox.  
The following keys are used in this section:  
[AUTO]  
[CYCLE START]



Tutor keypad.

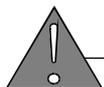
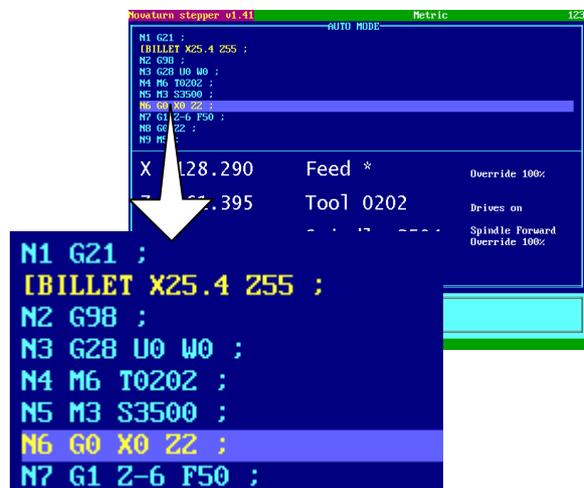
To run a program, the machine needs to be set in *AUTO MODE*. This is selected by pressing the [AUTO] key.

**AUTO MODE**



To start the program, press the [CYCLE START] key.

The program will now start to scroll, line by line, in the upper half of the display screen. At the same time, the co-ordinates, feedrate and spindle speed are shown, in the lower half of the display screen. These are updated, according to the point reached in the program. The particular program line reached is shown "highlighted" (yellow numbers and letters on a light blue line).



Please note -

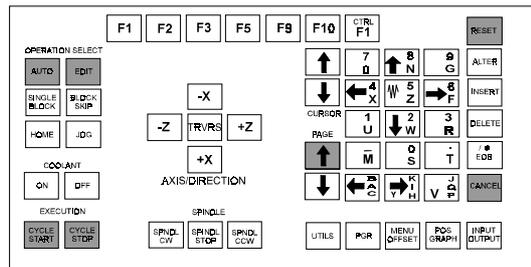
The co-ordinate values depicted on screenshots are used for illustrative purposes only.



# PAUSING PROGRAM.

Keys Helpbox.

The following keys are used in this section:  
[CYCLE STOP], [CYCLE START], [RESET]  
[CANCEL], [PAGE UP ARROW], [EDIT], [AUTO]



Tutor keypad.

To pause a program, press the [CYCLE STOP] key. This will set the current feedrate of the tool to zero, displaying the message 'Feed HOLD'.

Note, the spindle will still continue to rotate at a set speed.

To unpause the program, press the [CYCLE START] key.

**Feed Hold**

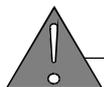
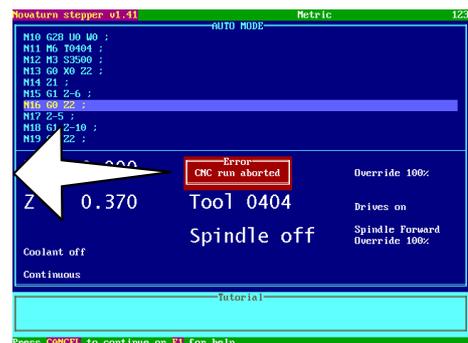


To stop a program, press the [CYCLE STOP] key to pause, then press the [RESET] or [CANCEL] key to abort the run.

The program will need to be manually reset back to its original starting point, if required. This can be achieved by two methods:

- 1) Use the [PAGE UP] arrow key until the original starting point of the program is reached.
- 2) Select *EDIT MODE* by pressing the [EDIT] key, then press the [RESET] key until the starting point of the program is reached. Press the [AUTO] key to return the machine back to *AUTO MODE*.

**Error  
CNC run aborted**



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



# OVERRIDING SPINDLE SPEED.

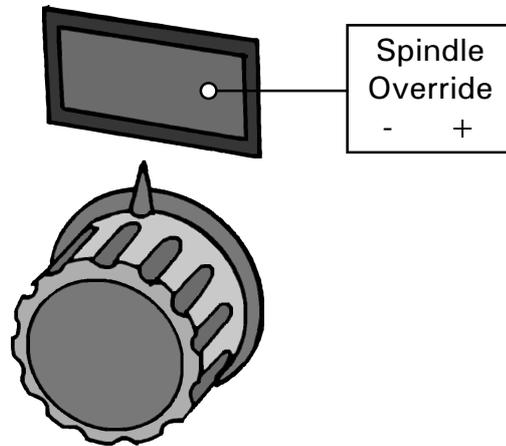
The spindle speed of the machine can be manually overridden during a machining operation using the potentiometer control dial fitted on the lower front panel of the machine (illustrated below).

On machines not fitted with this control dial, the tutor keypad is used to override the spindle speed (explained on the next page).

Note, the spindle speed override feature will only operate when speeds are actually being applied to the work (ie, during a machining operation).

# OVERRIDING SPINDLE SPEED - POTENTIOMETER CONTROLS.

On machines fitted with a potentiometer control dial, the spindle speed of the tool is manually adjusted by rotating the dial.



To increase the spindle speed, rotate the dial clockwise - as the dial is turned the 'faster' spindle speed will be updated and displayed on the control screen.

To decrease the spindle speed, rotate the dial anticlockwise - as the dial is turned the 'slower' spindle speed will be updated and displayed on the control screen.

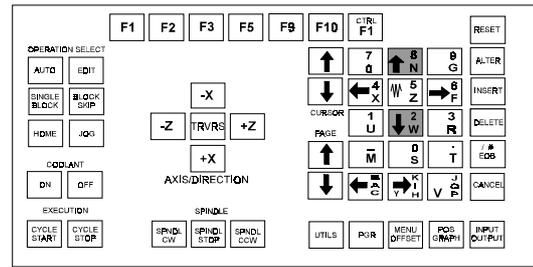
# OVERRIDING SPINDLE SPEED - TUTOR CONTROLS.

Keys Helpbox.

The following keys are used in this section:

[UP KEYPAD ARROW]

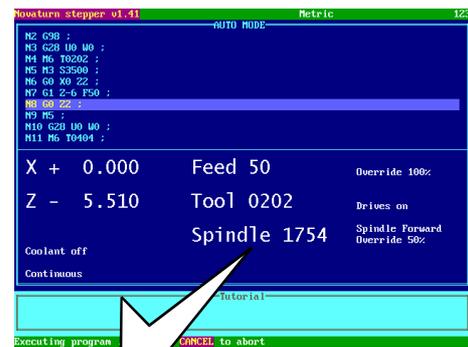
[DOWN KEYPAD ARROW]



Tutor keypad.

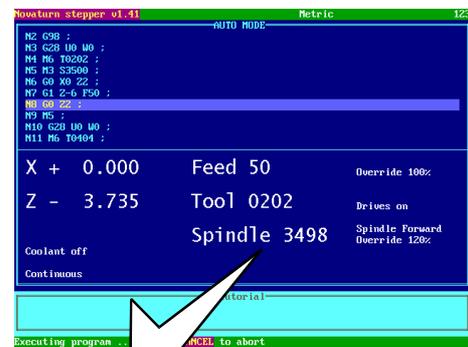
On machines not fitted with potentiometer controls, the spindle speed can be manually adjusted using the tutor keypad.

The spindle speed can be decreased down to a minimum value of 50%. To decrease the spindle speed, press the [DOWN KEYPAD ARROW] key until the desired value is displayed on screen.



Spindle 1754 Spindle Forward Override 50%

The spindle speed can be increased upto a maximum value of 120%. To increase the spindle speed, press the [UP KEYPAD ARROW] key until the desired value is displayed on screen.



Spindle 3498 Spindle Forward Override 120%

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

# OVERRIDING FEED RATE.

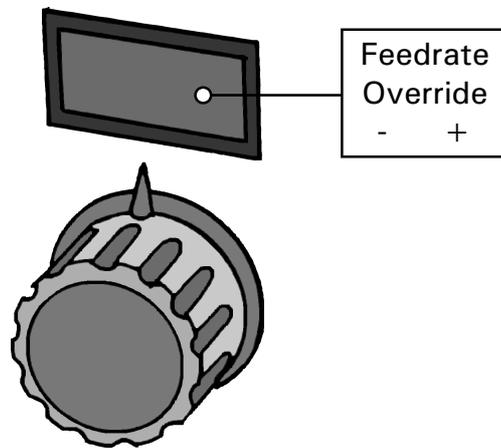
The feedrate of the machine can be manually overridden during a machining operation using the potentiometer control dial fitted on the lower front panel of the machine (illustrated below).

On machines not fitted with this control dial, the tutor keypad is used to override the feedrate (explained on the next page).

Note, the feedrate override feature will only operate when feedrates are actually being applied to the work (ie, during a machining operation).

# OVERRIDING FEED RATE - POTENTIOMETER CONTROLS.

On machines fitted with a potentiometer control dial, the feedrate of the tool is manually adjusted by rotating the dial.



To increase the feedrate, rotate the dial clockwise - as the dial is turned the 'faster' feedrate will be updated and displayed on the control screen.

To decrease the feedrate, rotate the dial anticlockwise - as the dial is turned the 'slower' feedrate will be updated and displayed on the control screen.

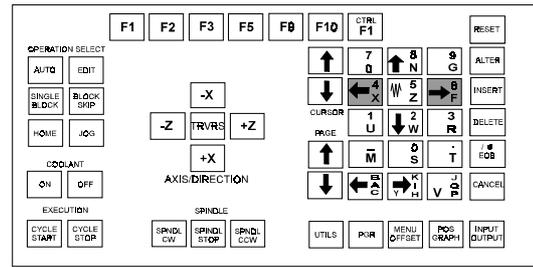
# OVERRIDING FEED RATE -TUTOR CONTROLS.

Keys Helpbox.

The following keys are used in this section:

[LEFT KEYPAD ARROW]

[RIGHT KEYPAD ARROW]



Tutor keypad.

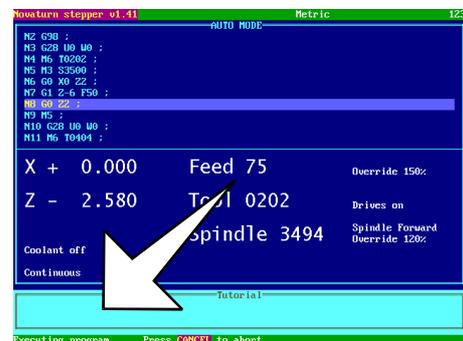
On machines not fitted with potentiometer controls, the feedrate can be manually adjusted using the tutor keypad.

The feedrate can be decreased down to a minimum value of 0%. To decrease the feedrate, press the [LEFT KEYPAD ARROW] key until the desired value is displayed on screen.



Feed 0 Override 0%

The feedrate can be increased upto a maximum value of 150%. To increase the feedrate, press the [RIGHT KEYPAD ARROW] key until the desired value is displayed on screen.



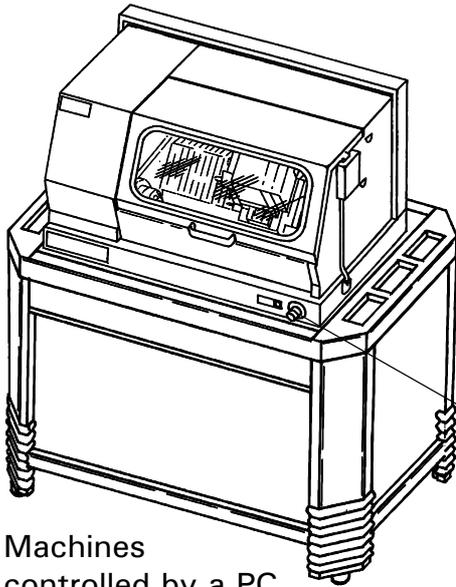
Feed 75 Override 150%

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

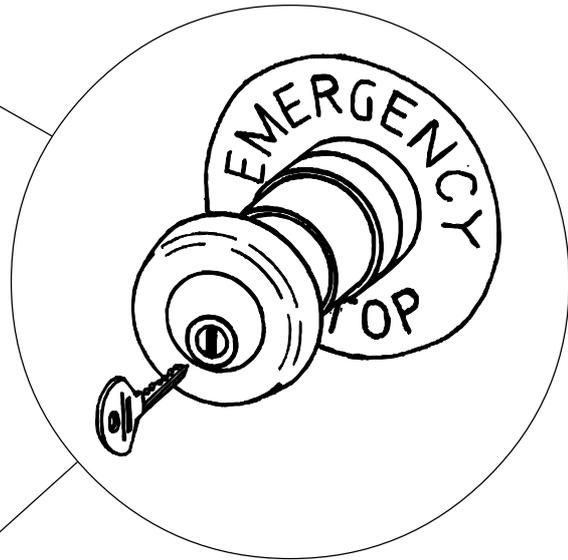
# EMERGENCY STOP.

The Emergency Stop button is located in two positions, according to the type of machine used:

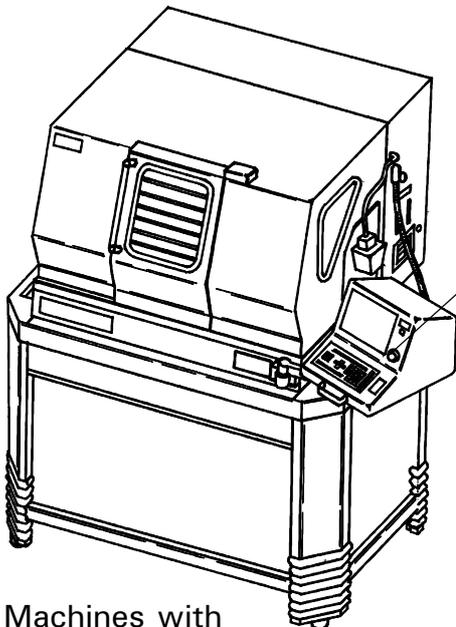
- 1) Machines controlled by a PC - Located on the front lower machine panel (at the far right-hand side).
- 2) Machines with an integrated controller - Located on the controller casing (at the right of the display screen).



Machines controlled by a PC.



LOCATION OF EMERGENCY  
STOP BUTTON.



Machines with  
an integrated controller.

# EMERGENCY STOP.

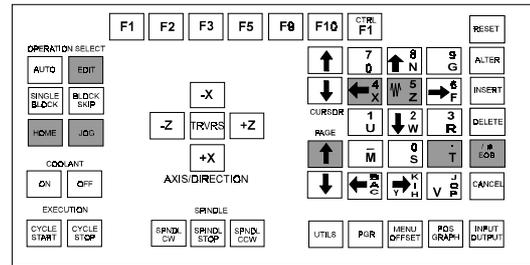
Keys Helpbox.

The following keys are used in this section:

[HOME], [X], [Z], [JOG], [T]

[NUMBERS] - not highlighted

[EOB], [EDIT], [PAGE UP ARROW]



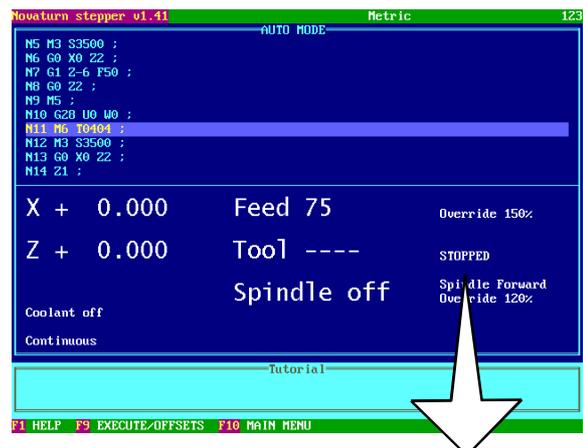
Tutor keypad.

When pressed, the Emergency Stop button will immediately halt the progress of a program along with any machine movement and cutting operations.

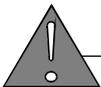
The Emergency Stop button will lock when fully pressed into its housing. Twist the button clockwise to release the lock. A key may be required to reset the button and allow access back to the machine.

To gain access back to the machine follow this procedure:

- 1) Home the machine by pressing [HOME], then [X], then [Z] to reset the datum point.
- 2) If a tool number is not present, reset the tool by pressing [JOG], then [T], then tool [NUMBER], then [EOB].
- 3) The program may also need resetting if it was interrupted whilst running. Press [EDIT] and then the [PAGE UP ARROW] until the first line of the program is reached.



**STOPPED**



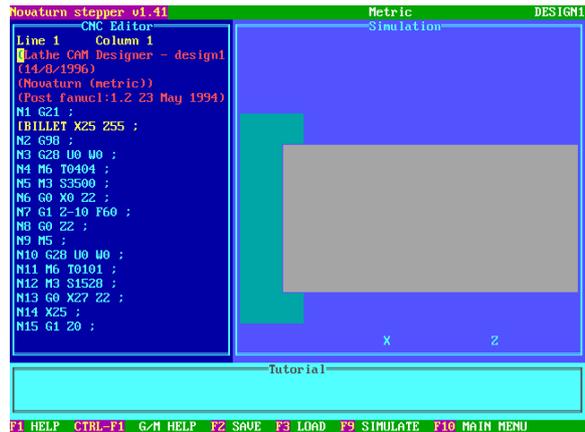
Please note -  
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

# EDITING A CNC FILE - SCREEN VIEW OPTIONS.

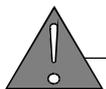
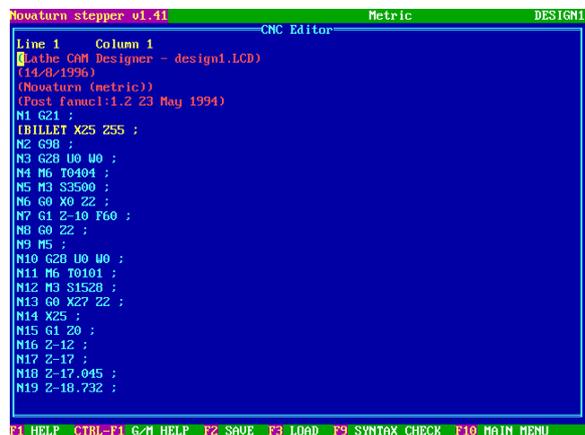
Once a CNC File has been loaded, its content can be altered using the Editor windows in the machine control software.

CNC Files can be edited in two main screen view options:

**EDIT AND SIMULATE.** The CNC File can be altered using the Editor side of the screen and then simulated using the graphical display(plan or 3d).



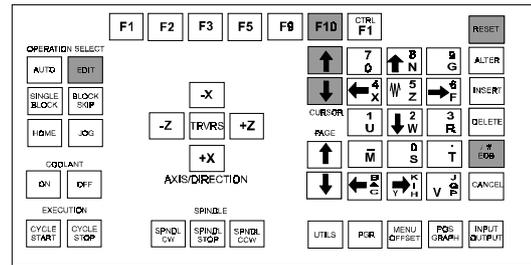
**EDIT ONLY.** The CNC File can be altered using a full screen Editor window.



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

# EDITING A CNC FILE - SCREEN VIEW OPTIONS.

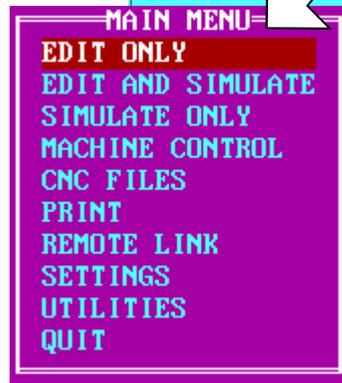
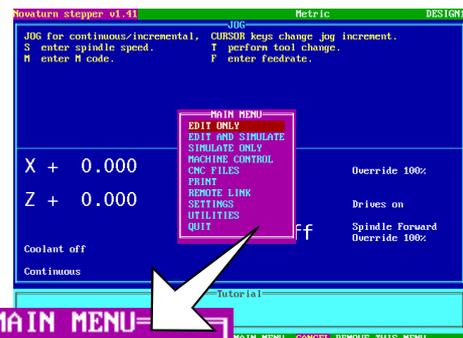
Keys Helpbox.  
The following keys are used in this section:  
[F10], [CURSOR ARROWS]  
[EOB], [RESET]  
[EDIT]



Tutor keypad.

Select the *MAIN MENU* by pressing the [F10] key. Highlight either the '*EDIT AND SIMULATE*' or '*EDIT ONLY*' options, using the [CURSOR ARROWS] keys and press the [EOB] key to select the highlighted option.

Alternatively, pressing the [EDIT] key will directly access the '*EDIT AND SIMULATE*' option.



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

# EDITING A CNC FILE - KEY FUNCTIONS.

Keys Helpbox.

The following keys are used in this section:

[CURSOR ARROWS]

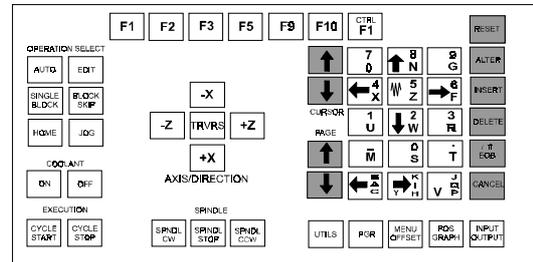
[NUMBERS/LETTERS] - not highlighted

[ALTER], [CANCEL]

[DELETE], [INSERT]

[PAGE UP ARROW], [PAGE DOWN ARROW]

[EOB]



Tutor keypad.

When editing a CNC File, the following keys are used:

[CURSOR ARROWS] keys.

These keys will cycle the flashing cursor up or down through each program word/character in the CNC File.

[DELETE] key.

This key will remove the program word/character highlighted by the cursor in the CNC File.

[INSERT] key.

This key will place a 'new' program word/character directly after the program word/character highlighted by the cursor in the CNC File.

[PAGE UP ARROW] and [PAGE DOWN ARROW] keys.

These keys will move the CNC File text up or down by one full screen page.

[EOB] key.

This key ends the program line, by placing an end of block character (; or /).

[CANCEL] key.

This key will clear any characters from the edit line.

[ALTER] key.

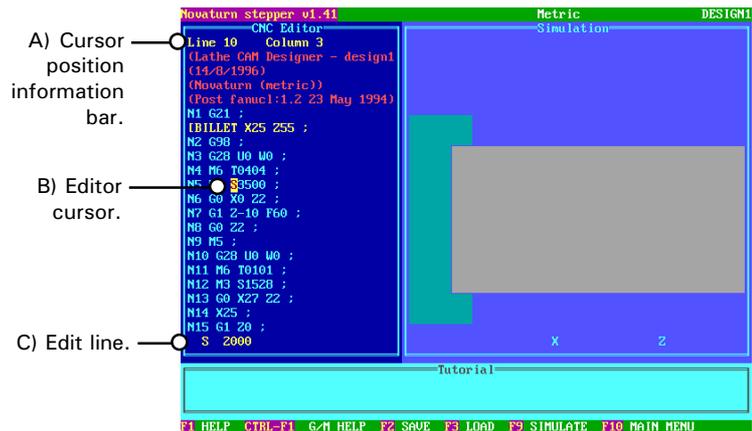
Pressing this key will replace the program word/character highlighted by the cursor in the CNC File with any 'new' text entered on the edit line.

# EDITING A CNC FILE - EDITOR WINDOW LAYOUT.

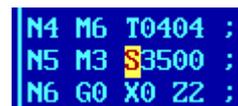
All example screenshots are shown using the 'EDIT AND SIMULATE' option.

The CNC File Editor window, in the 'EDIT AND SIMULATE' option, is displayed by default on the left side of the screen. To alter the screen display properties, see section 10.3 "Change Settings - Editor" and section 10.16 "Change Settings - Miscellaneous".

The general layout of the CNC File Editor window is shown below:



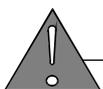
A) The Cursor position information bar displays the exact location of the cursor by program line and column.



B) The yellow Editor cursor highlights the program word or character that can be edited. In this example it highlights the program word 'S3500', referring to the spindle speed.



C) The Edit line (at the bottom of the Editor window) is the area of the display where 'new' program words/lines can be written, then inserted into the CNC File. In this example, a 'new' spindle speed of 'S2000' has been entered on the edit line, using the tutor keypad.

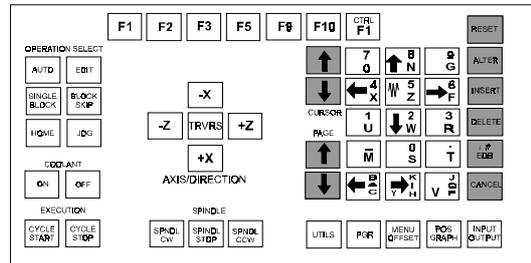


Please note -  
The co-ordinate values depicted  
on screenshots are used for  
illustrative purposes only.

# EDITING A CNC FILE.

## Keys Helpbox.

The following keys are used in this section:  
[CURSOR ARROWS]  
[NUMBERS/LETTERS] - not highlighted  
[ALTER], [CANCEL]  
[DELETE], [INSERT]  
[PAGE UP ARROW], [PAGE DOWN ARROW]  
[EOB]



Tutor keypad.

Using the '*EDIT AND SIMULATE*' screen shown on the previous page as an example, the CNC File could be edited in a variety of different ways.

## SELECTING PROGRAM TEXT.

```
N4 M6 T0404 ;  
N5 M3 S3500 ;  
N6 G0 X0 Z2 ;
```

Using the main screen displayed on the previous page as an example, the program word '*S3500*' is highlighted in the CNC File by using the [CURSOR ARROWS] keys. As the [CURSOR ARROWS] keys are pressed, the corresponding program word is highlighted. Note that the Editor window can only display a certain number of lines on the screen. To move through a larger CNC File quickly use the [PAGE UP ARROW] and [PAGE DOWN ARROW] keys.

```
S 2000
```

The '*new*' program word '*S2000*' is entered on the edit line using the tutor keypad.

Any '*new*' program word, or set of words, is always entered from the edit line.

The [CANCEL] key is used to clear unwanted text from the edit line.

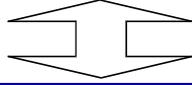
The [EOB] key will enter any text entered on the edit line as a '*new*' program line in the CNC File.

# EDITING A CNC FILE.

## INSERTING AND DELETING PROGRAM TEXT.

```
N4 M6 T0404 ;  
N5 M3 S3500 ;  
N6 G0 X0 Z2 ;
```

Edit line text.  
S 2000



```
N4 M6 T0404 ;  
N5 M3 S3500 S2000 ;  
N6 G0 X0 Z2 ;
```

If the [INSERT] key is pressed, any 'new' program word entered on the edit line is placed at the position of the cursor. The 'new' program line reads 'N5 M3 S3500 S2000 ;' as shown above.

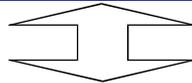
Any 'new' text is placed directly after the last program word/character highlighted by the cursor.

To remove the highlighted program word 'S2000', the [DELETE] key would be pressed whilst it is still highlighted.

## ALTERING A PROGRAM LINE.

```
N4 M6 T0404 ;  
N5 M3 S3500 ;  
N6 G0 X0 Z2 ;
```

Edit line text.  
S 2000



```
N4 M6 T0404 ;  
N5 M3 S2000 ;  
N6 G0 X0 Z2 ;
```

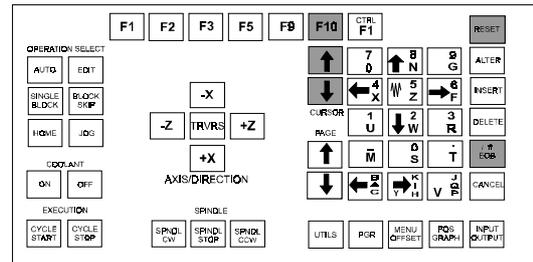
If the [ALTER] key is pressed when 'S2000' is entered on the edit line and 'S3500' is highlighted by the cursor, 'S3500' is replaced by 'S2000'.

If [ALTER] is pressed when the edit line contains a number of program words, they will all be placed over the program word/character last highlighted by the cursor.

# MAIN MENU

## - MACHINE CONTROL.

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

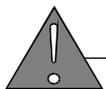
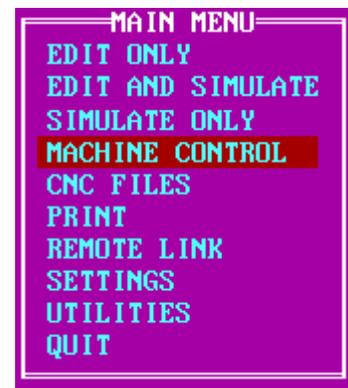


Tutor keypad.

The *MACHINE CONTROL* option will change an *EDIT MODE* screen display to the last *MACHINE CONTROL MODE* used (this will be either *JOG MODE*, *AUTO MODE* or *HOME MODE* depending on the last one used). The table and head of the machine can only be moved when the software is operating in one of the three *MACHINE CONTROL MODES*. If the screen is already set in one of these three *MACHINE CONTROL MODES*, nothing will happen to the main screen display.

Select the *MAIN MENU* by pressing the [F10] key. Highlight '*MACHINE CONTROL*' using the [CURSOR ARROWS] keys and press the [EOB] key.

Press the [RESET] key to remove any unwanted menus from the screen.



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

# MAIN MENU

## - PRINT.

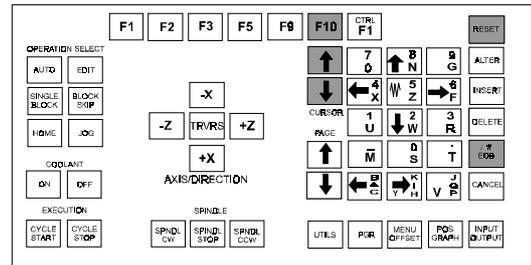
Keys Helpbox.

The following keys are used in this section:

[F10]

[CURSOR ARROWS]

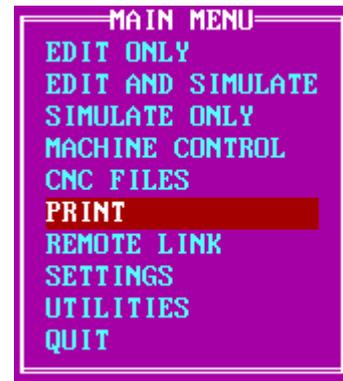
[EOB], [RESET]



Tutor keypad.

The *PRINT* option allows you to generate a paper copy of the currently loaded CNC File from a connected printer.

Select the *MAIN MENU* by pressing the [F10] key. Highlight '*PRINT*' using the [CURSOR ARROWS] keys and press the [EOB] key.



The *PRINTING SUB-MENU* contains two options :

- 1) *LISTING WITH ERRORS*. This option will print the CNC File as displayed in the Editor window with any errors highlighted.
- 2) *NO ERRORS ON LISTING*. This option will print the CNC File as displayed in the Editor window without highlighting any errors.

continued....



Please note -

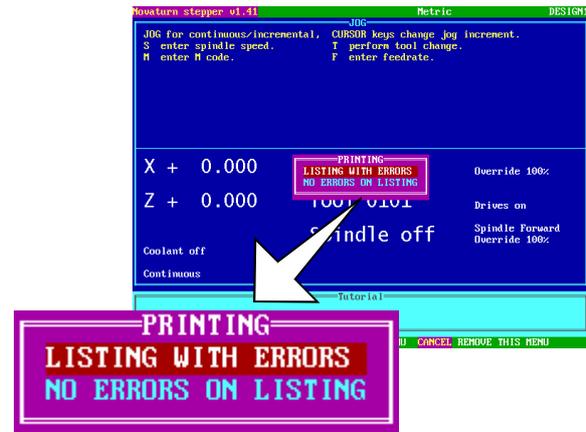
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

# MAIN MENU

## - PRINT.

continued....

Select the required option using the [CURSOR ARROWS] keys, then press the [EOB] key.



Press the [RESET] key to remove any unwanted menus from the screen.

## PRINTING ERRORS.

If the printer does not respond, check the following :

- 1) Is the cabling between the computer/control box and printer secure?
- 2) Is the printer set for Parallel or Serial communications?
- 3) Have the correct parameters been set in the 'Change Settings - Print Device' (see section 10.13)?
- 4) Is the printer switched 'on' and is there enough paper available for the printout?

## PAGE LAYOUT ERRORS.

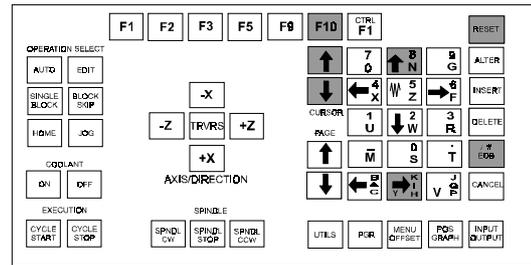
If the layout on the printout is incorrect, the page widths and linefeeds can be changed in the 'Change Settings - Print Page Layout' (see section 10.14).

# MAIN MENU

## - REMOTE LINK.

Keys Helpbox.

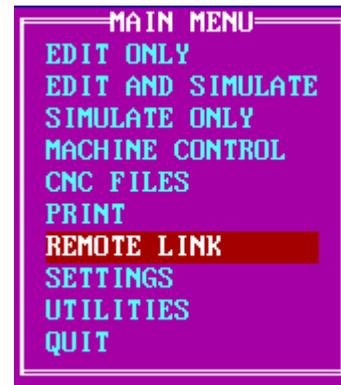
The following keys are used in this section:  
 [F10], [CURSOR ARROWS]  
 [EOB], [RESET]  
 [Y], [N]



Tutor keypad.

The *REMOTE DEVICE* option allows a CNC File to be sent or received from a remote device, such as another computer, paper tape puch reader, data carrier etc....

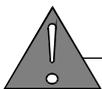
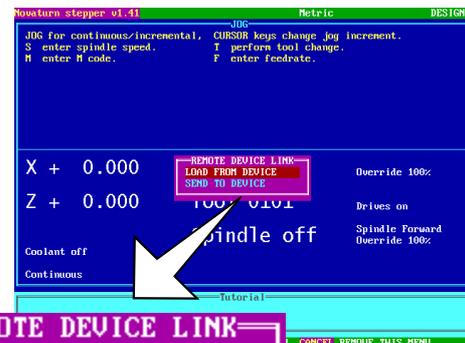
Select the *MAIN MENU* by pressing the [F10] key. Highlight '*REMOTE LINK*' using the [CURSOR ARROWS] keys and press the [EOB] key.



The *REMOTE LINK SUB-MENU* contains two options :

- 1) *LOAD FROM DEVICE*. This option will load the CNC File from the remote device.
- 2) *SEND TO DEVICE*. This option will send the CNC File to the remote device.

Select the required option using the [CURSOR ARROWS] keys, then press the [EOB] key.



Please note -

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

# MAIN MENU

## - REMOTE LINK.

### LOAD FROM DEVICE.

If there is a CNC File currently loaded in the control software, you will be asked whether to merge the CNC File when loading from the remote device.

Press the [Y] to merge both CNC Files into one and the [N] key to clear the current CNC File from the control software.

### SEND TO DEVICE.

You will be prompted with a '*READY TO SEND?*' message. Press the [Y] key to send the CNC File and the [N] key to abort the operation.

During CNC File transfer, a '*TRANSMITTING TO DEVICE*' message window will be displayed, showing the number of bytes and lines sent. A '*TRANSMISSION COMPLETED*' message will be displayed to confirm that the whole CNC File has been sent to the remote device.

Press the [RESET] key to remove any unwanted messages and menus from the screen.

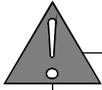
### COMMUNICATION ERRORS.

If an error is encountered, check the following :

- 1) Is the cabling between the computer/control box and remote device secure?
- 2) Is the cable connected to the correct ports with the correct pin connections?
- 3) Have the correct parameters been set in the "Settings Menu - Remote Link" (see section 10.15)?
- 4) Is the remote device switched 'on' and ready to send or receive data?

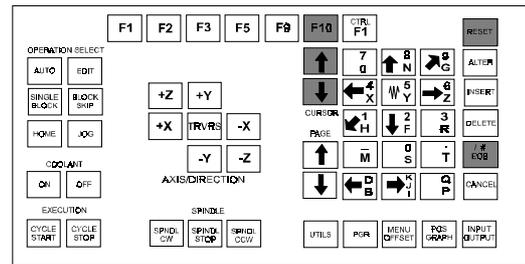
# MAIN MENU

## - UTILITIES.



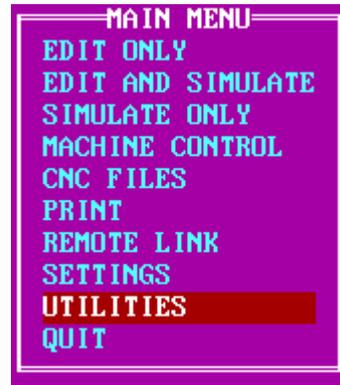
Please note -  
The DOS Access and LatheCAM Designer features are only available on machines controlled by an external PC. If an error message is displayed when trying to access DOS, press the [RESET] key.

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

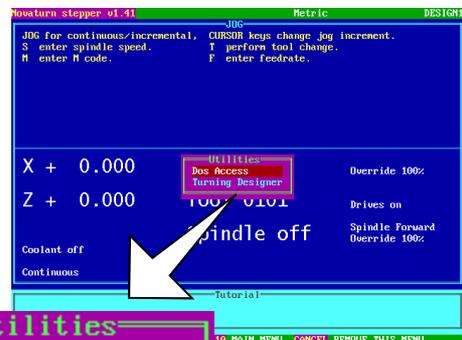


Tutor keypad.

Select the *MAIN MENU* by pressing the [F10] key. Highlight '*UTILITIES*' using the [CURSOR ARROWS] keys and press the [EOB] key.

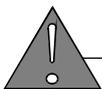
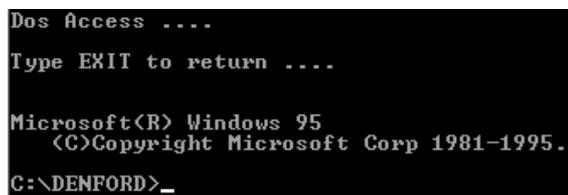


Press the [EOB] key when '*DOS ACCESS*' is highlighted to temporarily exit the control software.



The *UTILITIES MENU* will give access to other programs and DOS mode. To return to the control software, type '*EXIT*' at the DOS prompt or close the other program being used.

Press the [RESET] key to remove any unwanted menus from the screen.



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

# CHANGE SETTINGS.

The *CHANGE SETTINGS MENU* allows the software to be customised to suit the requirements of the end user. When all the options have been fully configured, the settings should be saved to disk. Each time the machine is started it will load these customised settings.

# CHANGE SETTINGS - MENU.

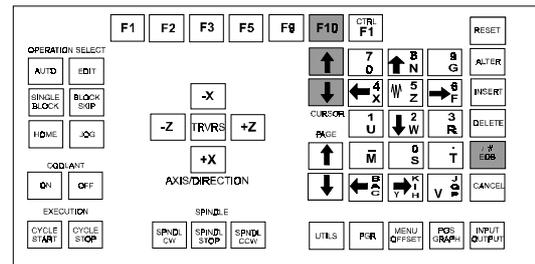
Keys Helpbox.

The following keys are used in this section:

[F10]

[CURSOR ARROWS]

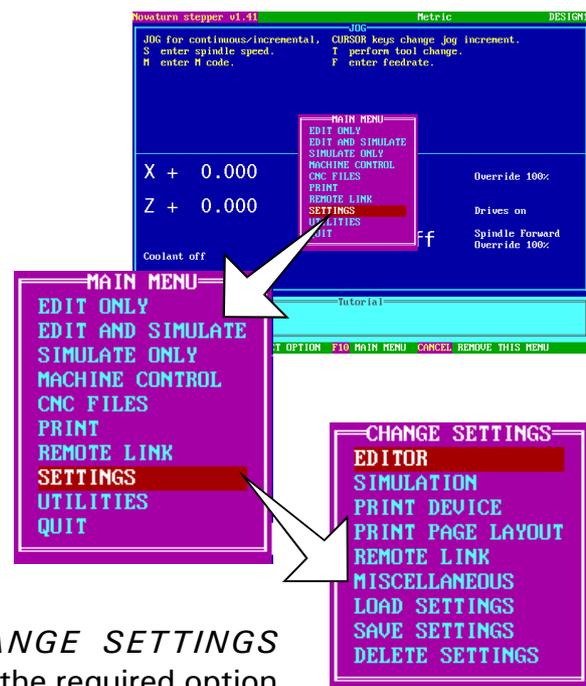
[EOB]



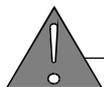
Tutor keypad.

The *CHANGE SETTINGS MENU* is used to switch on and off the various options available within the machine controlling software.

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



At the *CHANGE SETTINGS MENU*, select the required option and press the [EOB] key.



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

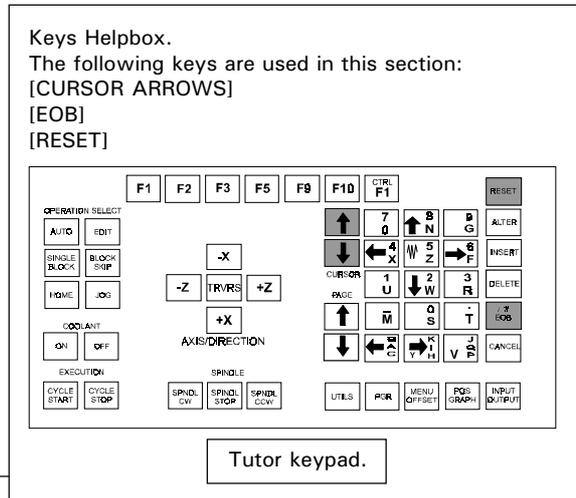
# CHANGE SETTINGS - MENU.



The *CHANGE SETTINGS MENU* contains nine options:

- 1) *EDITOR*. This option allows the CNC File Editor window to be customised.
- 2) *SIMULATION*. This option allows the graphics and views in the Simulation window to be customised.
- 3) *PRINT DEVICE*. This option allows any printers attached to the machine controller to be configured.
- 4) *PRINT PAGE LAYOUT*. This option allows the layouts of any printouts to be customised.
- 5) *REMOTE LINK*. This option allows the communication protocols between the machine controller and an external device to be configured.
- 6) *MISCELLANEOUS*. This option allows the units of measurement, user's name and screen text size to be customised.
- 7) *LOAD SETTINGS*. This option allows a collection of settings to be loaded from a disk.
- 8) *SAVE SETTINGS*. This option allows the current collection of settings to be saved to a disk.
- 9) *DELETE SETTINGS*. This option allows a collection of settings to be removed from a disk.

# CHANGE SETTINGS - EDITOR.



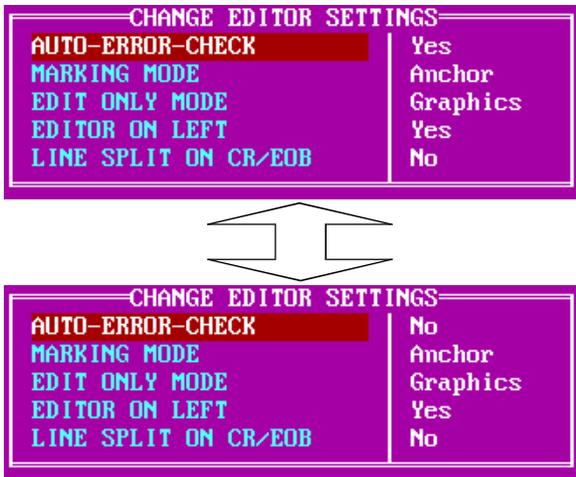
The *CHANGE SETTINGS (EDITOR) MENU* contains the following options:

1) *AUTO-ERROR CHECK*. This option, when set to 'YES', will check the validity of CNC program lines as they are manually entered. When the [EOB] key is pressed (to signify the end of a program line) an error description box will be displayed if an error is encountered. Press the [RESET] key to clear the error description box. The cursor will highlight where on the program line the error occurred.

To select this option, highlight '*AUTO-ERROR CHECK*' using the [CURSOR ARROWS] keys and press the [EOB] key.

The screen will indicate the current setting of the option. Continual pressing of the [EOB] key will toggle the option between 'YES' and 'NO'.

When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.



# CHANGE SETTINGS - EDITOR.

2) *MARKING MODE*. This option, will select between the '*DRAG*' or '*ANCHOR*' methods of cut and pasting parts CNC Files. This feature is **ONLY** available on qwerty (offline) versions of the control software.

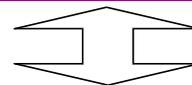
The *MARKING MODE* option has no effect on the machine controlling software.

3) *EDIT ONLY MODE*. This option toggles the preference of the *EDIT ONLY* display between '*TEXT*' or '*GRAPHICS*'. In *GRAPHICS MODE* preference is given to the appearance of the alphabet characters used by the control software (ie, the software drivers). In *TEXT MODE* preference is given to the computer video card, giving a slight speed increase when scrolling the text and direct access to specific character maps controlled by the video card (ie, the computer drivers).

To select this option, highlight '*EDIT ONLY MODE*' using the [CURSOR ARROWS] keys and press the [EOB] key.

The screen will indicate the current setting of the option. Continual pressing of the [EOB] key will toggle the option between '*TEXT*' and '*GRAPHICS*'.

When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.

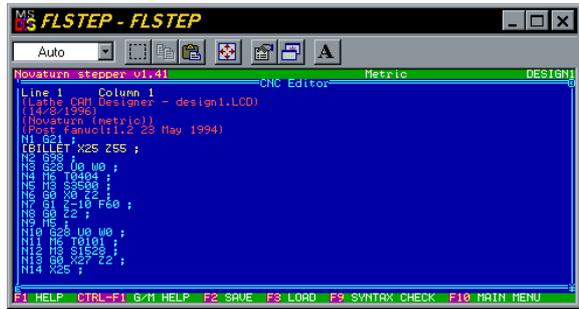


continued.....

# CHANGE SETTINGS - EDITOR.

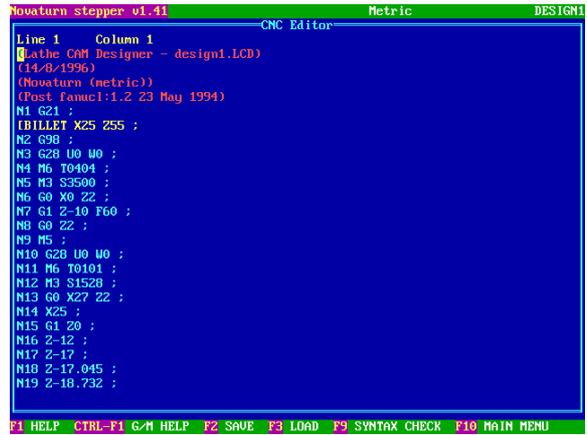
3) *EDIT ONLY MODE*. continued.....

'*EDIT ONLY MODE*' set to '*TEXT*'.



The screenshot shows a window titled 'FLSTEP - FLSTEP' with a menu bar containing 'Auto' and several icons. The main area is a text editor with a blue background and white text. The text is a CNC program for a lathe, starting with 'Novaturn stepper v1.41' and 'CNC Editor'. The program includes comments like '(Lathe CAM Designer - design1.LCD)' and '(14/8/1996)'. The code starts with 'N1 G21 ;', 'EBILLET X25 Z55 ;', and continues with various G and M codes. The status bar at the bottom shows 'F1 HELP CTRL-F1 G/M HELP F2 SAVE F3 LOAD F9 SYNTAX CHECK F10 MAIN MENU'.

'*EDIT ONLY MODE*' set to '*GRAPHICS*'.



The screenshot shows the same window as above, but the text is now in a larger font and includes more lines of code. The status bar at the bottom is the same as in the previous screenshot.

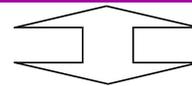
# CHANGE SETTINGS - EDITOR.

4) *EDITOR ON LEFT*. This option, when set to 'YES', will display the CNC editor window on the left side of the screen (when the control is set in 'EDIT AND SIMULATE' Mode). Selecting 'NO' will display the CNC editor window on the right side of the screen (when the control is set in 'EDIT AND SIMULATE' Mode).

To select this option, highlight 'EDITOR ON LEFT' using the [CURSOR ARROWS] keys and press the [EOB] key.

The screen will indicate the current setting of the option. Continual pressing of the [EOB] key will toggle the option between 'YES' and 'NO'.

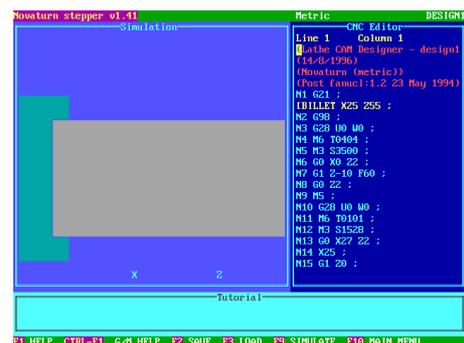
When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.



'Editor On Left' set to 'Yes'.



'Editor On Left' set to 'No'.



# CHANGE SETTINGS - EDITOR.

## 4) *LINE SPLIT ON CR/EOB.*

When this option is set to 'YES', the cursor can be used to split a program line apart. For example, when the cursor is positioned in the middle of a long program line, pressing the [EOB] key will create a new program line. Any program words which originally appeared after the cursor are now transferred to this new program line.

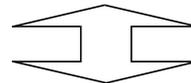
When this option is set to 'NO', pressing the [EOB] key will insert a new (blank) program line, after the program line on which the cursor is currently positioned. The original program line will be unaffected by this procedure.

To select this option, highlight '*LINE SPLIT ON CR/EOB*' using the [CURSOR ARROWS] keys and press the [EOB] key.

The screen will indicate the current setting of the option. Continual pressing of the [EOB] key will toggle the option between 'YES' and 'NO'.

When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.

CHANGE EDITOR SETTINGS	
AUTO-ERROR-CHECK	No
MARKING MODE	Anchor
EDIT ONLY MODE	Graphics
EDITOR ON LEFT	Yes
LINE SPLIT ON CR/EOB	Yes



CHANGE EDITOR SETTINGS	
AUTO-ERROR-CHECK	No
MARKING MODE	Anchor
EDIT ONLY MODE	Graphics
EDITOR ON LEFT	Yes
LINE SPLIT ON CR/EOB	No

# CHANGE SETTINGS - SIMULATION.

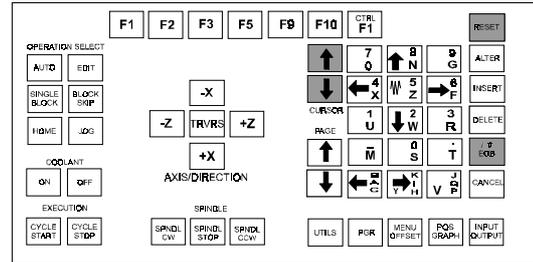
Keys Helpbox.

The following keys are used in this section:

[CURSOR ARROWS]

[EOB]

[RESET]



Tutor keypad.

The *CHANGE SETTINGS (SIMULATION) MENU* contains the following options:

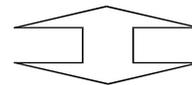
1) *SIMULATION WINDOW*. This option will switch the size of the graphical simulation window between wide and normal.

Normal splits the display screen to 50% editor window and 50% graphical simulation window. Wide increases the graphical simulation window, at the expense of the editor window size.

To select this option, highlight '*SIMULATION WINDOW*' using the [CURSOR ARROWS] keys and press the [EOB] key.

The screen will indicate the current setting of the option. Continual pressing of the [EOB] key will toggle the option between '*NORMAL*' and '*WIDE*'.

When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.



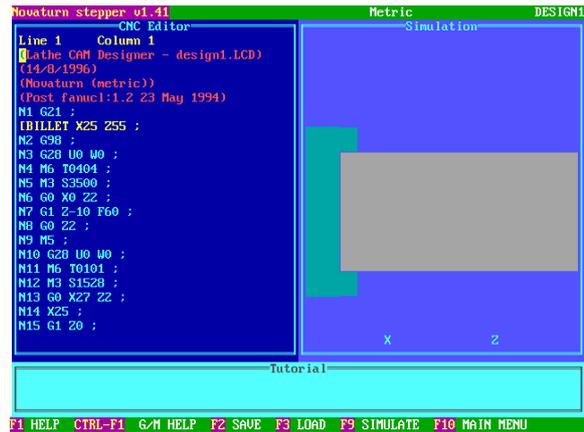
continued....

# CHANGE SETTINGS

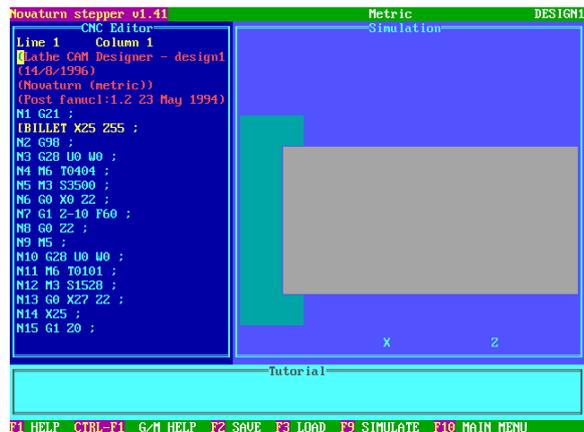
## - SIMULATION.

1) *SIMULATION WINDOW*. continued....

*SIMULATION WINDOW* set to 'NORMAL'.



*SIMULATION WINDOW* set to 'WIDE'.

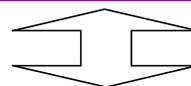


2) *SHOW 3D*. This option, when set to 'YES', will display the work as a three dimensional object.

To select this option, highlight '*SHOW 3D*' using the [CURSOR ARROWS] keys and press the [EOB] key.

The screen will indicate the current setting of the option. Continual pressing of the [EOB] key will toggle the option between 'ON' and 'OFF'.

When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.



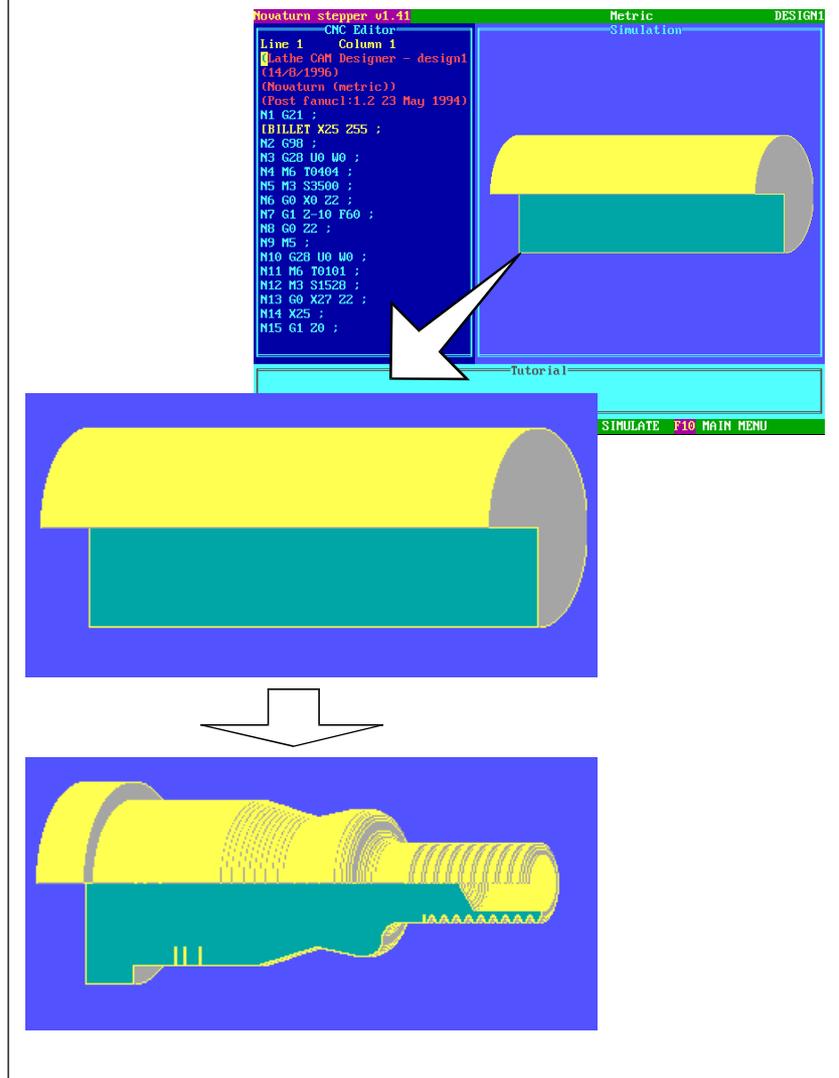
continued.....

# CHANGE SETTINGS

## - SIMULATION.

2) *SHOW 3D.* continued.....

Now, when '*RUN PROGRAM*' is selected, the work will be displayed in 3d, throughout the running of the program cycle.



# CHANGE SETTINGS

## - SIMULATION.

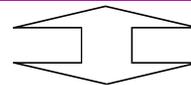
3) *DISPLAY FAST TRAVERSES*. This option is active when set to 'YES' (not available for 3d views).

*DISPLAY FAST TRAVERSES*, when set to 'YES', will show all fast traverse movements as yellow dotted lines. Lines cut into the billet are shown in white.

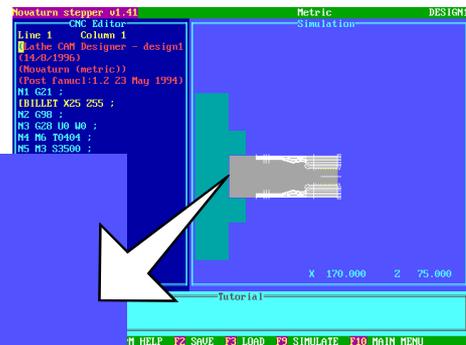
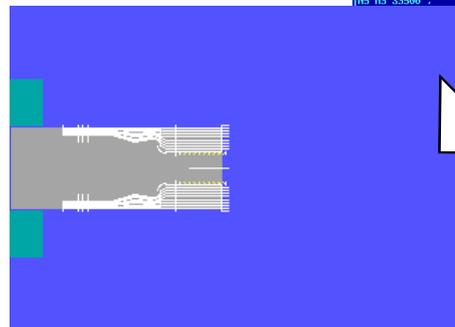
To select this option, highlight '*DISPLAY FAST TRAVERSES*' using the [CURSOR ARROWS] keys and press the [EOB] key.

The screen will indicate the current setting of the option. Continual pressing of the [EOB] key will toggle the option between 'YES' and 'NO'.

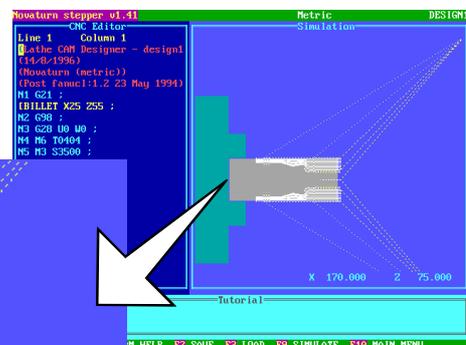
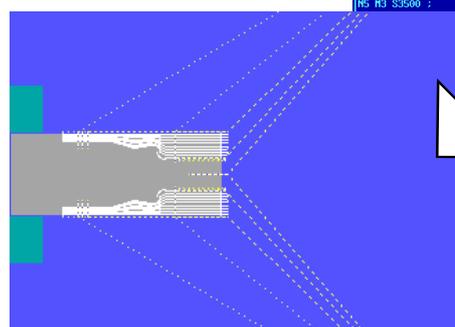
When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.



*DISPLAY FAST TRAVERSES* set to 'NO'.



*DISPLAY FAST TRAVERSES* set to 'YES'.



# CHANGE SETTINGS

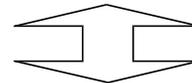
## - SIMULATION.

4) *TOOL MOTION*. This option, when set to 'ON' will display the movement and path of the tool as it cuts the material. In order for the correct tool profiles to be shown, the tool graphics must also be set - see section 3.9 "Simulation Menu - Set Tooling".

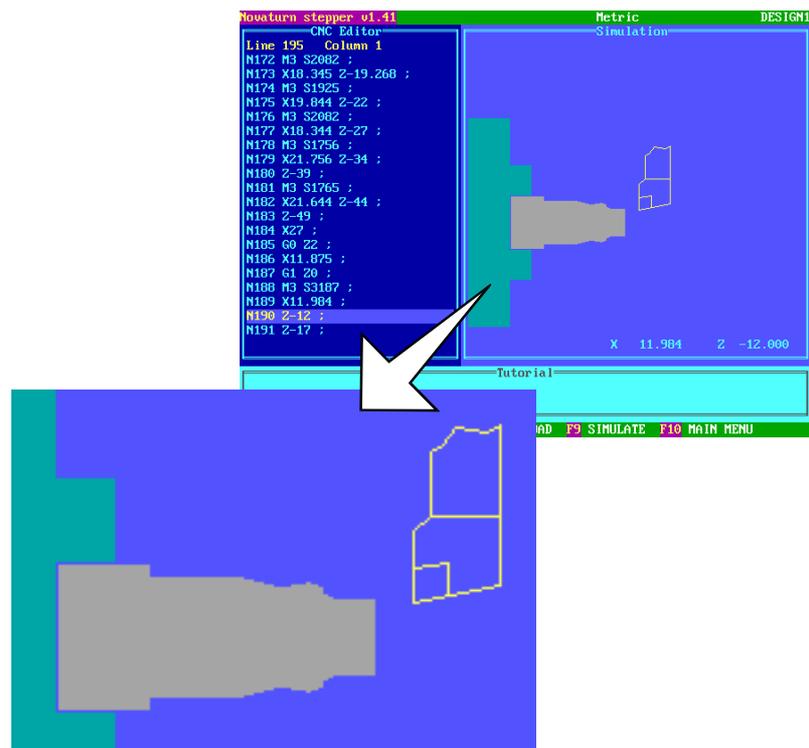
To select this option, highlight '*TOOL MOTION*' using the [CURSOR ARROWS] keys and press the [EOB] key.

The screen will indicate the current setting of the option. Continual pressing of the [EOB] key will toggle the option between 'ON' and 'OFF'.

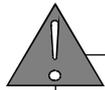
When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.



When '*RUN PROGRAM*' is selected, the tool motion will be shown, throughout the running of the program cycle. The program cycle may run slower with this option switched 'on'.

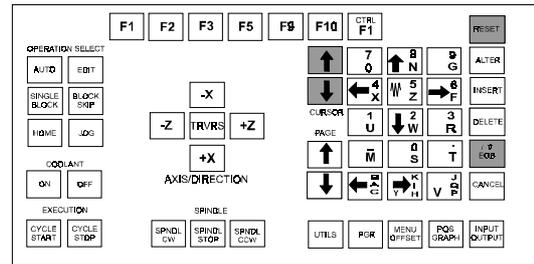


# CHANGE SETTINGS - PRINT DEVICE.



Please note -  
Any changes made to these settings should be carried out by either your IT Manager or computer technician.

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

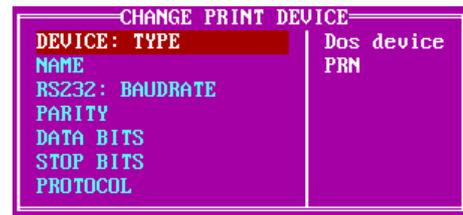


Tutor keypad.

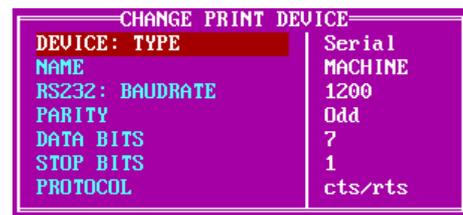
The *PRINT DEVICE* option is used to configure the settings for any printers attached to the machine controller.

There are three different printer options, selected by pressing the [EOB] key when the cursor is highlighting the '*DEVICE : TYPE*' :

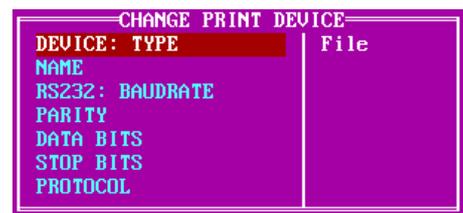
- 1) *DOS DEVICE* - A DOS Device is normally the parallel port which can be set to LPT1. LPT2 or PRN. Select this option if your printer has a parallel port.



- 2) *SERIAL DEVICE* - Select the individual settings using the [CURSOR ARROWS] keys and press the [EOB] key to toggle between the different values.

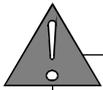


- 3) *FILE* - This option is used to save the CNC File on disk, for printing off at a later date. The current filename will be saved with an extension ".LST".



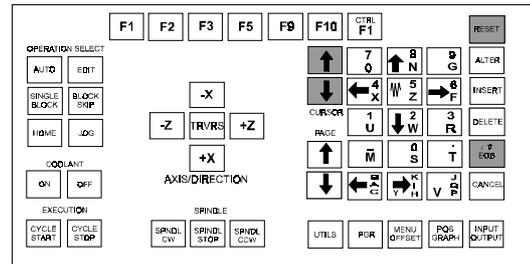
When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.

# CHANGE SETTINGS - PRINT PAGE LAYOUT.



Please note - Any changes made to these settings should be carried out by either your IT Manager or computer technician.

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



Tutor keypad.

The *PRINT PAGE LAYOUT* option is used to customise any printouts taken from the control software.

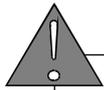
If several printers are available, save each individual setting with a different filename (see page 10.19).

Select the individual settings on the *CHANGE PRINT PAGE LAYOUT MENU* using the [CURSOR ARROWS]. Press the [EOB] key to move the cursor across, type in the required values, then press the [EOB] key to confirm the new value.

When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.

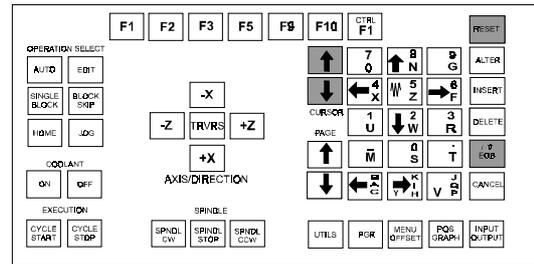
CHANGE PRINT PAGE LAYOUT	
PAGE WIDTH	80
PAGE DEPTH	60
LEFT MARGIN	0
TOP MARGIN	0
BOTTOM MARGIN	0
COLUMN WIDTH	80
CARRIAGE RETURN NULLS	0
LINE FEED NULLS	0
FORM FEED NULLS	0
PRINT LINE FEEDS	Yes

# CHANGE SETTINGS - REMOTE LINK.



Please note -  
Any changes made to these settings should be carried out by either your IT Manager or computer technician.

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



Tutor keypad.

The *REMOTE LINK* option is used to configure the communication protocols between the controller and an external device (such as a remote computer, paper tape punch, or printer).

On PC controlled machines, an unused serial (COM) port on the PC should be used (if available) to link to the external device.

On machines with an integrated controller, the port labelled "RS232" on the machine electrical control box should be used to link to the external device.

To select the individual settings use the [CURSOR ARROWS] keys and press the [EOB] key to toggle between the different values.

When the settings are correct, press the [RESET] key until all the menus have been removed from the screen.

CHANGE REMOTE LINK	
DEVICE NAME	MACHINE
BAUDRATE	4800
PARITY	Odd
DATA BITS	7
STOP BITS	1
SEND LINE-FEEDS	Yes
END OF-FILE	ctrl-2
SEND XOFF AT % FULL	100

# CHANGE SETTINGS

## - MISCELLANEOUS.

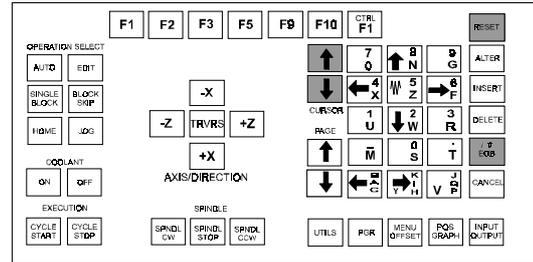
Keys Helpbox.

The following keys are used in this section:

[CURSOR ARROWS]

[EOB]

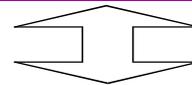
[RESET]



Tutor keypad.

The *CHANGE SETTINGS (MISCELLANEOUS) MENU* contains the following options:

1) *GLOBAL UNITS*. This option, when set to 'METRIC', will set the programming units of measurement as metric (millimetres). When set to 'IMPERIAL', the programming units of measurement are set as imperial (inches).  
 To select this option, highlight 'GLOBAL UNITS' using the [CURSOR ARROWS] keys and press the EOB] key. Continual pressing of the [EOB] key will toggle the option between the two settings.  
 When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.



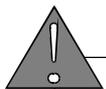
# CHANGE SETTINGS

## - MISCELLANEOUS.

2) *USER'S NAME*. This option allows the user's name to be printed out on any subsequent CNC File printouts.

To select this option, highlight '*USER'S NAME*' using the [CURSOR ARROWS] keys and press the [EOB] key to move the cursor across to the text '*A N Other*'. Enter the new text and press the [EOB] key to confirm.

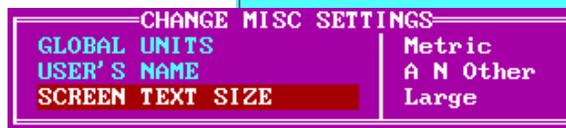
When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.



Please note -

The screen text size option will operate when the main screen display is set to "EGA" mode but will NOT operate when set to "VGA" mode. The screen mode can be altered by opening a DOS window and editing the "fanuc.go" file, found in the root of the machine directory. The screenshots in section 3) are taken from "EGA" displays.

3) *SCREEN TEXT SIZE*. This option, when set to '*LARGE*', will set the text size to 25 lines on screen :



When set to '*SMALL*', the text size is set to 43 lines on screen :



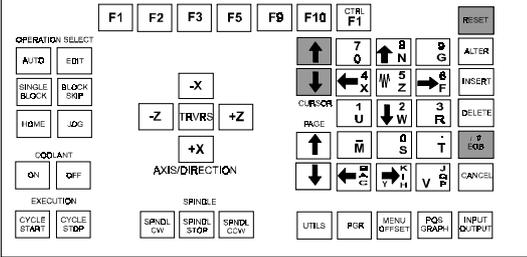
To select this option, highlight '*SCREEN TEXT SIZE*' using the [CURSOR ARROWS] keys and press the EOB key. Continual pressing of the [EOB] key will toggle the option between the two settings.

When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.

# CHANGE SETTINGS

## - LOAD SETTINGS.

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

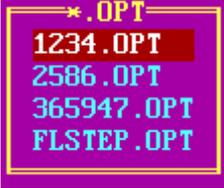


Tutor keypad.

The *LOAD SETTINGS* option is used to load any previously saved software settings onto the controller. When the *LOAD SETTINGS* option is selected, the default settings filename will appear in the edit window. The default filename is 'FLSTEP' :

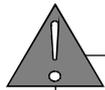


To list all of the available Settings Files, clear the edit window by pressing the [ALTER] key and press [EOB]. Select the required settings file from the list using the [CURSOR ARROWS] keys and press the [EOB] key to load the highlighted file :



When the Setting File has been loaded, press the [RESET] key until all the menus have been removed from the screen.

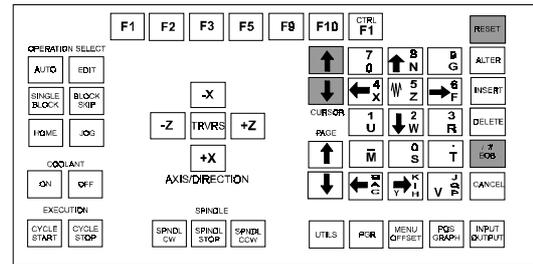
# CHANGE SETTINGS - SAVE SETTINGS.



Please note -

If the current settings file is to become the 'new' default settings file, save the file with the name 'FLSTEP' and ensure it is saved on the machines hard drive.

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



Tutor keypad.

The *SAVE SETTINGS* option is used to save the currently loaded software settings.

When the *SAVE SETTINGS* option is selected, the default settings filename will appear in the edit window. The default filename is 'FLSTEP' :



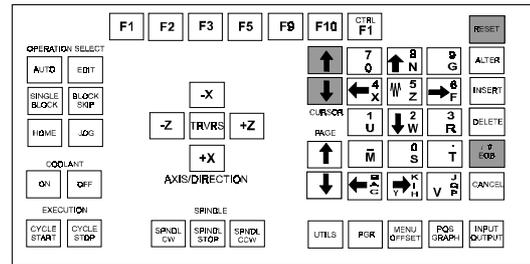
If the settings are saved with this ('FLSTEP') filename, they will be reloaded as the default settings whenever the machine software is restarted. To save the settings with a different name, press the [ALTER] key to clear the screen. Type in the 'new' filename and press the [EOB] key. The 'new' settings file will be saved on the currently selected drive :



When the settings file has been saved, press the [RESET] key until all the menus have been removed from the screen.

# CHANGE SETTINGS - DELETE SETTINGS.

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



Tutor keypad.

The *DELETE SETTINGS* option is used to permanently remove a settings file from a disk.

When the *DELETE SETTINGS* option is selected, an edit window will be displayed. Press the [EOB] key to delete the settings file shown. To enter a different settings filename, press the [ALTER] key to obtain a blank edit window and type in the required filename.



To list the settings files available, press the [EOB] key when the edit window is blank. Select the required settings file from the list using the [CURSOR ARROWS] keys and press the [EOB] key to delete the highlighted file :



When the settings file has been saved, press the [RESET] key until all the menus have been removed from the screen.

# PROGRAMMING TERMS AND CONVENTIONS.

This section describes the composition of a basic CNC part program, listing the following terms used:

- 1) Program Address characters.
- 2) G Codes used on Denford CNC Lathes.
- 3) M Codes used on Denford CNC Lathes.
- 4) Denford Directives (program codes specific to Denford CNC Machines).

# WHAT IS A PART PROGRAM ?

A *Part Program* is a list of coded instructions which describes how the designed component, or part, will be manufactured. This part program is also called the *CNC File*.

These coded instructions are called *data* - a series of letters and numbers. The part program includes all the geometrical and technological data to perform the required machine functions and movements to manufacture the part.

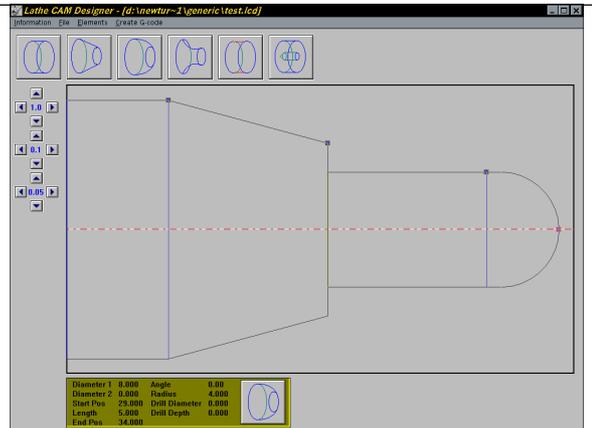
The part program can be further broken down into separate lines of data, each line describing a particular set of machining operations. These lines, which run in sequence, are called *blocks*.

A block of data contains *words*, sometimes called *codes*. Each word refers to a specific cutting/movement command or machine function. The programming language recognised by the CNC, the machine controller, is an I.S.O. code, which includes the *G and M code groups*.

Each program word is composed from a letter, called the *address*, along with a number.

These terms are illustrated on the next page....

# COMPOSITION OF A PART PROGRAM.



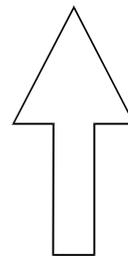
The component is designed "on-screen" using Denford's LatheCAM Designer software.

This CAD/CAM software package automatically generates a G code part program suitable for Denford CNC machines, listed on the bottom left of this page....

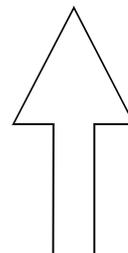
## PART PROGRAM EXAMPLE -

```
(Lathe CAM Designer - test.LCD)
(20/5/1997)
(Novaturn (metric))
(Post fanuc:1.2 23 May 1994)
N1G21
[BILLET X20 Z40
N2G98
N3G28U0W0
N4M6T0101
N5M3S1910
N6G0X20Z2
N7X22
N8X20
N9G1Z-33.5F60
N10X22
N11G0Z2
N12X15
N13M3S2546
N14G1Z-20.8
N15X17
N16G0Z2
N17X10
N18M3S3500
N19G1Z-15.3
N20X12
N21G0Z2
N22X5
N23G1Z-0.15
N24X7
N25X22
N26G0Z2
N27G1Z0
N28X0
N29G3X8Z-4K-4
N30G1Z-5
N31Z-16
N32X12
N33M3S2122
N34X18Z-27
N35Z-34
N36X22
N37G0Z2
N38M5
N39G28U0W0
N40M30
```

## ADDRESS EXAMPLE - G



## WORD EXAMPLE - G1



## BLOCK EXAMPLE - N23G1Z-0.15;

## DENFORD DIRECTIVE EXAMPLE - [BILLET

# LISTING OF ADDRESS CHARACTERS.

- N - Defines the program line (sequence) number.
- X - Defines a position movement perpendicular to the spindle centreline (along the X axis), programmed as a diameter value. Minus (-) movements of the tool are towards the centreline of rotation, whilst positive (+) movements are away from this centreline.
- Z - Defines a position movement parallel to the spindle centreline (along the Z axis), programmed as an absolute value. Minus (-) movements of the tool are away from the headstock, whilst positive (+) movements are towards the headstock..
- G - Defines preparatory functions.
- U - Defines an incremental distance (diameter value) for a position movement parallel to the X axis.
- W - Defines an incremental distance for a position movement parallel to the Z axis..
- R - Defines an arc radius as an absolute distance.
- I - Defines an arc radius as an incremental distance (with sign) parallel to the X axis, between the start of the arc and the arc centre (arc limited to quadrant).
- K - Defines an arc radius as an incremental distance (with sign) parallel to the Z axis, between the start of the arc and the arc centre (arc limited to quadrant).
- M - Defines miscellaneous or auxiliary functions.
- T - Defines tool numbers.
- S - Defines spindle speeds, speed constraints or constant surface speeds.
- F - Defines feed rates.
- Q - Defines final contour block in automatic cycles.
- P - Defines first contour block in automatic cycles or subprogram number.

# G CODES LISTING FOR DENFORD CNC FANUC LATHES.

NOTE: NOT ALL G CODES APPLY TO EACH MACHINE.

*Grp. Code and Description.*

1	G00 Positioning (Rapid Traverse)
1	G01 Linear Interpolation (Cutting Feed)
1	G02 Circular Interpolation CW
1	G03 Circular Interpolation CCW
Ø	G04 Dwell
Ø	G10 Offset Value Setting by Program
6	G20 Imperial Data Input (Inches)
6	G21 Metric Data Input (Millimetres)
9	G22 Stored Stroke Check On
9	G23 Stored Stroke Check Off
Ø	G27 Reference Point Return Check
Ø	G28 Reference Point Return
Ø	G29 Return from Reference Point
Ø	G30 Return to 2nd Reference Point
Ø	G31 Skip Function
1	G32 Thread Cutting
1	G34 Variable Lead Thread Cutting
Ø	G36 Automatic Tool Compensation X
Ø	G37 Automatic Tool Compensation Z
7	G40 Tool Nose Radius Compensation Cancel
7	G41 Tool Nose Radius Compensation Left
7	G42 Tool Nose Radius Compensation Right
Ø	G50 Work Co-ord. Change/Max. Spindle Speed Setting
Ø	G65 Macro Call
12	G66 Macro Modal Call
12	G67 Macro Modal Call Cancel
4	G70 Finishing Cycle
4	G71 Stock Removal in Turning
Ø	G72 Stock Removal in Facing
Ø	G73 Pattern Repeating
Ø	G74 Peck Drilling in Z Axis
Ø	G75 Grooving in X Axis
Ø	G76 Thread Cutting Cycle

# G CODES LISTING FOR DENFORD CNC FANUC LATHES.

- 1 G90 Cutting Cycle A
- 1 G92 Thread Cutting Cycle
- 1 G94 Cutting Cycle B
- 2 G96 Constant Surface Speed Control
- 2 G97 Constant Surface Speed Control Cancel
- 11 G98 Feed Per Minute
- 11 G99 Feed Per Revolution

## NOTES.

G codes from group 0 are non-modal (they must be programmed into every program block when required).

All other G codes are modal (they remain active through subsequent program blocks, until replaced or cancelled by a G code from their particular group).

# M CODES LISTING FOR DENFORD CNC FANUC LATHES.

NOTE: NOT ALL M CODES APPLY TO EACH MACHINE.

*Code. Description.*

M00*	Program Stop
M01*	Optional Stop
M02*	Program Reset
M03	Spindle Forward
M04	Spindle Reverse
M05*	Spindle Stop
M06	Auto Tool Change
M07	Coolant "B" ON
M08	Coolant "A" ON
M09*	Coolant OFF
M10	Chuck Open
M11	Chuck Close
M13	Spindle Forward and Coolant On
M14	Spindle Reverse and Coolant On
M15	Program Input Using "MIN P" (Special Function)
M16	Special Tool Call (Tool Call ignores Turret)
M19	Spindle Orientate
M20	Spindle Index A
M21	Spindle Index 2A
M22	Spindle Index 3A
M23	Spindle Index 4A
M25	Quill Extend
M26	Quill Retract
M29	Select "DNC" Mode
M30	Program Reset and Rewind
M31	Increment Parts Counter
M37	Door Open to Stop
M38	Door Open
M39	Door Close
M40	Parts Catcher Extend
M41	Parts Catcher Retract
M43	Swarf Conveyor Forward
M44	Swarf Conveyor Reverse
M45*	Swarf Conveyor Stop

# M CODES LISTING FOR DENFORD CNC FANUC LATHES.

M48	Lock % Feed and % Speed at 100%
M49	Cancel M48 (Default)
M50	Wait for Axis in Position Signal (cancels Continuous Path)
M51	Cancel M50 (Default)
M52	Pull-out in Threading = 90 Degrees (Default)
M53	Cancel M52
M54	Disable Spindle Fluctuation Testing (Default)
M56	Select Internal Chucking (from PLC Edition "F")
M57	Select External Chucking (from PLC Edition "F")
M62	Aux. 1. On
M63	Aux. 2. On
M64	Aux. 1. Off
M65	Aux. 2. Off
M66*	Wait for Input 1
M67*	Wait for Input 2
M68	Only Index with all Axes at Home Position
M69	Index Turret Anywhere
M7Ø	Mirror in X On
M76	Wait for Input 1 to go Low (from Revision C)
M77	Wait for Input 2 to go Low (from Revision C)
M8Ø	Mirror in X Off
M98	Sub-Program Call
M99	Sub-Program End

## NOTES.

Not all M codes listed are available, all M codes marked with an asterisk (\*) will be performed at the end of a program block (ie, after any axis movement).

# DENFORD DIRECTIVES.

Directives are program terms defined by Denford Limited.

They are used to help generate the '*Simulation Mode*' graphics used by the machine controlling software.

## [BILLET

This directive allows a billet that appears in a simulation window to be given a size. The billet definition should be placed at the start of a program, after the units of measurement have been set.

Example:

```
G21
```

```
[BILLET X5Ø.Ø Z1ØØ.Ø
```

This sets the measure to metric (Note - if set to Imperial the units would be inches) and defines the billet as 1ØØmm long with a diameter of 5Ø.Ø. Note that the length does not include the "extra" material required by the headstock chuck jaws.

## [SUBPROGRAM

This directive allows a program with a non-numeric name to be called as a subprogram.

Example:

```
[SUBPROGRAM Ø2ØØ FRED
```

```
M98 PØ2ØØ
```

This example assigns a subprogram number of Ø2ØØ to the program named FRED, then calls the subprogram Ø2ØØ.

## [STEP

This directive runs an on-screen program in single steps. This means the program will run one program line, then wait for the operator to prompt it to move to the next line; this continues until the program is instructed to stop this function.

The directive applies to both simulation and actual machining with a program.

# DENFORD DIRECTIVES.

## [NO STEP

This directive runs an on-screen program without single steps. This means the program will run as originally intended with no pausing, unless a pause is requested from within the program itself.

The directive applies to both simulation and actual machining with a program.

## [SHOW

This directive allows the machining operations to be graphically simulated on-screen.

## [NOSHOW

This directive stops the machining operations from being graphically simulated on-screen.

## !

An exclamation mark is used to display a message in the tutorial messages window (shown in the lower part of the screen). The message will be shown until it is either cleared or replaced by another message.

Tutorial messages are shown coloured green, within the program, on-screen. Messages must be entered off-line since text cannot be entered with the tutor keyboard.

Example:

**! NOW CUTTING THREAD**

This example would print the line "NOW CUTTING THREAD" in the tutorial messages window in the lower part of the screen.

# DENFORD DIRECTIVES.

**?**

A question mark is used to display a message in the tutorial messages window (shown in the lower part of the screen). When the message is displayed the program will stop. A keypress is required to set the program running again. Any messages will be shown until they are either cleared or replaced by another message.

Tutorial messages are shown coloured green within the program, on-screen. Messages must be entered off-line since text cannot be entered with the tutor keyboard.

Example:

**? CHECK SURFACE FINISH OF TAPER**

This example would print the line "CHECK SURFACE FINISH OF TAPER" in the tutorial messages window and stop the program. A key would need to be pressed to allow the program to continue.

**[CLEAR**

This clears any messages currently displayed in the tutorial messages window.

# GLOSSARY.

ABSOLUTE	In absolute programming, zero is the point from which all other dimensions are described.
ARC	A portion of a circle.
AUTOMATIC CYCLE	A mode of control operation that continuously runs a cycle or stored program until a program stop or end of program word is read by the controller.
AUXILIARY FUNCTION	The function of the CNC machine (ie, F, S, T, M codes etc.), other than co-ordinate based commands.
AXIS (AXES)	The planes of movement for the cutting tool, usually referred to as X (running at 90 degrees to the spindle centreline) and Z (running parallel to the spindle centreline).
BILLET	The actual material being machined, sometimes referred to as the "workpiece".
BLOCK	A collection of program words that collectively describe a machining operation.
CHARACTER	A number, letter or symbol as entered into a CNC program.
CIRCULAR INTERPOLATION	G-code term for a programmed arc movement.
COMMAND	A signal (or group of signals) instructing one step/operation to be carried out.
CNC	Computer Numerical control.
CYCLE	A sequence of events or commands.
DATUM	The point (co-ordinate) from which a series of measurements are taken.

DESKTOP TUTOR	The input control keypad for the machine. Keypad overlays are interchangeable according to the type of controller required.
DIRECTORY	An area of a disk containing the names and locations of the files it currently holds.
DISK	A computer information storage device, examples, C: (drive) is usually the computers hard (internal) disk and A: (drive) is usually the floppy (portable 3.5" diskette) disk.
DRIVE	The controller unit for a disk system.
DRY RUN	An operation used to test how a CNC program will function without driving the machine itself.
DWELL	A programmed time delay.
EDIT	The mode used for altering the content of a CNC program via the Desktop Tutor.
END OF BLOCK SIGNAL	A symbol or indicator that defines the end of a block of data. The 'pc' equivalent of the 'return' key.
ERROR	The deviation of an attained value from a desired value.
FEEDRATE	The rate, in mm/min or in/min, at which the cutting tool is advanced into the workpiece.
G-CODE	The programming language understood by the machine controller.
FILE	An arrangement of instructions or information, usually referring to work or control settings.
FORMAT	The pattern or way that data is organised.
G CODE	A preparatory code function in a CNC program that determines the control mode.

HARDWARE	Equipment such as the machine tool, the controller, or the computer.
HOME	The "Home" command (followed by an axis key) will send the toolpost to the extreme limits of the axis selected.
INCH (OR JOG) CONTROL	A manual control button, used when setting-up a machine, which permits the position of machine to be altered, either by very small pre-defined movements (called Jog steps) or larger continued movements (called continuous).
INCREMENTAL	Incremental programming uses co-ordinate movements that are related from the previous programmed position. Signs are used to indicate the direction of movement.
INPUT	The transfer of external information (data) into a control system.
INTERFACE	The medium through which the control/computer directs the machine tool.
M CODE	A miscellaneous code function in a CNC program used to indicate an auxiliary function (ie, coolant on, tool change etc.).
MACHINE CODE	The code obeyed by a computer or microprocessor system with no need for further translation.
MACHINE DATUM	Homing the machine will find the limits of movement for both axes. The toolpost will move to its "parked" (home) position and the tip of the currently selected tool at this point is taken as the Machine Datum.
MDI	MDI - Manual Data Input is the method used for inserting data into the control system (ie, Desktop Tutor, qwerty keyboard etc.).
MODAL	Modal codes entered into the controller by a CNC program are retained until changed by a code from the same modal group or cancelled.
NC	Numerical control.

PC	Personal computer.
PROGRAM	A systematic arrangements of instructions or information to suit a piece of equipment.
REFERENCE POINTS	The machine has 2 reference points (hidden microswitches) used in setting the limits of movement for its slides (axes).
RAPID TRAVERSE	Fast movement of the cutting tool through the 2 machine axes between cutting settings.
RPM	Revolutions per minute (rev/min) - a measure of spindle speed.
SLIDES	The 2 machine axes - see axis.
SOFTWARE	Programs, tool lists, sequence of instructions etc.....
TOOLPOST	The holder for the various cutting tools. Automatic Toolposts are mounted above the spindle centreline, whilst most manual toolposts are mounted below it.
TOOL OFFSETS	Tool Offsets "tell" the tool where the actual billet is positioned, in relation to the rest of the machine, so it can start cutting in the correct place. Tool Offsets will also account for the different sizes and types of cutting tool used on the billet.
TRAVERSE	Movement of the cutting tool through the 2 machine axes between cutting settings.
WORK (WORKPIECE)	The actual material being milled. Quite often, this work is also secured onto a sub-table. The work is sometimes referred to as the 'billet'.
WORD	A combination of a letter address and digits, used in a CNC program (ie, G42, M04 etc.).