

Novamill CNC
Milling Machine &
Mill CAM
Designer
Training Course

Denford Limited
Birds Royd
Brighouse
West Yorkshire
HD6 1NB
Tel: + 44 (0)1484 712264
Fax: + 44 (0)1484 722160
EMail: education@denford.co.uk
<http://www.denford.com>

CONTENTS.

Introduction

Contents	i
Welcome to Denford Limited	ii
Denford Limited Location Map	iii
Training Schedule	iv
Training Module Outcomes	v

Notes

A selection of graph paper and blank pages are included for your notes.

Bookmark Project

Equipment required and Mill CAM Designer Settings	1
Starting MillCAM Designer	2
The Material Selection Window	3
The Main Drawing Window	4
Opening a File and Page Set-up	5
Setting the Cutter and Depth of cut	6
Designing the Template	7
Designing and using the Template	8
Drawing the Design	9
The Drawing Element Buttons	10
Creating the G-code (CNC) File	11
Starting the CNC Machine Controlling Software	12
Setting the Machine Datum	14
Changing the Cutting Tool (ATC and Manually)	15
Clamping the Temporary MDF Bed with Miteebite Clamps	16
Setting the Tool Offsets	17
Setting the Work Datum X and Y Co-ordinates	18
Setting the Tool Offset Z Co-ordinates	20
Setting a Global Z Value for Material Thickness	22
Loading CNC Files	23
Changing the Default Load/Save Directory	24
Simulate a CNC File	25
Run/Pause a CNC File	29
Overriding the Spindle Speed	30
Overriding the Feedrate	31
Emergency Stop	32

- Please note that the information is correct at the time of printing but is subject to change -

WELCOME TO DENFORD LIMITED.

THE DENFORD MISSION STATEMENT.

"Denford is committed to providing quality, innovative and reliable technological solutions to support the education and training needs of current and future generations.

Using some of the world's most advanced technology, Denford aims to provide 'Total Packages', incorporating the highest level of technical assistance in terms of consultancy, equipment, training and courseware support."

TOTAL SOLUTIONS.

Denford is the world leader in the manufacture and supply of computerised machines, systems and software for education and training. A worldwide reputation for quality and technological excellence has enabled Denford to lead the field for over 50 years, tailoring its products to meet the ever-changing demands of schools, colleges, universities and training establishments.

Denford has established itself as a provider of "Total Solutions". Denford products span the complete learning curve, from the unique Computer Numerical Control (CNC) Desk-Top Tutor, revolutionary multimedia software and easy-to-use CAD/CAM packages for teaching the principles of design and manufacture, through to CNC Milling Machines, CNC Lathes, Advanced software and sophisticated Computer Integrated and Flexible Manufacturing Systems (CIM/FMS) enabling the teaching of complex engineering concepts and manufacturing techniques.

Denford is Currently Working in Partnership with BAe Schools Network, BESA, BT, CBI, DATA, DTI, EDEXCEL, EEF, EMTA, Engineering Council, ETEMA, Intel, Microsoft, Mitsubishi, MTTA, Nuffield, RCA, SCET and TC Trust.

THE DENFORD PROFESSIONAL TRAINING AND DEVELOPMENT CENTRE.

Opened in May 1995, the 600 sqm Denford Professional Training and Development Centre is an unrivalled high-tech facility which is active with new initiatives, projects and ideas. The Centre comprises a large training area equipped with computers and the very latest state-of-the-art educational software. There is also an extensive training and demonstration facility housing a wide range of CNC machines and a complete Computer Integrated Manufacturing System. Together with modern conference facilities, the Centre offers the ideal location for professional development programmes, training courses and demonstrations.

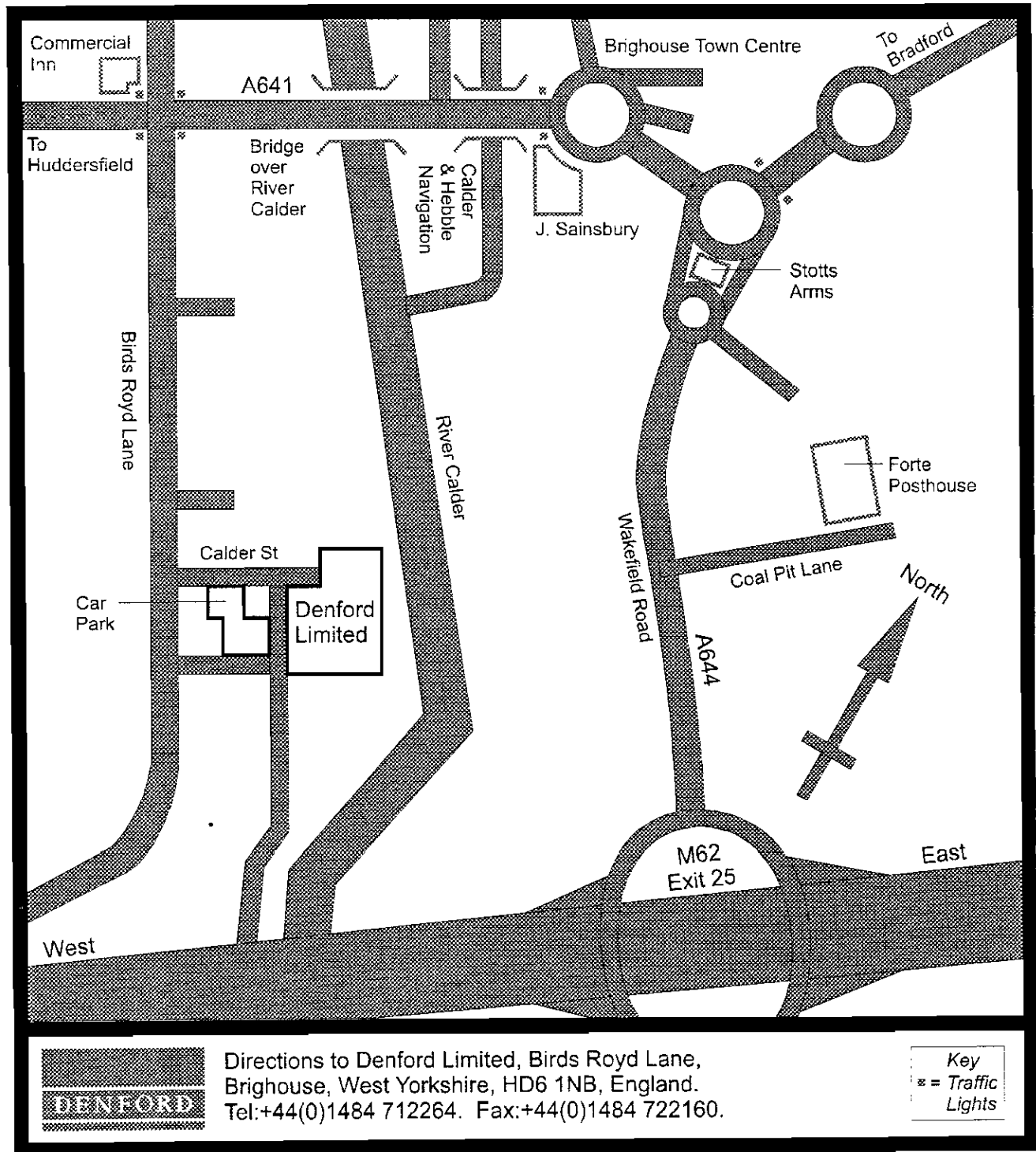


"We built the Centre to fulfil the growing need for in-service training (INSET) and other comprehensive training programmes. It is a measure of our 'Total Commitment' to education. We will endeavour to meet demands for up-to-date information and training on the latest developments in manufacturing technology attracting more students into design, manufacturing and engineering"

- Andrew Denford - Chief Executive

- Please note that the information is correct at the time of printing but is subject to change -

DENFORD LIMITED LOCATION MAP.



- Please note that the information is correct at the time of printing but is subject to change -

TRAINING SCHEDULE.

MODULE 1:

CNC MILLING MANUFACTURING SKILLS - STAGE 1

NOVAMILL CNC MILLING MACHINE & MILL CAM DESIGNER

Venue: Professional Training and Development Centre,
Denford Limited,
Birds Royd,
Brighouse,
West Yorkshire,
HD6 1NB.

Telephone: (01484) 712264.

Fax: (01484) 722160.

e-mail: customer_services@denford.co.uk

website: <http://www.denford.co.uk>

9.15am Registration.
9.30am Introduction to Mill Cam Designer.
11.00am Coffee.
11.15am Preparing a Design for conversion into a CNC file.
Introduction to a Novamill CNC Milling Machine.
12.30pm Lunch.
1.30pm Operating the CNC Milling Machine and simulating CNC
programs.
2.30pm Coffee.
2.45pm Practical Session.
3.30pm Finish.

Denford Customer Services Department:

Nicola Denford Customer Services Director
Kate Heal Customer Services Co-ordinator
Mark Colley Projects Co-ordinator
Peter Bond Training and Commissioning Engineer
Harvey Lamb Training and Commissioning Engineer
John Mitchell Training and Commissioning Engineer

- Please note that the information is correct at the time of printing but is subject to change -

TRAINING MODULE OUTCOMES.

Module Type:

MODULE 1: CNC MILLING MANUFACTURING SKILLS - STAGE 1

NOVAMILL CNC MILLING MACHINE & MILL CAM DESIGNER

Module Title:

Introduction to CAD/CAM (Computer Aided Design and Manufacture) software and CNC (Computer Numerically Controlled) milling machines equipped with Denford controllers.

Enables a trainee to understand the principles involved in CAD/CAM software and a CNC milling machine through 'hands-on' experience.

Previous experience:

No machining experience required, but a general familiarisation with computers (keyboard and mouse skills) and 'Windows 95/98' operating systems is useful.

Time required:

One day.

The trainee will be able to:

Work through a set project demonstrating the full CNC machining process:

- Use a CAD/CAM milling package (Denford MillCAM Designer) to create a design.
- Prepare their design for conversion into a CNC file.
- Identify and operate features on a CNC milling machine.
- Understand the importance of Tool Offsets.
- Manufacture their design on a CNC milling machine.
- Prepare and simulate CNC programs both on-line and off-line.

- Please note that the information is correct at the time of printing but is subject to change -

TRAINING MODULE OUTCOMES.

The trainee will gain knowledge about:

- The CAD/CAM milling package used to create a design (Denford MillCAM Designer).

Preparing the CAD/CAM software for generating a CNC file:

- Choosing the correct CNC milling machine.
- Choosing the correct size and type of material.
- Selecting the correct cutter diameter on elements of the design.
- Selecting the correct depth/s of cut for each tool.
- Organising the correct tool profiles for toolchanging sequences.
- Selecting the correct speeds, feedrates and stand-off heights.
- Understanding the types and uses of cutter available.

Operating the CNC milling machine:

- The layout and components within a CNC milling machine.
- Layout of the control keypad.
- Identifying Safety features.
- Switching on and starting the machine controlling software.
- Datuming the machine.
- Installation of tools / use of toolchanger.
- Familiarisation with the X, Y, Z co-ordinate system of moving.
- Understanding the principles and methods involved in Tool Offsets.
- Mounting and securing the workpiece safely.
- Loading a CNC file.
- Simulating the machining operation (on-line and off-line).

Operating the CNC milling machine:

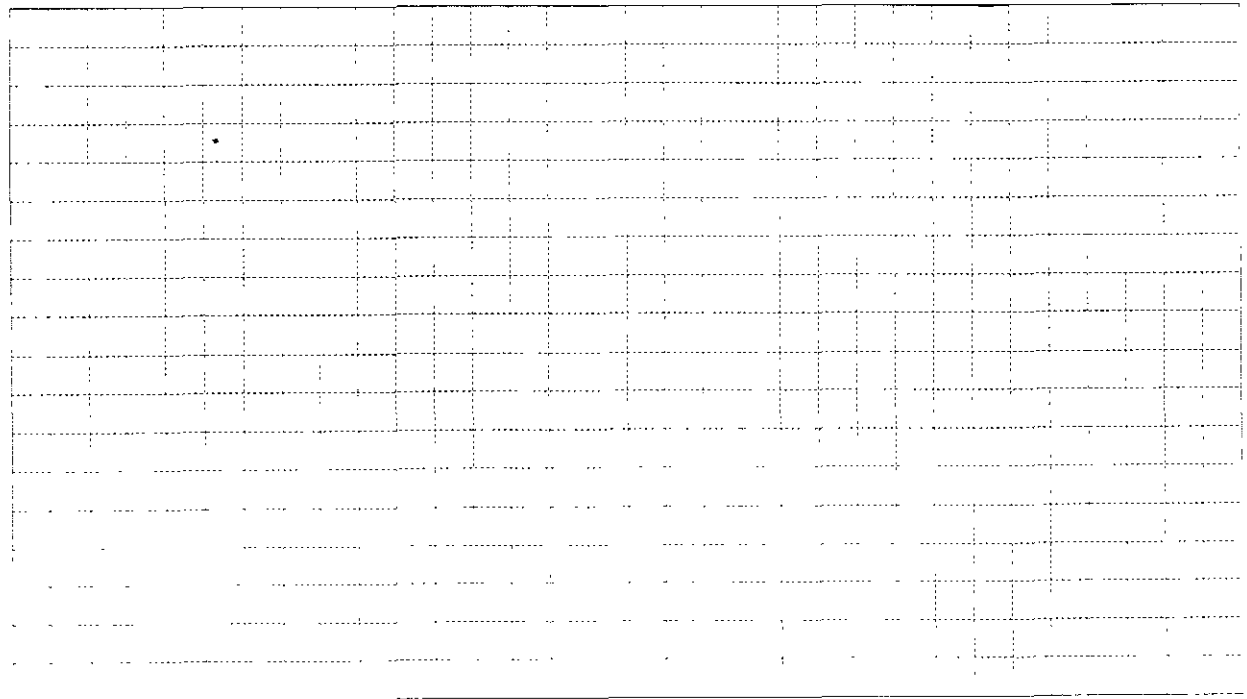
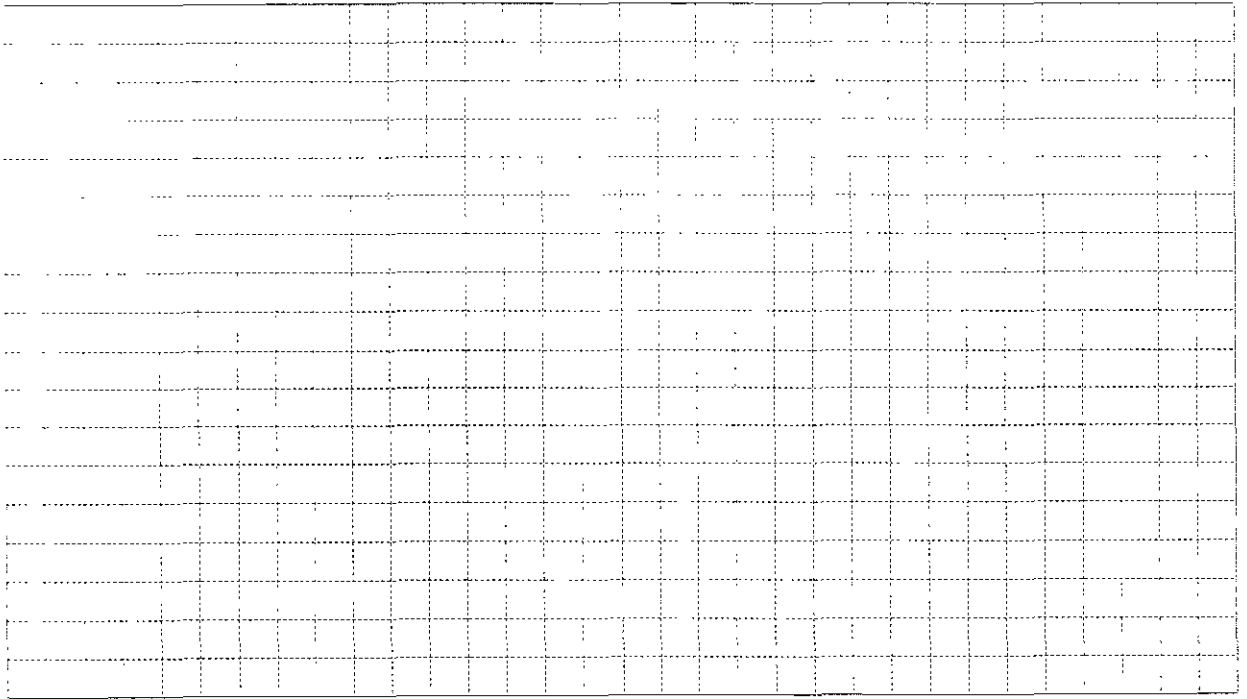
- Starting the machining cycle.
- Manually overriding speeds and feedrates.
- Using the cutting fluid.
- Common problems when machining and how to overcome them.

- The differences between off-line CNC software and on-line CNC machine controlling software.

- Please note that the information is correct at the time of printing but is subject to change -

MILL CAM DESIGNER GRAPH PAPER.

Grid size: 5mm squares.



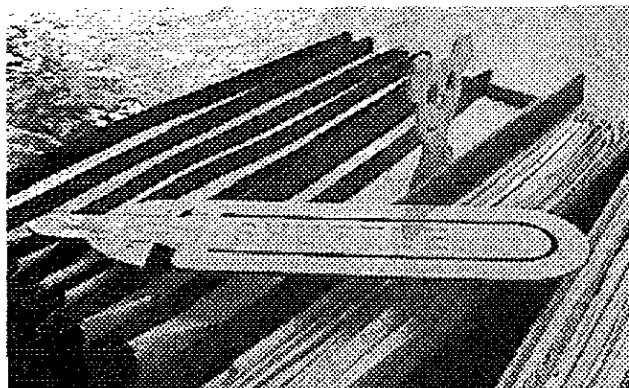
FOR FESD



Training Course
- Bookmark
Project

Denford Limited
Birds Royd
Brighouse
West Yorkshire
HD6 1NB
Tel: +44 (0)1484 712264
Fax: +44 (0)1484 722160
EMail: education@denford.co.uk
<http://www.denford.com>

PROJECT - BOOKMARK.



This project is designed to provide a practical introduction to CAD/CAM software and CNC Milling Machines.

You will use Denford Mill CAM Designer CAD/CAM software to design a simple plastic bookmark, then compile a CNC file.

You will set-up a Denford CNC Milling Machine, load your cnc file, simulate its production, then manufacture your design.

CONVENTIONS USED IN THIS PROJECT GUIDE.

Unless stated otherwise, all mouse clicks are performed using the left mouse button.

Text written in *italics* refers to text, menus or elements you can see on your computer monitor.

Eg, Click the *Line* button - would mean find the button with the word "Line" written on it, position your mouse cursor over the top of it and click once with the left mouse button.

Eg, Click *File|Open* - would mean left mouse click on the "File" menubar, move down the list that appears and left click on the "Open" option.

Text written in [square brackets] refers to keys you should press on your computer keyboard, or Desktop Tutor keypad.

Eg, Press [JOG] - would mean press the key labelled "JOG" on your Desktop Tutor keypad.

EQUIPMENT REQUIRED.

IBM compatible PC with Denford Mill CAM Designer software.

One 3½" floppy disk, containing the template file "bookmark.mcd".

One 2mm diameter slot cutter.

A Denford CNC Milling Machine, equipped with a datum plate, miteebite clamps and a temporary MDF (Medium Density Fibreboard) bed - 225mm x 115mm x 10mm.

One billet - High Density Polystyrene - 90mm x 160mm x 2mm.

MILL CAM DESIGNER SETTINGS.

Project Settings - press [Alt-S] :

Stand off height: 2.00 (mm)

Step size for pocket cycles: 1.00 (mm)

Maximum machining time: 15 (mins)

Grid size: 5.00 (mm)

Step size: 2.50 (mm)

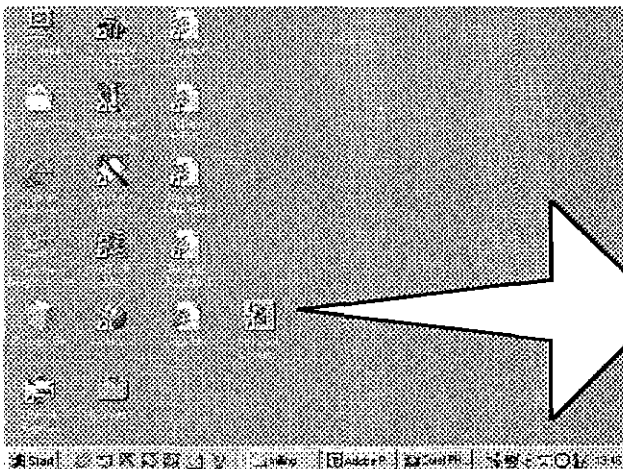
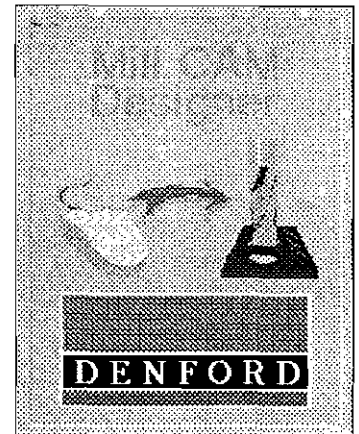
Cutter diameter: 2 (mm)

Cutter depth: 1 (ie, first set of data from list)

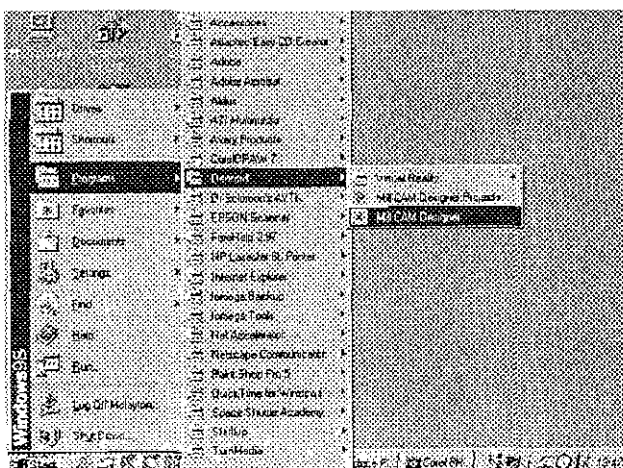
STARTING MILLCAM DESIGNER.

Denford Mill CAM Designer software is a simple to operate CAD/CAM software package. It is designed to help you prepare all the different pieces of information that the CNC Milling Machine will need, in order to successfully manufacture your design, including:

- The size of the piece of material you will use (called the billet).
- The type of material to be used. This will help determine the different feedrates and spindle speeds that will be used when your design is manufactured.
- The different types of cutting tool that will be used.
- The shape of your bookmark.
- The different depths of cut used on each part of your bookmark shape. You may want some parts of your design etched onto the surface of the billet and other parts completely removed.



To start the Mill CAM Designer software double-click the *Mill CAM Designer* shortcut icon on the computer desktop.



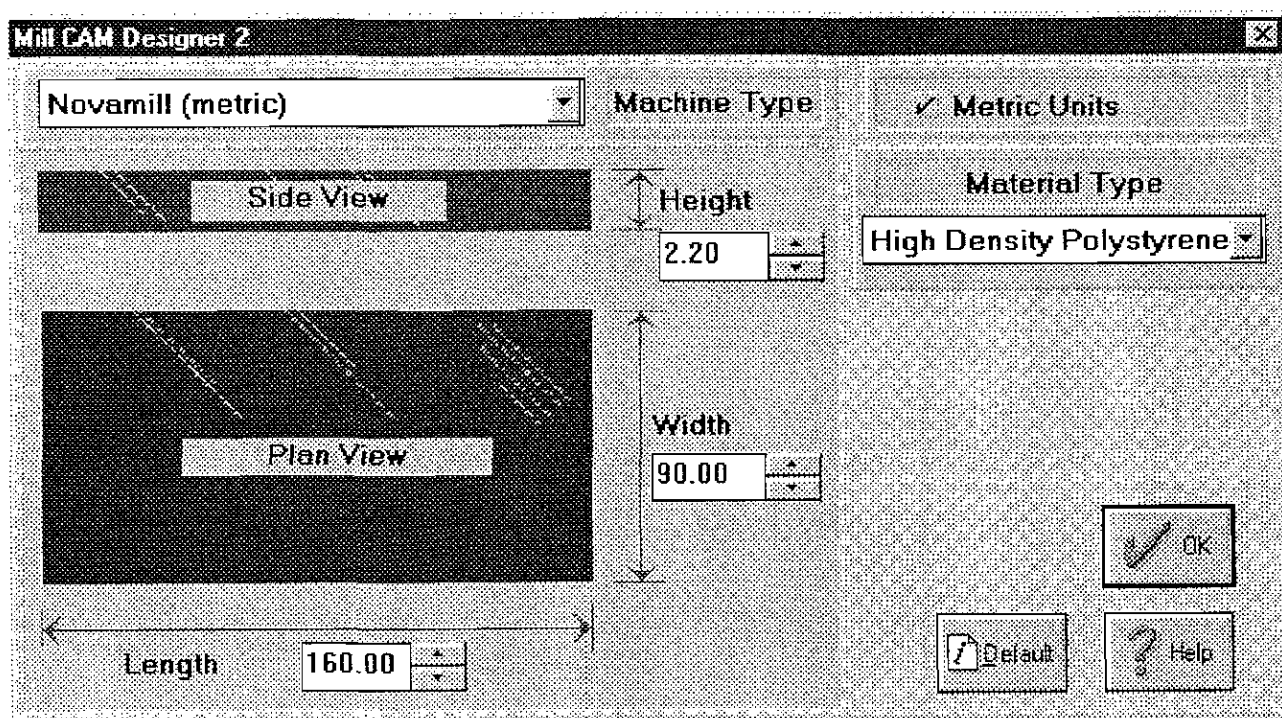
If this shortcut icon is not available, click *Start* on the Windows Startbar, followed by the *Programs* option, the program group *Denford* and finally the *Mill CAM Designer* icon.



MILLCAM - THE MATERIAL SELECTION WINDOW.

Enter the following details into the dialog boxes in the Material Selection window:

- 1) **Machine type** - Click the down arrow button and select the name of the CNC Milling Machine you will be using, together with the (*millimeteres*) units of measurement.
- 2) **Material Height** - Click the cursor in the numbers dialog box, just under the *Height* label and [Delete] any characters present. Retype with a new material height of 2.2 (even though the real billet thickness is 2mm).
You want to cut completely through the billet so this number must be greater than the thickness of the real billet. This overrides a safety feature in MillCAM that normally prevents the cutting tool from milling into the real machine table.
Since the billet will be secured on a temporary MDF bed, at least 10mm thick, the machine table itself will remain safe.
- 3) **Material Type** - Click the down arrow button and select *High Density Polystyrene*.
- 4) **Material Length and Width** - Using the up and down arrow buttons, enter the length as 160mm and width as 90mm.
- 5) **Confirm Details** - Click the OK button to confirm the details you have entered are correct and exit this window.



MILLCAM - THE MAIN DRAWING WINDOW.

The main drawing window has five main areas.

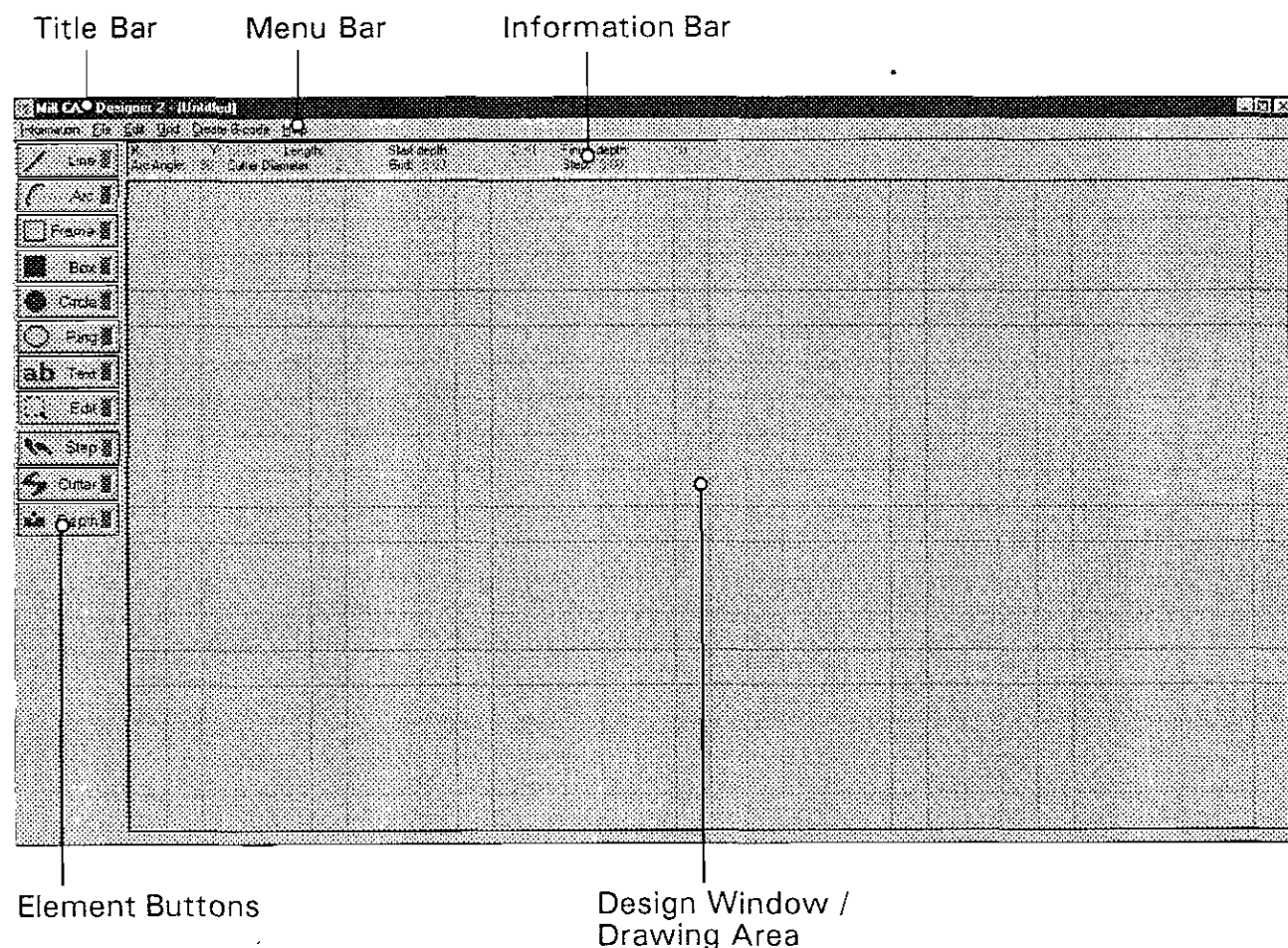
Design Window: Set at the start of the program by using the material size screen, as shown on the previous page. The Design window is where your design is drawn and displayed, in plan view.

Menu Bar: Click each title to display the dropdown menu, move your mouse cursor down the list and click on the required (highlighted) option.

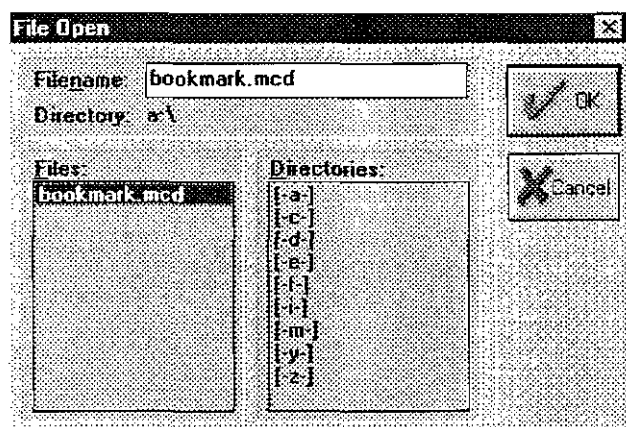
Title Bar: This shows the program name 'Mill CAM Designer 2' and the file name if you have previously saved the drawing.

Information Bar: This provides a continuously updated read-out of useful information. Details given include the current cursor position, depth of cut selected, diameter of the machine tool and grid size.

Element buttons: Each button represents a simple function or element that can be performed on your design.



MILLCAM - OPENING A FILE.



- 1) Insert your 3½" floppy disk into the floppy disk drive (A:).
- 2) From the menubar, click *File | Open*.
- 3) The *File Open* dialog window will appear.

Mill CAM Designer is usually set, by default, to check for files on 3½" floppy disk.

The message text *Directory a:* should be displayed under the *Filename* dialogue box.

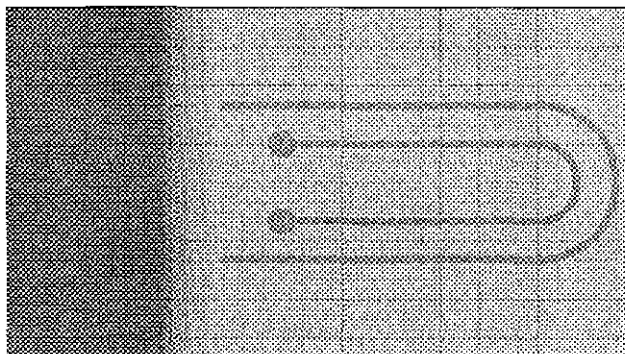
If not, double-click on [-a-] in the *Directories:* table.

- 4) Click the text *bookmark.mcd* in the *Files:* table.
- 5) Click the *OK* button to load the file.

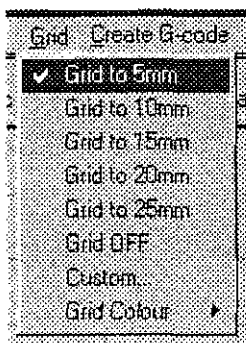
What you see on your screen is a part finished design.

The righthand two thirds of the design window contains the clip part of the design. This is the part that fits over the book pages in a similar way to a paper clip.

The lefthand third of the design window (shown dark grey on the screenshot) is left blank - this is where you will place your design !!



MILLCAM - PAGE SET-UP.



Changing the background grid size.

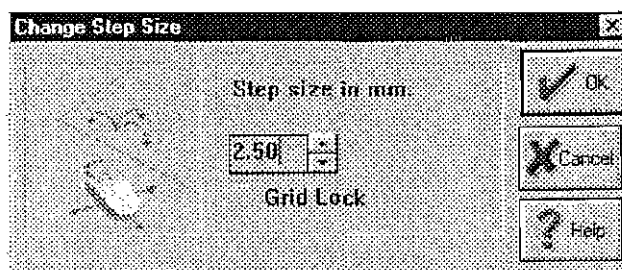
From the menubar click *Grid | Grid to 5 mm*. This will change the background grid to 5mm squares.

Changing the step size.

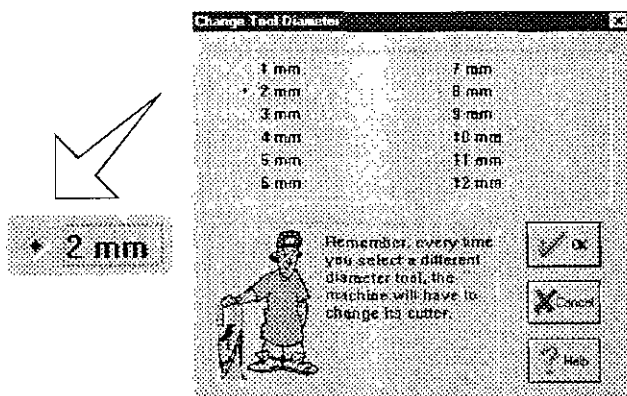
Click the *Step* button. The step size determines how accurately you can place lines on your design. The smaller the number, the more freedom you have over where to place any lines. Click inside the *numbers* dialog box, [Delete] the numbers and enter a new step size of 2.50.

Check that a tick mark is not present in the *Grid Lock* checkbox. Click the *OK* button to confirm.

Remember, if you need more accuracy when placing lines later on in your design, you can always make this



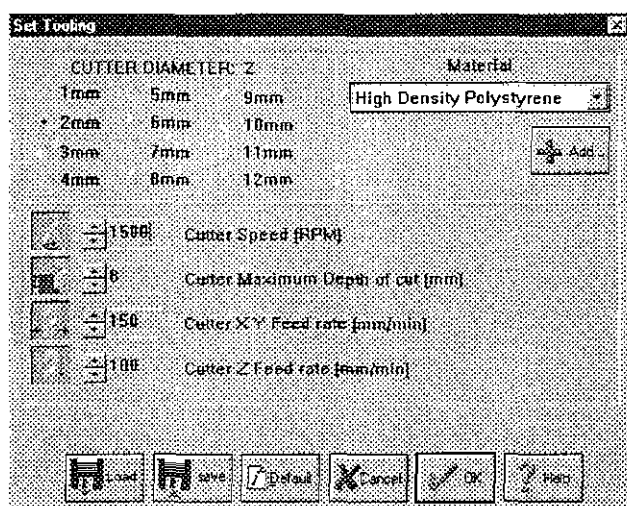
MILLCAM - SETTING THE CUTTER AND DEPTH OF CUT.



Changing the cutting tool size.

Click the *Cutter* button. In the *Change Tool Diameter* window, click the 2mm diamond marker so it is selected. Click the OK button to confirm.

Any lines placed in the design window will now be drawn using a 2mm line thickness (ie, the diameter of the cutter).



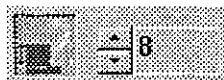
Make file

Ctrl+G

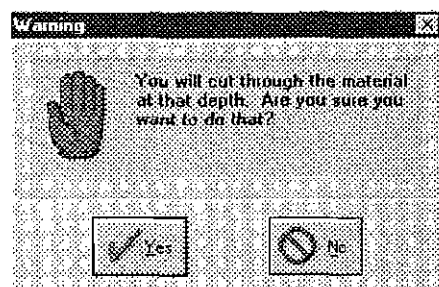
Set G-code parameters...

Setting the G-code Parameters.

From the menubar, click *Create G-code* | *Set G-code parameters* to display the Set Tooling window.



Using the up arrow button, increase the *Cutter Maximum Depth of cut (mm)* to 8. Click the OK button to confirm.



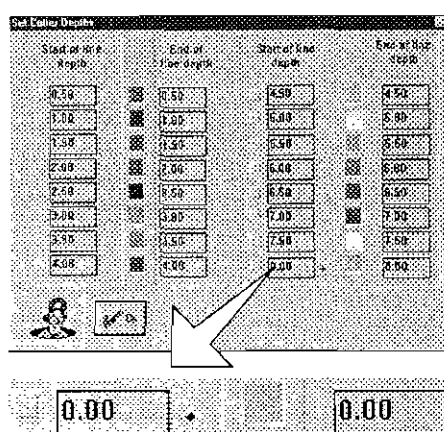
Changing the cutting tool depth.

Click the *Depth* button. In the *Set Cutter Depth* window, click the diamond marker next to the orange square so it is selected (this is the last colour in the list).

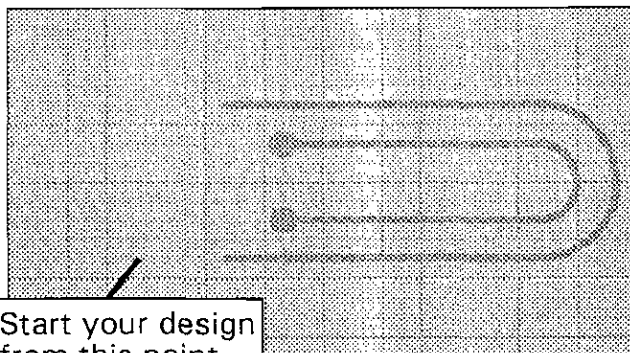
Click the Yes button on the Warning window (cutting through the billet) that is displayed.

Change the *Start* and *End* line depths so they both read 0.00 (zero). Click in each of the respective boxes, [Delete] the 8.00 numbers, then retype with the new numbers. Click the OK button to confirm.

Any lines now placed in the design window will be orange in colour to show they have a cutting depth of 0mm into the surface of the material. These lines will be used to draw a template - this will help you draw all the components of your design in the correct order.



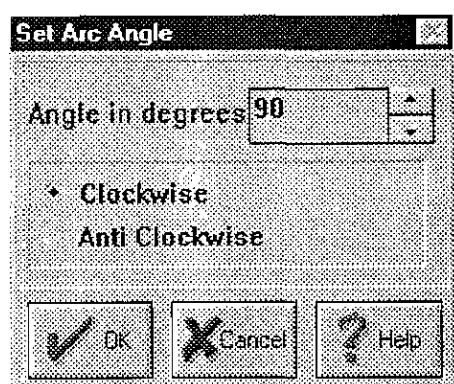
MILLCAM - DESIGNING THE TEMPLATE.



A template is useful for helping to plan and draw your complete design.

At the moment, all the elements you place in the drawing area will have a cutter depth of zero.

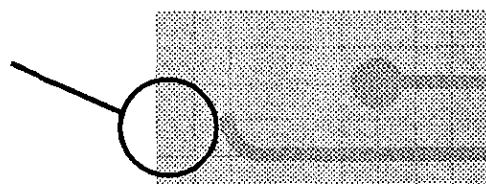
When the final g-code file is created these lines will be ignored, since the actual machine cutter wouldn't be performing any useful function by following these lines - they don't cut any of the material !!



Time to start drawing....

There are seven drawing element buttons, *Line*, *Arc*, *Frame*, *Box*, *Circle*, *Ring* and *Text* listed down the right-hand side of the drawing area. Click a drawing element button and start placing elements in the drawing area.

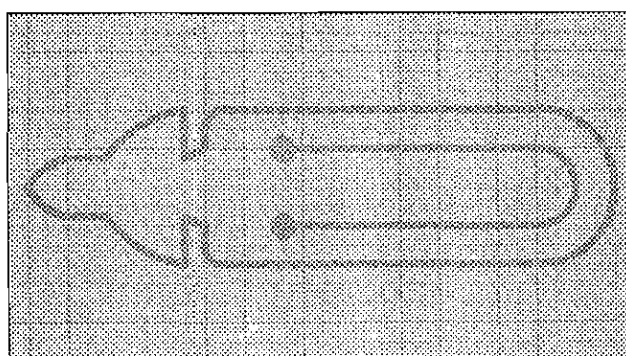
Example, drawing an arc.



For example, to draw an arc, click the *Arc* button. Click once in the drawing area. The *Set Arc Angle* window will appear. Select the angle of arc you require and its direction, then click the *OK* button. Position your mouse cursor at the starting point for the arc, click and hold the mouse button down. Drag the cursor out to the arc end point, then release the mouse button to finish the arc.

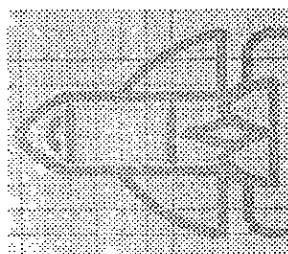
If you make a mistake, remove the last part of an element by clicking the right mouse button.

Try using all the different elements !! A guide to using the drawing element buttons is on page 11.



Draw your bookmark outline.

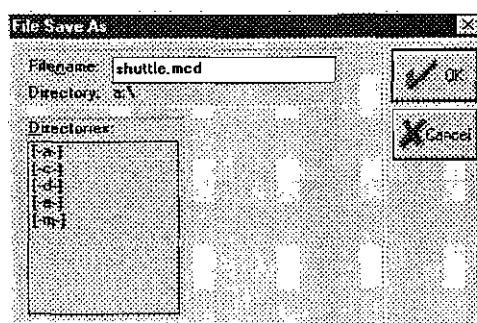
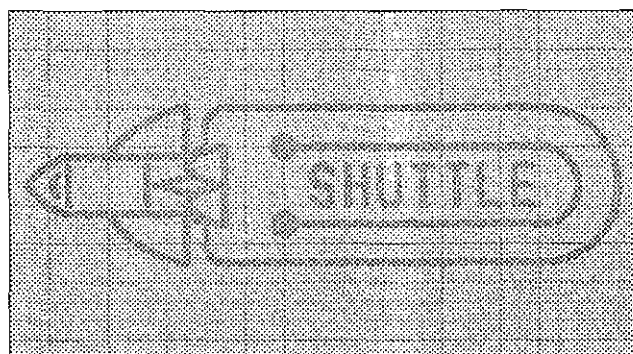
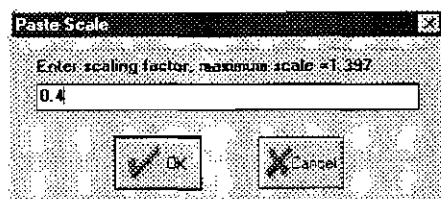
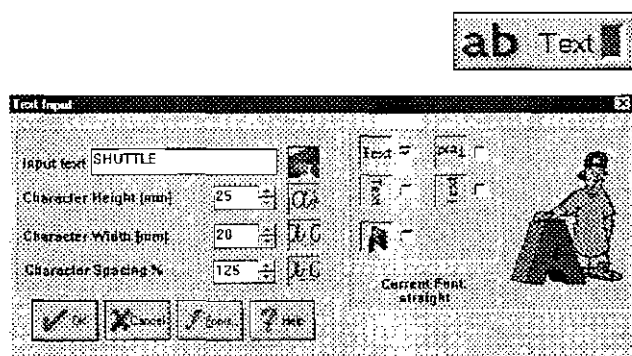
Add any internal details.



Build up the outline of your design. What shape do you want at the top of your bookmark? Our design shows the beginnings of a space shuttle design.

Next, fill in any detail areas to make your design look more realistic. In the example shown we've added a mixture of arcs and lines to depict the rockets, fuselage and windows of our shuttle design.

MILLCAM - DESIGNING AND USING THE TEMPLATE.



To add text to your design, click the *Text* button. Click once in the drawing area. The *Text Input* window will appear. Enter your text into the *Input Text* dialog box. You can alter the height and width of letters, spacing between letters, the direction in which the letters are drawn and the type of font (typeface) used, all from this window. Click the *OK* button to confirm your choices.

Note - text takes quite a long time to machine out, so only use one or two words !!

The *Paste Scale* window will appear. The maximum scale stated is the number you must type in if you wanted the text to fit exactly in the drawing area (ie, the largest possible size). Type in a suitable scale and click the *OK* button to continue.

The text will appear "ghosted" in the drawing area. Move the text to required location and click the left mouse button to place the text, or the right mouse button to discard the text.

Your template should now be complete. It makes sense at this stage to save your work.

From the menubar, click *File | Save As*. The *File Save As* window will appear. Type in a name for your Mill CAM Designer file. Note the location where your Mill CAM Designer file will be saved - you may need it later !!

Click the *OK* button to save your file. Note that Mill CAM Designer files are saved with the file extension .mcd, for example, "Shuttle.mcd".

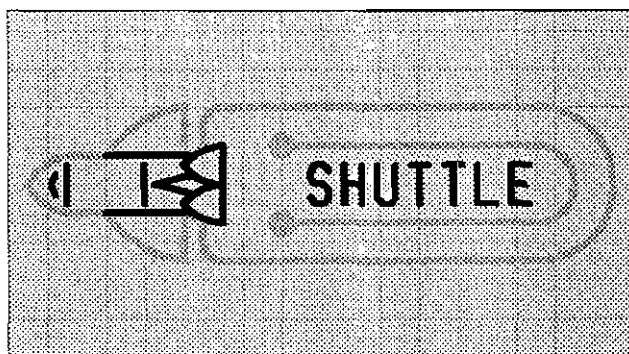
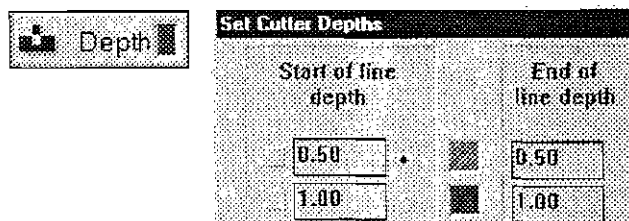
The most important aspect of creating a template is it allows you to redraw all the elements that make up your design, at their correct depths and in the correct order. **IMPORTANT !!**

The order you place all the different elements in the drawing area is the exact order that the milling machine will use when moving its cutting tool.

This includes all the start and end points for any elements such as lines and arcs. So it makes sense to plan your design very carefully - ideally the end point of any drawn lines should be the start point of the next drawn line.

Bearing this in mind, the very last part of your design that should be cut out is the outside profile, since only a much smaller area of double sided tape will be left to hold the plastic onto the temporary MDF bed.

MILLCAM - DRAWING THE DESIGN.

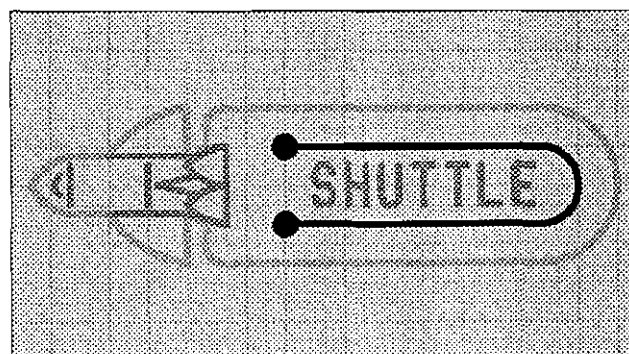
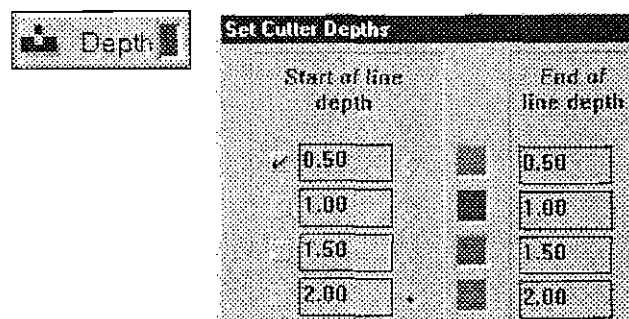


The first areas of your design that should be redrawn are any parts to be etched into the surface, along with any text areas.

Click the *Depth* button. In the *Set Cutter Depth* window, click the diamond marker next to the red square so it is selected. The *Start* and *End* line depths must read 0.50mm. Click the *OK* button to confirm.

All lines that you now draw will end up being cutting operations that etch 0.5mm into the surface of the plastic billet.

Redraw all the detail and text, using your template as a guide to positioning. On our example left, these areas are shown by the black lines. Remember to plan the best route for the cutting tool, to save machining time.



The last part of your design that should be cut out is the outside profile, since only a much smaller area of double sided tape will be left to hold this plastic to the temporary MDF bed.

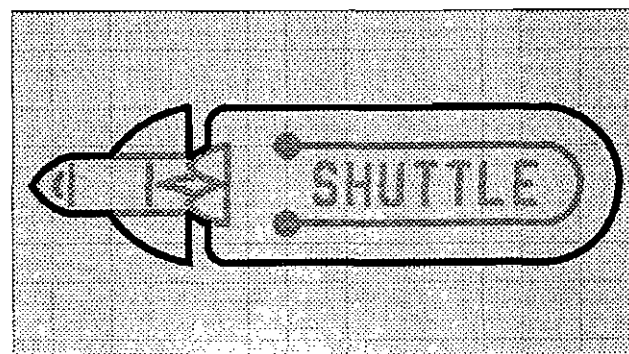
Click the *Depth* button. In the *Set Cutter Depth* window, click the diamond marker next to the green square so it is selected. The *Start* and *End* line depths must read 2.00mm. Click the *OK* button to confirm.

All lines that you now draw will end up being cutting operations that mill 2mm into the plastic, ie completely through the billet.

Draw the two rings and the U shape around the text first. Next, redraw the outside profile of your bookmark design - it should be one continuous line. On our examples, left, these areas are shown by the black lines.

Finally save your finished Mill CAM Designer file, using *File|Save* to overwrite the previously saved file.

Use *File|Save As* if you want to give a different name to your file. this is useful if you want to keep a backup of your original Mill CAM Designer template.

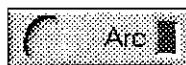


MILL CAM DESIGNER - THE DRAWING ELEMENT BUTTONS.

The Drawing Element Buttons:



Line: Click and hold at the starting point of the line, then drag the line out and release the mouse button when the end point is reached.



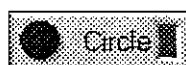
Arc: Click once in the drawing area and select the arc angle and direction from the dialog box, then click the *OK* button. Click and hold at the starting point of the arc, then drag the arc out and release the mouse button when the end point is reached. Right mouse click whilst dragging the arc will reverse the arc direction. Using the [up] and [down] arrow keys whilst dragging the arc will change the arc angle.



Frame: Click and hold at the starting point. Drag the frame out until it is the required size (the Information bar shows the distance between opposite corners), then release the mouse button.



Box: Same as the frame element (above), only the entire area will be milled out.



Circle: Same as the ring element (below), only the entire area will be milled out.



Ring: Click and hold the when the pointer is at the desired centre point of the circle. Drag the circle outwards until the correct radius is achieved (the radius is shown on the Information Bar), then release the mouse button.



Text: Click once in the drawing area. A dialog box will appear. Enter the required text. Select the checkboxes for position, mirroring, height, width and spacing. Click the fonts button and select a font. Click the *OK* button and choose a scale (the max scale size is shown). Position the text and left mouse click when the text is over the desired location.

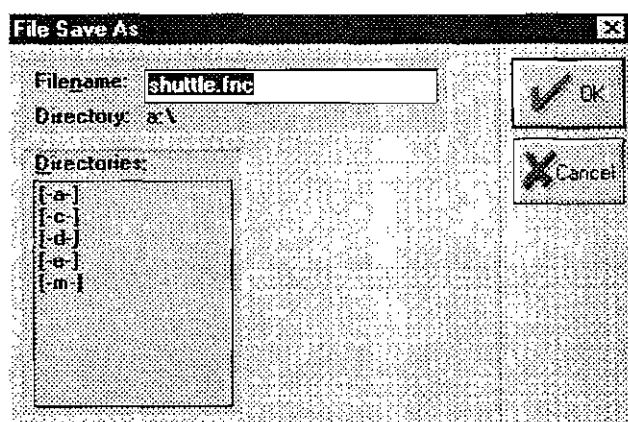


Edit: Elements can be edited using a number of different methods.

- 1) Click on the drawing area, hold the mouse button down and drag the dotted line marquee box around the elements to select them.
- 2) Right click on the drawing area. This cycles around all the individual parts of the elements, in the order they have been drawn.
- 3) From the *Edit* menu, click *Select All* to highlight all the elements on the screen.

From the menubar, click *Edit|Copy* to copy the elements to the clipboard, or *Edit|Cut* to cut the elements to the clipboard. To paste from the clipboard click *Edit|Paste*, select a new scale, then use the left mouse button to place when over the desired location, or the right mouse button to discard the elements.

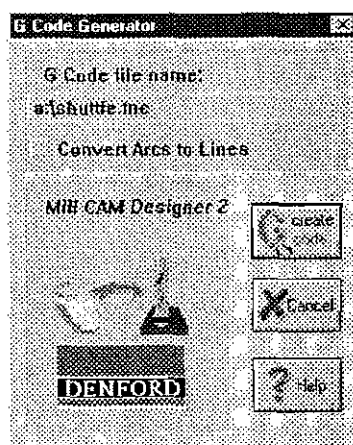
MILLCAM - CREATING THE G-CODE (CNC) FILE.



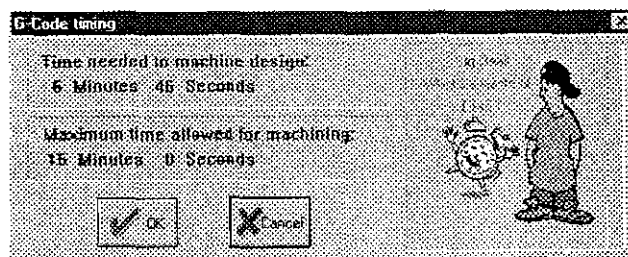
The G-code file, also called the CNC file, is a series of instructions that the cnc machine can understand. Essentially, it relates to all the different types of cutting tool movements, feedrates and speeds required to manufacture your design.

To create a G-code file, from the menubar, click *Create G-code | Make File*.

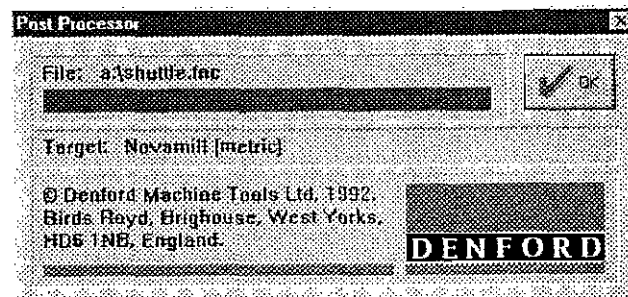
The *File Save As* window will appear. Type in a name for your G-code file. If you have saved your Mill CAM Designer file previously, its name will appear in the dialog box. Click the *OK* button to begin the process of converting your design into a g-code file. Note that G-code files are saved with the file extension .fnc, for example, "Shuttle.fnc".



The *G Code Generator* window will appear. Confirm that you want to send your design to the post-processor (the software that converts your design) by clicking the *Create G-code* button. Note the location where your G-code file will be saved - you will need it when you want to machine your design !!



A *G Code Timing* window will appear. This window will display the time that the cnc machine will need in order to manufacture your design. Click the *OK* button to continue.



The *Post Processor* window will display the conversion progress of your design into a G-code file. Click the *OK* button when the process has finished.

You are now ready to take your g-code file to the CNC Milling Machine.

MANUFACTURING YOUR BOOKMARK DESIGN.

You will need your cnc file, generated using the Mill CAM Designer post-processor, on 3½" floppy disk.

This section will help you:

- Start your CNC Milling Machine.
- Set the Machine Datum point and prepare the tooling.
- Prepare and clamp the billet.
- Set the Tool Offsets.
- Load and simulate the production of your design.
- Manufacture your design.

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

STARTING THE CNC MACHINE CONTROLLING SOFTWARE.

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

To switch **on** the 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 will automatically load if installed on the computer hard disk.

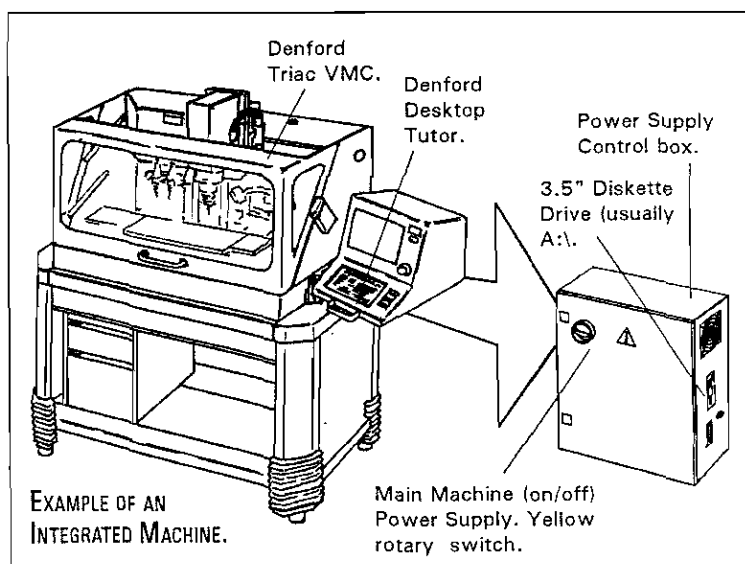
If the controller software is supplied on floppy (3.25 inch) disk, insert the disk into the floppy drive on the power supply control box before switching on the machine.

To switch **off** the machine, exit the machine control software.

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.

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....



STARTING THE CNC MACHINE CONTROLLING SOFTWARE.

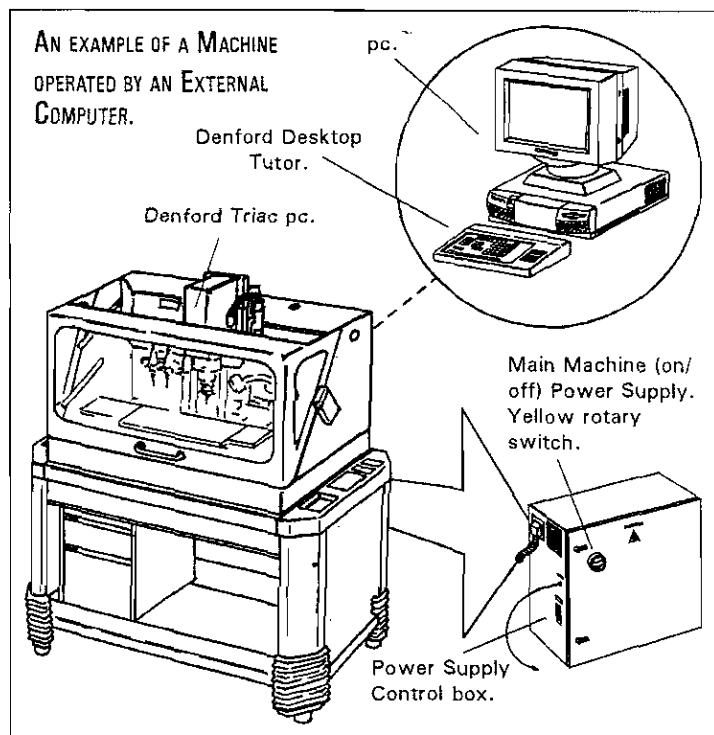
A **pc operated machine** has 3 main components:

- 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 right).

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 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.

To switch **off** the machine, exit the machine control software.

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.

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....

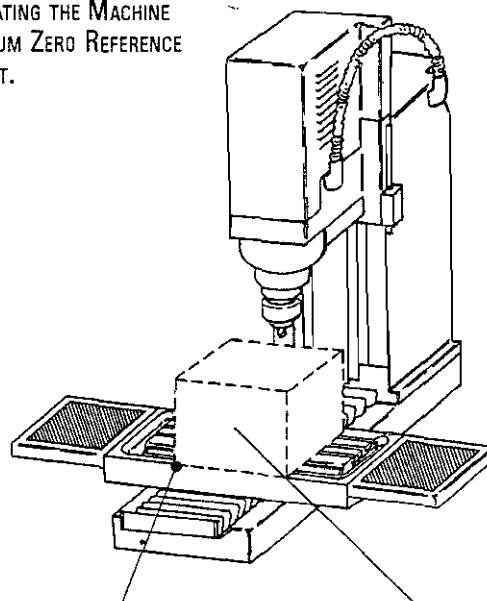
SETTING THE MACHINE DATUM.

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.

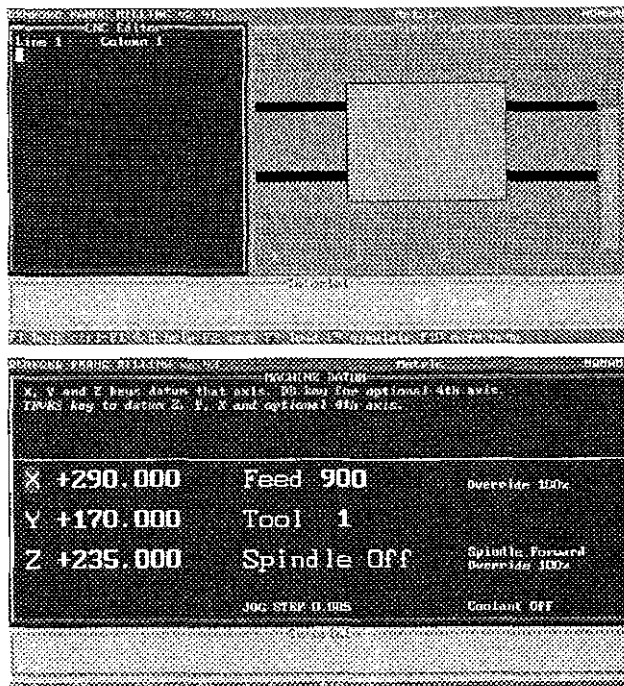
The machine datum point is used when taking any measurements from future co-ordinates we load or program. This is fine, if the start co-ordinates of our work coincide with the machine datum zero reference point. If not, this point must be moved when setting the tool offset for the machine.

LOCATING THE MACHINE
DATUM ZERO REFERENCE
POINT.



Front, left, lower corner of the block is the machine datum zero reference point, with the co-ordinates $X=0$, $Y=0$, $Z=0$.

Maximum machining size shown by 'imaginary' block.



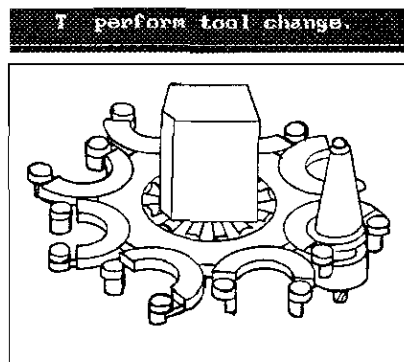
On loading up the DENFORD FANUC MILLING software, the start up screen will be displayed.

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

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.

CHANGING THE CUTTING TOOL - ATC.



The automatic tool changer (ATC) 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 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. Note there should never be a cutting tool in the carousel pocket directly facing the spindle.

CHANGING THE CUTTING TOOL - MANUALLY.

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) and press the [T] key to obtain the Tool number screen prompt. Enter the number of the tool you wish to change to and press the [EOB] key to confirm this operation is correct.

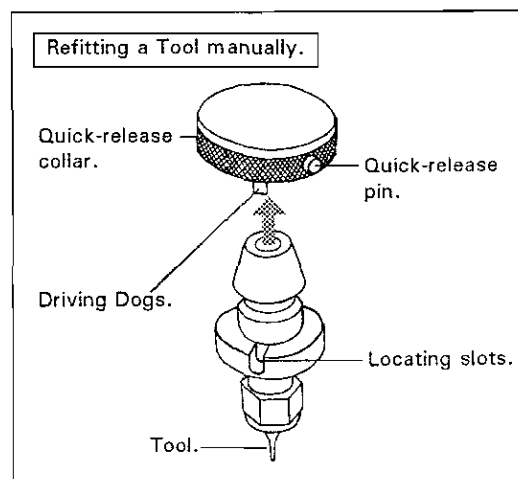
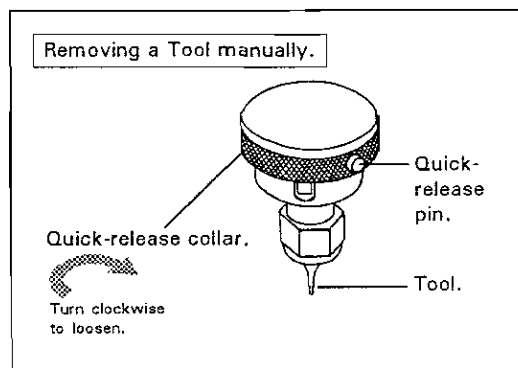
Note that when a *program* using more than one tool is run in future, the software will prompt you to manually change tools before continuing.

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.

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

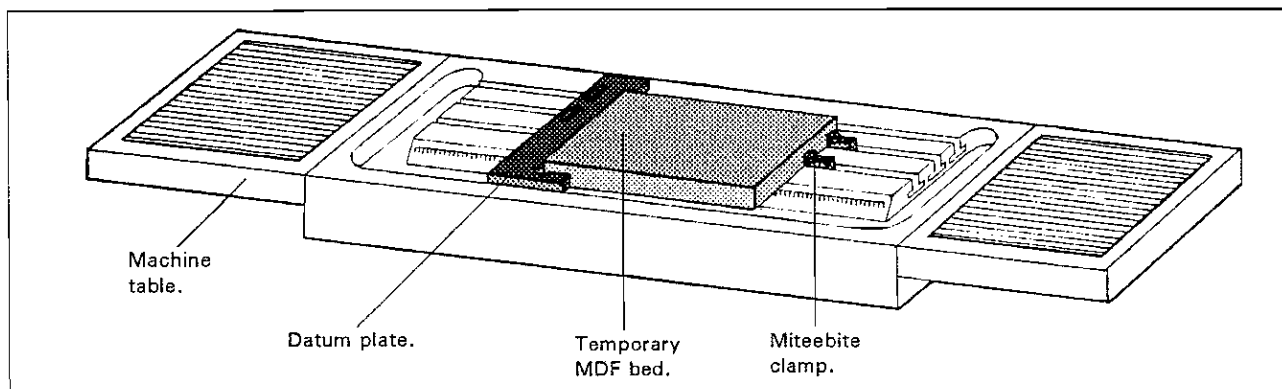
Finally, press the [EOB] key to confirm that the tool change is now complete.

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.



CLAMPING THE TEMPORARY MDF BED WITH 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 - the plastic billet will be held on the temporary MDF bed using double sided tape.



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').

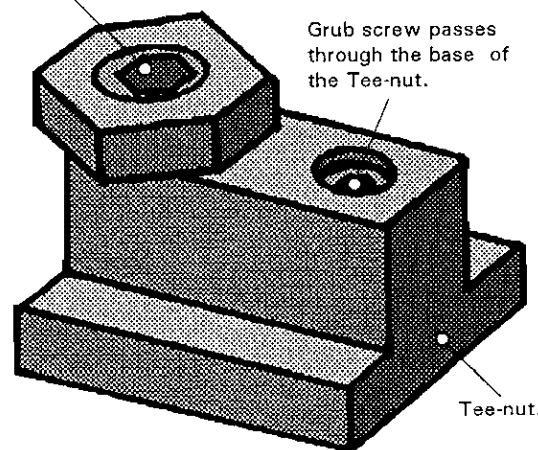
Continual turning of the bolt is unnecessary, since the full range of movement for the hexagon washer is covered in a single 360 degree rotation of the bolt. In this respect, the hexagon washer will not tighten further if the bolt is continually turned clockwise.

Miteebites are ideal for small repeatable projects using a temporary MDF bed to hold the workpiece. The temporary MDF bed can be continually withdrawn from the machine table, then replaced, always to the same position.

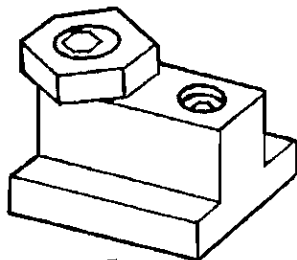
Hexagon washer.
Bolt - both the head and the allen key holes are machined slightly 'off centre'.

To loosen grubscrew turn anticlockwise.

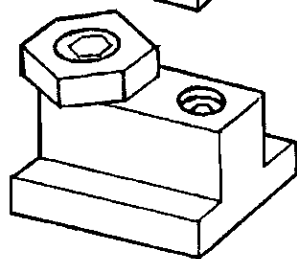
To tighten grubscrew turn clockwise.



Hexagon washer set in open position.



Hexagon washer set in closed position.



SETTING THE TOOL OFFSETS - INTRODUCTION.

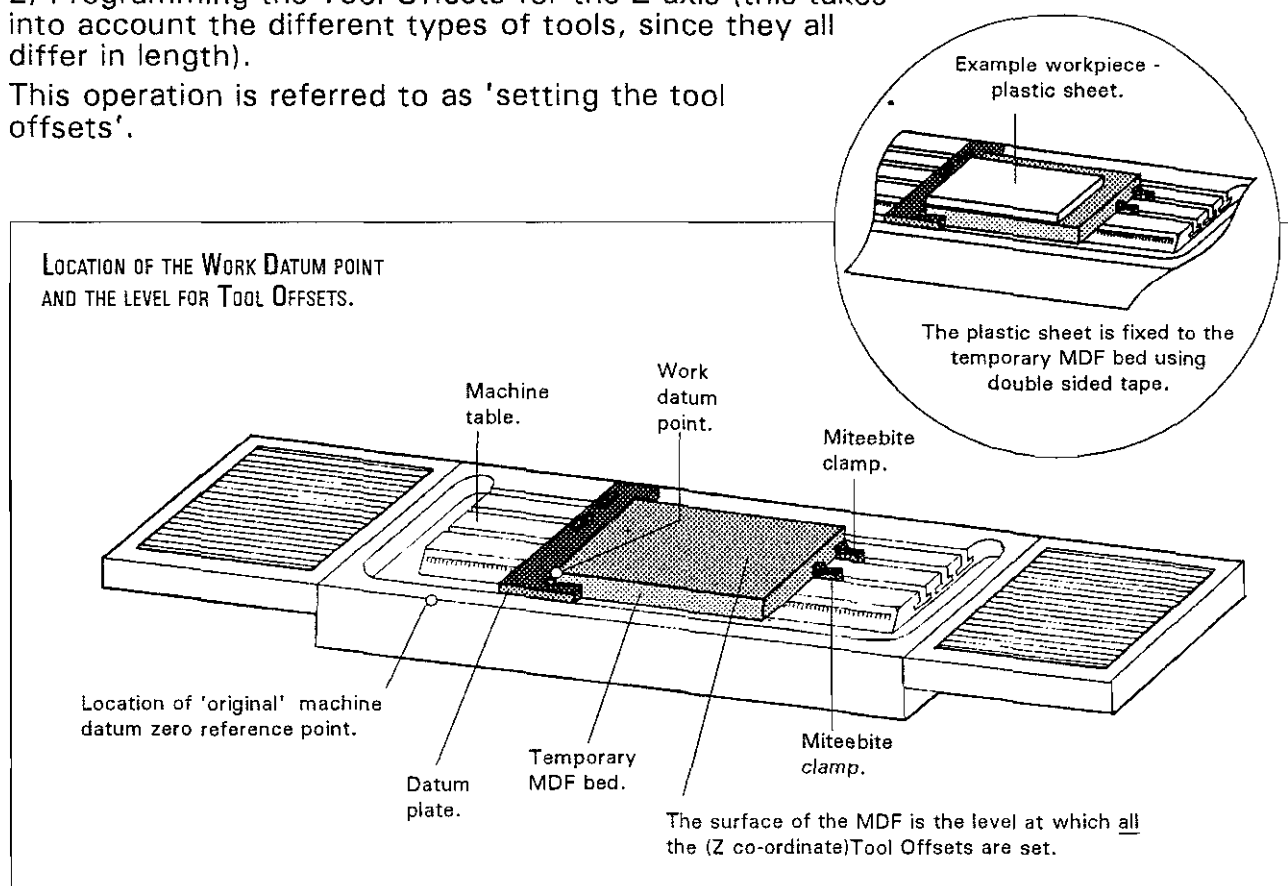
Quite often, the workpiece is much smaller than the 'imaginary block' used in setting the machine datum point. For example, a typical piece of work could be a small sheet of plastic fixed to a temporary MDF bed with double sided tape (to prevent damage occurring to the table when machining) as shown 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 referred to as 'setting the tool offsets'.



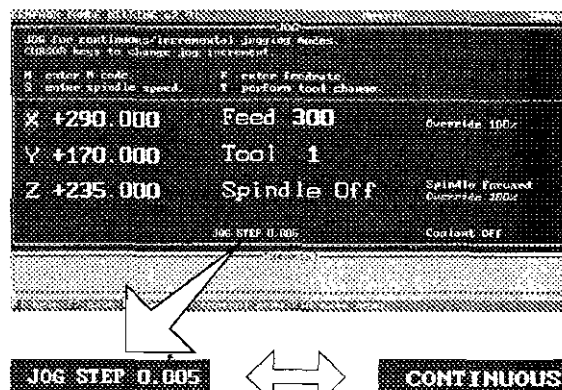
SETTING THE WORK DATUM X AND Y CO-ORDINATES.

Ensure that a 2mm tool is set in the machine head spindle (this needs to be tool 1) before proceeding to set the tool offsets. For details on how to change tools, please see the page 15.

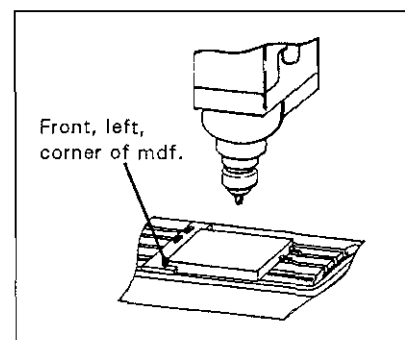
Warning - Before beginning to set the X and Y Offsets, ensure the X and Y values in the Tool Offsets Table are set to Ø (zero).

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 *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).

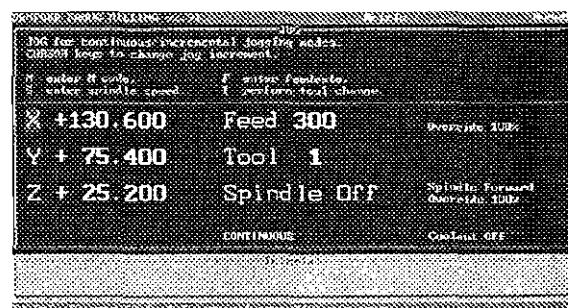


Before moving the table or head in any direction, you need to set the movement to 'continuous' (ie, the table or head will move continuously, so long as one of the movement keys is being pressed). To do this, press the [JOG] key again until the screen shows the words 'continuous' in the lower area of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

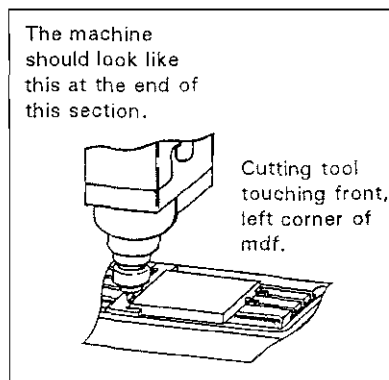


2.

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 not touching. Pressing the [TRVRS.] key at the same time will increase this rate of movement.



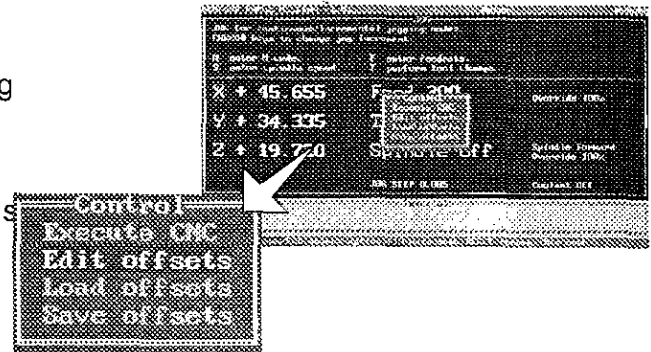
Next, move the table so that the centre of the cutter is positioned exactly over the front, left corner of the temporary MDF bed. The [JOG] key can be used to switch from 'continuous' to a more accurate degree of adjustment using stepped movements, called 'Jog Step'. This degree of movement is adjusted using the blue [CURSOR ARROWS] keys. Jog steps can be selected between a movement of 0.005mm minimum to 5mm maximum.



SETTING THE WORK DATUM X AND Y CO-ORDINATES.

3.

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).



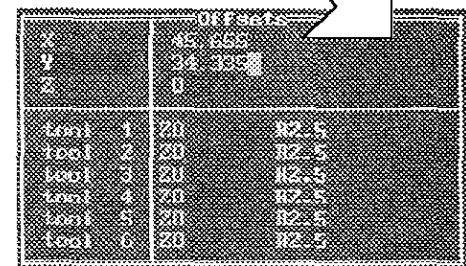
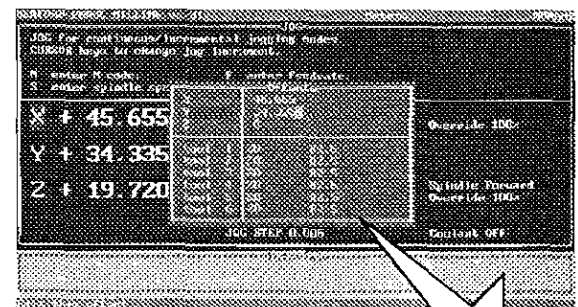
4.

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'.

Note - At this stage, both the X and Y offsets table co-ordinates should read zero, otherwise any new co-ordinates transferred from the main screen will be incorrect.

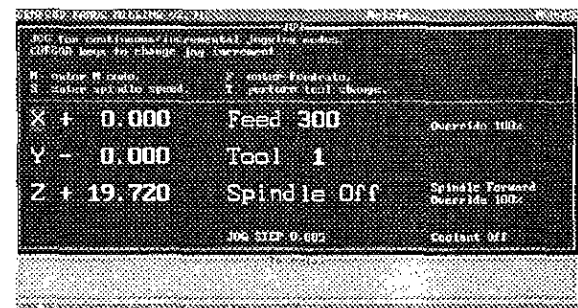
Type in the new value (shown on the blue main screen, just to the left of the offsets table) using the [NUMBERS] keys; use the [DELETE] key to remove any incorrect characters.

Press the [EOB] key to confirm this new figure.



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').

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).



SETTING THE TOOL OFFSET Z CO-ORDINATES.

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 would need to be repeated for each tool if we wanted to use more than one tool on the workpiece.

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

1.

Select *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.

2.

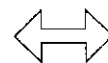
Using the [+Z] and [-Z] keys, make the tip of the cutter just touch the top surface of the bed.

Remember that *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.

JOG STEP 0.005



CONTINUOUS

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 Mode Steps*.

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

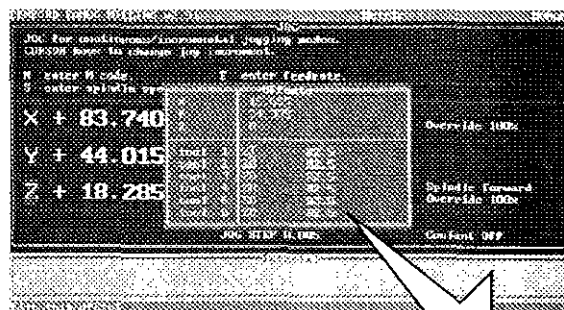
Cutting tool touching mdf surface.

SETTING THE TOOL OFFSET Z CO-ORDINATES.

3.

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 right).

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



Offsets			
X		45.655	
Y		34.335	
Z		0	
tool	1	20	R2.5
tool	2	20	R2.5
tool	3	20	R2.5
tool	4	20	R2.5
tool	5	20	R2.5
tool	6	20	R2.5

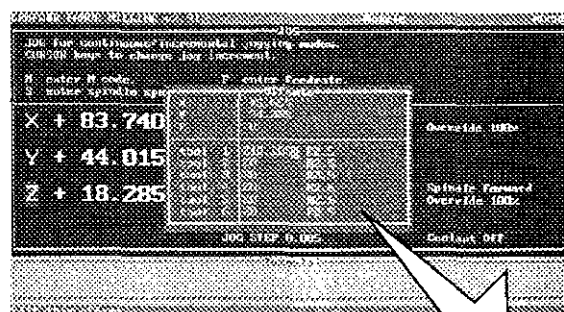
4.

Note - At this stage, all the Z co-ordinates in the offsets table should all read zero, otherwise any new co-ordinates transferred from the main screen will be incorrect.

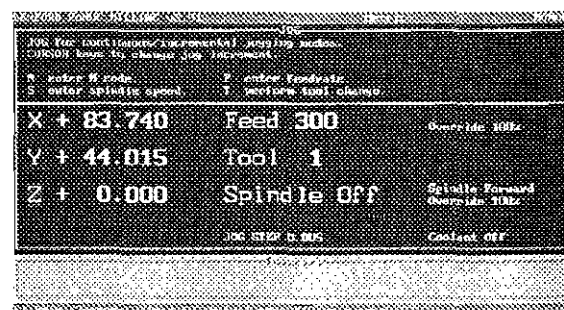
Enter the new Z value, using the [NUMBERS] keys; use the [DELETE] key to remove any incorrect characters. The Z value you want to copy is shown on the blue main screen, just to the left of the offsets table.

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).



Offsets			
X		45.655	
Y		34.335	
Z		0	
tool	1	20	R2.5
tool	2	20	R2.5
tool	3	20	R2.5
tool	4	20	R2.5
tool	5	20	R2.5
tool	6	20	R2.5



SETTING A GLOBAL Z VALUE FOR MATERIAL THICKNESS.

Now that the Tool Offsets have been set to coincide with the surface of the temporary MDF bed, fix the high density polystyrene billet into position. Press the billet firmly onto the temporary MDF bed in the orientation shown in 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 (2mm). Rather than program this value individually for each tool as described when setting the Z co-ordinate Tool Offsets, we can program a 'global' Z value which will be applied to every tool

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.

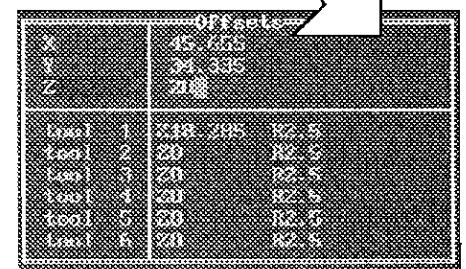
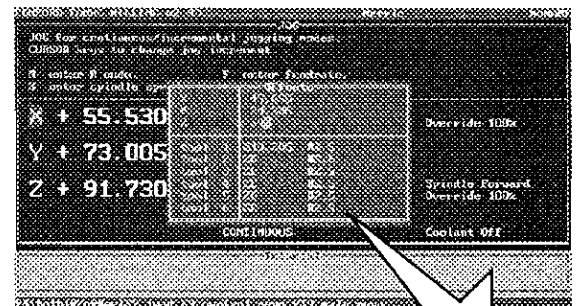
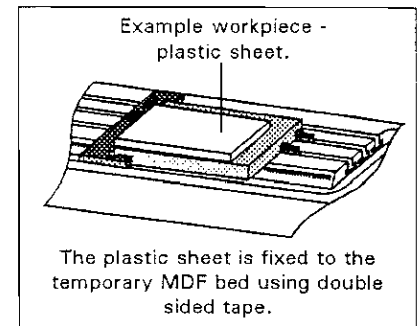
From the list of offsets shown highlight the 'Z' co-ordinate (not any of the Tool number co-ordinates!) and press the [Z] key to move the cursor across to where it reads '0'.

Enter the 'global' Z value - this should be the thickness of your work material (2). Use the [DELETE] key to remove any incorrect characters.

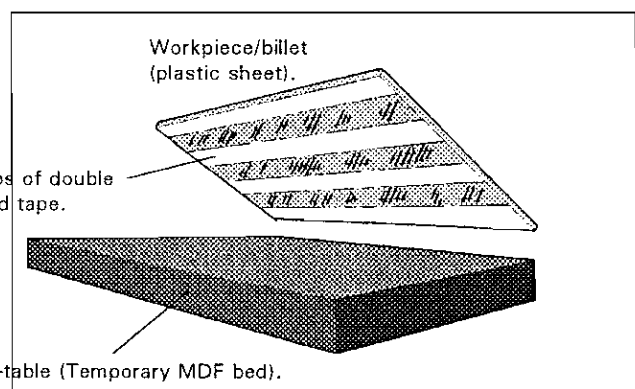
Finally, press [EOB] to confirm this 'global' Z value.

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, ie 2mm.

Double sided tape can be used to hold thin soft material, like plastic. Ensure that the entire underneath surface of the billet is covered with double sided tape and press it firmly to the temporary bed. It has to be stuck to a flat surface, such as MDF (Medium Density Fibreboard), which is then used as a smaller type of machine table, called a sub-table or temporary bed. This sub-table can then be clamped to the machine table using tee bolts, miteebite clamps or held in a machine vice. Sub-tables are also a useful protection against the tool cutting completely through the workpiece, since they are cheap to replace (unlike the machine table itself!).



DOUBLE SIDED TAPE.



LOADING CNC FILES.

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

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

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.

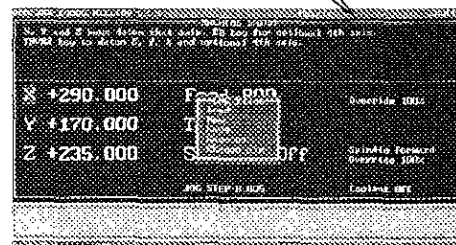
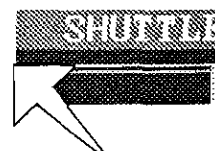
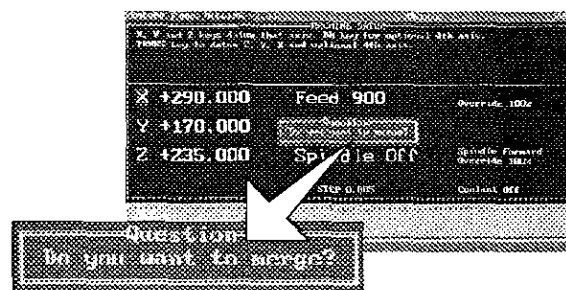
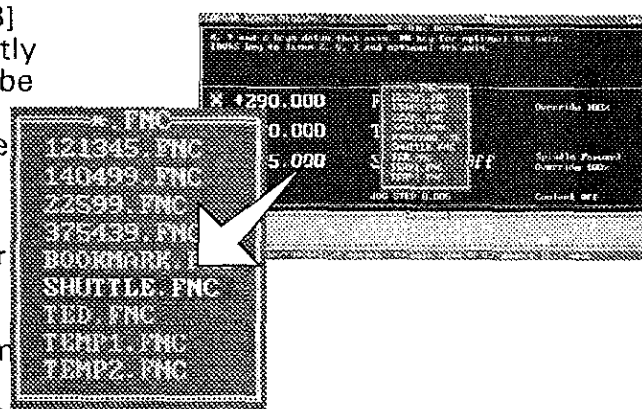
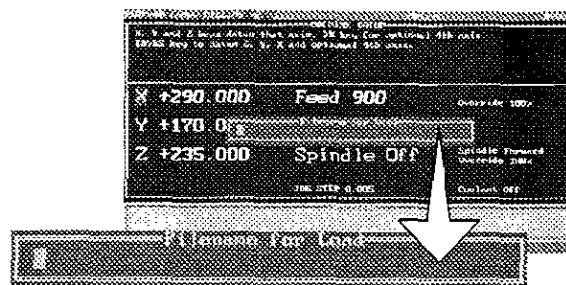
If the filename is unknown, press the [EOB] key to list all the files stored on the currently selected drive. Files within these lists can be loaded by highlighting them using the [CURSOR ARROWS] keys and pressing the [EOB] key.

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 [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.

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.



CHANGING THE DEFAULT LOAD/SAVE DIRECTORY.

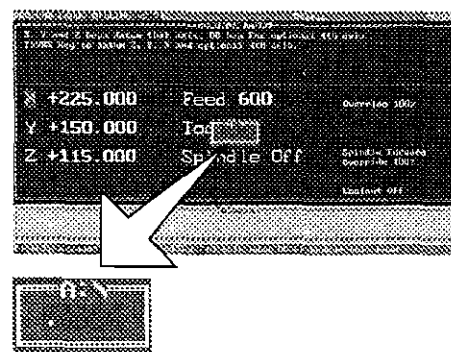
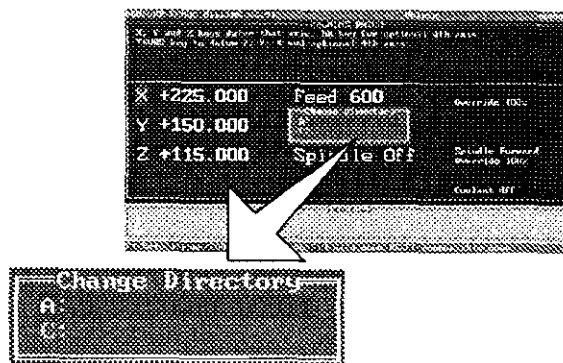
Select the *Main Menu* by pressing the [F10] key. Highlight '*CNC Files*' using the [CURSOR ARROWS] keys and press the [EOB] key to confirm this choice.

Highlight '*Change dir*' with the [CURSOR ARROWS] keys and press the [EOB] key.

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.



To load CNC Files when the directory has just been changed, 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.

To load Offset Files when the directory has just been changed, select the *Offset Control Menu* by pressing the [MENU OFFSET] key. Highlight the '*Load offsets*' option and press the [EOB] key.

Note - the screen may display the previous setting for the drive. 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 new file 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 file required using the [CURSOR ARROWS] keys and press the [EOB] key to load the file.

An Error message window stating *There are no matching files* will be displayed, if there are no matching files on the drive that is being read. To clear this message, press the [RESET] key.

To save CNC Files when the directory has just been changed, select the *Main Menu* by pressing the [F10] key. Highlight '*CNC Files*' and press the [EOB] key. Highlight the '*Save As*' option and press the [EOB] key.

To save Offset Files when the directory has just been changed, select the *Offset Control Menu* by pressing 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. 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 new file will now be saved on the new drive.

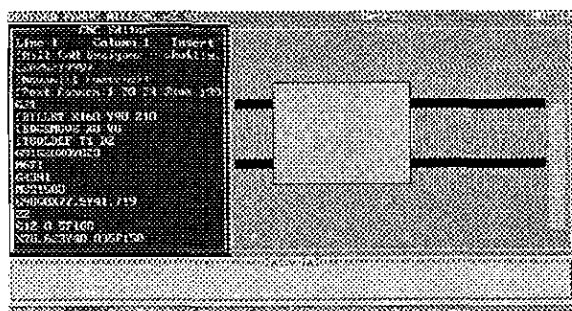
Note - CNC Files are stored in ".fnc" format. Offset Files are stored in ".fao" format.

SIMULATE A CNC FILE.

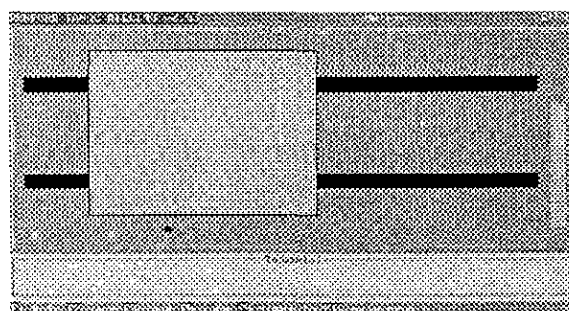
Once a CNC File has been loaded, its action can be simulated on screen. Remember that the name of the CNC File is displayed in the top right-hand corner of the display. In the example screenshots, a program called 'Shuttle' has been loaded.

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.

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 (shown) or 3d.



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

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

Note -

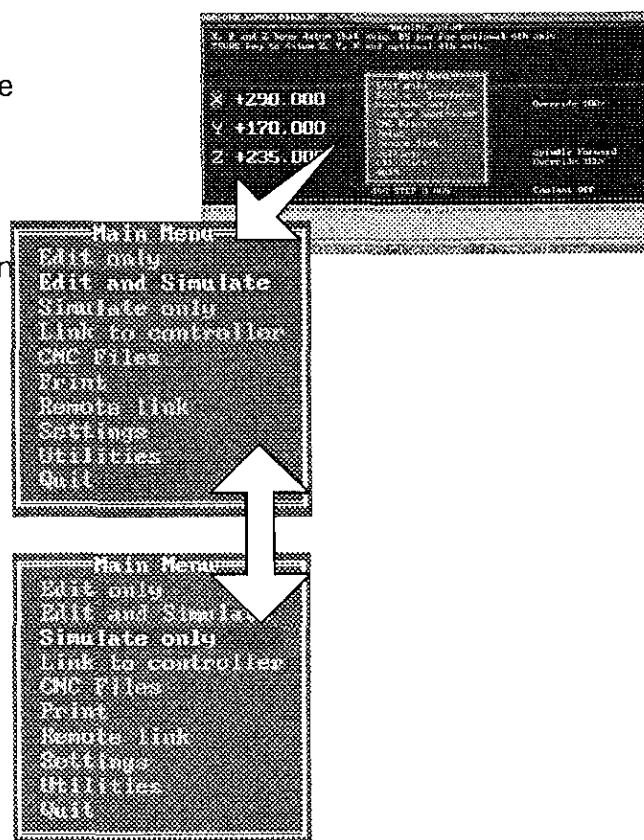
The function of the [F9] key will depend on the machine mode:

AUTO MODE = Control Menu.

EDIT MODE = Simulation Menu.

HOME MODE = Control Menu.

JOG MODE = Control Menu.



SIMULATE A CNC FILE.

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.

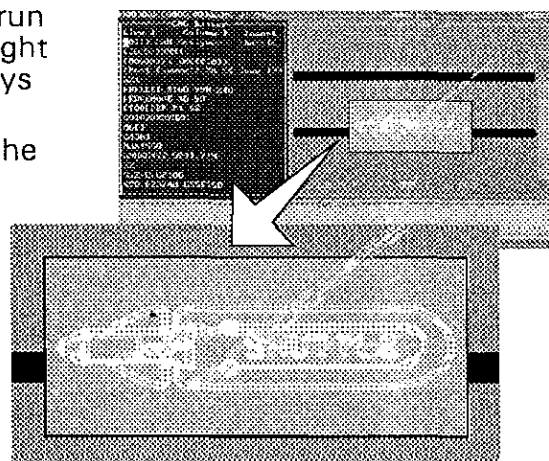


2) *Run Program*. This instructs the computer to run through the CNC File. To start this option, highlight '*Run Program*' using the [CURSOR ARROWS] keys and press [EOB].

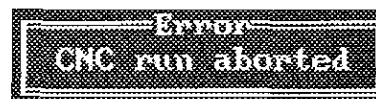
The '*Run Program*' option is set to run through the entire CNC File cycle, from start to finish. Whilst the CNC File is running, the written text will scroll down the screen and the pictorial view will be simultaneously updated.

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.



Note - The [CYCLE STOP] key can also be used to pause a run. The remainder of the CNC File will be run as a separate 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.



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.

SIMULATE A CNC FILE.

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.

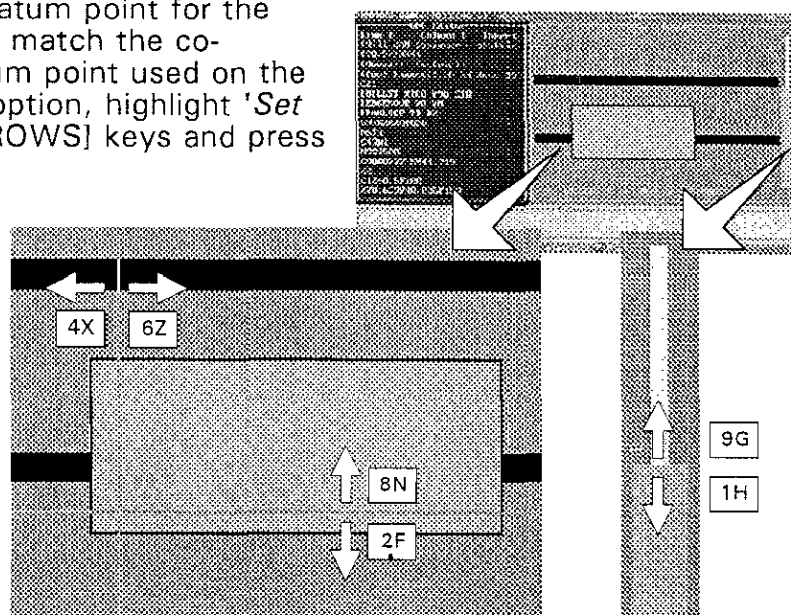
[4X LEFT ARROW] = Purple/blue Crosshair left.

[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.

Note - the graphical datum does not need to be set when using cnc files generated from Mill CAM Designer.



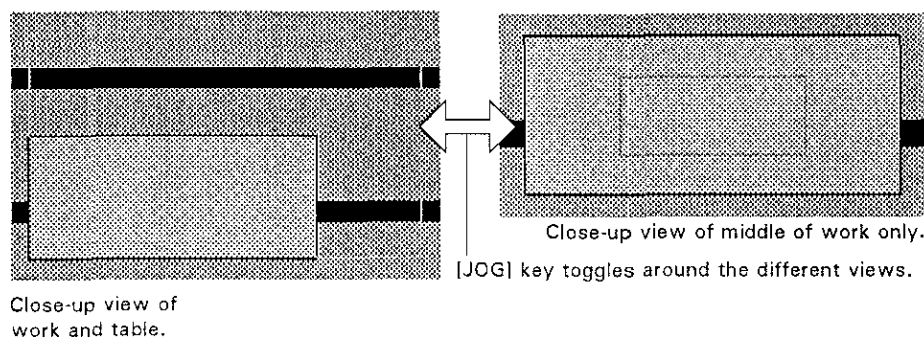
5) *Set View*. This allows the detail level of the plan pictorial view to be selected in the view editor. To select this option, highlight '*Set View*' using the [CURSOR ARROWS] keys and press [EOB].

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

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.

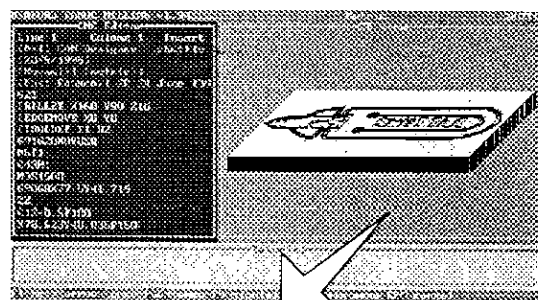


SIMULATE A CNC FILE.

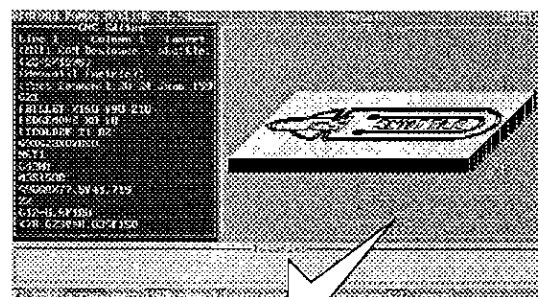
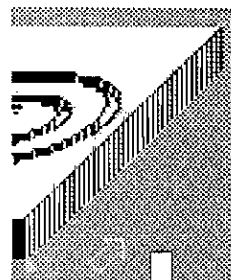
6) *3d View*. This shows the simulation graphic as a 3 dimensional view, rather than a plan view. To select this option, highlight '*3d View*' using the [CURSOR ARROWS] keys and press [EOB].

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

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.

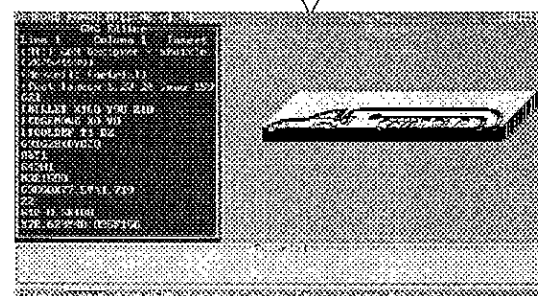
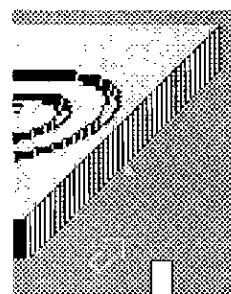


7) *Post Process*. This feature requires the optional Denford Universal Post Processor software in order to function. The Desktop Tutor can be used in conjunction with the Denford Universal Post Processor to translate a CNC File into different control languages.

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.

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



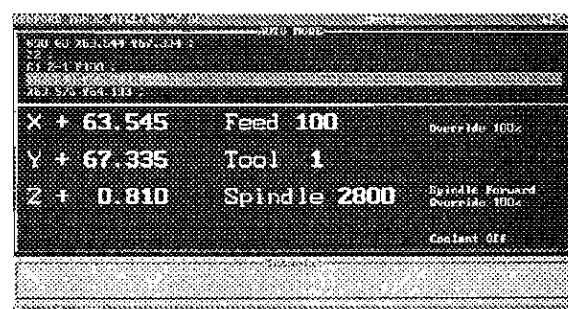
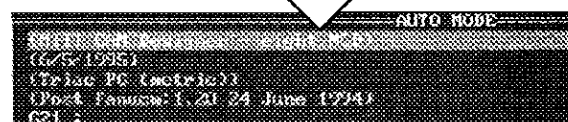
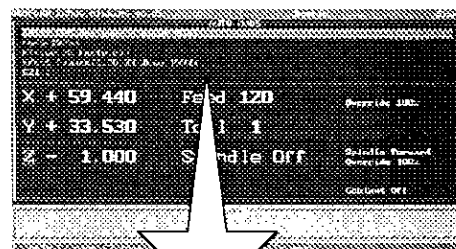
RUN/PAUSE A CNC FILE.

RUN A CNC FILE.

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').



PAUSE A CNC FILE.

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.

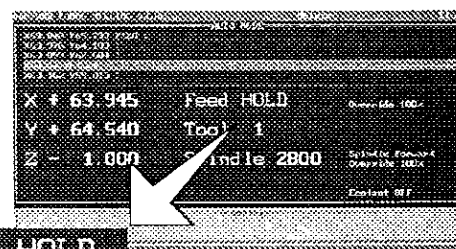
To resume the CNC File, press the [CYCLE START] key.

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

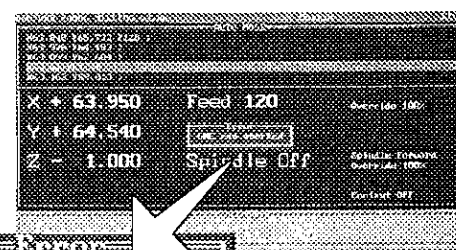
Press the [RESET] key to clear any error messages.

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*.



Feed HOLD



Error
CNC run aborted

OVERRIDING THE 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.

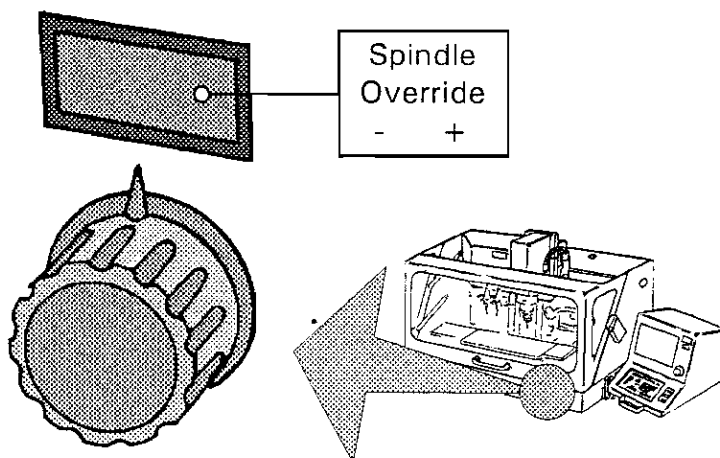
On machines not fitted with this control dial, the tutor keypad is used to override the spindle speed.

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

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

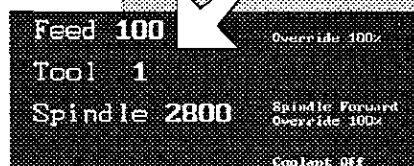
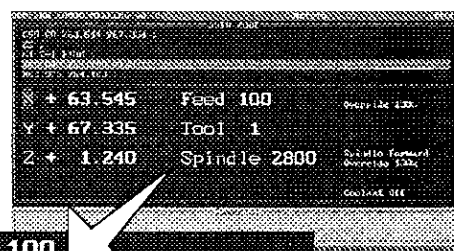
To increase the spindle speed, rotate the dial clockwise - as the dial is turned the 'faster' spindle speed will be updated and displayed on the control screen.

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.



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 up to a maximum value of 120%. To increase the spindle speed, press the [UP KEYPAD ARROW] key until the desired value is displayed on screen.



OVERRIDING THE FEEDRATE.

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.

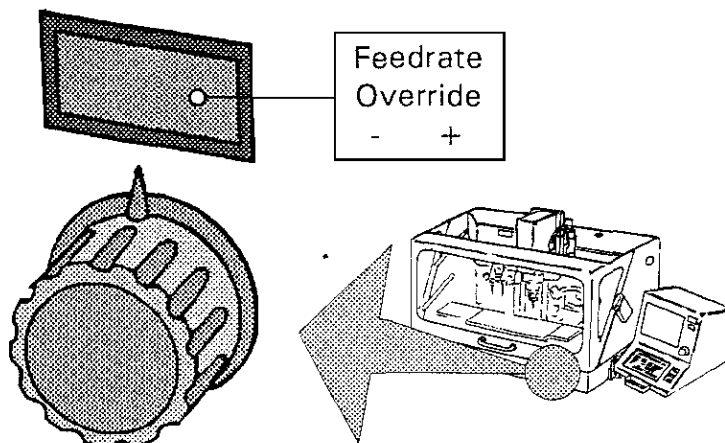
On machines not fitted with this control dial, the tutor keypad is used to override the feedrate.

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

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

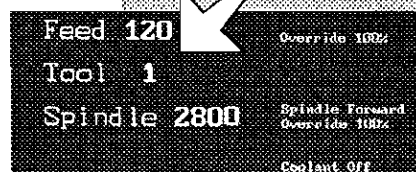
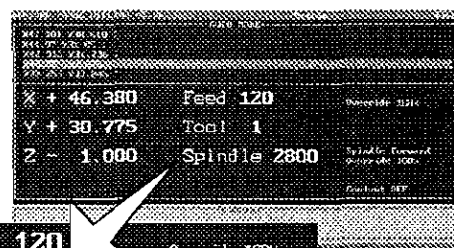
To increase the feedrate, rotate the dial clockwise - as the dial is turned the 'faster' feedrate will be updated and displayed on the control screen.

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.



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.



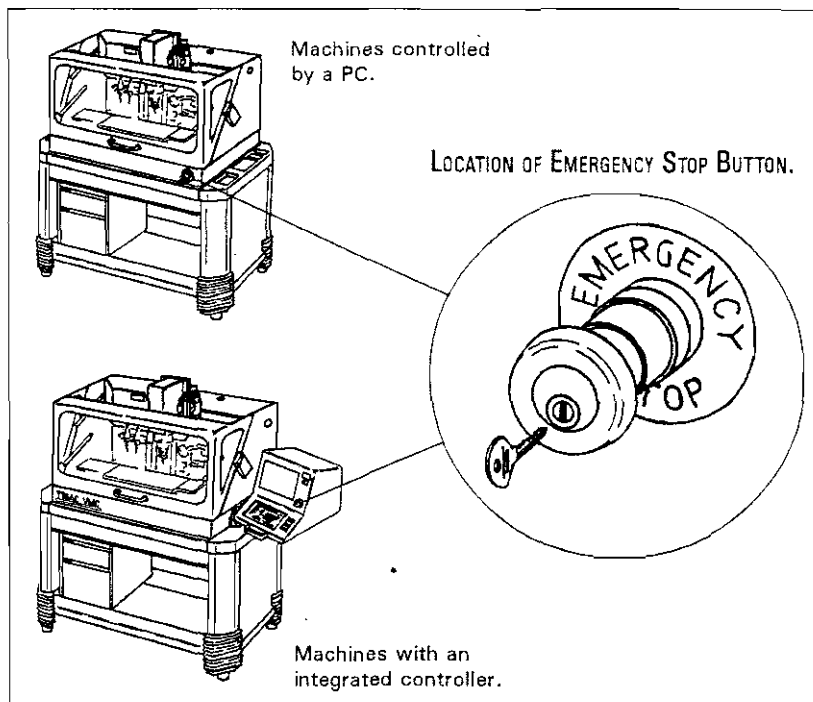
The feedrate can be increased up to a maximum value of 150%. To increase the feedrate, press the [RIGHT KEYPAD ARROW] key until the desired value is displayed on screen.



EMERGENCY STOP.

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

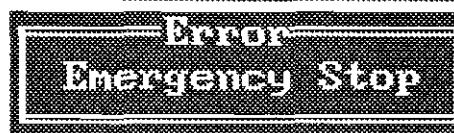
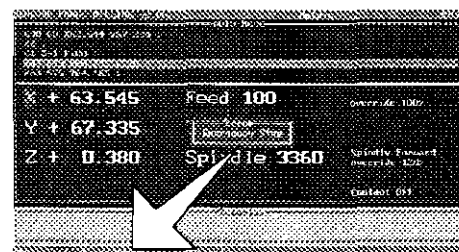
- 1) Machines controlled by a PC - Located on the front of the lower machine panel.
- 2) Machines with an integrated controller - Located on the controller casing (at the right of the display screen).



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). Following an emergency stop, the machine axes will need to be homed (datum the machine).