DENFORD

COMPUTERISED MACHINES AND SYSTEMS

Generic Milling 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

Notes.

Contents.

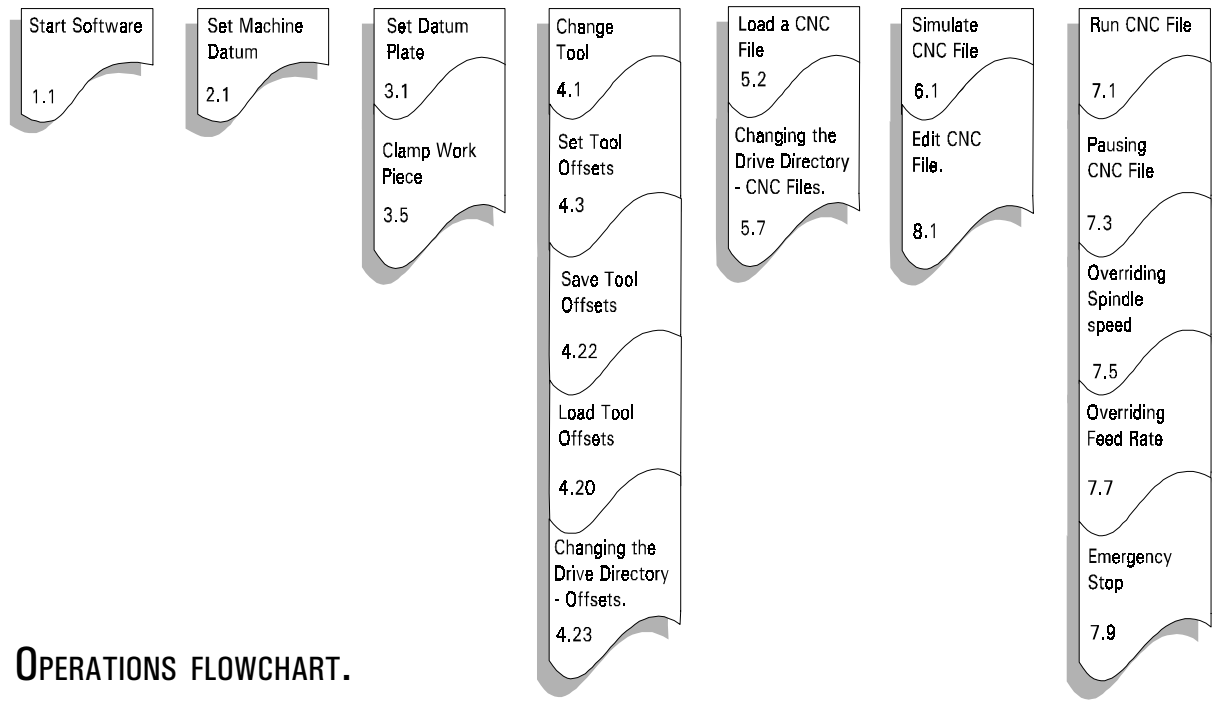
Contents.

INTRODUCTION.

Using a new piece of hardware, such as your Denford Miller, 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 Vertical Machining Centres (VMCs) and Milling Machines.



OPERATIONS FLOWCHART.

The Operations flowchart lists the progress of a typical operating procedure for a Milling Machine or Vertical Machining Centre. Sections in this manual correspond to each of the operations listed above.

USING THE MANUAL.....

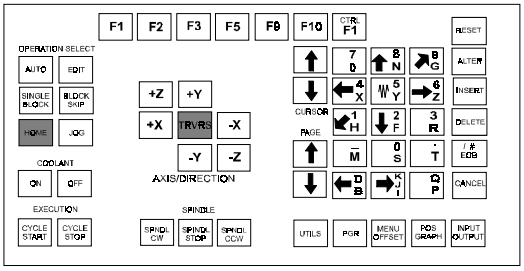
Conventions used in this manual follow this format :

[.....] - 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.....

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



Tutor keypad.

The helpbox above would show that the [HOME] and [TRVRS] keys are used at some point during that particular section.

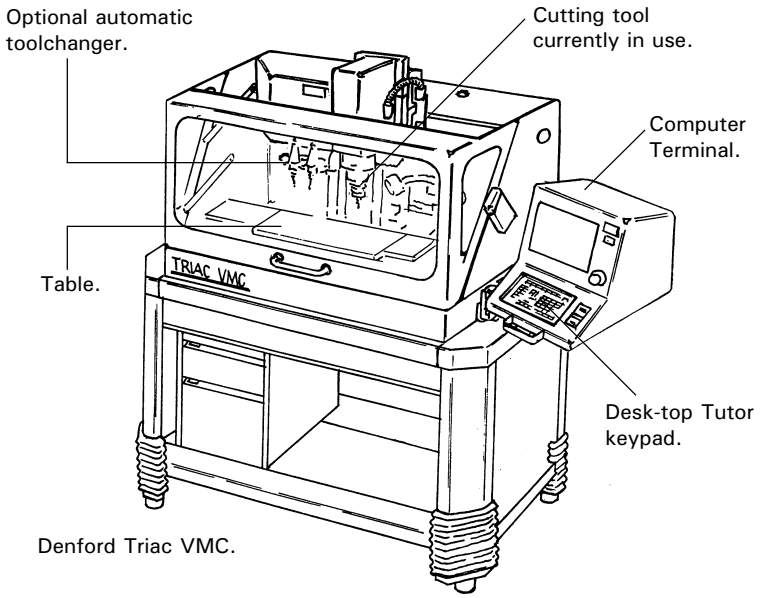
When relevant, screenshots and graphics are also provided to accompany the text. Specific screenshots and drawings of machines are only used for illustrative purposes. All the operations described in this manual relate to any Denford VMC or Milling Machine.

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

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

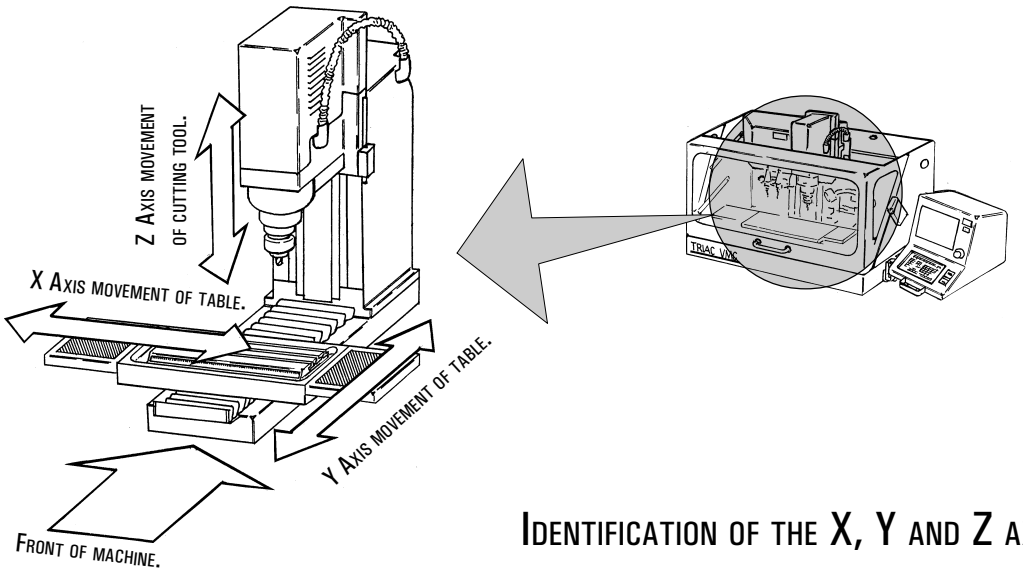
For Machine specific features and operations, please consult the seperate Installation Manual supplied with your machine.

BASIC COMPONENTS....



BASIC LAYOUT OF COMPONENTS ON AN EXAMPLE VMC.

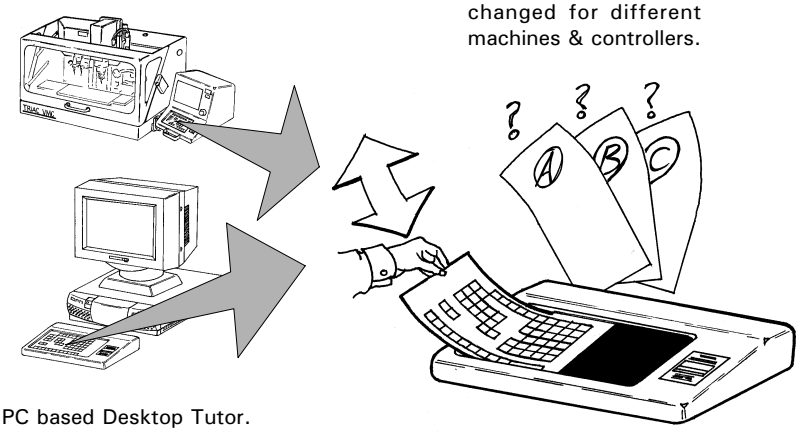
A milling machine is used to make holes and slots in workpieces. In order to machine different parts of the workpiece, both the table and the cutting tool of the milling machine are moveable. On a CNC (Computer Numerical Control) milling machine, they are moved by motors controlled by the computer. The working envelope has a set of co-ordinates labelled X,Y and Z. This allows the computer to control and send the cutting tool to areas automatically, once they have been programmed into the machines computer, via the Desk-top Tutor keypad.



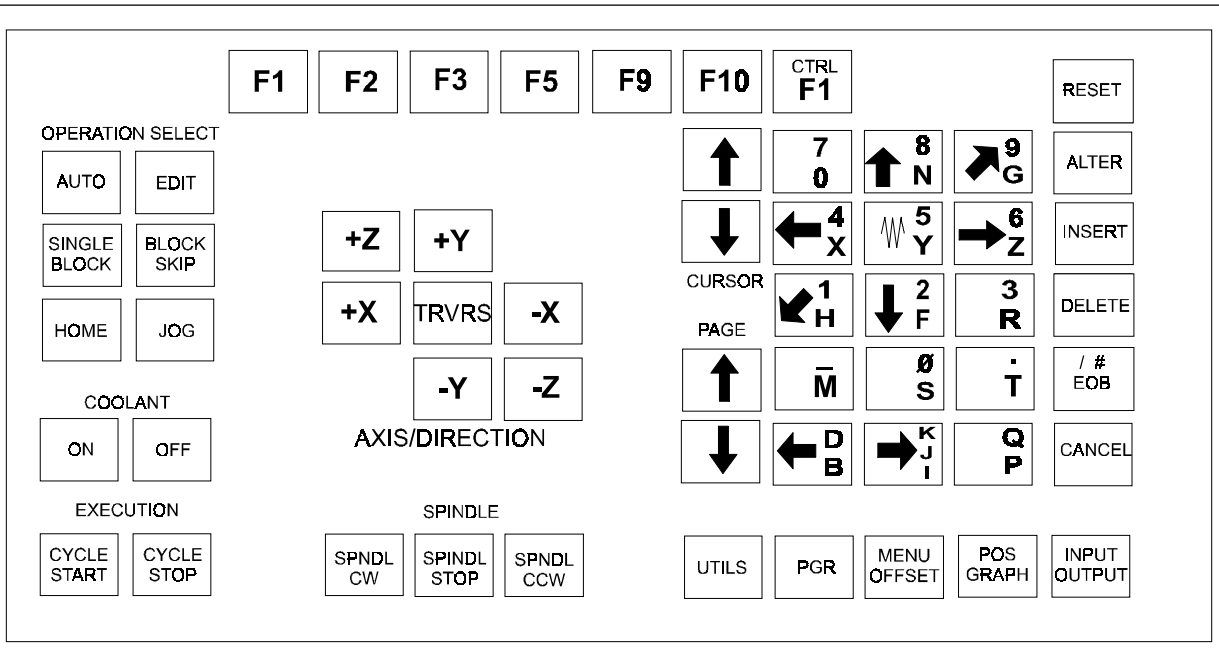
IDENTIFICATION OF THE X, Y AND Z AXES.

The X axis of the machine is the movement of the table left and right (parallel to the front edge of the machine).
The Y axis of the machine is the movement of the table forwards and backwards (parallel to the side edge of the machine).
The Z axis of the machine is the movement of the cutting tool up and down.

Operating the Desktop Tutor Control Panel.



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



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

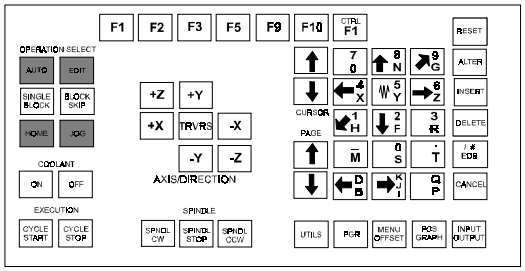
The Operation Select keys will set the mode in which the machine and controller will run:

The [AUTO] key is used to select *Auto Mode* - pressing this key will allow the user to run any program loaded into the machine.

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.

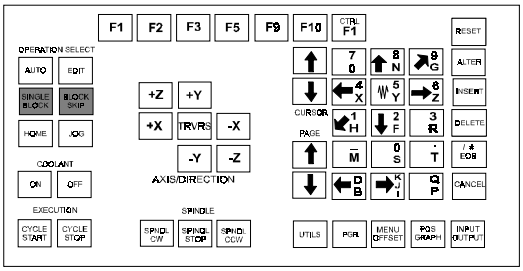
Many of the other keys on the controller have multiple functions, according to the machine mode that is set.

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.

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

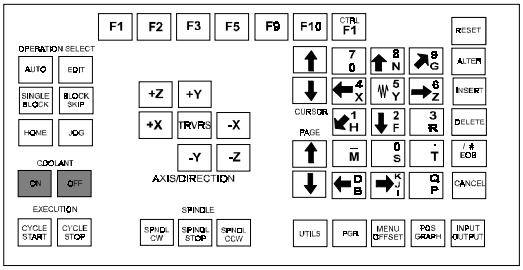
AUTO EDIT HOME JOG The [BLOCK SKIP] key is used select the option to ignore, or include, specific program blocks (activated by a "/" character in front of the block).



Tutor keypad.

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

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



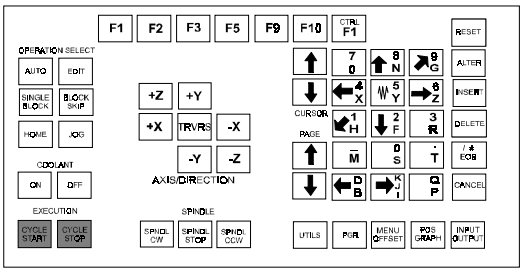
Tutor keypad.

The [CYCLE START] key is used to:

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

The [CYCLE STOP] key is used to:

1) AUTO EDIT Stop a program currently running on the machine.
2) AUTO EDIT 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 [-Y] key is used to control axis movement in the -Y direction.

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

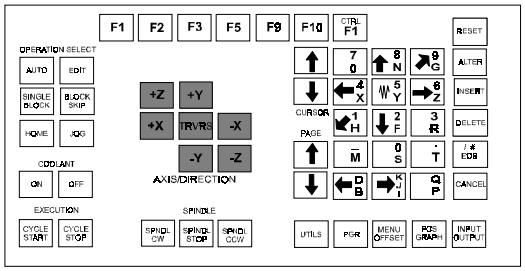
JOG The [+Y] key is used to control axis movement in the +Y direction.

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.

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

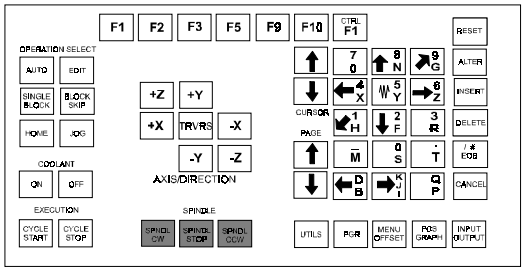


Tutor keypad.

HOME JOG The [SPNDL.CW] key is used to turn the spindle in a clockwise (forward) direction, when viewed from looking down onto the top of the machine head.

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 (reverse) direction, when viewed from looking down onto the top of the machine head.



Tutor keypad.

The blue [CURSOR ARROWS] keys are used to:

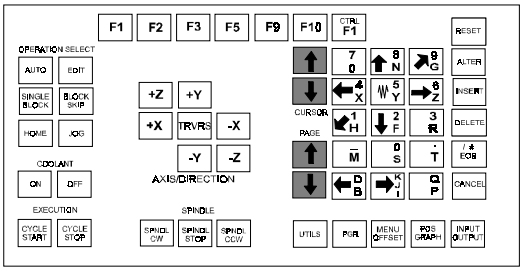
1) [EDIT] [HOME] [JOG] Move up or down lines of text on purple Menu Selection screens and when editing programs.

2) [AUTO] Move up or down the program lines before machining is started.

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

1) [EDIT] [HOME] [JOG] Move between the top and bottom lines of purple Menu Selection screens and programs.

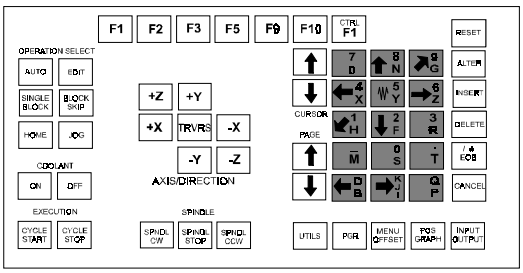
2) [AUTO] Move between up or down the pages of the program before machining is started.



Tutor keypad.

[EDIT] [JOG] 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.

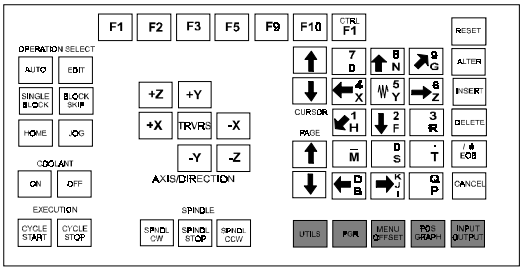
[EDIT] The [UTILS] key is used to display any directives within a program, shown on screen as [YELLOW LINES]. Directives are Denford definitions for tool sizes, billet sizes etc....

[AUTO] [EDIT] [HOME] [JOG] The [PGR.] key is used to toggle between these screen modes: *Simulate Only*, *Edit only* or *Edit and Simulate*.

[AUTO] [HOME] [JOG] The [MENU OFFSET] key is used to select the *Control Options Menu* (Execute CNC, Edit Offsets, Load Offsets or Save Offsets).

[AUTO] [HOME] [JOG] 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).

[EDIT] The [INPUT OUTPUT] key is used to select the *Remote Device Link Menu* (this menu allows data to be sent or received from external peripherals).



Tutor keypad.

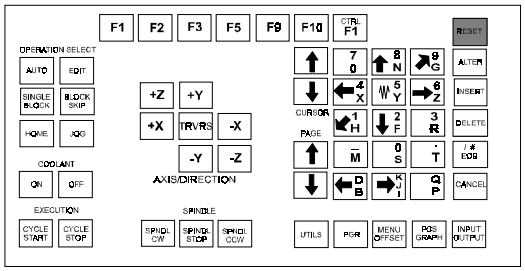
The [RESET] key is used to:

1) [AUTO] [EDIT] [HOME] [JOG] Move back through/clear any menu screens accessed, one by one, until the highest (start) level is reached.

2) [AUTO] [EDIT] [HOME] [JOG] Clear any red warning or error messages from the screen.

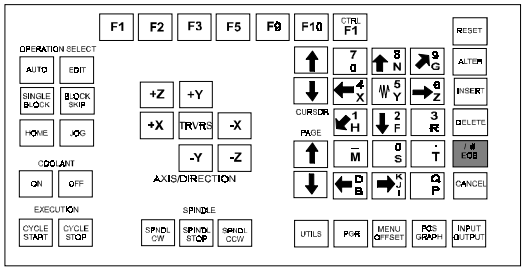
3) [EDIT] Move to the top of a program which is not being run.

4) [AUTO] [EDIT] Aborts a program which is running.



Tutor keypad.

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



Tutor keypad.

The [ALTER] key is used to:

1) [EDIT] Change any words (made from an address letter and a number) in a program line.

2) [AUTO] [EDIT] [HOME] [JOG] Delete information in a text entry box (ie, load/save boxes).

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

The [DELETE] key is used to:

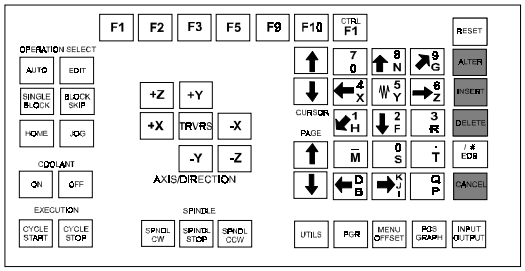
1) [EDIT] Remove a word from a program line.

2) [AUTO] [EDIT] [HOME] [JOG] Remove unwanted characters that have been typed in.

The [CANCEL] key is used to:

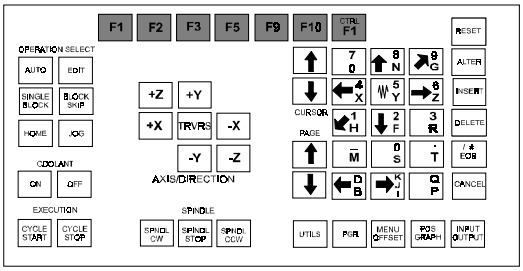
1) [EDIT] Remove a word from a program data entry line.

2) [AUTO] Abort a running program.

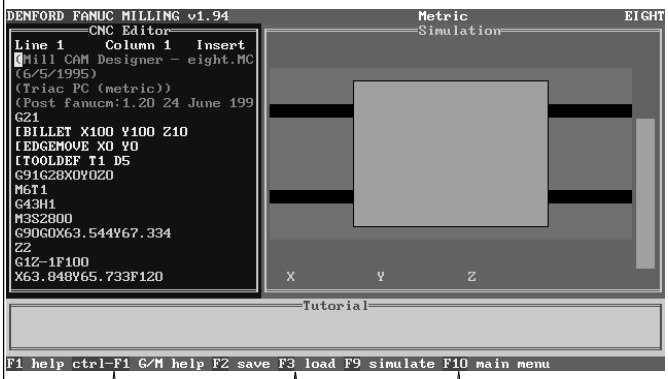


Tutor keypad.

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



Tutor keypad.



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

AUTO EDIT HOME JOG The [F1] key is used to display *General Help* screens concerning software features, commands and menus.

AUTO EDIT HOME JOG The [F2] key is used to save a program to the default drive.

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

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

The [F9] key is used to:

1) AUTO EDIT HOME JOG Display the *Control Menu*.

2) EDIT Display the *Simulation Menu*.

AUTO EDIT HOME JOG The [F10] key is used to display the *Main Menu*.

EDIT The [CTRL F1] key is used to display *G and M code Help* screens concerning the uses and definitions of G and M codes.

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:

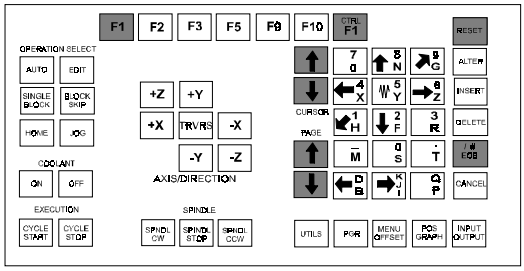
Manual Feedrate Override - Use the key labelled [4X LEFT ARROW] to decrease the feedrate and the [6Z RIGHT ARROW] to increase the feedrate.

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*.

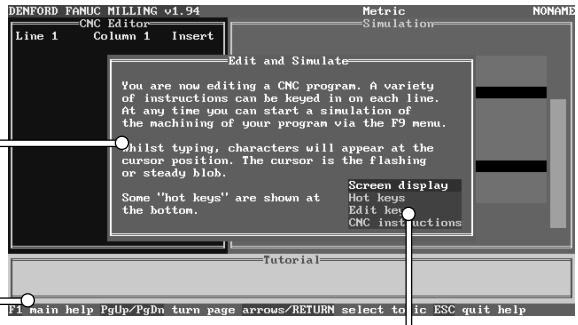
CONTEXT SENSITIVE HELP.

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



Tutor keypad.

Both the *General Help* screens (press the [F1] key) and the *G and M code Help* screens (press the [CTRL F1] key) are context sensitive. This means the help screens displayed will relate directly to the last known position of the main screen cursor (when information is available). The layout of *Help* screens are as follows:



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.

Pressing the [F5} key will access the *Data Information Help* screens (including date/time, users name, software and device settings).

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!

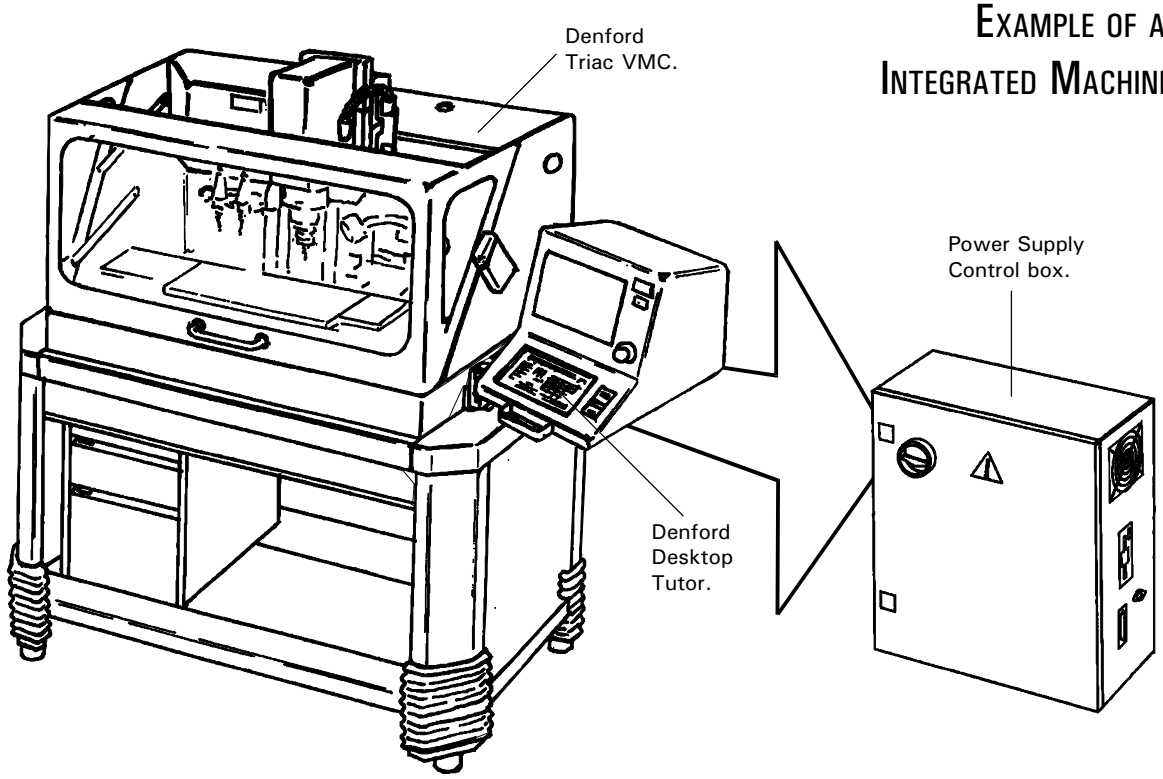
The Power Supply controls for integrated machines are sited on either,

(i) free standing metal power supply cabinets

(as illustrated below).

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

Example of an Integrated Machine.



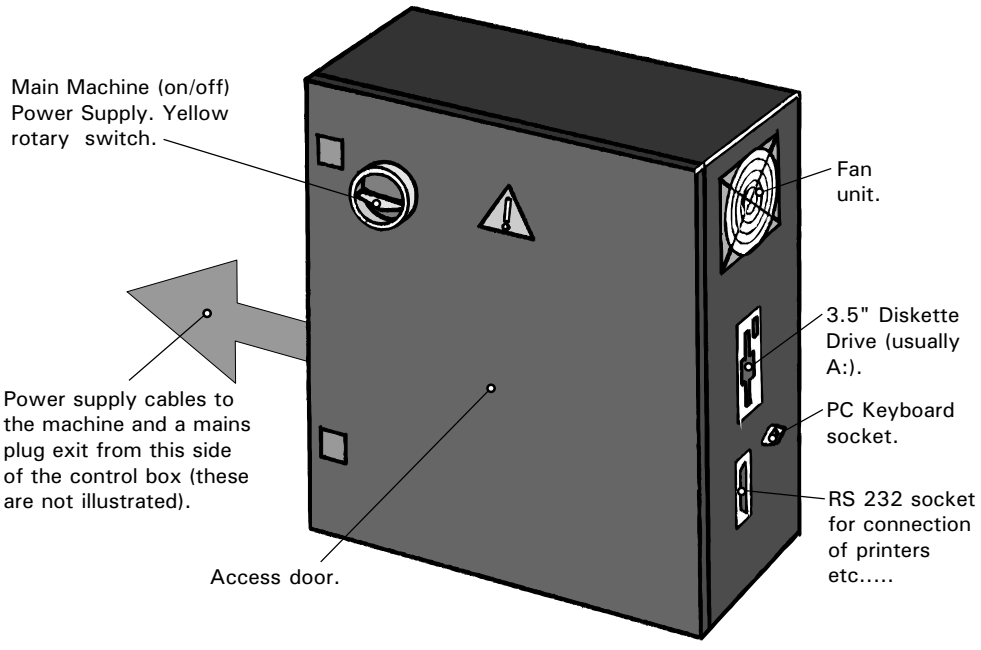
Switching on a Machine with an Integrated Controller.

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.

The machine control software and necessary drivers wil 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.

Exit the machine control software (see page 1.7). Turn the yellow rotary power supply switch to the 'off' position. The machine must not be turned off if a milling program is running, or the machine is cutting work....



Integrated Machine, Power Supply control box.

Switching on a Machine controlled by a PC.

A pc operated machine has 3 main components:

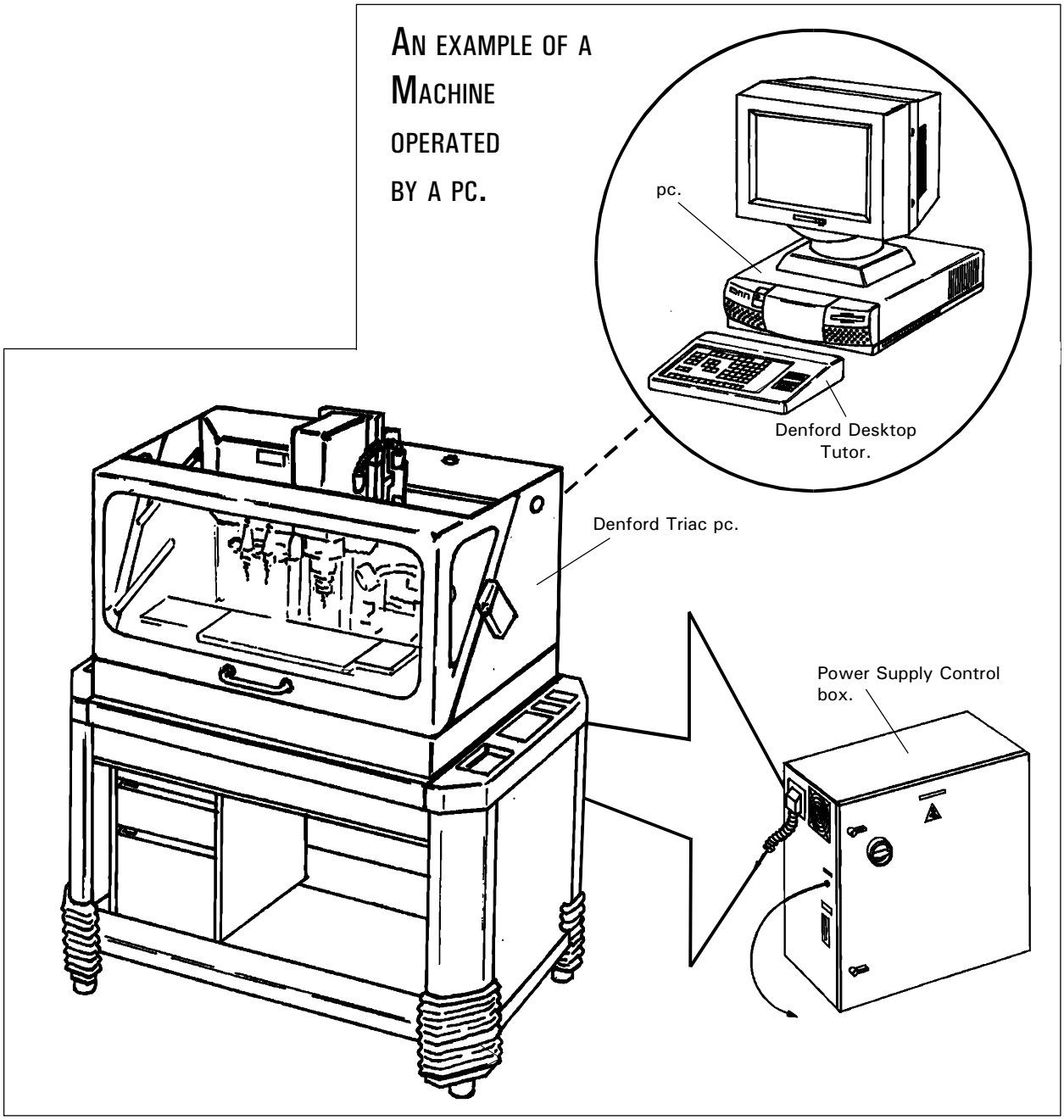
1) The pc (personal computer) itself, with an attached Denford Desktop Tutor.

2) The CNC milling machine.

3) The machine power supply unit.

The pc and CNC milling machine 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 free standing metal power supply cabinets (as illustrated below).

An example of a Machine operated by a PC.



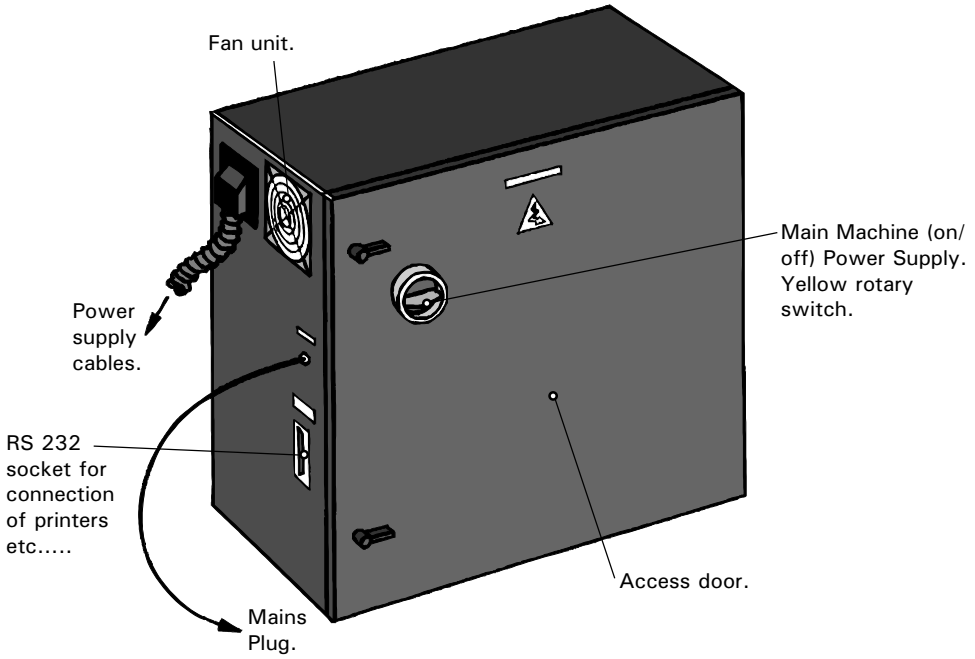
Switching on a Machine controlled by a PC.

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

.

Exit the machine control software (see page 1.7). Turn the yellow rotary power supply switch to the 'off' position. The machine must not be turned off if a milling program is running, or the machine is cutting work....

PC operated Machine, Power Supply control box.



PC Controlled Machines - Loading the Control Software from a Hard Disk.

To load the machine control 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.
Micromill	/DENFORD	FANUCSMD
Novamill	/NOVAMILL	FANUCMD
Triac PC	/TRIACPC	FANUCMD

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

To start the machine control 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 keyboard.

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

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

The machine control software will now load.

On machines operated with a Desktop Tutor, the pc keyboard is disabled during software use.

PC Controlled Machines - Loading the Control Software from a Floppy Disk.

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.
Micromill	FANUCSMD
Novamill	FANUCMD
Triac PC	FANUCMD

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).

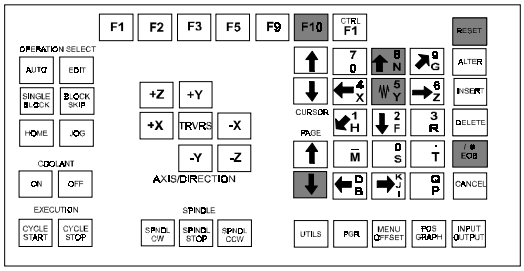
Press the [ENTER] / [RETURN] key on the pc keyboard.

The machine controlling software will now load.

On machines operated with a Desktop Tutor, the pc keyboard is disabled during software use.

Closing the Control Software.

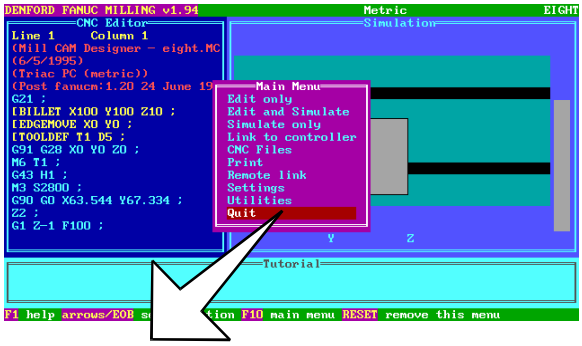
Desktop Tutor Keys Helpbox.
The following keys are used in this section:
[F10], [PAGE DOWN], [N], [Y], [EOB]



Tutor keypad.

Select the *Main Menu* by pressing the [F10] key.

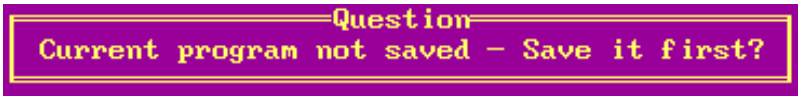
Press the [PAGE DOWN] key to highlight '*Quit*', then press the [EOB] key to close the control software.



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

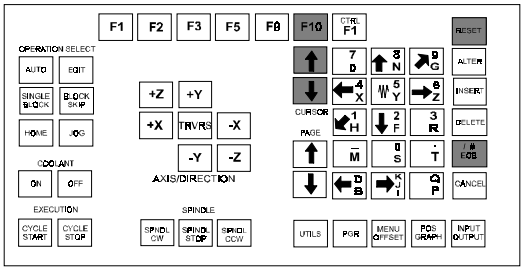
If this program does not need to be saved, press the [N] key.

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.



CONTROL SOFTWARE - MAIN MENU.

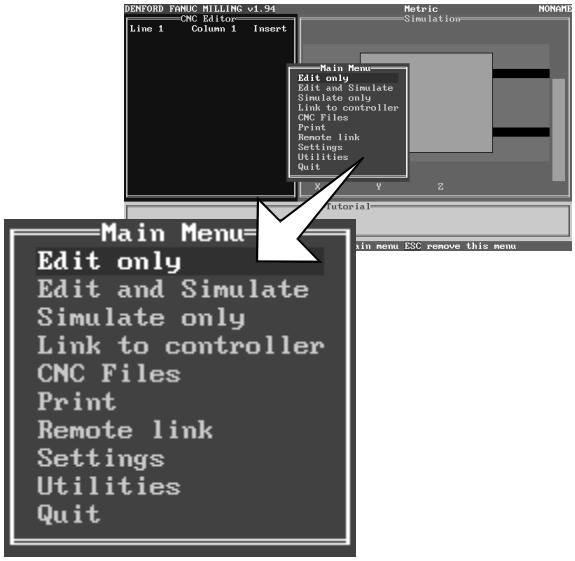
Keys Helpbox.
The following keys are used in this section:
[F10]
[CURSOR ARROWS]
[EOB], [RESET]



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 ben 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) *Link to Controller*. This option allows a CNC File to be downfed or loaded from an external FANUC controller.

5) *CNC Files*. This option gives access to a sub-menu allowing CNC Files to be loaded, created, 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 control software and returns to DOS.

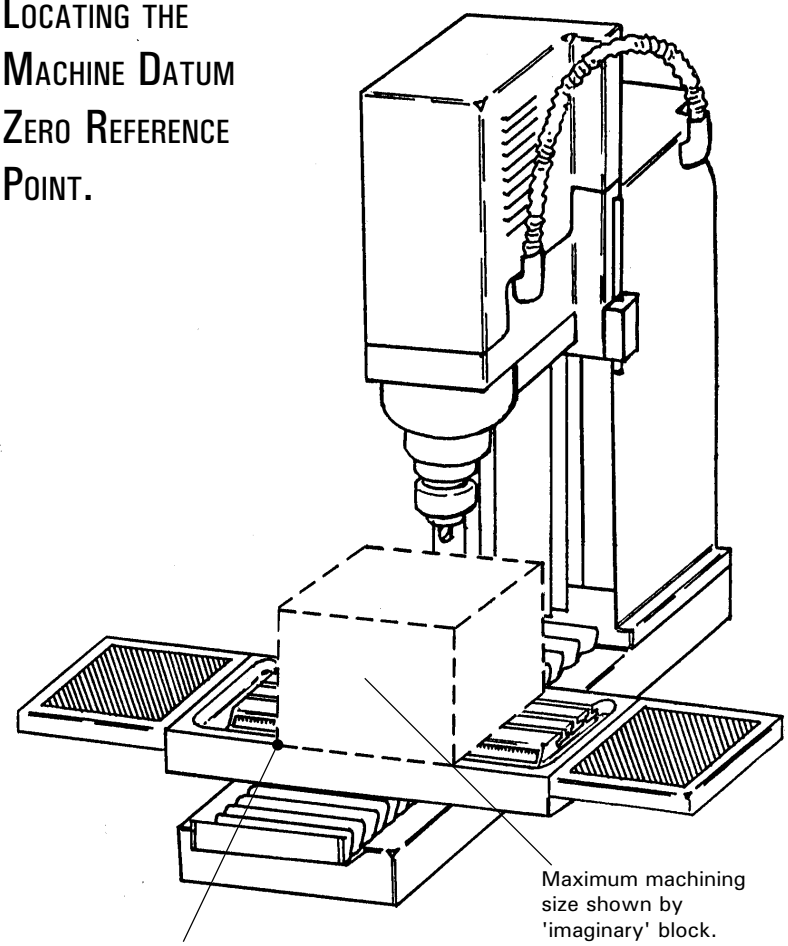
Automatic Search for the Machine Datum Point.

It is necessary to home the machine whenever it is switched on, to find the machine datum point - this is used as a zero reference for describing other co-ordinates on the machine.

The machine datum zero reference point (co-ordinates X = 0, Y = 0, Z = 0) is the front, left, lower corner of an imaginary block placed on the table. The block itself represents the largest possible size of workpiece the miller could manage to machine.

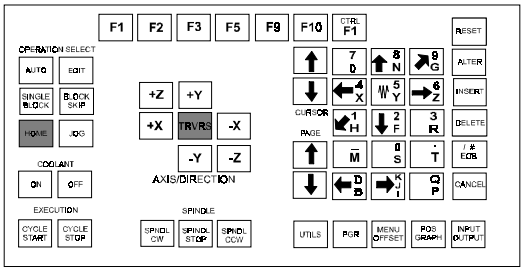
The machine datum point is used when taking any measurements from future co-ordinates we load or program. This is fine, if the start co-ordinates of our work coincide with the machine datum zero reference point. If not, this point must be moved - see section 4.3 "Setting the Tool Offset of the Machine".

Locating the Machine Datum Zero Reference Point.



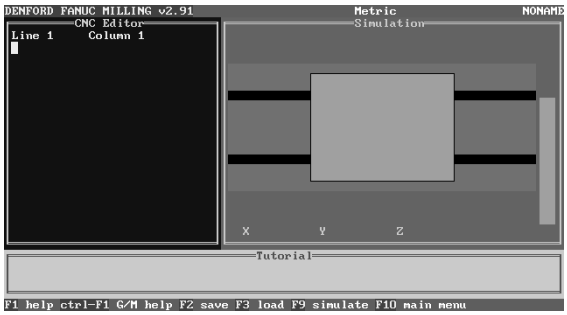
Automatic Search for the Machine Datum Point.

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

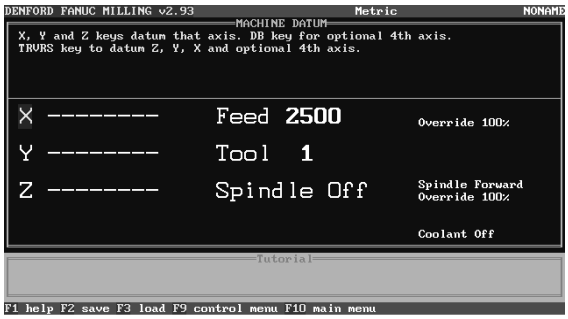


Tutor keypad.

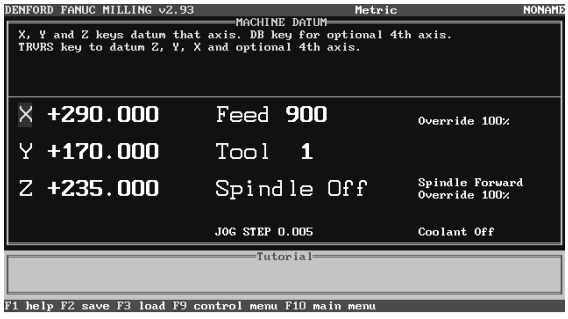
On loading up the DENFORD FANUC MILLING software, the start up screen will be displayed.....



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



Next, press the [TRVRS] key. The machine table and cutting tool will move until the 3 reference points are located. Upon completion of this procedure, the screen will display a set of co-ordinates, relating to the maximum limits of travel for each axis*.

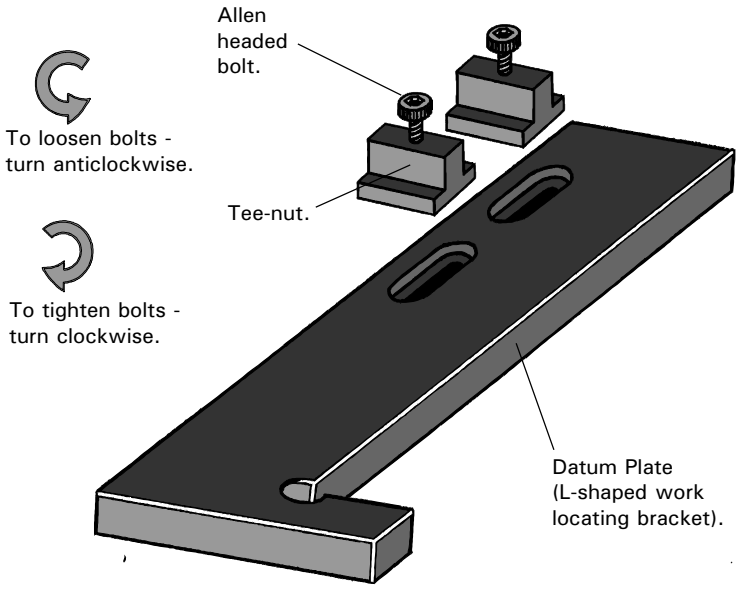


Note - Novamill machines will datum with an X co-ordinate display of 'zero' due to the position of their microswitch. The X axis limit of 225 can be viewed by moving the table in *Jog Mode* to the far left of the machine. Press [JOG], then [+X].

Please note - The numerical figures depicted on screenshots will differ slightly according to the machine being used. In this example, the co-ordinates shown refer to setting the Datum point on a Denford Triac machine.

Setting the Datum Plate.

The Datum Plate is an 'L-shaped' bracket used in helping to locate work to be positioned on the machine table. It is fixed in position using two Tee-nuts (tightened with allen headed bolts), which locate into two of the three available T shaped channels* which run horizontally (i.e. parallel to the X axis) under the surface of the machine table.



Removal of Datum Plate.

To remove the datum plate, the 2 allen headed bolts need to be undone, by turning them in an anti clockwise direction. Next, slide the whole unit along until the Tee-nuts are released from their channels, then withdraw the datum plate from the machine table.

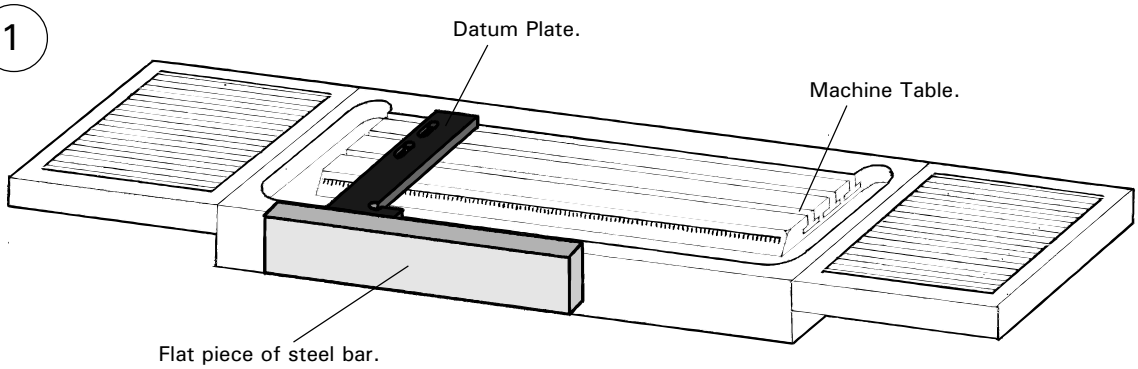
Fitting of Datum Plate.

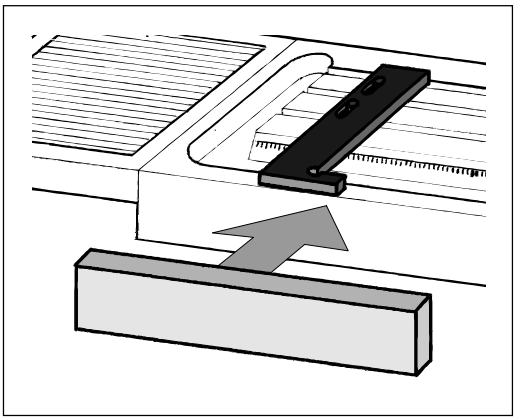
To fit the datum plate in position on the machine table, place it at the end of the T shaped channels. Line up each Tee-nut with its respective channel and slide it into position. Next, move the whole unit along to the required position on the machine table - note that the datum plate can be slightly adjusted forwards and backwards (i.e. parallel to the Y axis), but for larger movements the whole unit must be withdrawn and moved to the next T channels along*. Once the datum plate has been approximately positioned in the correct place, tighten each of the allen headed bolts, by turning them in a clockwise direction until they just begin to grip the plate to the table surface. It must still be possible to move the datum plate, since it may require final adjustments if it needs to be lined up 'square' on the machine table.

* Please Note - Novamill machines only have two T shaped channels.

The following diagrams illustrate a number of methods for making final adjustments to the datum plate. Each method varies according to the accuracy required, when the datum plate needs to be positioned 'square' with respect to the machine table (i.e. the edges of the datum plate run exactly parallel with the 3 machine axes).

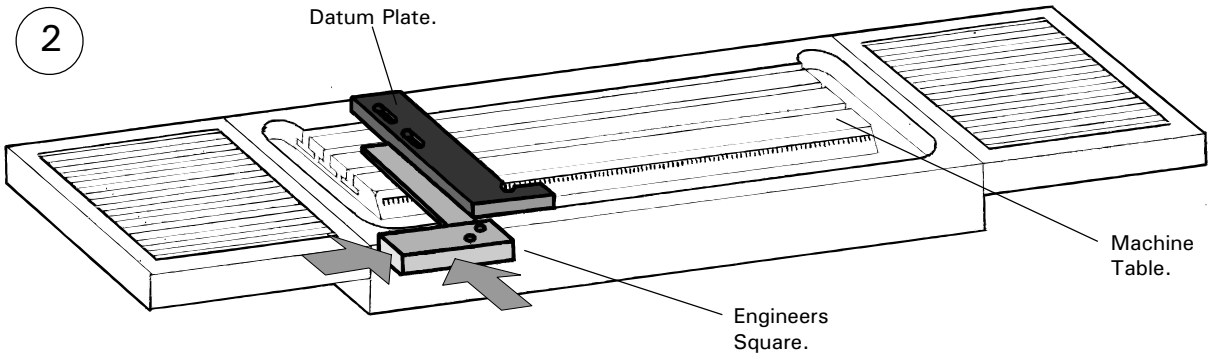
1





This method is useful if the front face of the datum plate can be positioned exactly level with the front edge of the machine table. Use the 'true' flat face (i.e. accurately level and straight) of a section of material, such as a piece of flat steel bar. Press the steel bar firmly against the front edge of the table and adjust the datum plate so *its* front face also touches the surface of the steel bar. Tighten the allen headed bolts. Note that although this method is quick, it is also fairly inaccurate.

2

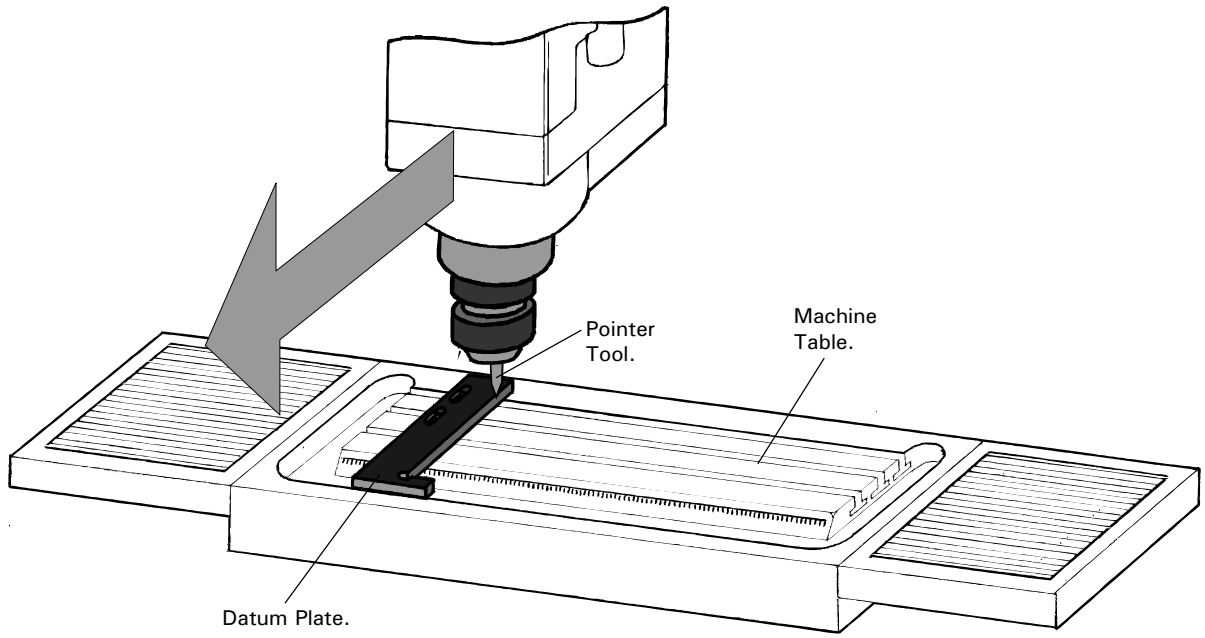


To obtain a better degree of accuracy, use an engineers square lined up against the front edge of the machine table. Adjust the datum plate so it touches the engineers square and tighten the allen headed bolts. This method has the added advantage of allowing the datum plate to be fixed further 'into' the machine table (i.e. anywhere on the machine table running parallel with the Y axis).

3

Set up the machine so a pointer is held in place of the cutting tool (the [T] key will change tools on an automatic tool changer). Align the pointing tool so it is positioned slightly above one of the 2 edges of the datum plate, which run parallel with the Y axis. Use the [+X], [-X], [+Y], [-Y], [+Z] and [-Z] keys to move the pointing tool, with the *Jog Mode* set to read '*continuous*' on screen (this can be selected using the [JOG] key). This can only be done with the guard closed, since the axes will only move in *Incremental Jog Mode* with the guard open.

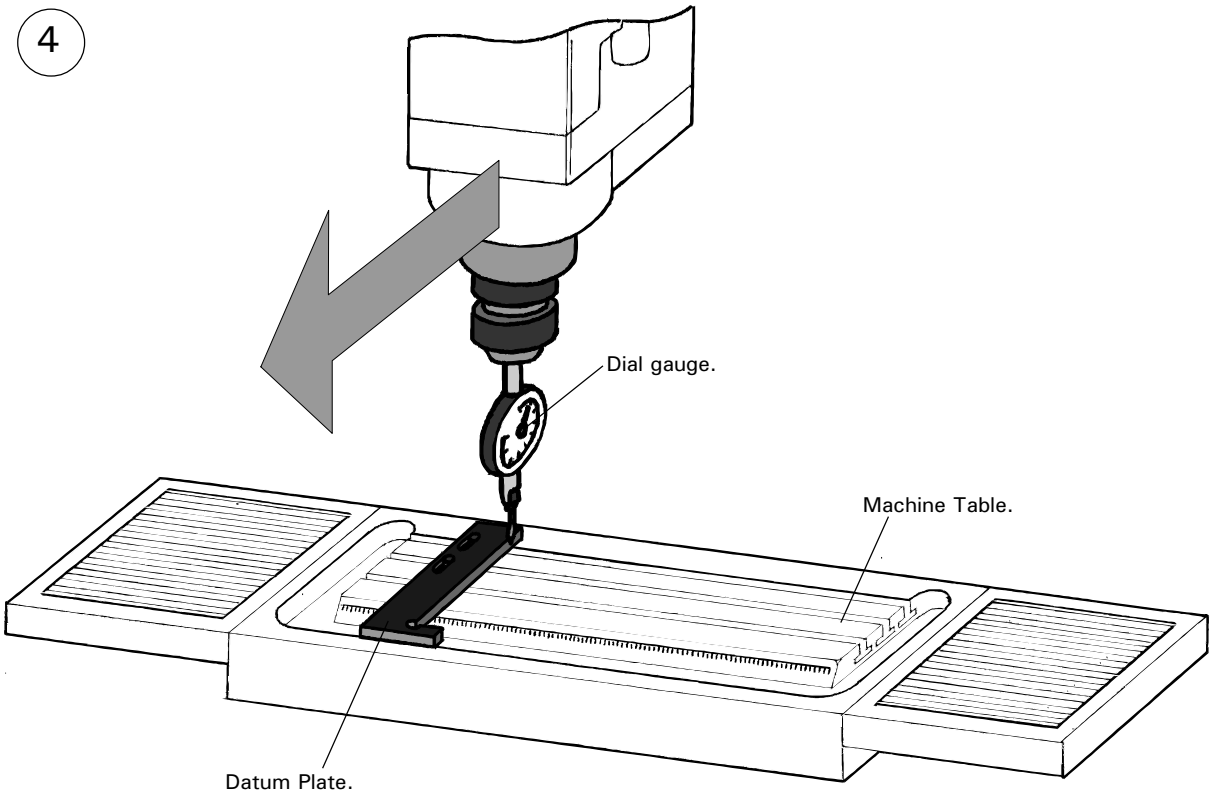
Be careful not to hit the edge of the datum plate with the pointer when manoeuvring it into position above the chosen edge.



Start with the pointer near the back of the datum plate edge you have chosen. Use the [-Y] key to move the pointer towards the front of the datum plate, checking that the tip of the pointer is still lined up exactly over the edge you have chosen. If the pointer does not line up, then manually adjust the position of the datum plate until it does. Keep repeating these steps, moving the pointer forwards and backwards, along the datum plate edge, until a suitable degree of accuracy has been obtained.

For a final check, the pointer can be moved above and along one of the datum plate edges which run parallel to the X axis. Finally, tighten the allen headed bolts to fix the datum plate firmly in place.

4

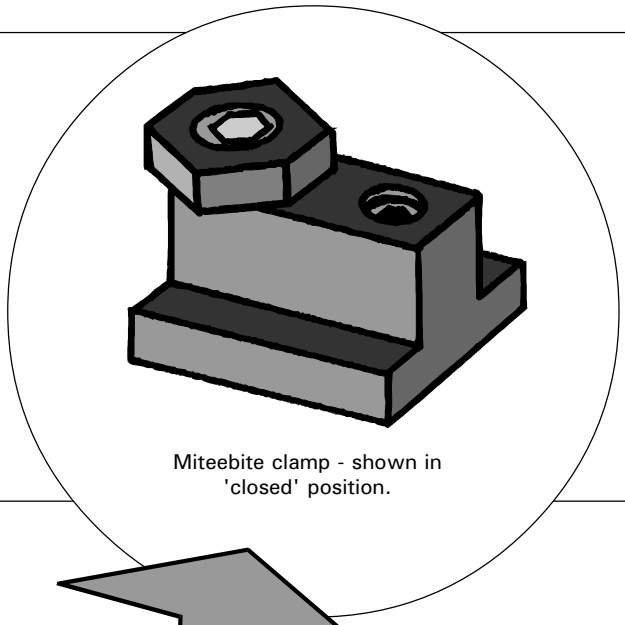


Set up the machine so a dial gauge is held in place of the cutting tool (the [T] key will change tools on an automatic tool changer). Align the dial gauge so it is positioned along one of the 2 sides of the datum plate, which run parallel with the Y axis. Use the [+X], [-X], [+Y], [-Y], [+Z] and [-Z] keys to move the dial gauge, with the *Jog Mode* set to read '*continuous*' on screen (this can be selected using the [JOG] key).

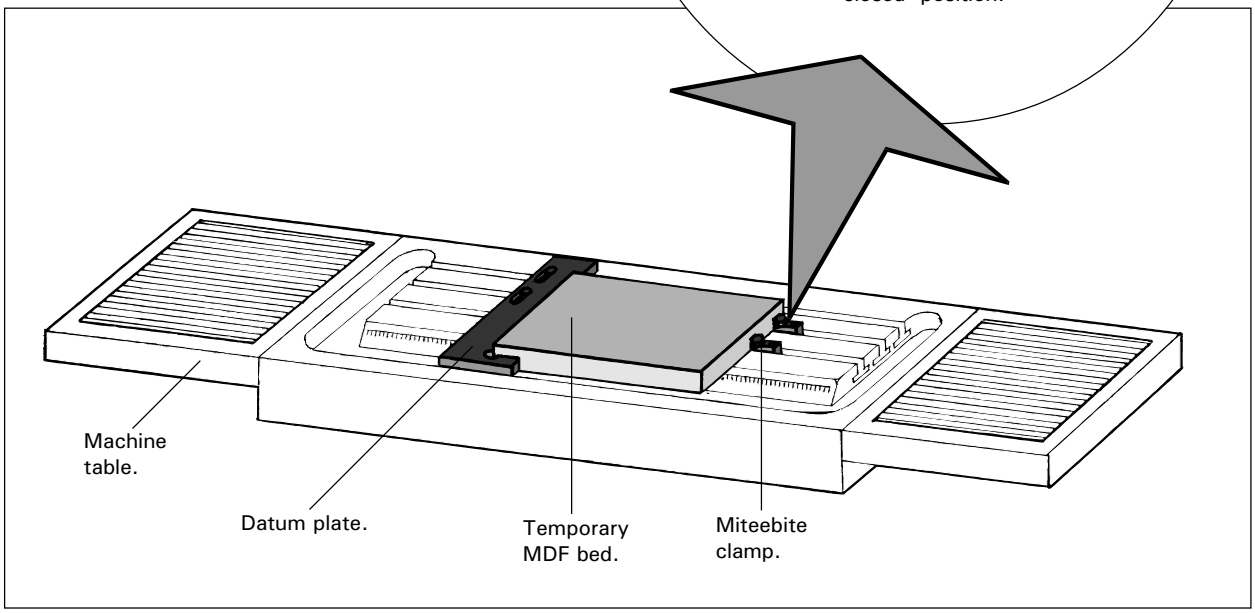
Start with the dial gauge near the back of the datum plate edge you have chosen. Use the [-Y] key to move the dial gauge towards the front of the datum plate, checking that the values indicated on the dial gauge do not alter. If the values do alter, then manually adjust the position of the datum plate until the values are constant. Keep repeating these steps, moving the dial gauge forwards and backwards along the datum plate edge, using the [+Y] and [-Y] keys, until a suitable degree of accuracy has been obtained. Finally, tighten the allen headed bolts to fix the datum plate firmly in place.

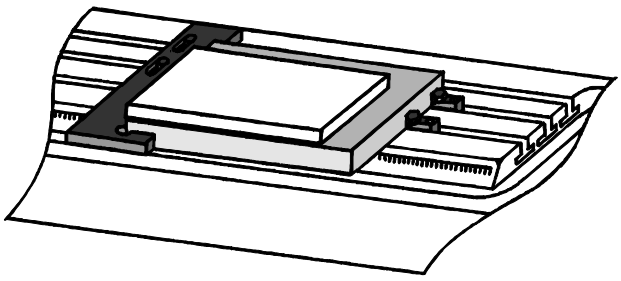
Clamping the Work Piece - Using Miteebite Clamps.

Miteebite clamps are a quick and versatile method of securing most pieces of work to the machine table. In the example shown below, a temporary MDF bed is used as a 'sub machine table'. This bed is clamped down and used as a safety measure to prevent damage occurring to the machine table itself, should a problem occur when milling.



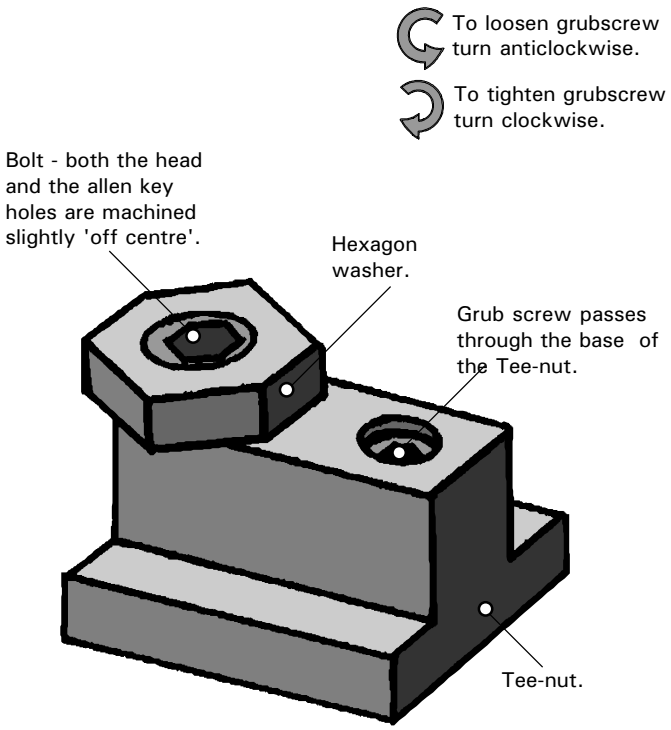
Miteebite clamp - shown in 'closed' position.





The actual workpiece, such as a sheet of plastic, would be held in place on the temporary MDF bed using double sided tape.

How does a Miteebite Clamp work?



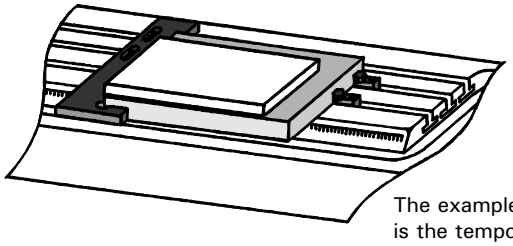
The base of the Miteebite clamp consists of a Tee-nut, with 2 threaded holes passing right through its section from top to bottom.

One of these threaded holes contains a grubscrew. When this is tightened, the base of the grubscrew pushes against the surface of the T channel in which it has been placed, thus securing the Tee-nut in position.

The other threaded hole contains a bolt which has its head and allen key hole machined slightly 'off centre'. A hexagon washer spins freely around this bolt head. The bolt behaves in a similar way to a cam when rotated. If the allen key hole is facing away from the grubscrew, then the hexagon washer is 'slack' against the work (i.e. the miteebite is 'open'). If the bolt is then turned through 180 degrees so that the allen key hole is now facing towards the grubscrew, then the hexagon washer will be 'tight' against the work (i.e. the miteebite is 'closed').

Continual turning of the bolt is unnecessary, since the full range of movement for the hexagon washer is covered in a single 360 degree rotation of the bolt. In this respect, the hexagon washer will not tighten further if the bolt is continually turned clockwise.

Clamping a piece of work using the Miteebite Clamps.



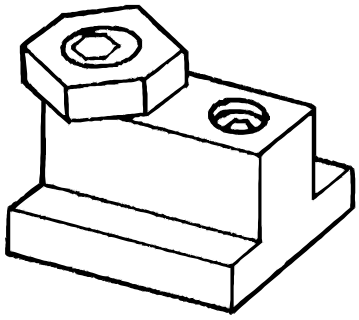
The example used in this section is the temporary MDF bed with a sheet of plastic retained using double sided tape.

Set the Datum Plate into position, as described in section 3.1, then place the workpiece onto the machine table, so it is located correctly against the datum plate.

Next, position the miteebites into their respective T channels and slide them along until they touch the temporary MDF bed. Ensure that it is one of the six flat sides of the hexagon washers which press against the workpiece, not one of their points.

Note that the hexagon washers should be positioned at this stage so they are 'open' (i.e. the off-centre allen key holes on the bolts should be facing away from the grubscrews) as shown in the illustration directly to the left.....

Hexagon washer set in open position.

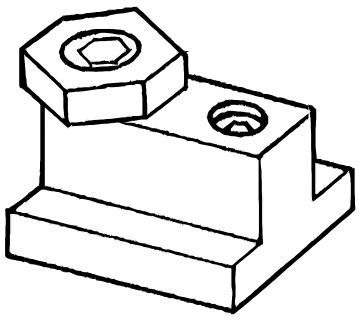


Now tighten the grubscrews in each miteebite to lock them firmly in position. At this stage, it should still be possible to remove the temporary MDF bed.

Remember, the grubscrews only lock the miteebites in position on the machine table - it is the hexagon washers which actually lock the workpiece in position.

To finally lock the workpiece firmly in place, turn the bolts with the off-centre allen key holes 180 degrees so the hexagon washers are in the 'closed' position (i.e. the off-centre allen key holes on the bolts should now be facing towards the grubscrews) as shown in the illustration directly to the left.....

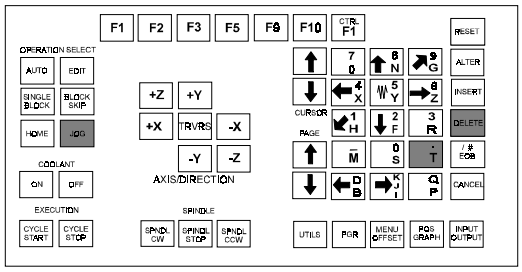
Hexagon washer set in closed position.



Now that the miteebites have been set, the temporary MDF bed can be continually withdrawn from the machine table, then replaced, always to the same position. This is an advantage for jobs involving the repeat milling of pieces of work, such as a small 'production run' or a 'college class/group project'.

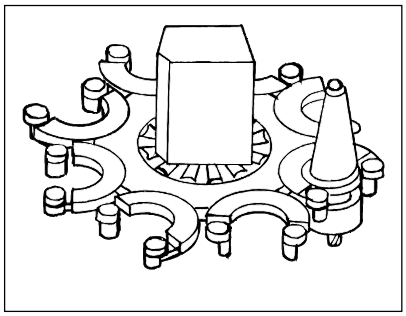
Changing the Tool with an Automatic Tool Changer.

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



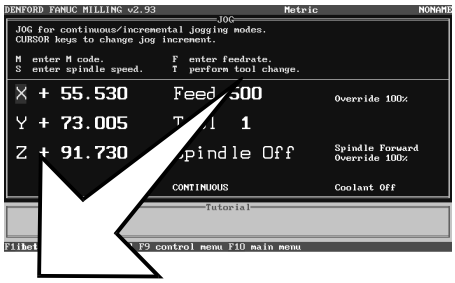
Tutor keypad.

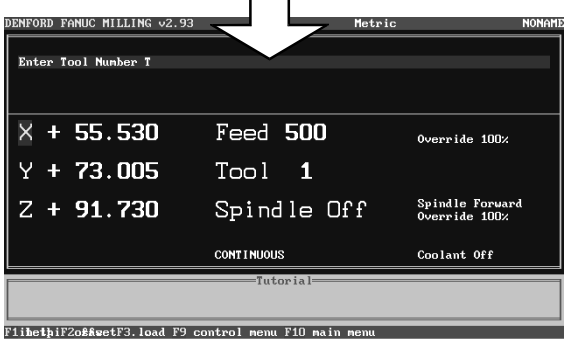
Note there should never be a cutting tool in the carousel pocket directly facing the spindle.



The automatic tool changer is used to transfer different tool holders between the head of the machine and the carousel (at the back of the machine) where they are stored when not in use. Each slot on the carousel is assigned a 'number'. The tool holders can also be referred to by these numbers, if the they are always kept in the same carousel slot, with the same tool profile present in the holder.

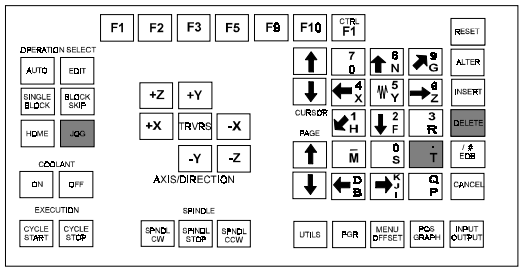
To change to a different tool number, first check the machine is set in *Jog Mode* (selected by pressing the [JOG] key). Next, press the [T] key to obtain the Tool number prompt, then type in the number of the tool you wish to change to, using the [NUMBER] keys. Any incorrect numbers typed in can be removed by pressing the [DELETE] key. Press the [EOB] key to confirm this tool number. The tool numbers will now be changed automatically on the screen as the carousel changes the tool holders.





CHANGING THE TOOL MANUALLY.

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



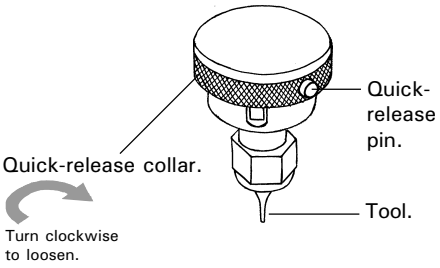
Tutor keypad.

Note that when a *program* using more than one tool is run in future, the software will prompt you to manually change tools before continuing.

If the machine is not equipped with an automatic tool changer, then each tool needs to be changed manually.

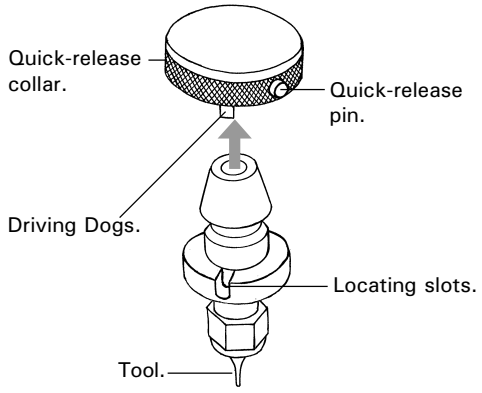
The software commands and prompts remain the same as changing the tool, automatically. Check the machine is set in *Jog Mode* (selected by pressing the [JOG] key), press the [T] key to obtain the Tool number screen prompt, then enter the number of the tool you wish to change to and press the [EOB] key to confirm this operation is correct.

Removing a Tool manually.



To physically remove a tool from the machine, grip the tool holder quick-release collar, so that the quick-release pin in it is fully depressed. Hold the tool itself still and rotate the quick-release collar clockwise until it stops. Remove the tool whilst keeping the quick-release pin still depressed - this prevents the quick-release mechanism from closing.

Refitting a Tool manually.



To refit, align the two locating slots of the tool with the two driving dogs on the quick-release collar. Push the tool up into the holder. The quick-release mechanism should now spring closed and grip the 'new' tool securely.

Finally, press the [EOB] key to confirm that the tool change is now complete.

Note - the [EOB] key needs to be pressed twice for manual tool changes, once to confirm the request for a tool change operation and a second time to confirm the tool holder has physically been changed.

Setting the Tool Offsets - Introduction.

Quite often, the workpiece is much smaller than the 'imaginary block' used in setting the machine datum point (see section 2.1). For example, a typical piece of work could be a small sheet of plastic fixed to a temporary MDF bed with double sided tape (to prevent damage occurring to the table when machining) as shown in the diagram below. Due to its size, the work is usually clamped somewhere in the middle of the table.

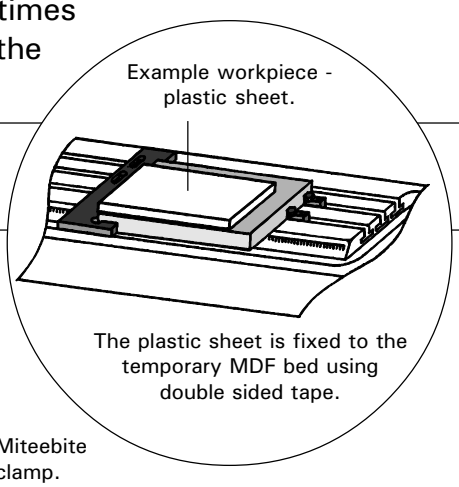
The miller, however, will still recognise its original machine datum point and take all its measurements from here. Therefore, we must move this machine datum point to coincide exactly with the 'new' starting co-ordinates of our piece of work - this 'new' datum is called the 'Work Datum point'. It must also take account of all the different types of tools we wish to use.

This operation consists of two stages :

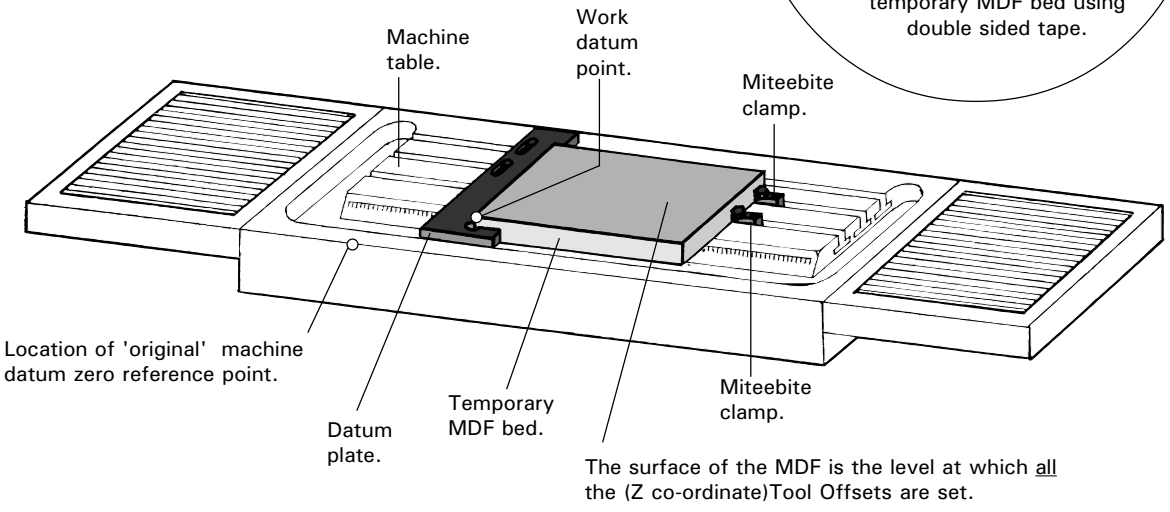
1) Programming a Work Datum point for the X and Y axes only.

2) Programming the Tool Offsets for the Z axis (this takes into account the different types of tools, since they all differ in length).

This operation is sometimes referred to as 'setting the tool offsets'.



Location of the Work Datum point and the level for Tool Offsets.

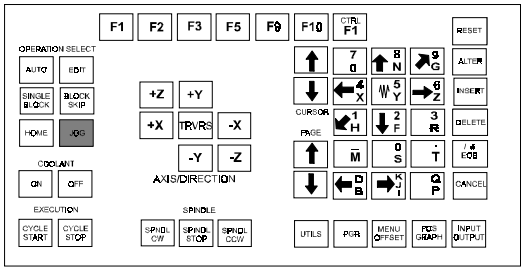


Setting the Work Datum X and Y Co-ordinates.

Instructions refer to setting up the X and Y co-ordinates for the example workpiece described on the previous page - 'Setting Tool Offsets - Introduction' (Section 4.3).

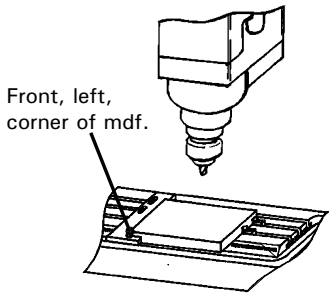
Warning - Before beginning to set the X and Y Offsets, ensure the X and Y values in the Tool Offsets Table are set to Ø (see page 4.17) .

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



Tutor keypad.

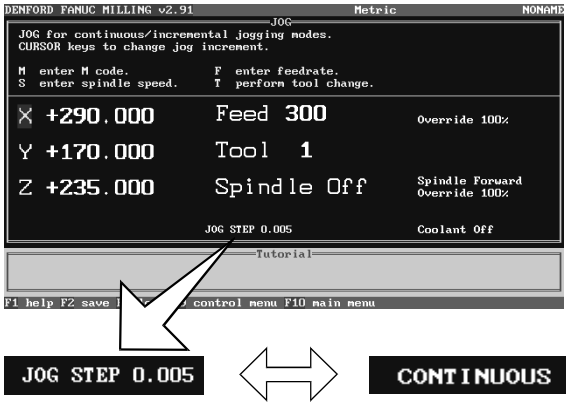
1.



To set the X and Y co-ordinates, the position of the front, left, corner of the temporary MDF bed needs to be found.

Select the *Jog Mode* by pressing the [JOG] key. When in *Jog Mode*, the miller can be controlled manually. The screen will display the X, Y and Z co-ordinates, the tool selected (this should be tool 1) and the spindle action (this should be off).

Before moving the table or head in any direction, you need to set the movement to '*continuous*' (ie, the table or head will move continuously, so long as one of the movement keys is being pressed). To do this, press the [JOG] key again until the screen shows the words '*continuous*' in the lower area of the display.

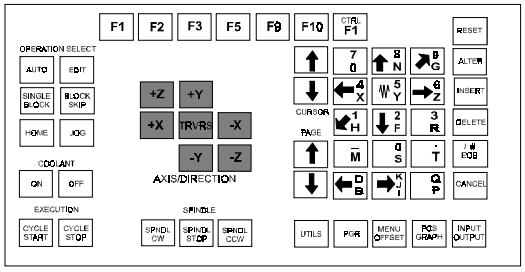


JOG STEP 0.005 ⇔ CONTINUOUS

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

SETTING THE WORK DATUM X AND Y CO-ORDINATES.

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

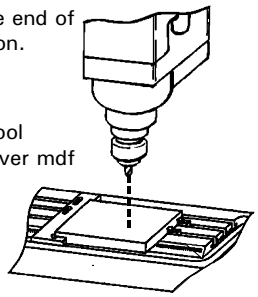


Tutor keypad.

2.

The machine should look like this at the end of this section.

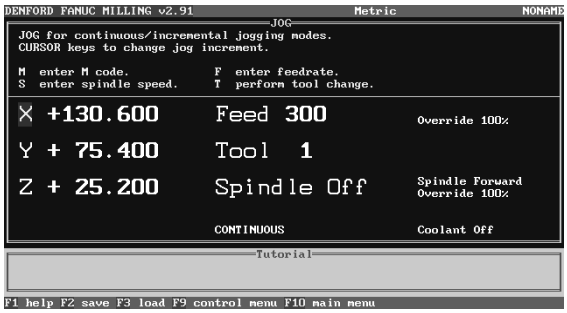
Cutting tool directly over mdf surface.



Using the blue [+X], [-X], [+Y], [-Y], [+Z] and [-Z] keys, move the cutter so it is positioned fairly close to anywhere on the MDFs upper surface, but <u>not</u> touching.

Pressing the [TRVRS.] key at the same time will increase this rate of movement.

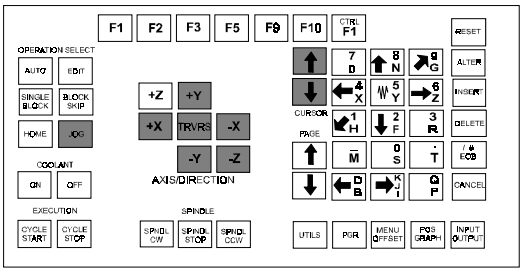
You should notice that as the table or head are moved, the X, Y or Z co-ordinates displayed on the screen will change. These numbers are the co-ordinates of the cutting tool tip, relative to its previously set datum point.



[+X] - Moves the table left.
[-X] - Moves the table right.
[+Y] - Moves the table towards the front.
[-Y] - Moves the table towards the back.
[+Z] - Moves the cutter up.
[-Z] - Moves the cutter down.

Setting the Work Datum X and Y Co-ordinates.

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

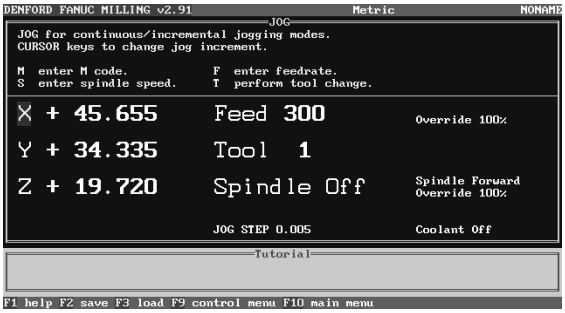


Tutor keypad.

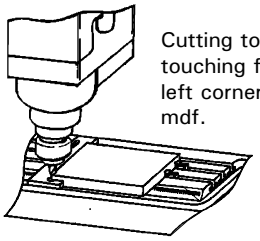
3.

Next, move the table so that the centre of the cutter is positioned exactly over the front, left corner of the temporary MDF bed. Use the blue [+X] , [-X] , [+Y], [-Y] and [-Z] keys to do this.

continued....



The machine should look like this at the end of this section.



Cutting tool touching front, left corner of mdf.

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

SETTING THE WORK DATUM X AND Y CO-ORDINATES.

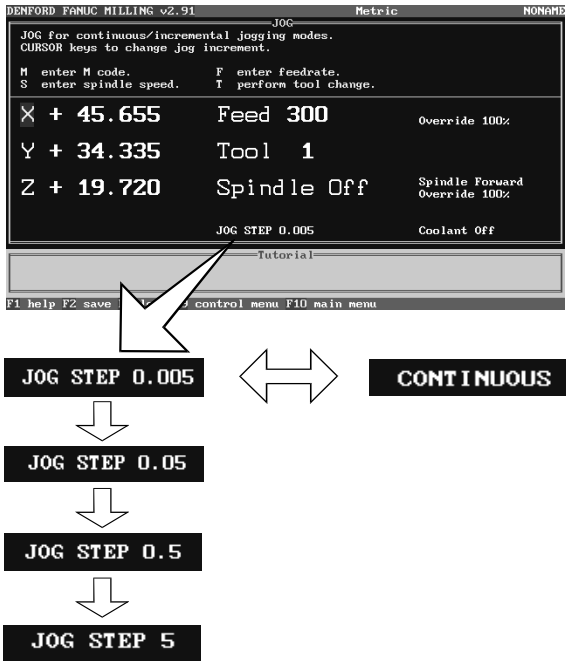
3.

continued....

When you are fairly close to this point the [JOG] key can be used to switch from '*continuous*' to a more accurate degree of adjustment using stepped movements (ie, the table or head will move a set amount each time you press a movement key).

The size of movement will be displayed, for example '*JOG STEP 0.005*' is a movement of 0.005mm every time you press a movement key.

Depending on the accuracy of the movement required, this can be selected using the blue [CURSOR ARROWS] keys. Jog steps can be selected between a movement of 0.005mm minimum to 5mm maximum.



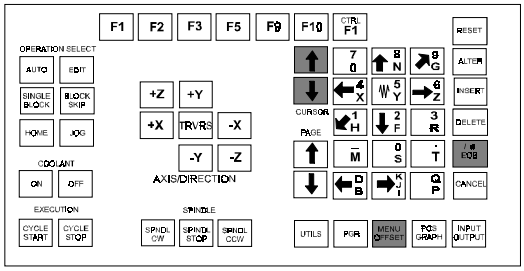
When fine adjustments are being made, open the machine guard to check the accuracy of the cutter position.

Note - as a safety feature the machine will not operate in *Jog Mode continuous* with the guard open, only in the last setting for *Jog steps*.

[+X] - Moves the table left.
[-X] - Moves the table right.
[+Y] - Moves the table towards the front.
[-Y] - Moves the table towards the back.
[+Z] - Moves the cutter up.
[-Z] - Moves the cutter down.

Setting the Work Datum X and Y Co-ordinates.

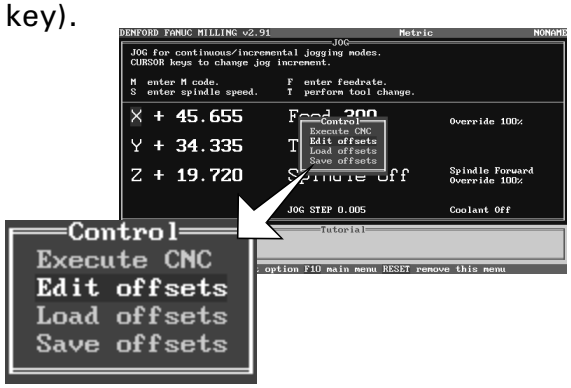
Keys Helpbox.
The following keys are used in this section:
[MENU OFFSET]
[CURSOR ARROWS]
[EOB]



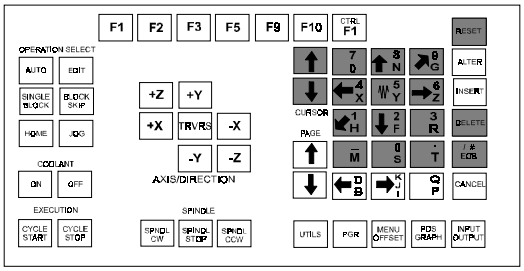
Tutor keypad.

4.

Select the *Offset Control Menu* by pressing the [MENU OFFSET] key and highlight the '*Edit offsets*' option using the blue [CURSOR ARROWS] keys. Select this command by pressing the [EOB] key (this is the End Of Block, or enter key).



Keys Helpbox.
The following keys are used in this section:
[NUMBERS], [X], [Y]
[CURSOR ARROWS]
[EOB], [RESET], [DELETE]



Tutor keypad.

5.

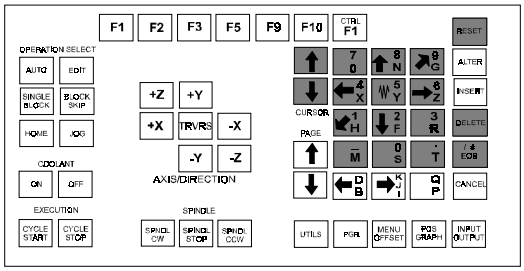
From the list of Offsets shown, highlight the '*X co-ordinate offset*' using the blue [CURSOR ARROWS] keys. To enter the new X value, press the [X] key to move the screen cursor across to where it reads '*0*'. Type in the new value using the [NUMBERS] keys; use the [DELETE] key to remove any incorrect characters.

continued....

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

Setting the Work Datum X and Y Co-ordinates.

Keys Helpbox.
The following keys are used in this section:
[NUMBERS], [X], [Y]
[CURSOR ARROWS]
[EOB], [RESET], [DELETE]



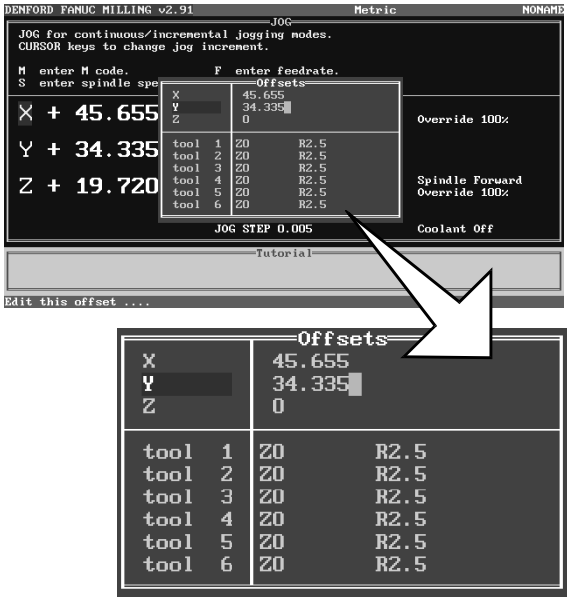
Tutor keypad.

5.

Please note -
Clear (zero) any previously stored offsets, before entering any new values.

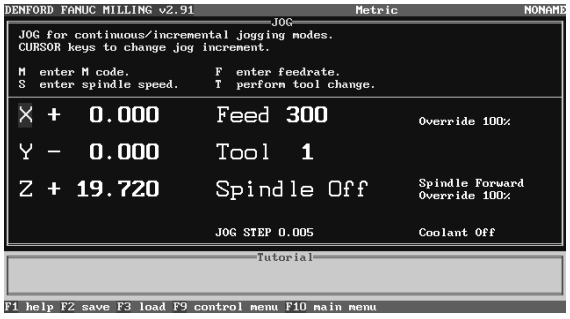
Press the [EOB] key to confirm this new figure. The X value you want to copy is displayed to the left of the offset screen.

Repeat the same steps for the '*Y co-ordinate offset*' (pressing the [Y] key to move the screen cursor across to where the Y value reads '*0*').



Please note -
Tool Radius values (*R*) are only required for CNC Files using cutter compensation codes (G41 and G42). They do not require setting for a machine using CNC Files generated with MillCAM Designer.

To check if this has registered correctly, press the [RESET] key and look at the main screen to see if the X and Y co-ordinates have returned to zero (ie, the cutter recognises this point as its new datum for X and Y).



[+X] - Moves the table left.
[-X] - Moves the table right.
[+Y] - Moves the table towards the front.
[-Y] - Moves the table towards the back.
[+Z] - Moves the cutter up.
[-Z] - Moves the cutter down.

Setting the Tool Offset Z Co-ordinates.

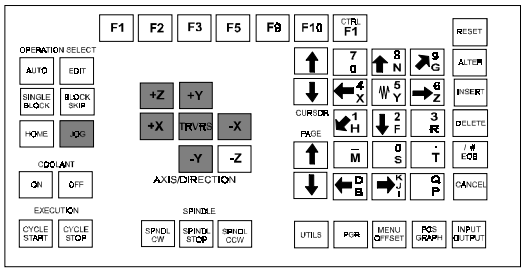
Instructions refer to setting up the Z co-ordinate for the example workpiece described in section 4.3 - 'Setting Tool Offsets - Introduction'.

To set the Tool Offset Z co-ordinate, the cutter needs to be placed so it is just touching the surface of the temporary MDF bed.

When this new Z figure is combined with the X and Y figures previously entered, the co-ordinates found will be our new datum point but only for the tool that is currently selected (at the moment this should be tool 1). This process must be repeated for each tool we wish to use on the workpiece.

Warning - Before beginning to set the Z Offsets, ensure the Z values in the Tool Offsets Table are set to Ø (see page 4.17) .

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



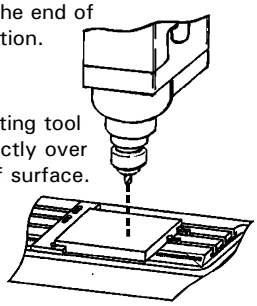
Tutor keypad.

1.

Select the *Jog Mode* by pressing the [JOG] key and check that it is set to '*continuous*'. Before moving the table in any direction, raise the head using the [+Z] key so that it is comfortably above any surfaces it could hit when moving across.

Now move the table so that the cutter is positioned somewhere over the surface of the temporary MDF bed using the [+X] , [-X] , [+Y] and [-Y] keys.

The machine should look like this at the end of this section.

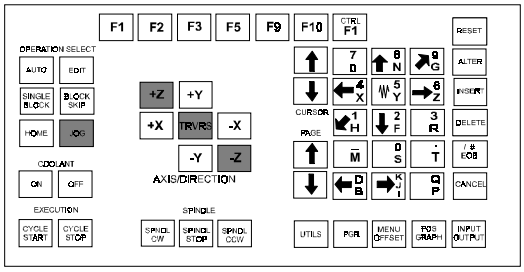


Cutting tool directly over mdf surface.

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

Setting the Tool Offset Z Co-ordinates.

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



Tutor keypad.

2.

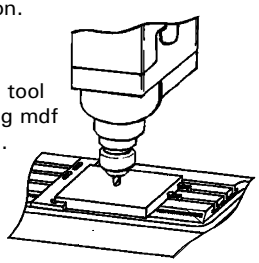
Using the [+Z] and [-Z] keys, make the tip of the cutter just touch the top surface of the bed.

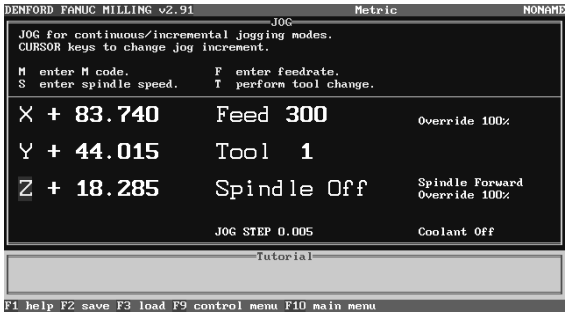
Remember that the *Jog Mode* can be changed to stepped movements for greater accuracy, once the tip is quite close to the surface.

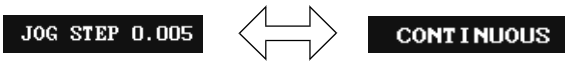
As the cutter approaches the surface of the bed, spin it slowly by hand to help check when it makes contact.

The machine should look like this at the end of this section.

Cutting tool touching mdf surface.





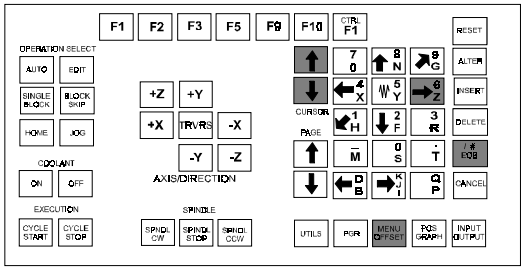


Note - as a safety feature the machine will not operate in *Jog Mode continuous* with the guard open, only in the last setting for *Jog steps*.

[+X] - Moves the table left.
[-X] - Moves the table right.
[+Y] - Moves the table towards the front.
[-Y] - Moves the table towards the back.
[+Z] - Moves the cutter up.
[-Z] - Moves the cutter down.

Setting the Tool Offset Z Co-ordinates.

Keys Helpbox.
The following keys are used in this section:
[MENU OFFSET]
[Z], [CURSOR ARROWS]
[EOB]



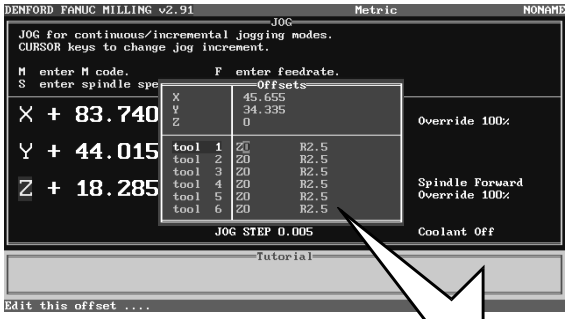
Tutor keypad.

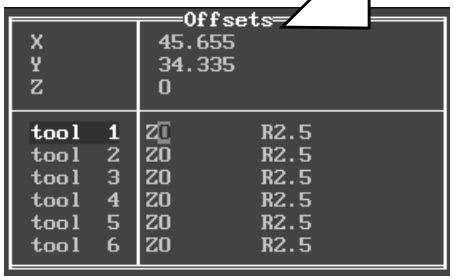
3.

Please note -
Clear (zero) any previously stored offsets, before entering any new values.

Next, select the *Offset Control Menu* by pressing the [MENU OFFSET] key, highlight the '*Edit offsets*' option and select it by pressing the [EOB] key. From the list of Offsets shown, highlight the '*tool 1 offset*' using the [CURSOR ARROWS] keys (see screenshot below).

Note - do <u>not</u> highlight the '*Z co-ordinate offset*' by mistake! To move the screen cursor across to where it reads '*Z0*', press the [Z] key.

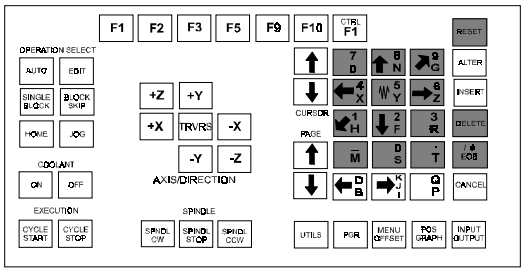




Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

Setting the Tool Offset Z Co-ordinates.

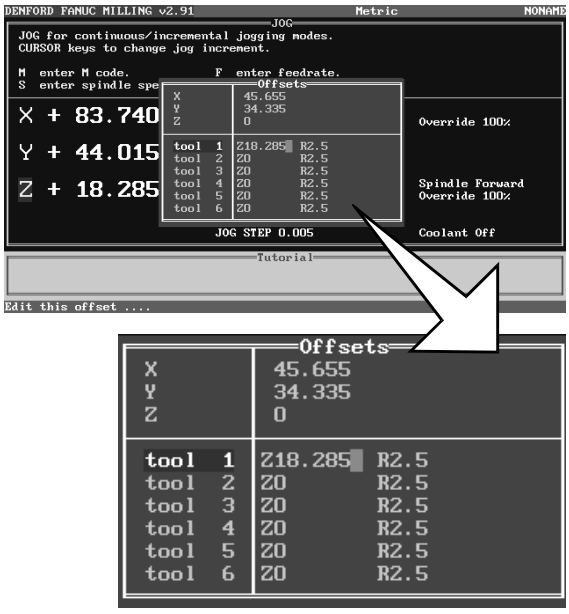
Keys Helpbox.
The following keys are used in this section:
[NUMBERS]
[EOB]
[RESET], [DELETE]



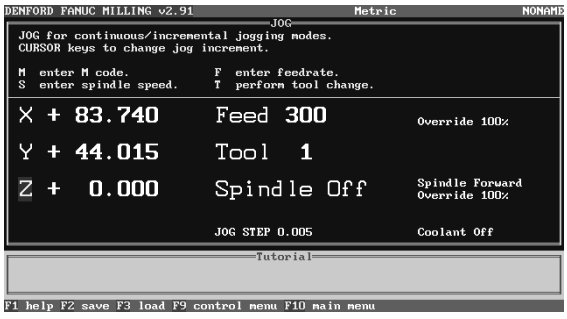
Tutor keypad.

4.

Enter the new Z value, using the [NUMBERS] keys; use the [DELETE] key to remove any incorrect characters. The Z value you want to copy is displayed to the left of the offset screen. Press the [EOB] key to confirm this value.



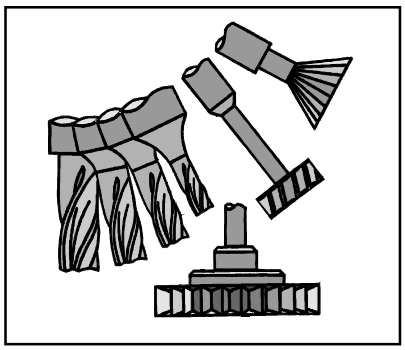
To check if this has registered correctly, press the [RESET] key and look at the main screen to see if the Z co-ordinate has returned to zero (ie, the cutter recognises this point as its tool offset for Z, when using tool 1).



Please note -
Tool Radius values (*R*) are only required for CNC Files using cutter compensation codes (G41 and G42). They do not require setting for a machine using CNC Files generated with MillCAM Designer.

[+X] - Moves the table left.
[-X] - Moves the table right.
[+Y] - Moves the table towards the front.
[-Y] - Moves the table towards the back.
[+Z] - Moves the cutter up.
[-Z] - Moves the cutter down.

5.



Tool Selections.

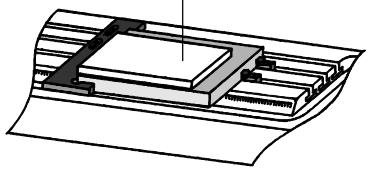
Each cutting tool is a different length, so the Z tool offset co-ordinates you have entered for tool 1 will not necessarily be correct when using other tool numbers.

Therefore, the 'Z co-ordinate Tool Offsets' stage must be carried out and the values entered, for each tool you wish to use when machining.

For example, if you want to use tool 3, you would have to change from your current tool (number 1) to tool 3. Then, you would position this cutter so it touched the surface of the bed again, select *'Edit offsets'* and transfer the values for this cutter into the tool 3 offset menu.

Setting a 'Global' Z value for work material thickness.

Example workpiece - plastic sheet.



The plastic sheet is fixed to the temporary MDF bed using double sided tape.

Now that the Tool Offsets have been set to coincide with the surface of the temporary MDF bed, the workpiece is fixed into position (in this example it is a sheet of plastic - refer to the diagram opposite). If the miller starts cutting it will take all its Z co-ordinate measurements from the surface of the temporary MDF bed (i.e. the Z Tool Offsets).

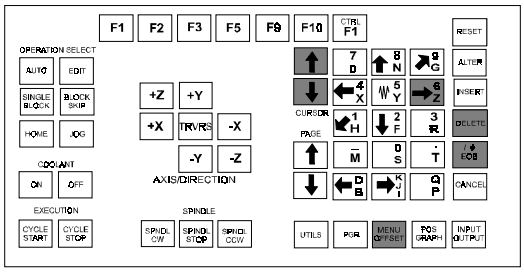
Due to the way the programs are written, we require this Z co-ordinate measurement to be taken from the surface of the work itself, otherwise the miller will start cutting into the temporary MDF bed and not our workpiece.

Each Z co-ordinate for each tool must therefore be moved up, the distance moved up equating to the thickness of our work material. Rather than program this value individually for each tool as desribed when setting the Z co-ordinate Tool Offsets, we can program a 'global' Z value which will be applied to each tool automatically.

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

SETTING A 'GLOBAL' Z VALUE FOR WORK MATERIAL THICKNESS.

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



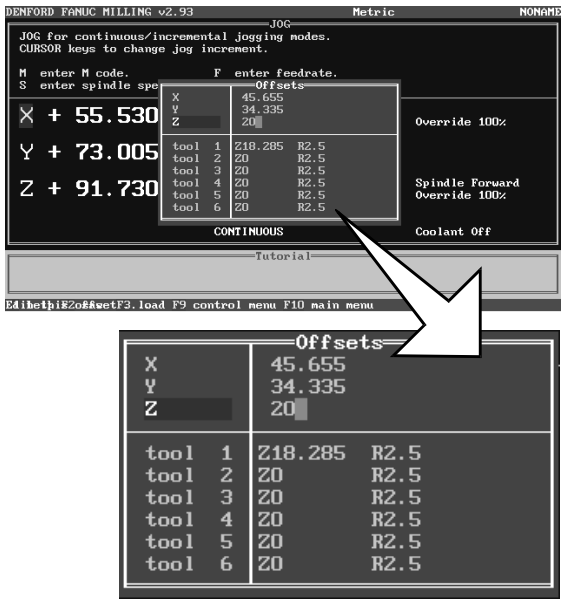
Tutor keypad.

Select the *Offset Control Menu* by pressing the [MENU OFFSET] key, highlight the '*Edit Offsets*' option using the blue [CURSOR ARROWS] keys and press the [EOB] key.

From the list of offsets shown highlight the '*Z*' co-ordinate (<u>not</u> any of the Tool number co-ordinates!) and press the [Z] key to move the cursor across to where it reads '*0*'.

Enter the 'global' Z value - this should be the thickness of your work material. Use the [DELETE] key to remove any incorrect characters.

Finally, press [EOB] to confirm this 'global' Z value.

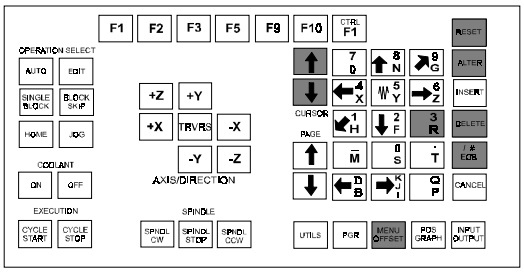


To check if this has registered correctly, press the [RESET] key and look at the main screen, the Z co-ordinate value will reduce by the value entered into the global Z offset.

[+X] - Moves the table left.
[-X] - Moves the table right.
[+Y] - Moves the table towards the front.
[-Y] - Moves the table towards the back.
[+Z] - Moves the cutter up.
[-Z] - Moves the cutter down.

Tool Offsets - Setting the Tool Radius value.

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



Tutor keypad.

The tool radius, '*R*',may be typed into the collection of tool offsets. The radius or diameter value is usually stamped on the cutting tool.

The software will assign a default value of 2.5 to a tool radius.

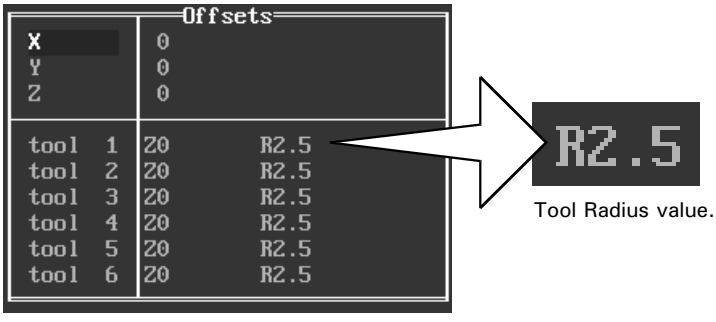
Tool radius values are required by programs that use the G41 (Cutter Compensation Left) and G42 (Cutter Compensation Right) codes.

They do not need to be set for any CNC Files generated using Denford MillCAM Designer.

To alter the tool radius values, press the [MENU OFFSET] key and use the [CURSOR] keys to highlight the '*Edit Offsets*' option. Press the [EOB] key to display the Table of Tool Offsets.

Use the [CURSOR] keys to highlight the tool number required, then press the [R] key to switch the cursor to the '*R*' value. Type the tool radius value into the table using the [NUMBER] keys (any incorrect characters can be removed by pressing the [DELETE] key, or use the [ALTER] key to remove the existing values), then press the [EOB] key.

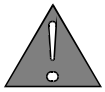
Press the [RESET] key to remove the tool offsets table and any unwanted menus from the screen.



Tool Radius value.

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

Tool Offsets - Warning.



Warning - Check the machine does not contain any previous offset values before beginning to set your own !!

If you use a machine which has been left switched on, or uses default offsets loaded from the batch files, check that no other offset values are present (ie, the values in the Tool Offset Table should all read zero) before beginning the process of setting your own offsets.

The X, Y and Z co-ordinates on the main screen display will include any offsets already present on the machine in their values. If offsets are already present and you begin the process of setting the offsets for a new piece of work, these "old" offset values need to be taken into account when entering "new" values from the main screen display into the Tool Offset Table. This is important since any CNC Files run with incorrect offset values may result in the machine sending the cutter to the wrong area of your work, or into the machine table itself !!

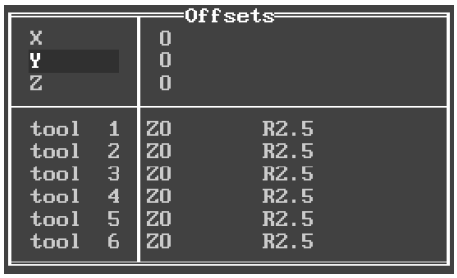
The easiest method of dealing with this problem is to save the "old" offset values (if required) and then reset all the values in the Tool Offset Table back to zero.

continued....

Tool Offsets - Warning.

If you suspect that a set of offsets may already be present on the machine, follow this procedure :

Press the [MENU OFFSET] key and select '*Edit Offsets*' followed by [EOB] to display the Tool Offsets Table.. All the numerical values present in the '*X,Y,Z or Tool No.*' sections of the display must read ZERO. If they do not, then change each one.... entering values into the Tool Offset Table is covered in an earlier part of this section. The default Tool Offset Table screen is shown below.

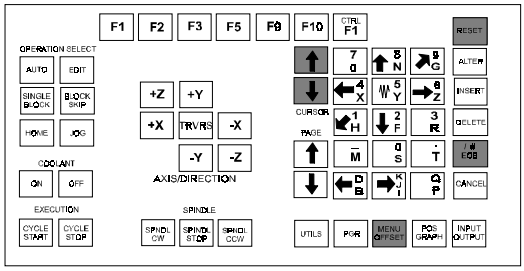


When all the values have been set to ZERO, press the [RESET] key. Notice that the main screen co-ordinates will change indicating that the new offsets (Ø,Ø,Ø) have been applied. Your own set of "new" work offsets can now be manually entered, safely.

If you regularly use a machine which others have access to, it may save time if you save your own "default" offset file with all the co-ordinates set at ZERO.

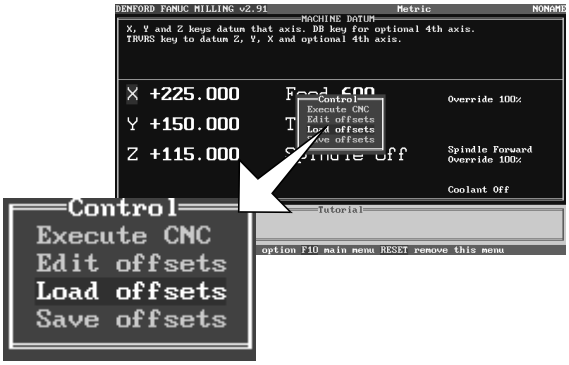
OFFSET FILES
- CONTROL MENU.

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



Tutor keypad.

Select the *Offset Control Menu* by pressing the [MENU OFFSET] key.



The *Offset Control Menu* contains 4 options:

i) *Execute CNC* - run the currently loaded CNC File on the machine.

ii) *Edit Offsets* - the Tool Offset Table is displayed, from which individual Offset values can be altered.

iii) *Load Offsets* - load a collection of Tool Offsets.

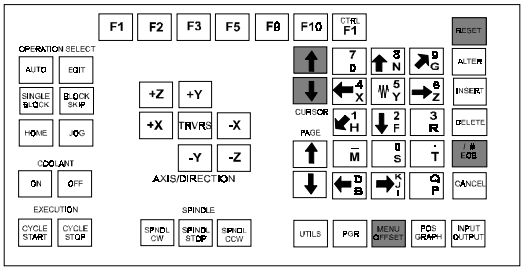
iv) *Save Offsets* - save a collection of Tool Offsets.

Press the [RESET] key to clear any menus.

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

OFFSET FILES - EXECUTE CNC.

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

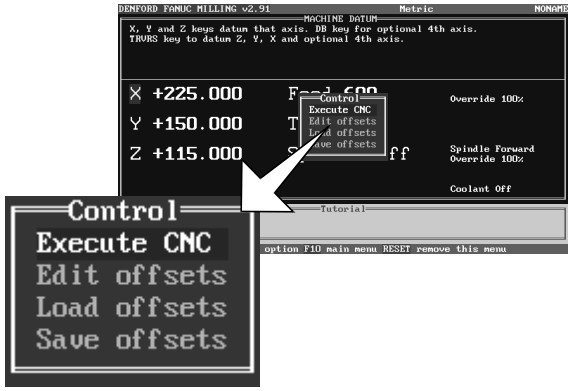


Tutor keypad.

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 [MENU OFFSET] key.

From the four choices, highlight the *'Execute CNC'* option using the blue [CURSOR ARROWS] keys and press [EOB].



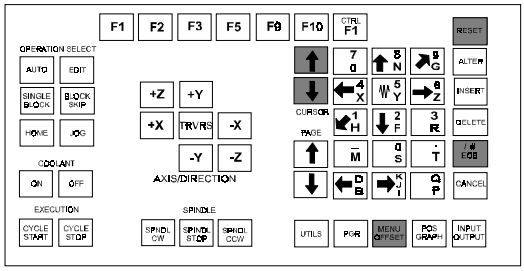
The machine will switch to *Auto Mode* and will allow the user to run the currently loaded CNC File.

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.

Offset Files
- Edit Offsets.

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

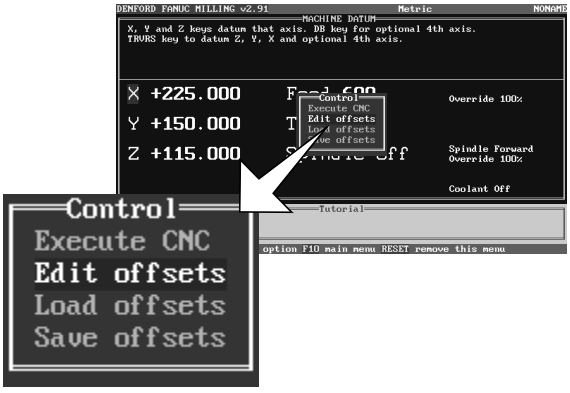


Tutor keypad.

The '*Edit Offsets*' command will display the Tool Offsets Table.

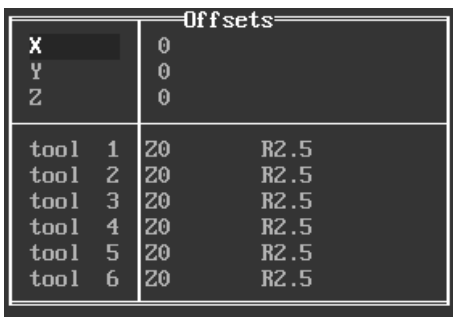
To edit or view the current collection of tool offset values, select the *Offset Control Menu* by pressing the [MENU OFFSET] key.

From the four choices, highlight the *'Edit Offsets'* option using the blue [CURSOR ARROWS] keys and press [EOB].



The Tool Offset values are displayed; to edit their values, see section 4.3 - "Setting Tool Offsets".

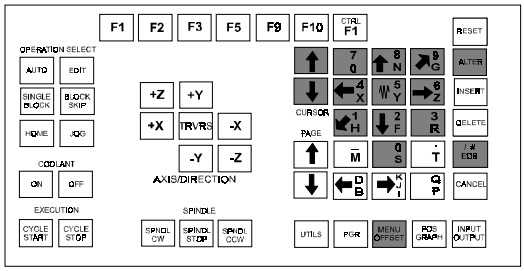
The default Tool Offset Table (ie, no offsets are entered) is shown below.



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

Offset Files - Load.

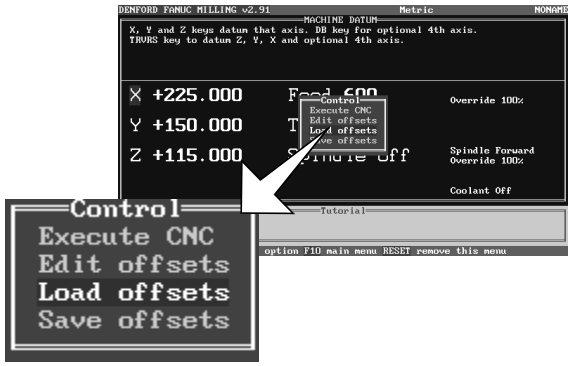
Keys Helpbox.
The following keys are used in this section:
[MENU OFFSET]
[EOB], [ALTER]
[CURSOR ARROWS]
[NUMBERS]



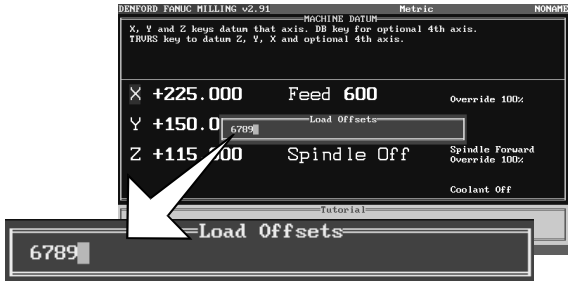
Tutor keypad.

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 offset files from the default drive, then the drive destination must be changed. See page 4.25 "Changing the Drive Directory - Offset Files".

To load a collection of offset values, select the *Offset Control Menu* by pressing the [MENU OFFSET] key. From the four choices, highlight the *'Load offsets'* option using the blue [CURSOR ARROWS] keys and press [EOB].



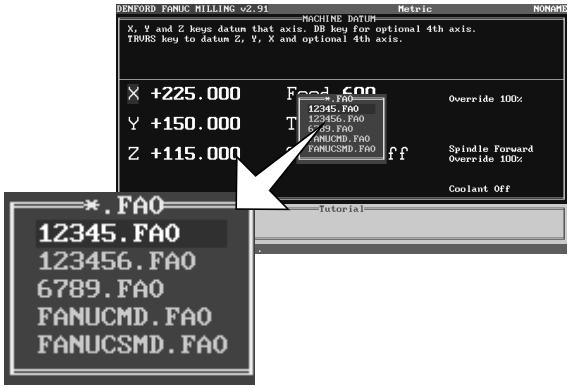
Enter the file name (number) for the offsets using the [NUMBER] keys and confirm this by pressing the [EOB] key.



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

OFFSET FILES - LOAD.

If a previous, or incorrect, filename (number) for the offsets is displayed, press the [ALTER] key to clear it. If the filename 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 [EOB].



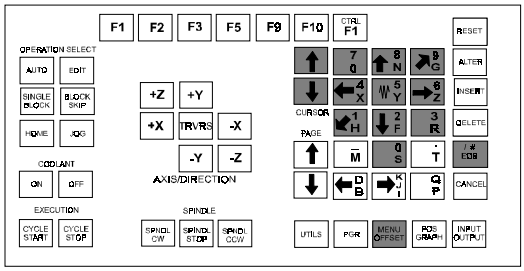
The selected collection of Tool Offsets will now be loaded.

To edit or view the individual offset values, select the '*Edit Offsets*' command in the *Offset Control Menu*. This will display the Tool Offsets Table.

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

Offset Files - Save.

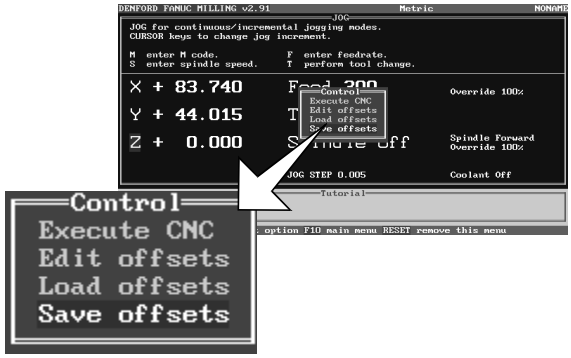
Keys Helpbox.
The following keys are used in this section:
[MENU OFFSET]
[EOB]
[CURSOR ARROWS]
[NUMBERS]



Tutor keypad.

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 offset files to the default drive, then the drive destination must be changed. See page 4.25 "Changing the Drive Directory - Offset Files".

To save a selection of offset values, select the *Offset Control Menu* by pressing the [MENU OFFSET] key. From the four choices, highlight the *'Save offsets'* option using the blue [CURSOR ARROWS] keys and press [EOB].



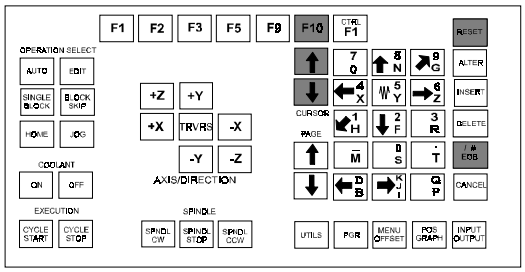
Enter a number (representing the file name for the offsets) using the [NUMBER] keys and confirm this by pressing the [EOB] key. Note that the file will be saved on the currently selected drive. Make a note of the name of your new offset file for future reference.

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

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

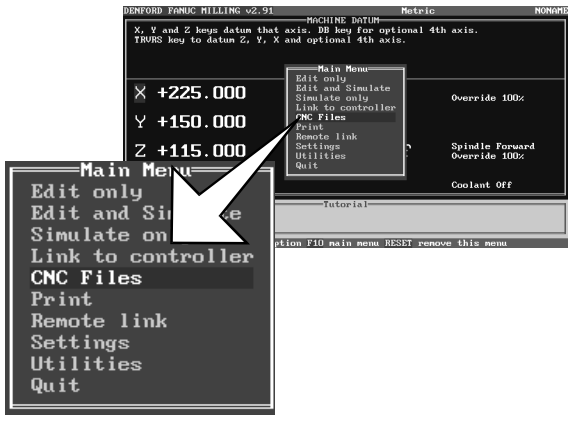
Changing the Drive Directory - Offset Files.

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

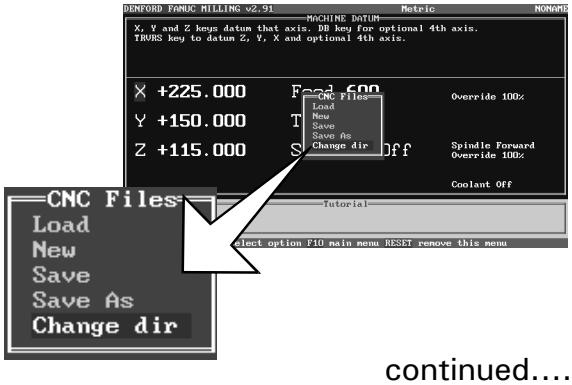


Tutor keypad.

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.

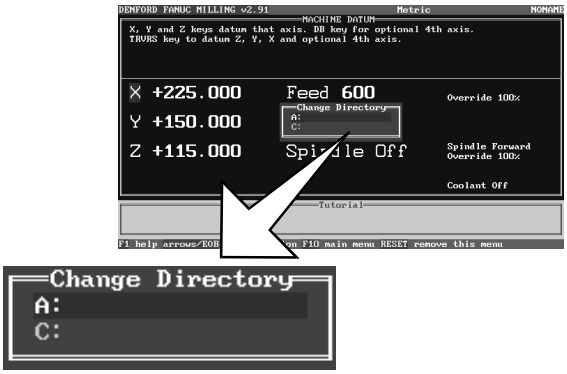


continued....

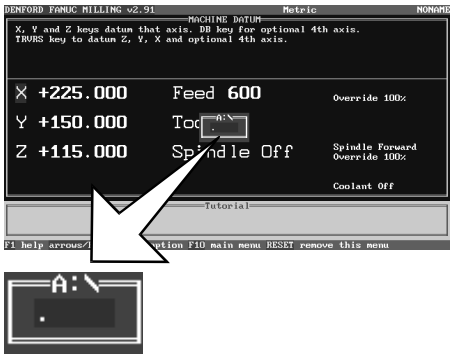
Changing the Drive Directory - Offset Files.

continued....

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



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

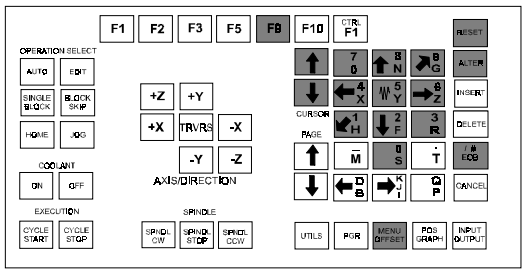


Press the [RESET] key to clear the screen of any unwanted menus or information.

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

Saving Offset Files on a Changed Drive.

Keys Helpbox.
The following keys are used in this section:
[F9], [MENU OFFSET]
[CURSOR ARROWS], [EOB], [ALTER]
[NUMBERS], [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 or the [MENU OFFSET] key. Highlight the '*Save Offsets*' option 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 the [EOB] key to confirm this. The offset file (.fao files) will now be saved on the new drive.

Please Note -
CNC Files are stored in ".fnc" format.
Offset Files are stored in ".fao" format.

Loading 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 the '*Load Offsets*' option 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 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 the [EOB] key to confirm this. The offset file (.fao files) will now be loaded from the new drive.

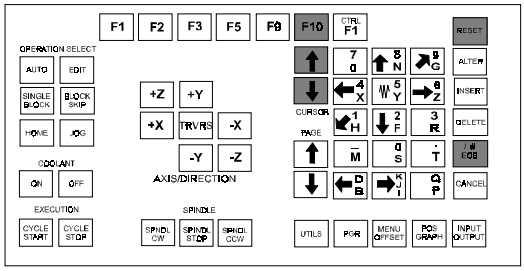
If the filename is unknown, the list of files stored on the drive can be accessed by pressing the [EOB] key.

The message window below 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" format.
Offset Files are stored in ".fao" format.

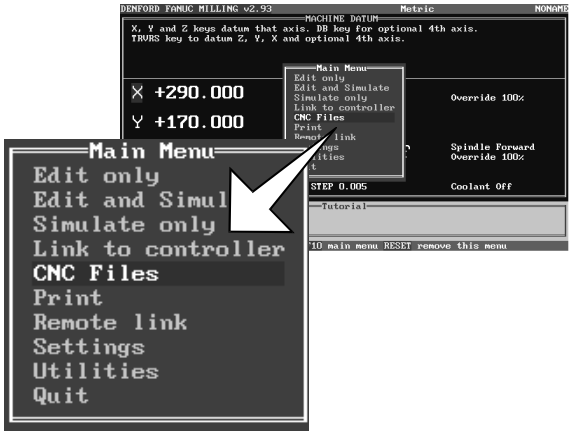
CNC Files - Menu.

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 options:

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.

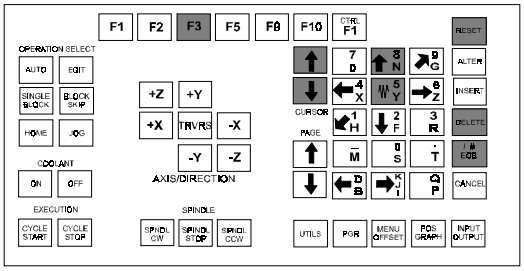
v) *Change dir* - change the drive and/or directory used to load and save CNC (or Offset) Files.

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 - Load.

Keys Helpbox.
The following keys are used in this section:
[CURSOR ARROWS], [EOB], [F3]
[NUMBERS] - not highlighted
[DELETE], [N], [Y], [RESET]



Tutor keypad.

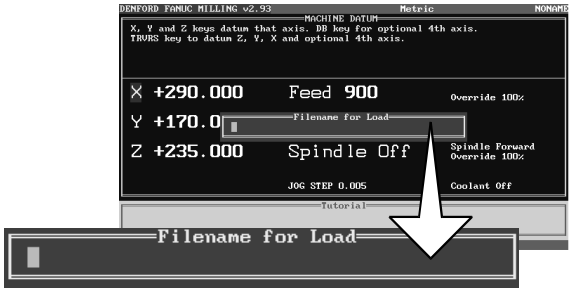
Please Note - your software will be set to read either the computers hard drive (usually C:) or the floppy disk drive (usually A:) by default. If you do not want to load the CNC Files from the default drive, then the drive destination must be changed. See page 5.7 "Changing the Drive Directory - CNC Files".

Highlight '*Load*' in the *CNC Files Menu* using the [CURSOR ARROWS] keys and press the [EOB] key.

Note, the [F3] key can be used as a short-cut to load CNC Files from the default drive in most areas of the control software.

Type in the name of the 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.

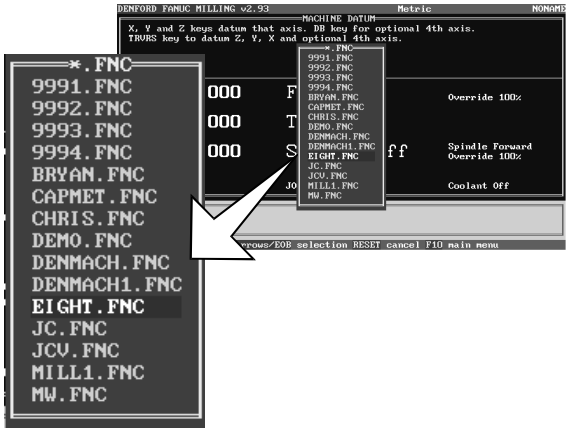
If the filename contains any alphabet characters, it can only be loaded from a directory listing (see the next page).



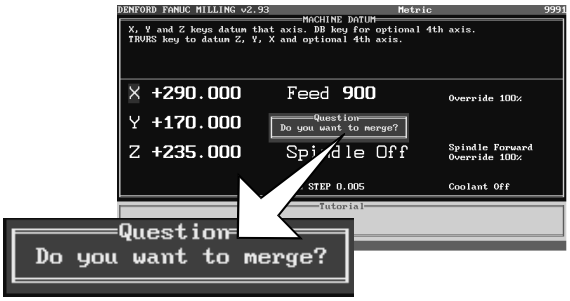
Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

CNC Files - Load.

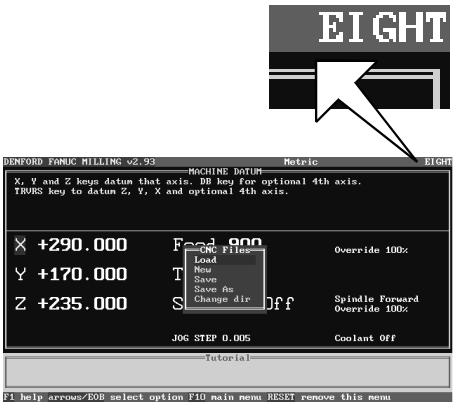
If the filename is unknown, press the [EOB] key to list all the files stored on the currently selected drive. Files within these lists can be loaded by highlighting them using the [CURSOR ARROWS] keys and pressing the [EOB] key.



If there is a CNC File currently in the editor, 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 [Y] key, otherwise, press the [N] key to clear the current CNC File and load the selected CNC File into the editor.



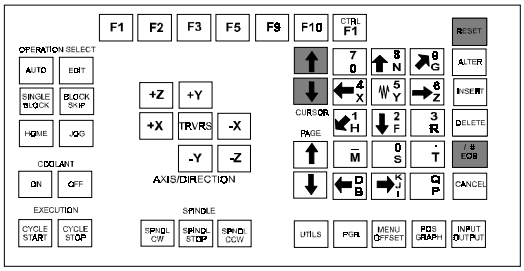
The name of the CNC File you have just loaded is displayed in the top right hand corner of the screen.



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

CNC Files - New.

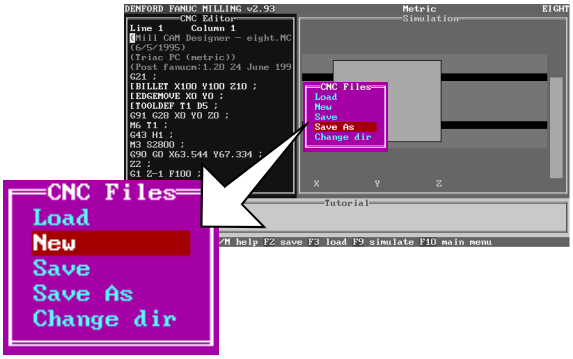
Keys Helpbox.
The following keys are used in this section:
[CURSOR ARROWS]
[EOB]
[RESET]



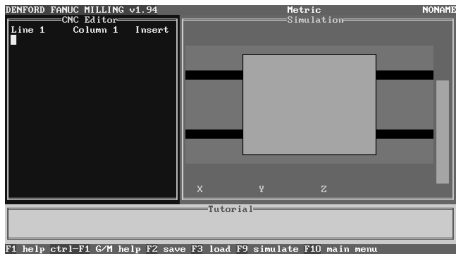
Tutor keypad.

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.

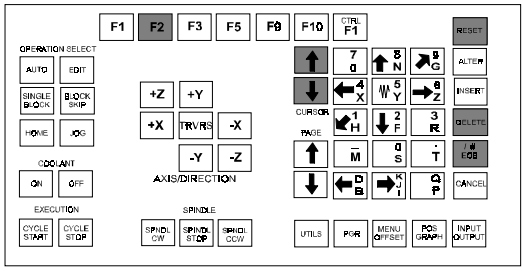


Press the [RESET] key to clear the screen of any unwanted menus or information.

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

CNC Files - Save.

Keys Helpbox.
The following keys are used in this section:
[CURSOR ARROWS], [F2]
[NUMBERS] - not highlighted
[DELETE], [EOB], [RESET]

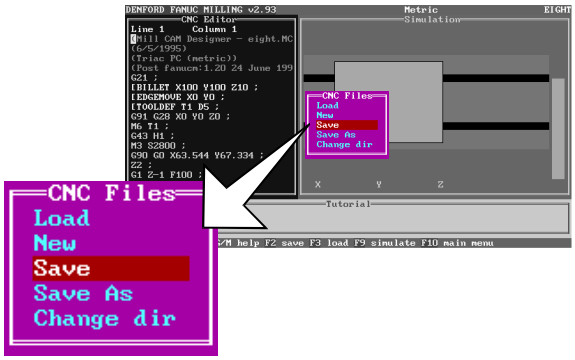


Tutor keypad.

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 5.7 "Changing the Drive Directory - CNC Files".

Highlight '*Save*' in the *CNC Files Menu* using the [CURSOR ARROWS] keys and press the [EOB] key.

Note, the [F2] key can be used as a short-cut to save CNC Files from to default drive in most areas of the control software.



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.

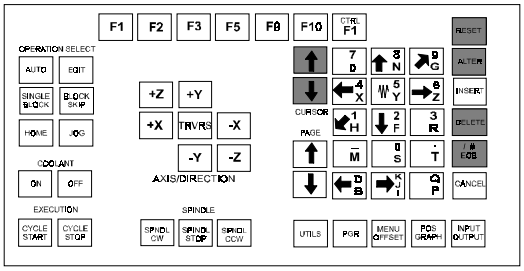
Note that CNC Files can only be saved with alphabet characters when using a Qwerty keyboard.

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 - Save as.

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



Tutor keypad.

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 5.7 "Changing the Drive Directory - CNC Files".

To save a newly edited 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 [ALTER] key.

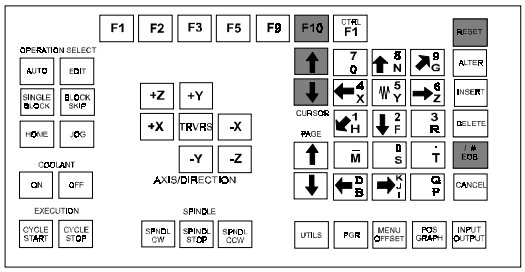
Press the [NUMBERS] keys.

Press the [EOB] key to save this new file to the currently selected drive. 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.

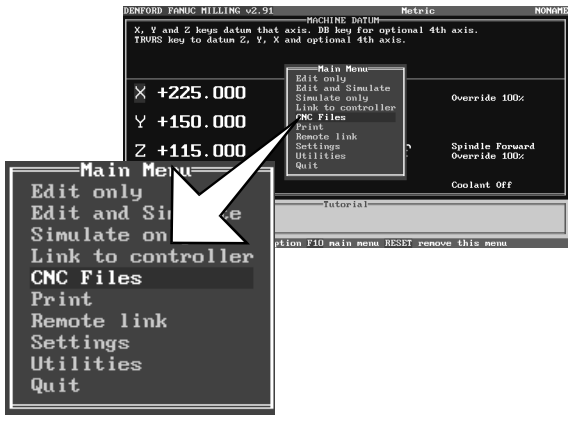
CHANGING THE DRIVE DIRECTORY - CNC FILES.

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

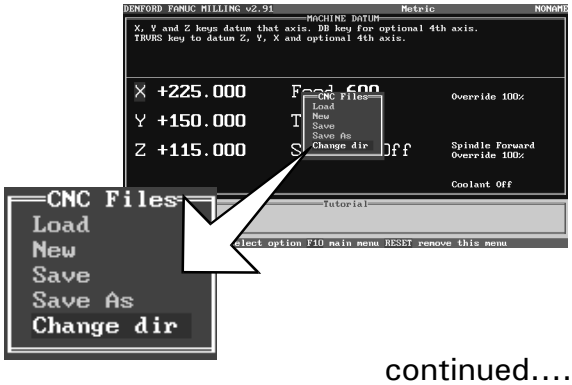


Tutor keypad.

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.

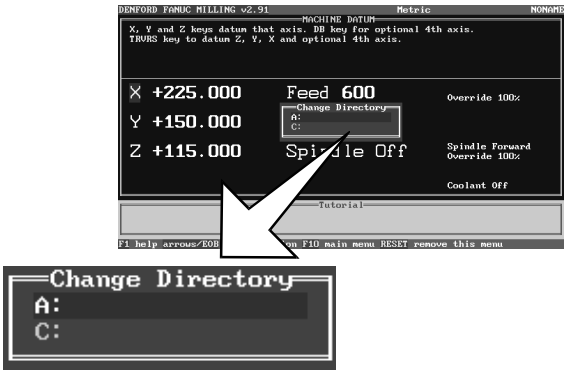


continued....

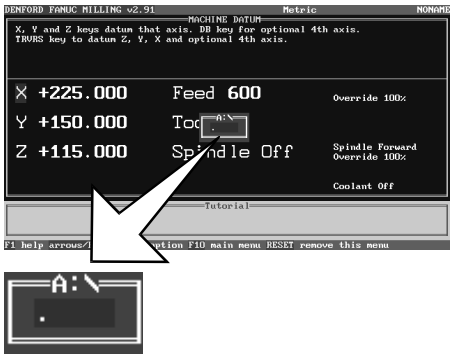
Changing the Drive Directory - CNC Files.

continued....

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



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

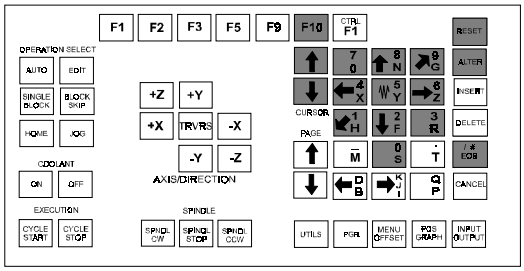


Press the [RESET] key to clear the screen of any unwanted menus or information.

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

Saving Files on a Changed Drive.

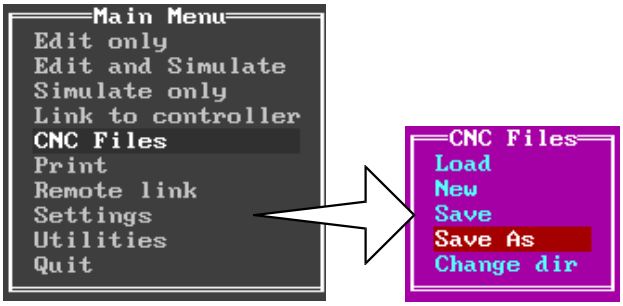
Keys Helpbox.
The following keys are used in this section:
[F10], [CURSOR ARROWS]
[EOB], [ALTER]
[NUMBERS], [RESET]



Tutor keypad.

Please note -
Do not use the *Save* option, since this will save the CNC File to the original, rather than the new, drive.

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.



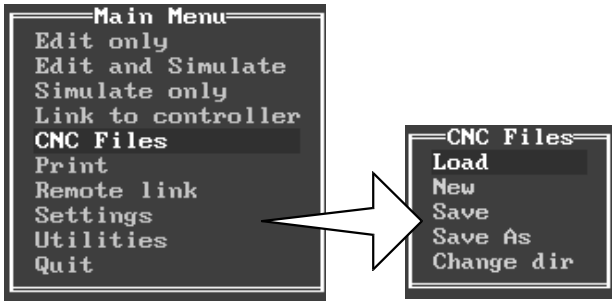
Note - the screen may display the previous setting for the drive. In the example below, the screen displays the drive as '*C:*' , even though it has just been changed to save on '*A:*' , as shown in the last section. If this occurs, press the [ALTER] key to reset to the new drive.

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

Please Note -
CNC Files are stored in ".fnc" format.
Offset Files are stored in ".fao" format.

Loading 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 below, 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 (.fnc files) will now be loaded from the new drive.

If the filename is unknown, the list of files stored on the drive can be accessed by pressing the [EOB] key. Highlight the CNC File required using the [CURSOR ARROWS] keys and press the [EOB] key to load the file.

The message window below 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" format.
Offset Files are stored in ".fao" format.

Simulate CNC File.

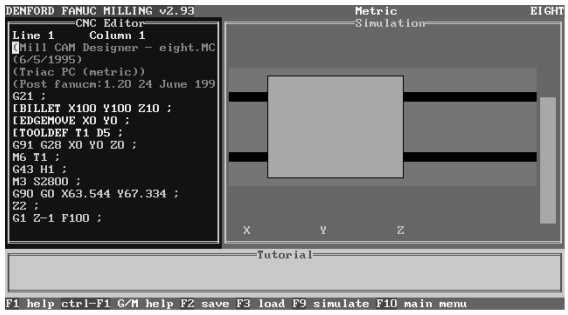
Once a CNC File has been loaded, its action can be simulated on screen. Remember that the name of the CNC File is displayed in the top right-hand corner of the display. In the example screenshots, a program called 'Eight' has been loaded.

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 to the table at this stage, since the machine will not cut during a simulation exercise.

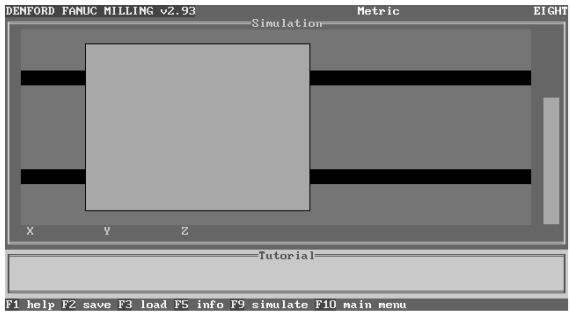
Simulate - Screen View Options.

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).



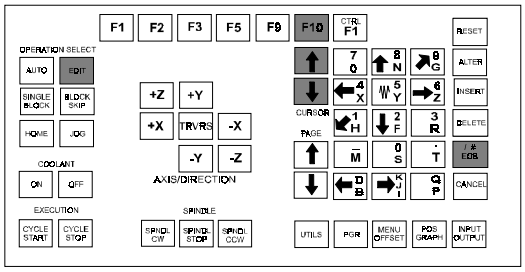
Simulate only. The CNC File can be simulated using a full screen graphical display(plan or 3d).



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

Simulate - Select Screen View.

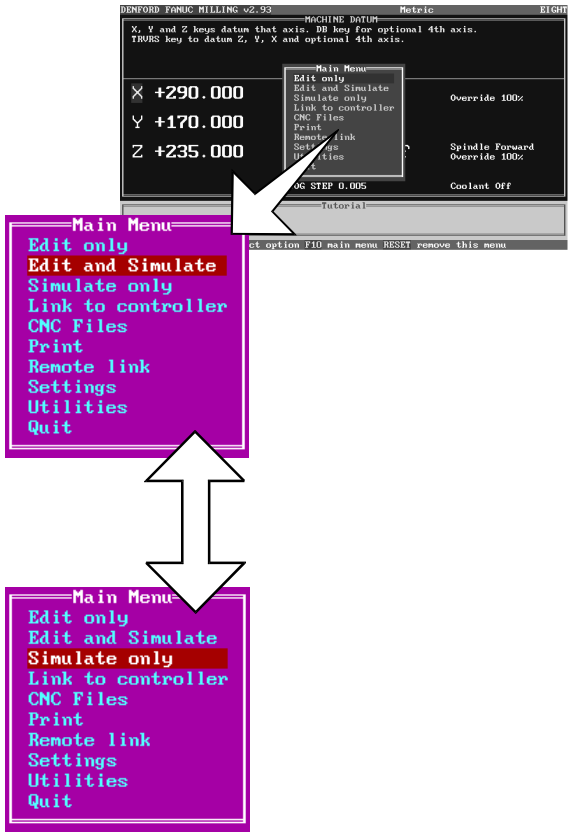
Keys Helpbox.
The following keys are used in this section:
[F10], [CURSOR ARROWS]
[EOB], [EDIT]



Tutor keypad.

Select the *Main Menu* by pressing the [F10] key. Highlight either the '*Edit and Simulate*' or '*Simulate only*' options, using the [CURSOR] keys and press the [EOB] key to select the highlighted option.

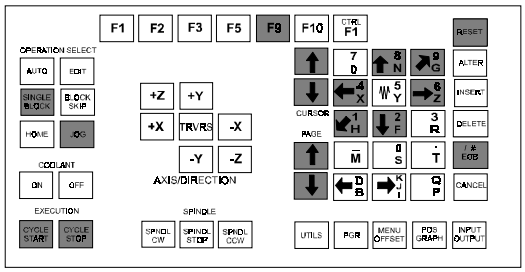
Alternatively, pressing the [EDIT] key will directly access the '*Edit and Simulate*' option.



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

SIMULATION MENU.

Keys Helpbox.
The following keys are used in this section:
[F9][CURSOR ARROWS]
[EOB], [JOG], [PAGE UP], [PAGE DOWN]
[CYCLE START], [CYCLE STOP], [RESET]
[8N UP ARROW], [2F DOWN ARROW]
[6Z RIGHT ARROW], [4X LEFT ARROW]
[9G UPPER RIGHT ARROW], [1H LOWER LEFT ARROW]
[SINGLE BLOCK]

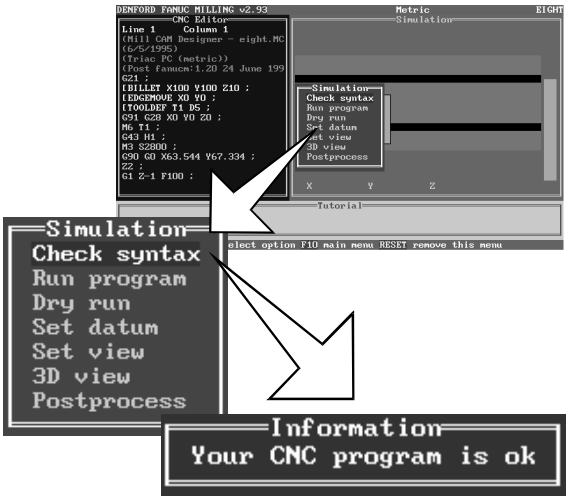


Tutor keypad.

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.

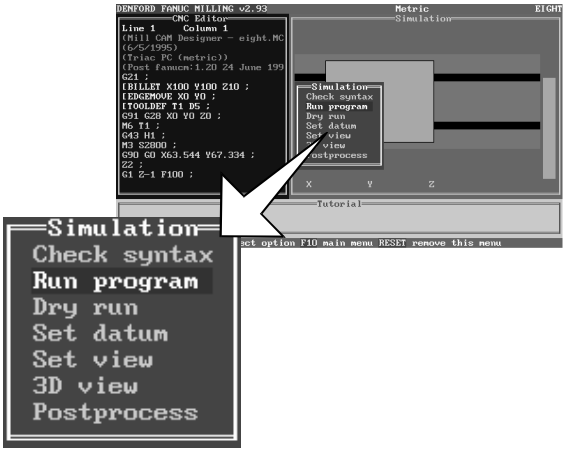
To start simulating the CNC File in the chosen screen view, select the *Simulation Menu* by pressing the [F9] key. All example screenshots are shown using the *'Edit and Simulate'* option.

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 this exercise is cleared by pressing the [RESET] key.

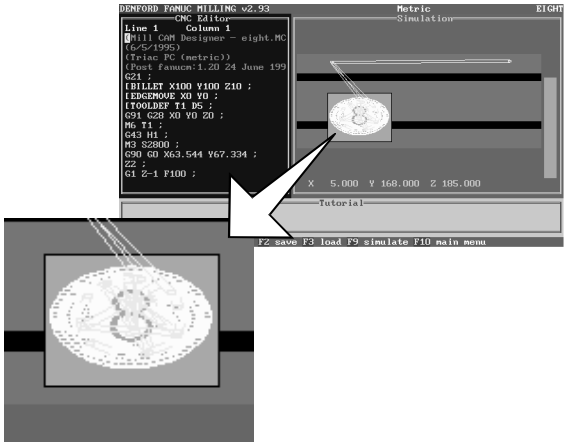


SIMULATION MENU.

2) *Run Program*. This instructs the computer to run through the CNC File. 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 CNC File cycle, from start to finish. Whilst the CNC File is running, the written text will scroll down the screen and the pictorial view will be simultaneously updated.



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

Pressing the [SINGLE BLOCK] key, executes one block of the program and updates the pictorial view simultaneously. The entire program can be simulated block by block using the [SINGLE BLOCK] key.

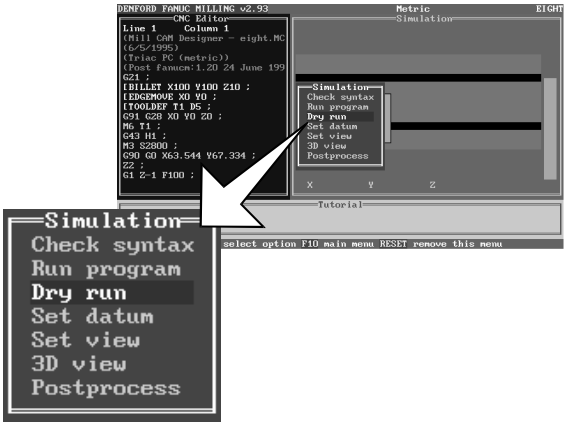
Please note -
The [CYCLE STOP] key can also be used to pause a run. The remainder of the CNC File will be run as a seperate cycle by pressing the [CYCLE START] key. Once this smaller cycle has finished press the [RESET] key to return to the start of the CNC File.

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

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.

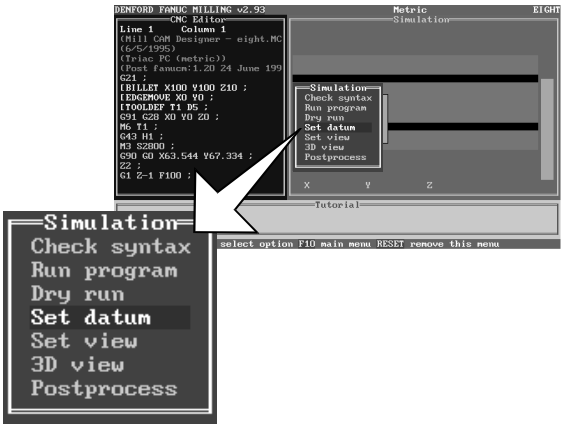
To run this option, highlight '*Dry Run*' using the [CURSOR ARROWS] keys and press [EOB]. On short programs it may appear that nothing has happened, since the *Dry Run* operation may take less than a second to complete.



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 message windows which are displayed can be cleared by pressing the [RESET] key.

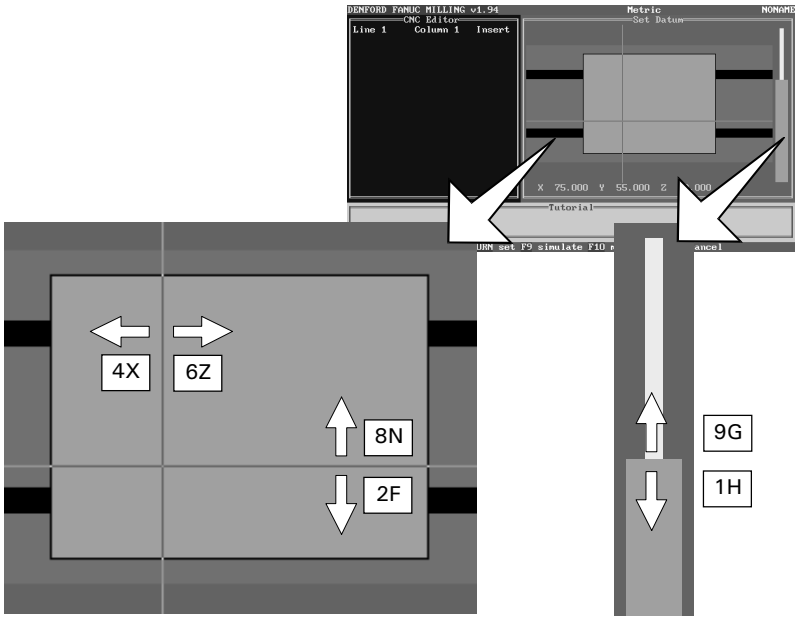
SIMULATION MENU.

4) *Set Datum*. This allows the datum point for the graphical simulation to be set to match the co-ordinates of the 'real' work datum point used on the billet and CNC File. To run this option, highlight '*Set Datum*' using the [CURSOR ARROWS] keys and press [EOB].



Use the following keys to move the position of the datum point:

[8N UP ARROW] = Purple/blue Crosshair up.
[2F DOWN ARROW] = Purple/blue Crosshair down.
[6Z RIGHT ARROW] = Purple/blue Crosshair right.
[4X LEFT ARROW] = Purple/blue Crosshair left.
[9G UPPER RIGHT ARROW] = Yellow tool depth up.
[1H LOWER LEFT ARROW] = Yellow tool depth down.

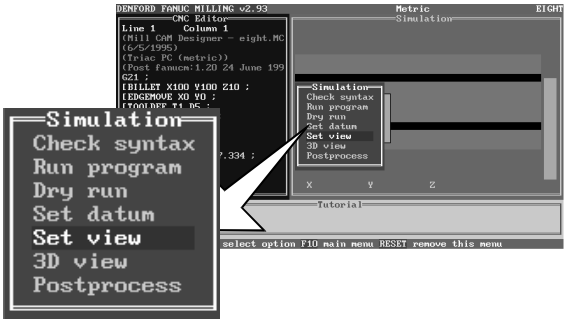


Press the [EOB] to set the graphical datum point, then the [RESET] key to clear any unwanted menus from the screen.

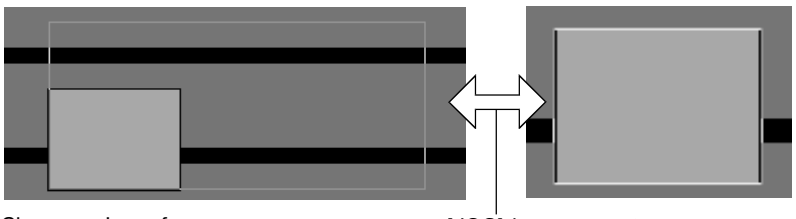
Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

SIMULATION MENU.

5) *Set View*. This allows the detail level of the plan pictorial view to be selected in the view editor. To select this option, highlight '*Set View*' using the [CURSOR ARROWS] keys and press [EOB].



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

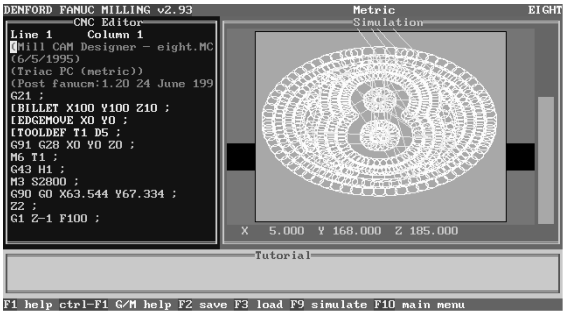


Close-up view of work and table.

[JOG] key toggles around the different views.

Close-up view of work only.

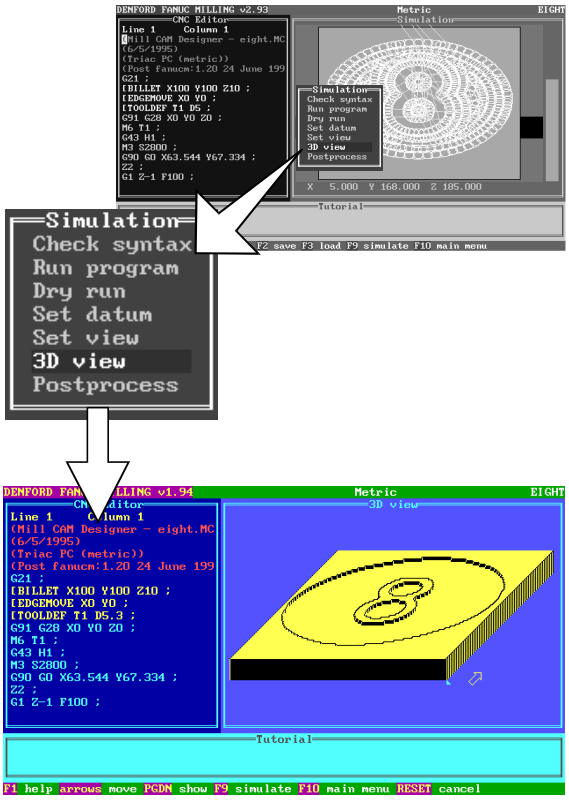
Now, when *'Run Program'* is selected, the plan pictorial view shown will be the one previously chosen in the *'Set View'* option.



If the simulation window is set to display 3d views, the view editor will operate in plan view when the '*Set View*' option is selected. The next time the CNC File is run, the 3d view will be that chosen in the view editor.

SIMULATION MENU.

6) *3d View*. This shows the simulation graphic as a 3 dimensional view, rather than a plan view. To select this option, highlight '*3d View*' using the [CURSOR ARROWS] keys and press [EOB].



The [PAGE UP] key can be used to toggle between a back or front 3d view of the design.

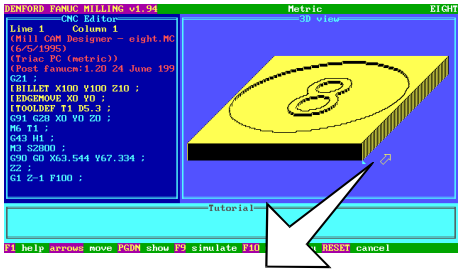
continued....

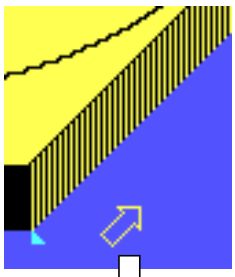
Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

Simulation Menu.

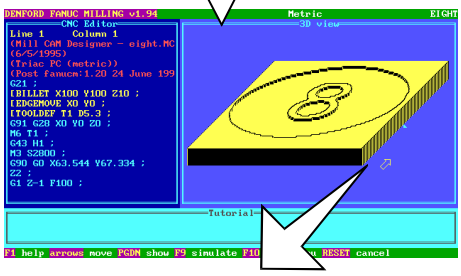
6) *3d View*. continued....

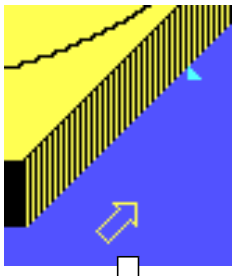
The [8N UP ARROW] key and [2F DOWN ARROW] keys can be used to establish a cross section point through the "Y" plane (indicated by a small light blue triangle on the righthand side if the 3d billet). To display the cross section at the point selected, press the [PAGE DOWN] key.



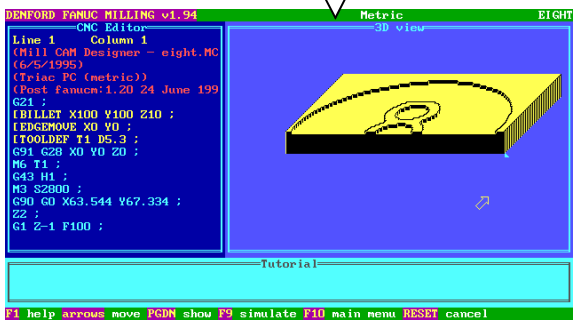


Use [8N UP ARROW] and [2F DOWN ARROW] keys to move blue triangle marker.





Use [PAGE DOWN] key to switch to cross sectional 3d view.

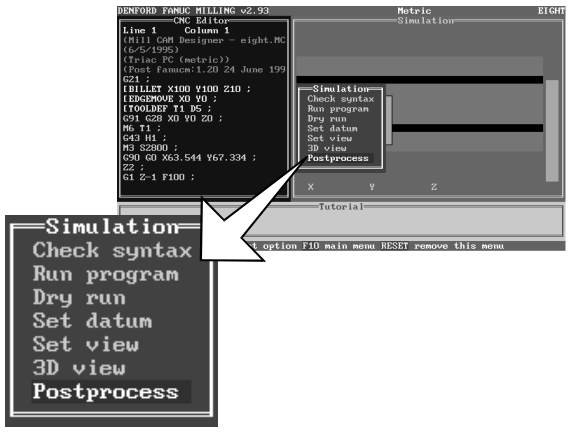


SIMULATION MENU.

7) *Post Process*. This feature requires the optional Denford Universal Post Processor software in order to function. The Desktop Tutor can be used in conjuction with the Denford Universal Post Processor to translate a CNC File into different control languages.

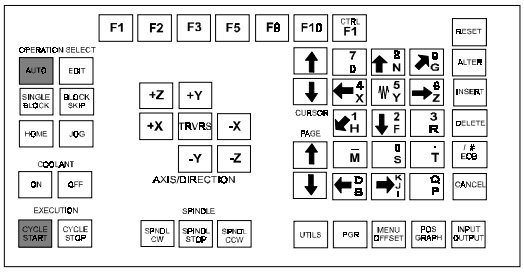
To select this option, highlight '*Post Process*' using the [CURSOR ARROWS] keys and press [EOB].

When the [EOB] key is pressed, a new file is created for the Denford Universal Post Processor software, based upon the original CNC File currently loaded in the controller. This new file is saved to the currently selected drive and directory with the same name as the CNC File currently loaded in the controller, but a different file extension, ".tnc". The original CNC File is unaffected by this operation. Note - no message window will be displayed to indicate that this new ".tnc" file has been created.



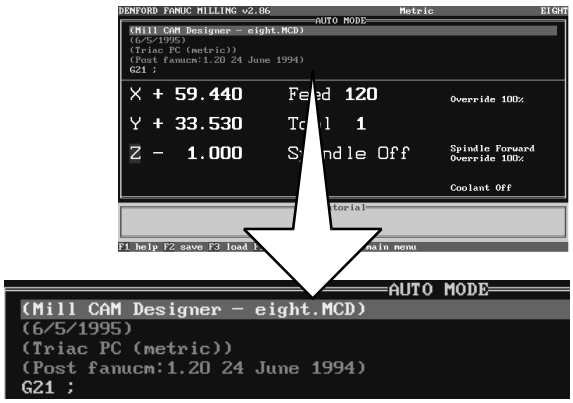
RUN CNC FILE.

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



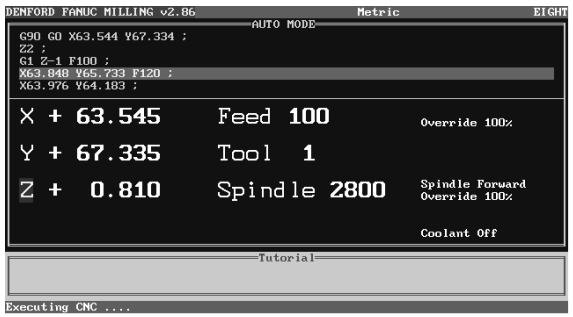
Tutor keypad.

To run a CNC File, the machine needs to be set in *Auto Mode*. This is selected by pressing the [AUTO] key.



To start the CNC File, press the [CYCLE START] key.

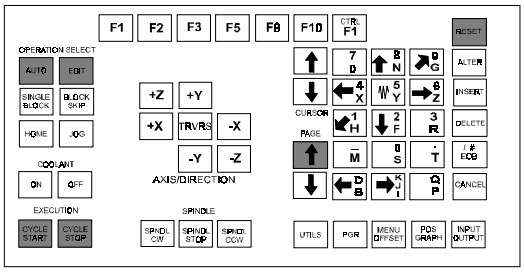
The CNC File will now start to scroll, line by line, in the upper half of the display screen. At the same time, the co-ordinates, feedrate and spindle speed are shown, in the lower half of the display screen. These are updated, according to the point reached in the CNC File (the particular CNC File line reached is shown 'highlighted').



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

PAUSING CNC FILE.

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

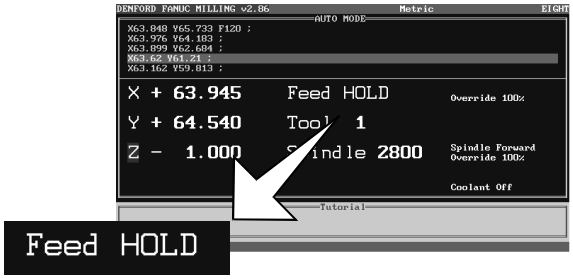


Tutor keypad.

To pause a CNC File, 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 resume the CNC File, press the [CYCLE START] key.



To stop a CNC File, press the [CYCLE STOP] key to pause, then press the [RESET] key to abort the run.

Press the [RESET] key to clear the error message shown above.

The CNC File 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 CNC File is reached.

2) Select *Edit Mode* by pressing the [EDIT] key, then press the [RESET] key. This will reset the CNC File to its start point. Press the [AUTO] key to return the machine back to *Auto Mode*.

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

Overriding Spindle Speed.

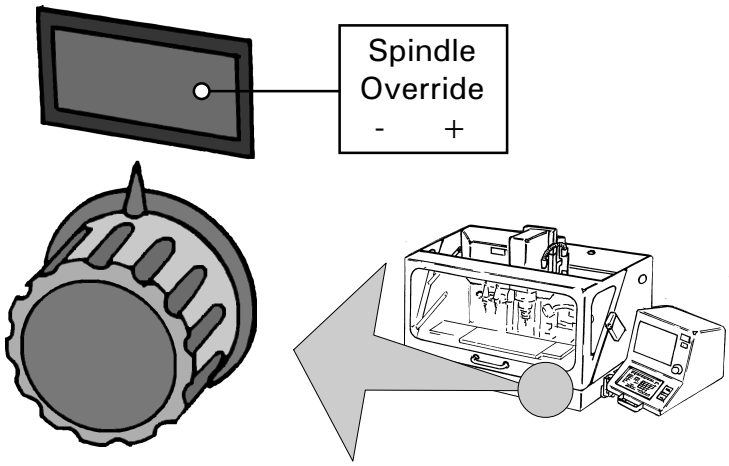
The spindle speed of the machine can be manually overridden during a machining operation using the potentiometer control dial fitted on the lower front panel of the machine (illustrated below).

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.

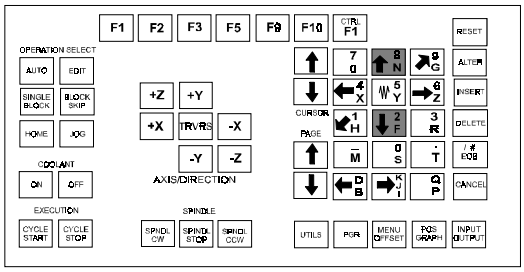


To increase the spindle speed, rotate the dial clockwise - as the dial is turned the 'faster' spindle speed will be updated and displayed on the control screen.

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.

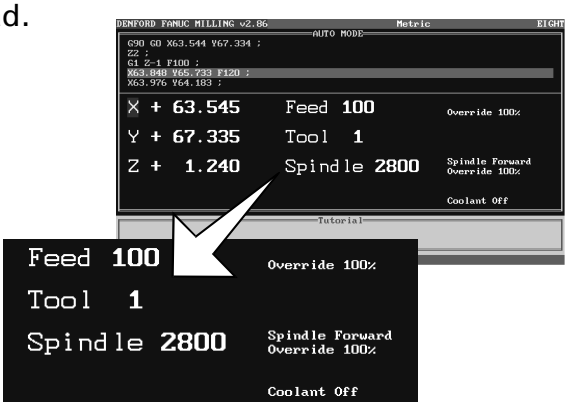
Overriding Spindle Speed - Tutor Controls.

Keys Helpbox.
The following keys are used in this section:
[UP KEYPAD ARROW]
[DOWN KEYPAD ARROW]

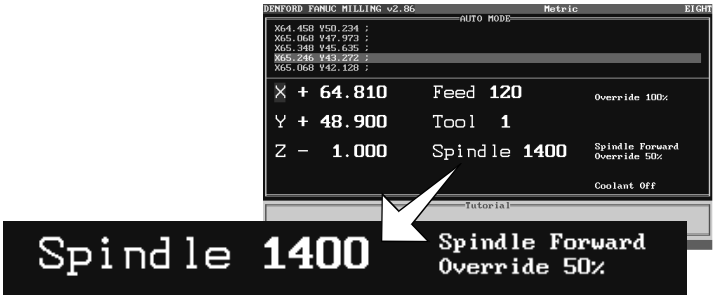


Tutor keypad.

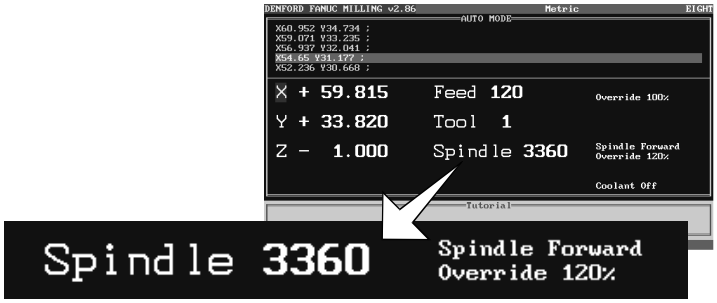
On machines not fitted with potentiometer controls, the spindle speed of the tool is manually adjusted using the tutor keypad.



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.



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.



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

Overriding Feed Rate.

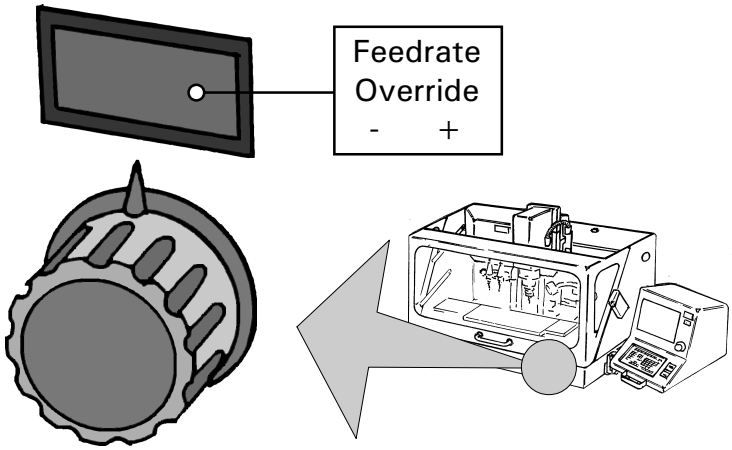
The feedrate of the machine can be manually overridden during a machining operation using the potentiometer control dial fitted on the lower front panel of the machine (illustrated below).

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.

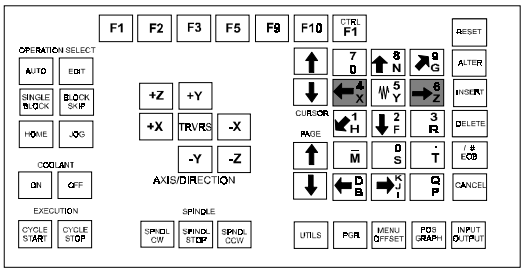


To increase the feedrate, rotate the dial clockwise - as the dial is turned the 'faster' feedrate will be updated and displayed on the control screen.

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.

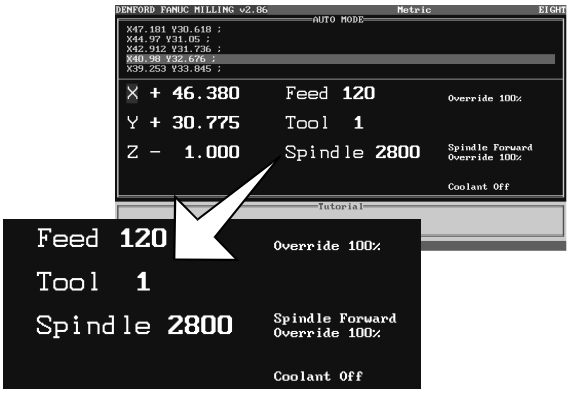
Overriding Feed Rate - Tutor Controls.

Keys Helpbox.
The following keys are used in this section:
[LEFT KEYPAD ARROW]
[RIGHT KEYPAD ARROW]



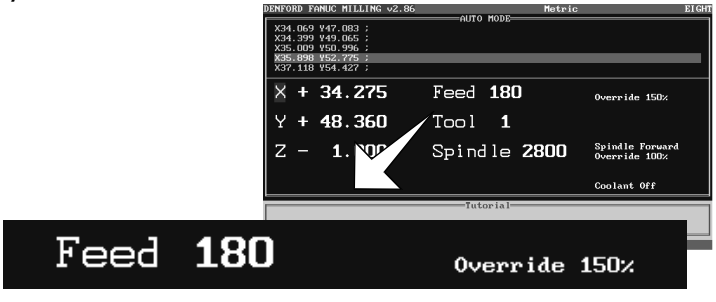
Tutor keypad.

On machines not fitted with potentiometer controls, the feedrate of the tool is manually adjusted using the tutor keypad.



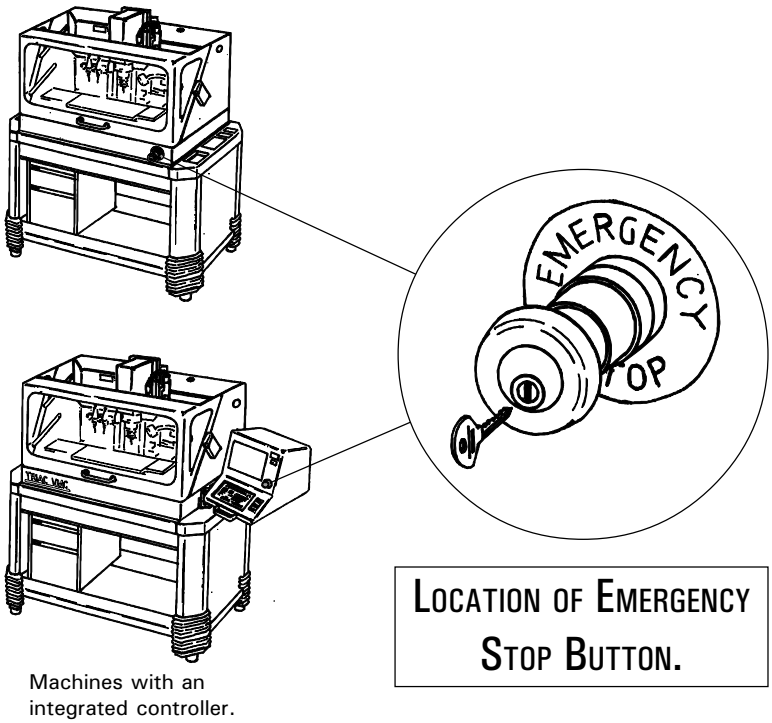
The feedrate can be decreased down to a minimum value of 0%. To decrease the feedrate, press the [LEFT KEYPAD ARROW] key until the desired value is displayed on screen.

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.



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

EMERGENCY STOP.



LOCATION OF EMERGENCY STOP BUTTON.

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

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

EMERGENCY STOP.

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

The Emergency Stop button will lock when fully pressed into its housing. To reset the Emergency Stop button, turn it clockwise or anticlockwise depending upon the type of switch fitted to release the locking mechanism. A key may be required on certain machines to reset the button and allow access back to the machine.

To clear any messages displayed after pressing the Emergency Stop button, press the [RESET] key.

The program will also need to reset back to its original starting position (enter *Edit Mode* by pressing the [EDIT] key and press the [RESET] key until the first lines of the program are visible).

The machine axes will need to be homed - see section 2.1 "Automatic Search for the Machine Datum Point".

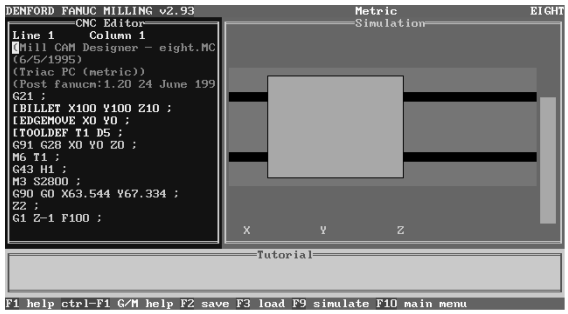
Please note -
The co-ordinate values depicted on screenshots are used for illustrative purposes only.

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).

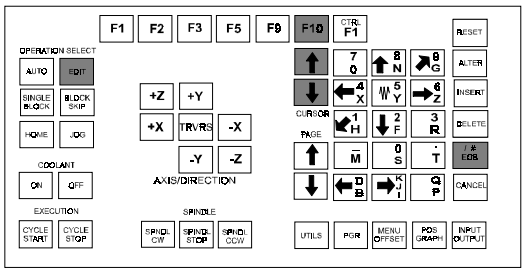


Edit only. The CNC File can be altered using a full screen Editor window.

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

EDITING A CNC FILE - SCREEN VIEW OPTIONS.

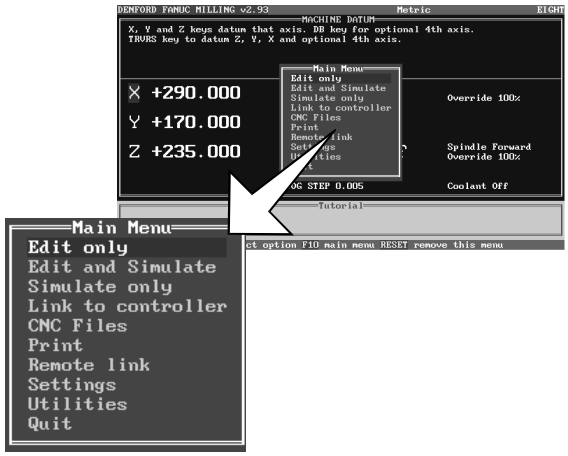
Keys Helpbox.
The following keys are used in this section:
[F10], [CURSOR ARROWS]
[EOB], [EDIT]



Tutor keypad.

Select the *Main Menu* by pressing the [F10] key. Highlight either the '*Edit and Simulate*' or '*Edit only*' options, using the [CURSOR] keys and press the [EOB] key to select the highlighted option.

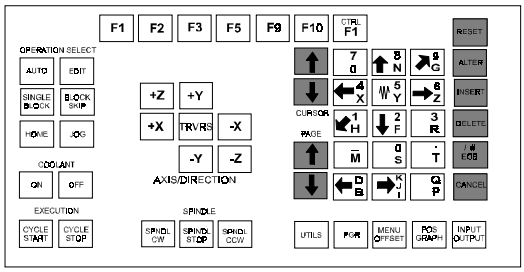
Alternatively, pressing the [EDIT] key will directly access the '*Edit and Simulate*' option.



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 entered on the edit line 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 /) when entered on the edit line.

[CANCEL] key.
This key will clear any characters from the edit line.

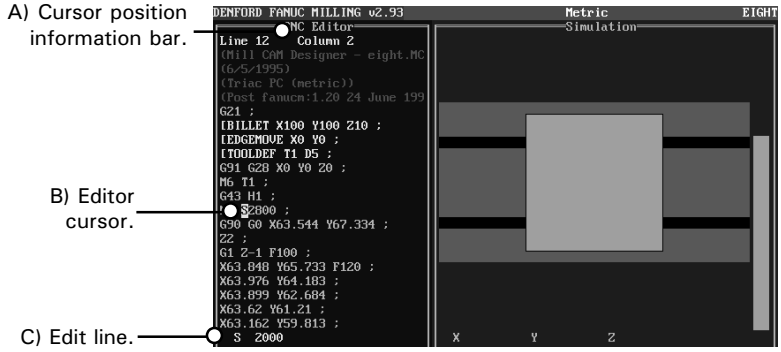
[ALTER] key.
Pressing this key will replace the program word/character highlighted by the cursor in the CNC File with any 'new' text entered on the edit line.

EDITING A CNC FILE - EDITOR WINDOW LAYOUT.

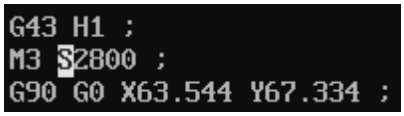
All example screenshots are shown using the *'Edit and Simulate'* option.

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 10.15 '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.

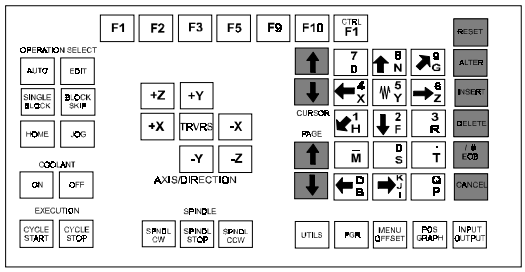


B) The yellow Editor cursor highlights the program word or character that can be edited. In this example it highlights the program word '*S2800*', referring to the spindle speed.

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.

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.

```
G43 H1 ;
M3 S2800 ;
G90 G0 X63.544 Y67.334 ;
```

The required section of the program, the program word '*S2800*' is highlighted using the [CURSOR ARROWS] keys. To move through a large 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.

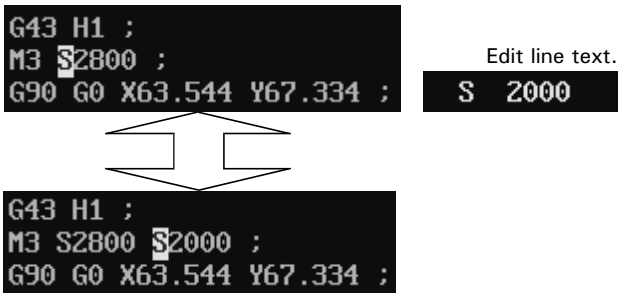
Any 'new' program word, or set of words, is always entered from the edit line.

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.

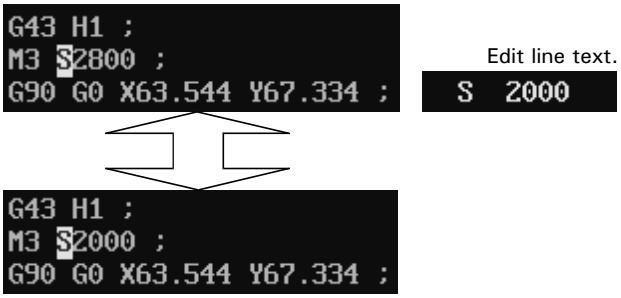
INSERTING AND DELETING PROGRAM TEXT.



If the [INSERT] key is pressed, any 'new' text entered on the edit line is placed directly after the last program word/character highlighted by the cursor. The 'new' program line reads '*M3 S2800 S2000 ;*' as shown above.

To remove the highlighted program word '*S2000*', the [DELETE] key would be pressed.

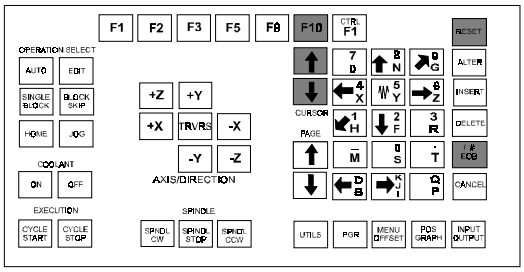
ALTERING A PROGRAM LINE.



If the [ALTER] key is pressed when '*S2000*' is entered on the edit line and '*S2800*' is highlighted by the cursor, '*S2800*' is replaced by '*S2000*'.

Main Menu - Link to Controller.

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



Tutor keypad.

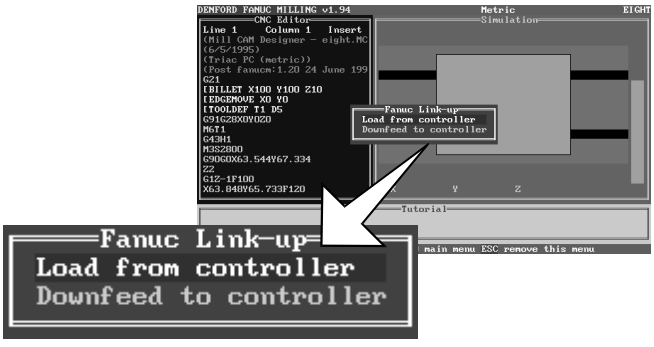
The *Link to Controller* option allows a CNC File to be sent or received from an external FANUC controller. This option is useful for CNC Files that are designed and generated in 'clean' working areas before being sent down to the machining area, or CNC Files that are larger than the capacity of one floppy disk.

Select the *Main Menu* by pressing the [F10] key. Highlight '*Link to Controller*' using the [CURSOR ARROWS] keys and press the [EOB] key.

The *Link to Controller Sub-menu* contains two options :

1) *Load from Controller*. This option will load the CNC File from the external FANUC controller.
2) *Downfeed to Controller*. This option will send the CNC File to the external FANUC controller.

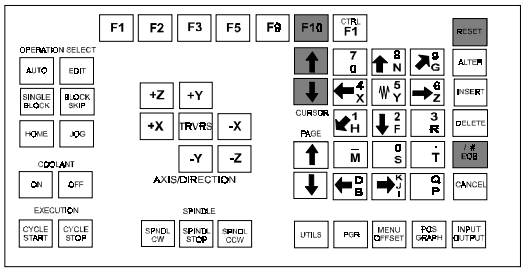
Select the required option using the [CURSOR ARROWS] keys, then press the [EOB] key.



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

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 :

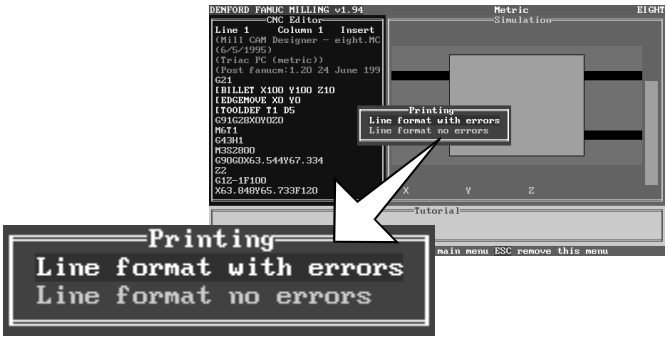
1) *Line format with errors*. This option will print the CNC File as displayed in the Editor window with any errors highlighted.
2) *Line format with no errors*. This option will print the CNC File as displayed in the Editor window without highlighting any errors.

continued....

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 :

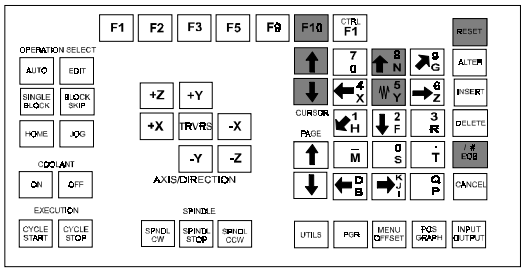
1) Is the cabling between the computer/control box and printer secure?
2) Is the printer set for Parallel or Serial communications?
3) Have the correct parameters been set in the 'Settings Menu - Print Device' (see section 10.11) ?
4) Is the printer switched 'on' and is there enough paper available for the printout?

Page Layout Errors.

If the layout on the printout is incorrect, the page widths and linefeeds can be changed in the 'Settings Menu - Print Page Layout' (see section 10.12).

Main Menu
- Remote Link.

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



Tutor keypad.

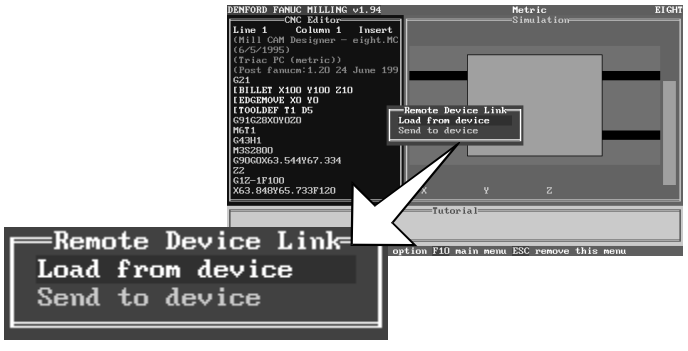
The *Remote Device* option allows a CNC File to be sent or received from a remote device, such as another computer, paper tape punch reader, data carrier etc....

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.



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.

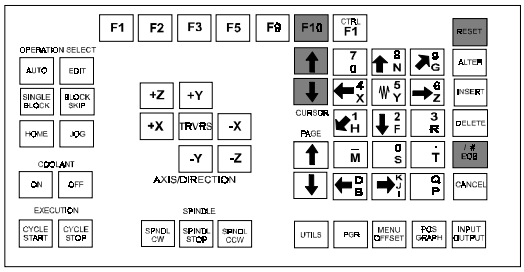
Communication Errors.

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 10.14)?
4) Is the remote device switched 'on' and ready to send or receive data?

Main Menu - Utilities.

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



Tutor keypad.

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

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.

Other external DOS programs can be run when the computer is set in DOS mode. To return to the control software, type '*EXIT*' at the DOS prompt, then press the [EOB] key, when prompted.

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

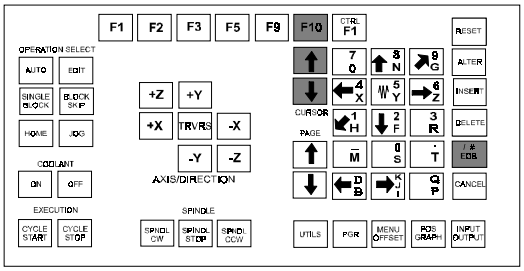
Change Settings.

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.

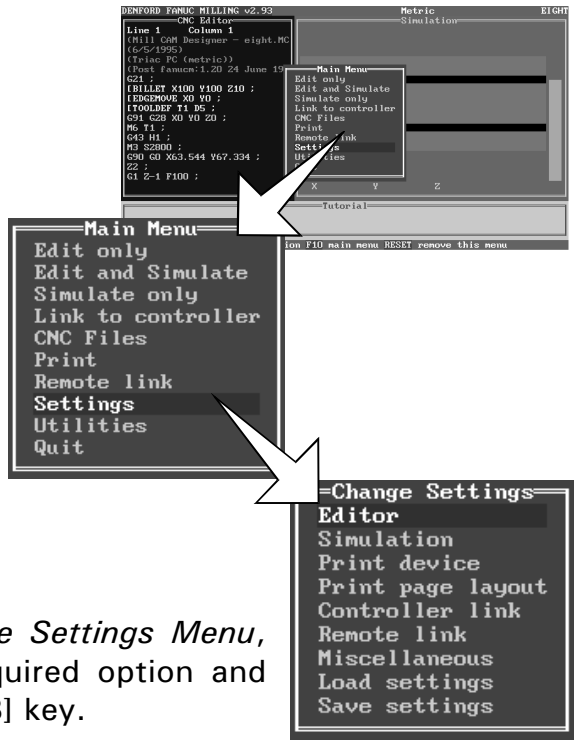
Keys Helpbox.
The following keys are used in this section:
[F10]
[CURSOR ARROWS]
[EOB]



Tutor keypad.

The *Change Settings Menu* is used to switch on and off the various options available within the machine control 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.

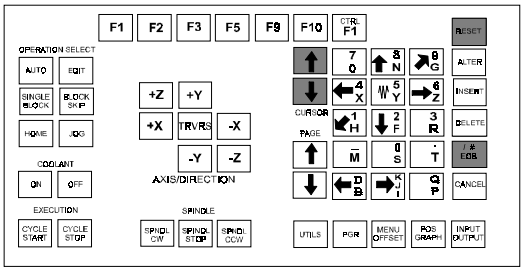
Change Settings - Menu.

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 layout of any printouts to be customised.
5) *Controller Link*. This option allows the communication protocols between the machine and the controller to be configured.
6) *Remote Link*. This option allows the communication protocols between the machine controller and an external device to be configured.
7) *Miscellaneous*. This option allows the units of measurement, user's name and screen text size to be customised.
8) *Load Settings*. This option allows a collection of settings to be loaded.
9) *Save Settings*. This option allows the current collection of settings to be saved.

CHANGE SETTINGS - EDITOR.

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



Tutor keypad.

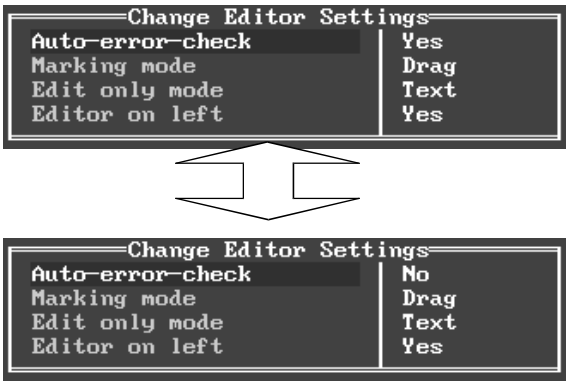
The *Change Settings (Editor) Menu* contains the following options:

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 occured.

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 part 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.

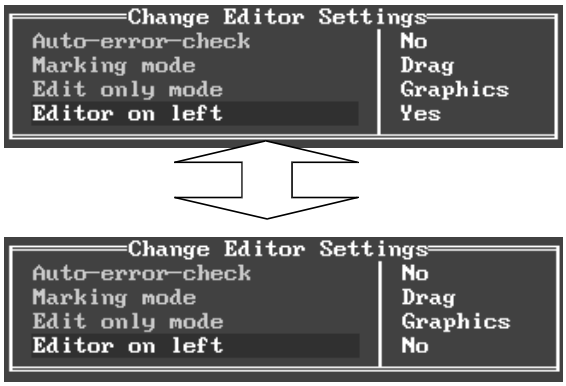
CHANGE SETTINGS - EDITOR.

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.

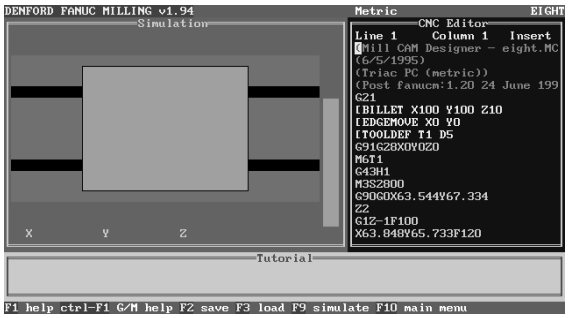
The screen will indicate the current setting of the option. Continual pressing of the [EOB] key will toggle the option between '*Yes*' and '*No*'.

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



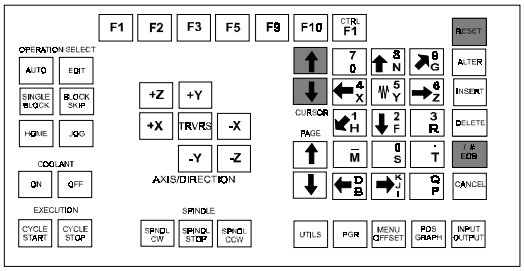
'*Editor On Left*' set to '*Yes*'.

'*Editor On Left*' set to '*No*'.



CHANGE SETTINGS - 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:

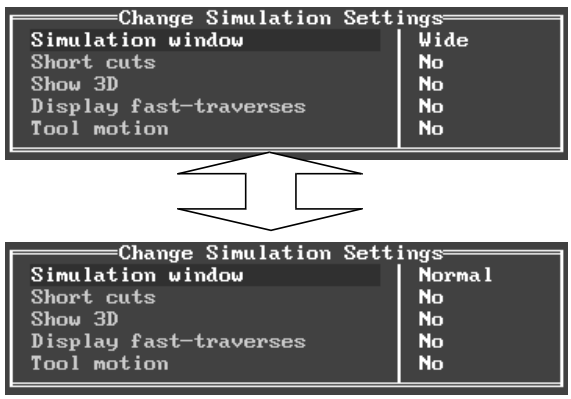
1) *Simulation Window*. This option will switch the size of the graphical simulation window between wide and normal.

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.

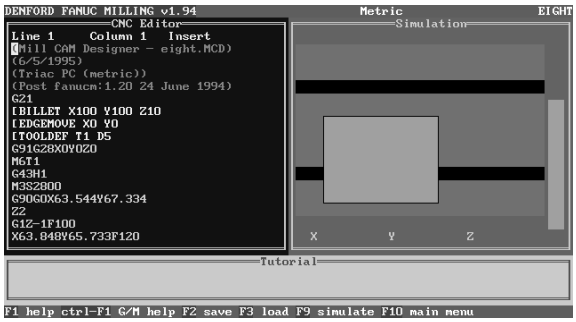


continued....

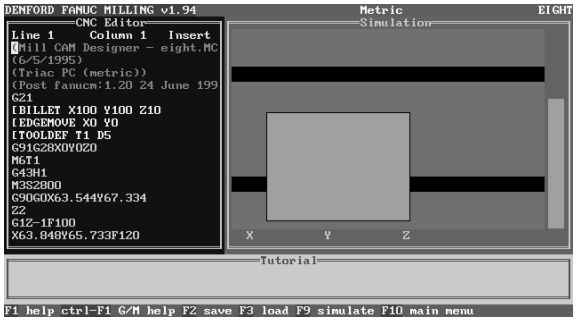
CHANGE SETTINGS - SIMULATION.

1) *Simulation Window*. continued....

Simulation Window set to '*Normal*'.



Simulation Window set to '*Wide*'.



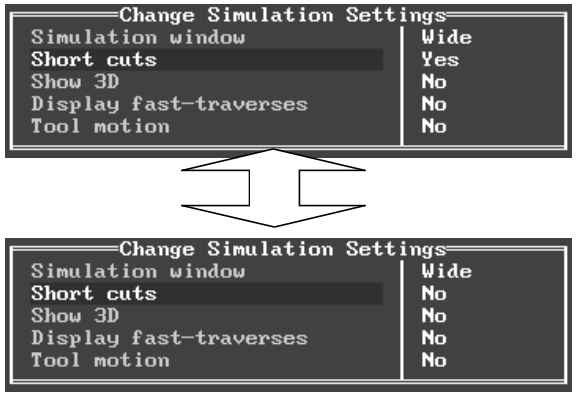
2) *Short Cuts*. This option is active when set to '*Yes*'.

Shorts Cuts, when set to '*Yes*', will speed up the simulation graphics by shortening any repetitive cycle commands (eg, peck drilling). The numbers of cycles required to complete the command are shown on screen by one single operation, rather than a series of repeat operations.

To select this option, highlight '*Short Cuts*' 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.



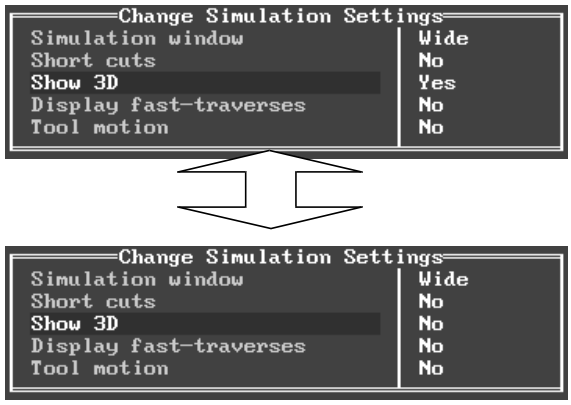
CHANGE SETTINGS - SIMULATION.

3) *Show 3d*. This option, when set to '*Yes*', will display the work as a three dimensional object.

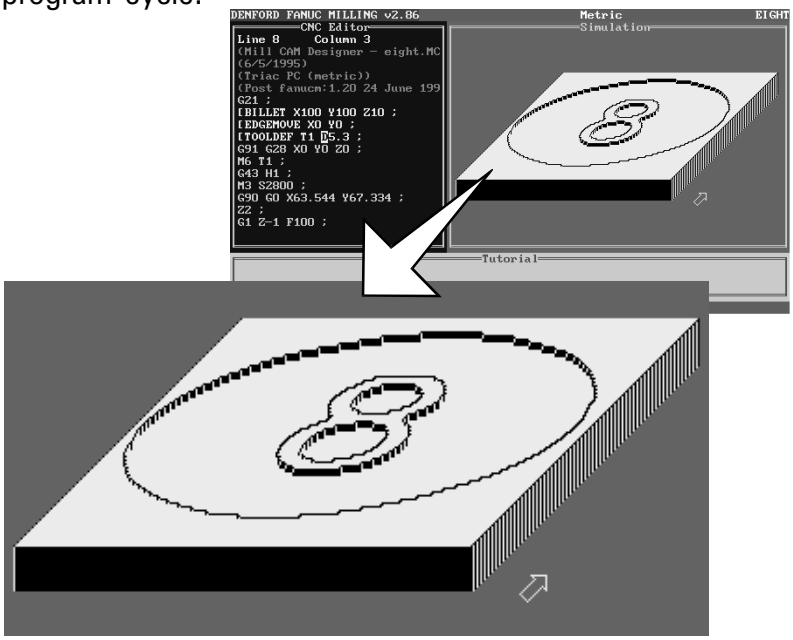
To select this option, highlight '*Show 3d*' using the [CURSOR ARROWS] keys and press the [EOB] key.

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

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



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



CHANGE SETTINGS - SIMULATION.

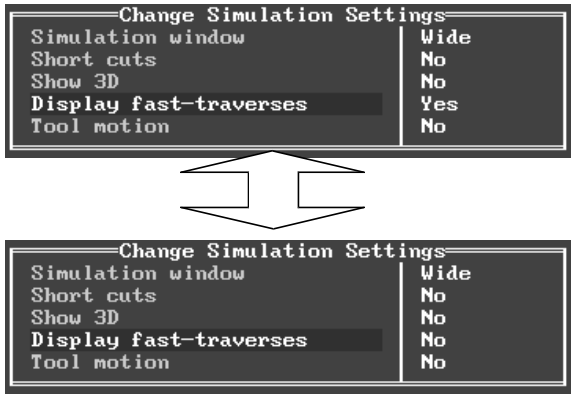
4) *Display Fast Traverses*. This option is active when set to '*Yes*' (not available for 3d views).

Display Fast Traverses, when set to '*Yes*', will show all fast traverse movements as yellow lines. Lines cut into the billet are shown in white.

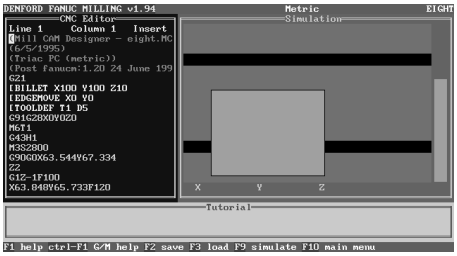
To select this option, highlight '*Display Fast Traverses*' using the [CURSOR ARROWS] keys and press the [EOB] key.

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

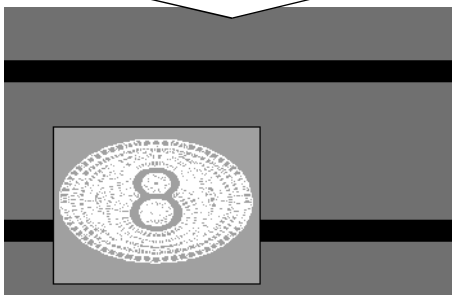
When the setting is correct, press the [RESET] key until all the menus have been removed from the screen.



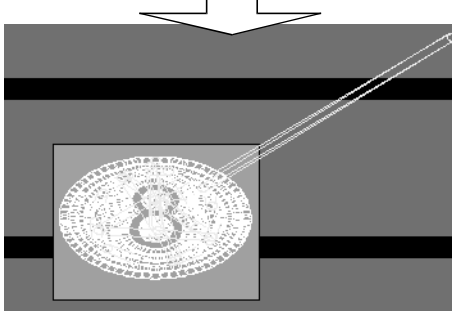
Run CNC File with....



Display Fast Traverses set to '*No*'.



Display Fast Traverses set to '*Yes*'.



CHANGE SETTINGS - SIMULATION.

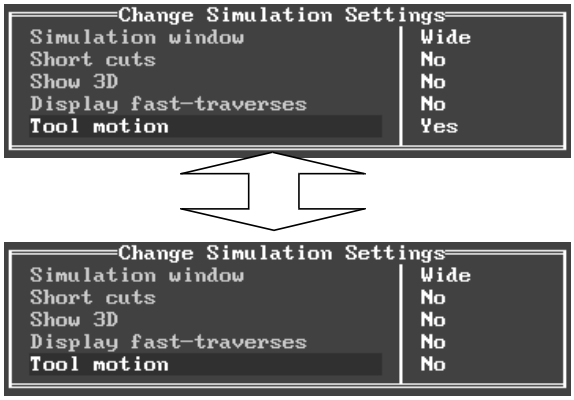
Please note -
The Tool Motion feature is not available when CNC Files use cutter diameters of less than 3mm.

5) *Tool Motion*. This option, when set to '*Yes*' will display the movement and path of the tool as it cuts the material.

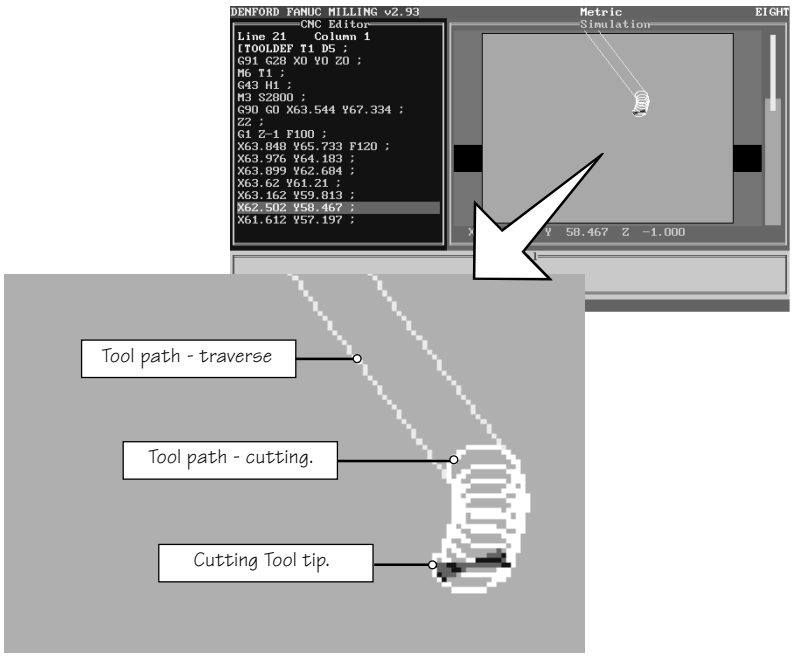
To select this option, highlight '*Tool Motion*' using the [CURSOR ARROWS] keys and press the [EOB] key.

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

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

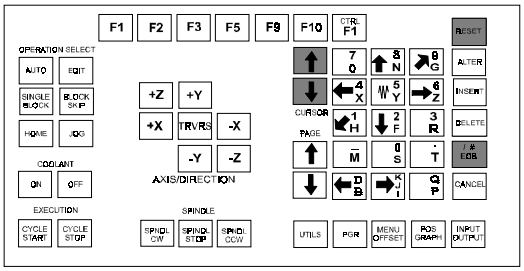


Now, when *'Run Program'* is selected, the tool motion will be shown , throughout the running of the program cycle. The program cycle will run considerably slower with this option switched 'on'.



CHANGE SETTINGS - PRINT DEVICE.

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



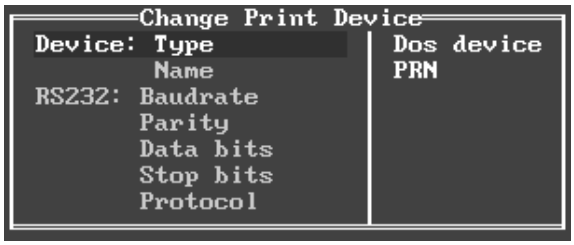
Tutor keypad.

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

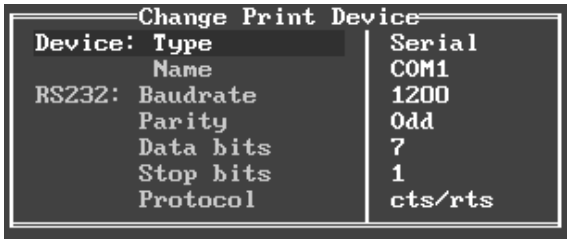
The *Print Device* option is used to configure the settings for any printers attached to the machine controller.

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

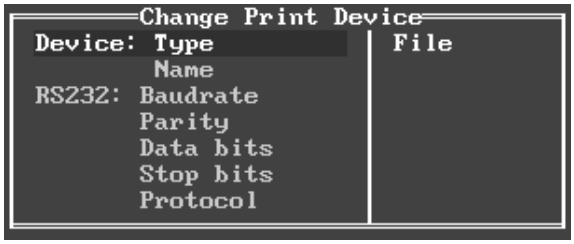
1) *DOS Device* - A DOS Device is normally the parallel port which can be set to LPT1. LPT2 or PRN. Select this option if your printer has a parallel port.



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



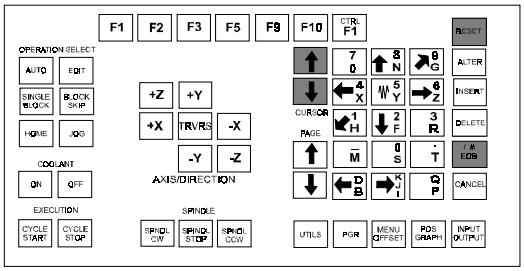
3) *File* - This option is used to save the CNC File on disk, for printing off at a later date. The current filename will be saved with an extension ".LST".



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

Change Settings - Print Page Layout.

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



Tutor keypad.

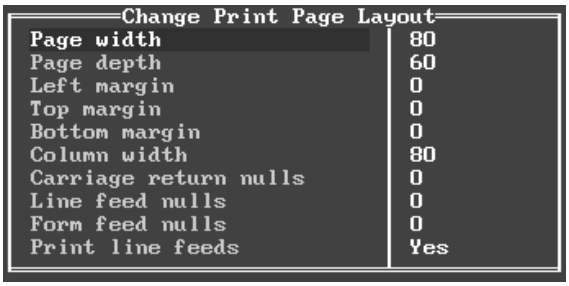
Please note -
Any changes made to these settings should be carried out by either your IT Manager or computer technician.

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 10.18).

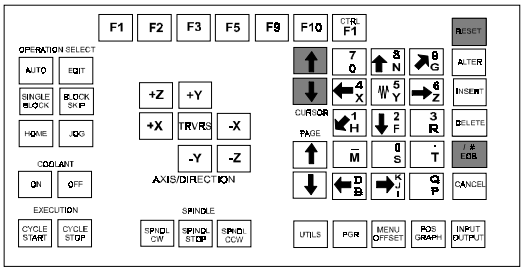
Select the individual settings on the *Change Print Page Layout Menu* using the [CURSOR ARROWS]. Press the [EOB] key to move the cursor across, type in the required values, then press the [EOB] key to confirm the new value.

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



CHANGE SETTINGS - CONTROLLER LINK.

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



Tutor keypad.

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

The *Controller Link* option is used to configure the communication protocols between the machine and the controller.

On PC controlled machines, The PC serial (COM) ports are used for communicating. Usually, COM 1 is used to connect the PC to the desktop tutor keypad and COM 2 is used to connect the PC to the port labelled "RS232" on the machine electrical control box. In this case, COM 2 is the *controller link*.

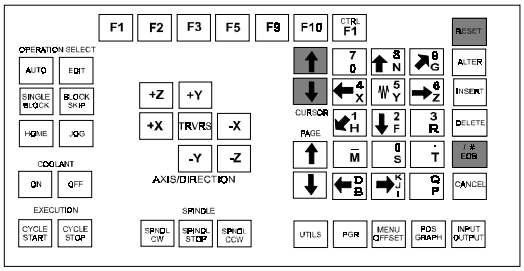
On machines with an integrated controller, the *controller link* is "hidden" inside the electrical control box. In this case, the option is ineffective and should NOT be altered.

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 SETTINGS - REMOTE LINK.

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



Tutor keypad.

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

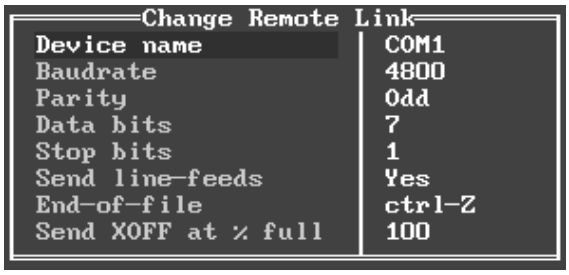
The *Remote Link* option is used to configure the communication protocols between the controller and an external device (such as a remote computer, paper tape punch, or printer).

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.

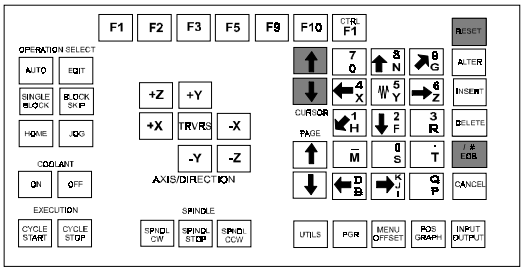
To select the individual settings use the [CURSOR ARROWS] keys and press the [EOB] key to toggle between the different values.

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



CHANGE SETTINGS - MISCELLANEOUS.

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



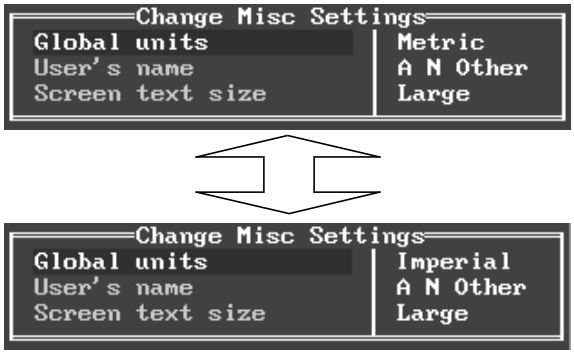
Tutor keypad.

The *Change Settings (Miscellaneous) Menu* contains the following options:

1) *Global Units*. This option will display the axes co-ordinates and feedrate in either Metric units (millimetres) or Imperial units (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.

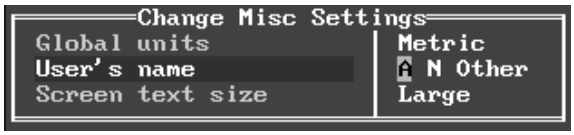


CHANGE SETTINGS - MISCELLANEOUS.

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

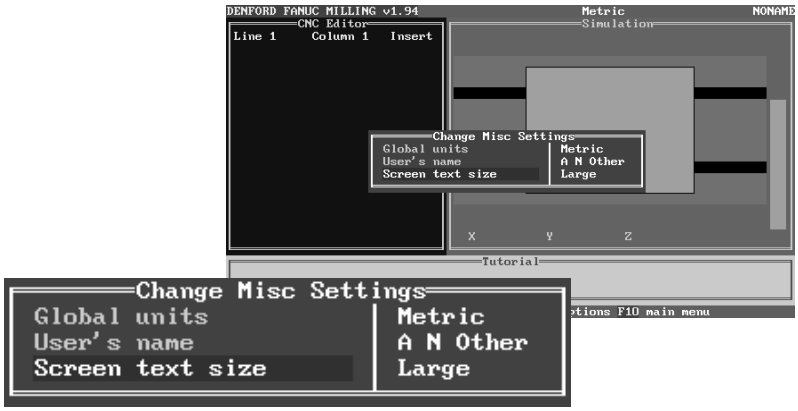
To select this option, highlight '*User's Name*' using the [CURSOR ARROWS] keys and press the [EOB] key to move the cursor across to the text '*A N Other*'. Enter the new text using the [NUMBERS] keys and press the [EOB] key to confirm. Note that alphabet characters can only be entered when using a qwerty keyboard.

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.

3) *Screen Text Size*. This option, when set to '*Large*', will set the text size to 25 lines on screen :



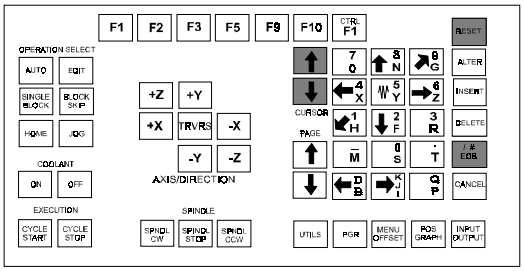
When set to '*Small*', the text size is set to 43 lines on screen :

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.

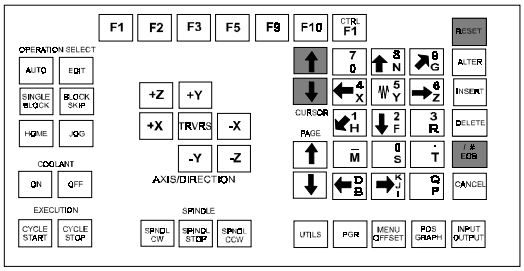
When the *Load Settings* option is selected,the default settings filename will appear in the edit window. The default filename is '*FANUCM*' :

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 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.

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



Tutor keypad.

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

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 '*FANUCM*' :

If the settings are saved with this ('*FANUCM*') 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.

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 Milling Machines.

3) M Codes used on Denford CNC Milling Machines.

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.

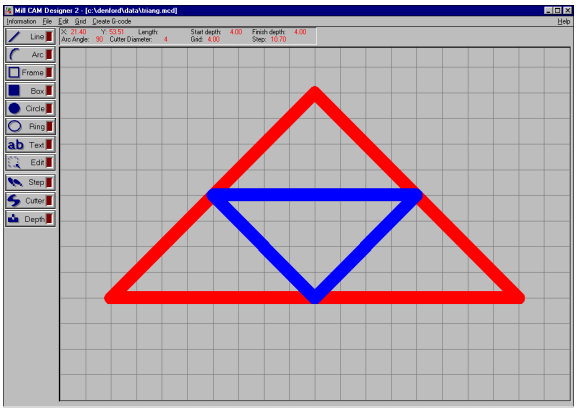
The part program can be further broken down into separate lines of data, each line describing a particular set of machining operations. These lines, which run in sequence, are called *blocks*.

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 Mill CAM Designer.

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 -

```
(Mill CAM Designer - triang.MCD)
(3/3/1997)
(Novamill (metric))
(Post fanucm:1.20 24 June 1994)
G21
[BILLET X80 Y55 Z10
[EDGEMOVE X0 Y0
[TOOLDEF T1 D2
N0010 G91G28X0Y0Z0;
N0020 M6T1;
N0030 G43H1;
N0040 M3S1500;
N0050 G90G0X40Y48;
N0060 Z2;
N0070 G1Z-0.5F100;
N0080 X72Y16F150;
N0090 X8;
N0100 X40Y48;
N0110 G0Z2;
N0120 X24Y32;
N0130 G1Z-1F100;
N0140 X56F150;
N0150 X40Y16;
N0160 X24Y32;
N0170 G0Z2;
N0180 M5;
N0190 G91G28X0Y0Z0;
N0200 M30;
```

Address example - G

Word example - G01

Block example - N0130 G1Z-1F100;

Denford Directive Example - [BILLET

Listing of Address Characters.

N	- Program Sequence (line) number.
X	- Primary motion in X axis.
Y	- Primary motion in Y axis.
Z	- Primary motion in Z axis.
G	- Preparatory functions.
I	- Incremental distance parallel to X axis.
J	- Incemental distance parallel to Y axis.
K	- Incremental distance parallel to Z axis.
R	- Radius.
M	- Miscellaneous functions.
T	- Tool numbers.
S	- Spindle speeds.
F	- Feed rates.

G Codes Listing for Denford CNC Milling Machines.

Note - Not all G codes apply to each machine.

G Code.	Group.	Function.
GØØ	1	Positioning (Rapid Traverse)
GØ1	1	Linear Interpolation (Cutting Feed)
GØ2	1	Circular Interpolation CW
GØ3	1	Circular Interpolation CCW
GØ4	Ø	Dwell, Exact Stop
G2Ø	6	Imperial Data Input (Inches)
G21	6	Metric Data Input (Millimetres)
G28	Ø	Reference Point Return
G4Ø	7	Cutter Compensation Cancel
G41	7	Cutter Compensation Left
G42	7	Cutter Compensation Right
G73	9	Peck Drilling Cycle
G74	9	Counter Tapping
G76	*9*	*Fine Boring*
G8Ø*	9	Canned Cycle Cancel
G81	9	Drilling Cycle, Spot Boring
G82	9	Drilling Cycle, Counter Boring
G83	9	Peck Drilling Cycle
G84	9	Tapping
G85	9	Boring Cycle
G86	9	Boring Cycle
G87	*9*	*Back Boring Cycle*
G89	9	Boring Cycle
G9Ø*	3	Absolute Zero
G91	3	Incremental Command
G94*	5	Feed per Minute
G95	5	Feed per Revolution
G98*	1Ø	Return to Initial Point in Canned Cycle
G99	1Ø	Return to R in Canned Cycle
G17Ø	Ø	Circular Pocket
G171	Ø	Circular Pocket
G172	Ø	Rectangular Pocket
G173	Ø	Rectangular Pocket

Code listing full and correct at the time of printing.

Notes.

G codes from group Ø 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).

The G codes indicated by an asterisk (*) are reactivated as defaults when the machine started.

M Codes Listing for Denford CNC Milling Machines.

Note - Not all M codes apply to each machine.

M code.	Function.
MØØ*	Program Stop
MØ1*	Optional Stop
MØ2*	Program Reset
MØ3	Spindle Forward (clockwise)
MØ4	Spindle Reverse (counter clockwise)
MØ5*	Spindle Stop
MØ6	Automatic Tool Change
MØ8	Coolant On
MØ9*	Coolant Off
M1Ø	Vice/Work Clamp Open
M11	Vice/Work Clamp Close
M13	Spindle Forward and Coolant On
M14	Spindle Reverse and Coolant On
M19	Spindle Orientation
M2Ø	ATC Arm In
M21	ATC Arm Out
M22	ATC Arm Down
M23	ATC Arm Up
M24	ATC Drawbar Unclamp
M25	ATC Drawbar Clamp
M27	Reset Carousel to Pocket One
M3Ø	Program Reset and Rewind
M32	Carousel CW
M33	Carousel CCW
M38	Door Open
M39	Door Close
M62	Auxiliary Output 1 On
M63	Auxiliary Output 2 On

continued....

M Codes Listing for Denford CNC Milling Machines.

continued....

M code.	Function.
M64	Auxiliary Output 1 Off
M65	Auxiliary Output 2 Off
M66*	Wait for Auxiliary Output 1 On
M67*	Wait for Auxiliary Output 2 On
M7Ø	Mirror in X On
M71	Mirror in Y On
M76	Wait for Auxiliary Output 1 Off
M77	Wait for Auxiliary Output 2 Off
M8Ø	Mirror in X Off
M81	Mirror in Y Off
M98	Sub Program Call
M99	Sub Program End and Return

Code listing full and correct at the time of printing.

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 X1ØØ.Ø Y9Ø.Ø Z2Ø.Ø

This sets the measure to metric (Note - if set to Imperial the units would be inches) and defines the billet as1ØØmm long by 9Ømm wide, with a depth of 2Ømm.

[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.

[TOOLDEF

This directive sets the length and diameter of a cutting tool. The length of a tool is the distance from the spindle nose to the bottom of the cutter.

Example:

G21

[TOOLDEF T1 D8 Z65

This example defines tool number 1 as being 8mm in diameter, and 65mm long.

DENFORD DIRECTIVES.

[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.

[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.

[EDGEMOVE

This directive will move the edges of the billet relative to the workpiece datum point. For the purposes of graphical simulation, the workpiece datum point is usually set as the lower front corner of the billet.

[EDGEMOVE is useful when a workpiece datum point written into a G-code program does not match this default (lower lefthand front corner) position on the billet.

It moves the graphic display of the billet so the workpiece datum point in the graphical simulation aligns with the workpiece datum point in the program.

Example:

G21

[EDGEMOVE XØ Y-2Ø

This example moves the billet Ømm on X and -2Ømm on Y from the workpiece datum point.

DENFORD DIRECTIVES.

!

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 1Ømm BORDER

This example would print the line "NOW CUTTING 1Ømm BORDER" in the tutorial messages window in the lower part of the screen.

?

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 THAT A 6mm SLOT DRILL IS PRESENT

This example would print the line "CHECK THAT A 6mm SLOT DRILL IS PRESENT" 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.

ALLEN HEAD	A hexagon shaped hole on the head (top)of a set screw. These are tightened/loosened using 'allen keys'.
AXIS (AXES)	The planes of movement for the cutting tool, usually referred to as X (horizontal left and right, parallel to the front edge of the table), Y (horizontal forward and backwards, parallel to the side edge of the table) and Z (directly vertical). Combinations of all 3 allow precise co-ordinates to be described.
CNC	Computer Numerical Control.
CO-ORDINATES	Positions or relationships of points or planes. Co-ordinates are usually described using three numbers referring to the (X,Y,Z) axes, e.g. the co-ordinate (23,35,45) means X axis = +23 units, Y axis = +35 units and Z axis = +45 units.
CUTTER SPEED	The velocity of the cutting edge of the tool relative to the workpiece. With circular tools, the cutting speed is related to the tool when new (maximum cutting diameter). Usually the effect of feedrate is ignored.
CYCLE	A sequence of events or commands.
DATUM	The point (co-ordinate) from which a series of measurements are taken.
DATUM PLATE	The L-shaped bracket used to help locate pieces of work in position on the machine table.
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.
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. For milling and drilling, the feedrate applies to the reference point on the end of the axis of the tool.
FILE	An arrangement of instructions or information, usually referring to work or control settings.
HARDWARE	Equipment such as the machine tool, the controller, or the computer.
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).
MACHINE CODE	The code obeyed by a computer or microprocessor system with no need for further translation.
MACHINE DATUM	The Machine Datum (or reference) point is the co-ordinate set automatically by the machine so it can relate the position of its cutting tool to the 3 slides (axes). The machine must be instructed to search for this point when it is first switched or the power supply is interrupted during a machining session. When set, it is the point from which the machine takes all its measurements (displayed on screen).
MACHINE TABLE	See table.

MITEEBITE CLAMP	Method of securing work to the machine table, using the series of machine table T channels, see section 3.5.
NC	Numerical control.
PC	Personal computer.
PROGRAM	A systematic arrangements of instructions or information to suit a piece of equipment.
RAPID TRAVERSE	Fast movement of the cutting tool through the 3 machine axes between cutting settings.
REFERENCE POINTS	The machine has 3 reference points (hidden microswitches) used in setting the limits of movement for its slides (axes).
RPM	Revolutions per minute (rev/min) - a measure of spindle speed.
SLIDES	The 3 machine axes - see axis.
SPINDLE SPEED	The rate of rotation (velocity) of the machine head/ cutting tool, measured in RPM.
SOFTWARE	Programs, tool lists, sequence of instructions etc.....
SUB-TABLE	A secondary table, clamped to the actual machine table. The work is then fastened to this secondary table. Used as a safety feature to prevent damage occurring to the actual machine table, should a problem occur when milling. E.g. A sheet of MDF. Sometimes referred to as a 'temporary bed'.
T CHANNEL	There are three slots, or channels (upsidedown 'T' shapes), which run horizontally along the machine table (parallel with the X axis) just under the surface. They are used when fitting the datum plate and clamps in position on the machine table.

TABLE	The horizontal platform upon which work is secured, sometimes referred to as the 'bed' or the 'machine table'.
TEE-NUT	An upsidedown 'T' shaped block found on clamps which fit into the T channels on the machine table.
TOOL OFFSET	When machining, allowances must be made for the size of tools being used, since they all differ in length. The tool offset is the amount the Z value must be moved (or offset), so that all the different cutting tool tips used line up with each other, on the surface of the piece of work being machined.
TRAVERSE	Movement of the cutting tool through the 3 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'.
WORK DATUM	Before machining can commence, the starting co-ordinate (or datum) for the cutting tool must be moved so it exactly matches the starting point position on the piece of work. This point is the Work Datum (X and Y values only since the Z value depends on the length of tool being used - see Tool Offset).