DENFORD

Total Commitment to Education and Training WorldWide.

Micromill
CNC Milling Machine
User's Manual.

CE

approved

Denford Limited reserves the right to alter any specifications and documentation without prior notice. No part of this manual or its accompanying documents may be reproduced or transmitted in any form or by any means, electronic or mechanical, for any purpose, without the express written permission of Denford Limited.
All brands and products are trademarks or registered trademarks of their respective companies.
Copyright Denford Limited - Version 1.0 - 2000. All rights reserved.

Contact Information

Address:	Denford Limited, Birds Royd, Brighouse, West Yorkshire, HD6 1NB, UK.
Telephone:	General Enquiries +44 (0) 1484 712264 Customer Services +44 (0) 1484 722733
Fax:	+44 (0) 1484 722160
e-mail:	for sales enquiries contact, sales@denford.co.uk for machine servicing enquiries contact, service@denford.co.uk for customer services and technical support contact, customer_services@denford.co.uk

Notes

Contents

Preface

Contact Information 2
Contents 3
Warning Notices 6
About this Manual 7
Conventions used in this Manual 8

Section 1: Introduction

Introducing your Micromill CNC machine 9
What is CNC? 10
What is the VR CNC Milling software? 11
Before Installing... 12

Section 2: Safety Features

Safety Features Overview and Precautions 13
Safety Features - Emergency Stop Button 14
Safety Features - Interlock Guard Switch 15

Section 3: Machine and Software Installation

Unpacking & Lifting your Micromill 16
Deciding on a Site for your Micromill 17
Levelling your Micromill 18
Connecting your PC to the Micromill 19
Connecting the Mains Supply 20
Removal of Protective Coatings 21
Component Connection Schematic Diagram 22
PC Specifications for your Micromill Controller 23
VR CNC Milling Software Installation 24

Section 4: Getting Started

Using your Micromill - Overview 25
General Layout of Micromill Components 26
Micromill Front Panel Layout 27
Switching your Micromill On and Off 28
Starting the Machine Controlling Software 29
General Layout of the Software 30

Section 5: Loading or Creating a CNC File

Loading and Saving a CNC File 31
The "Editor" window. 32

Contents

Section 6: Configuring Tools in the Software

Using the Tool Library 33
Viewing Tool Data 34
Adding New Tool Profiles 35
Using the Machine Tooling window 36
Transferring Tools into "Machine Tooling" 37

Section 7: Simulating a CNC File

Running a 2D Simulation of a CNC File 38
Running a 3D Simulation of a CNC File 39

Section 8: CNC Machine Control

Machine Communications 40
Homing the CNC Machine - Home Mode 41
Co-ordinate System Display Modes 42
Jogging the Axes - Jog Mode 43
Selecting M Codes 44
Jogging the Axes - Movement Keys 45

Section 9: Preparing Tooling Hardware

The Drawbar and Collet Tooling System 46
Removing a Tool from the Drawbar / Collet 47
Installing a Tool into the Drawbar / Collet 48
The Quick Change Tooling System 49
Manual Tool Changing with the Software 50

Section 10: Work Holding

The Datum Plate 52
Setting the Datum Plate 53
Miteebite Clamps 56
How does a Miteebite Clamp work? 57
Using Miteebite Clamps 58

Section 11: Configuring Offsets

Introducing Offsets 59
Configuring a Workpiece Offset 61
Creating a new Workpiece Offset File 62
Configuring the Tool Length Offset 64

Section 12: Part Manufacture

Manufacturing your Part - Auto Mode 67
Stopping the Machining Process 68
Overriding Feedrates and Spindle Speeds 69

Contents

Section 13: Maintenance

Maintenance Schedule and Lubrication Chart 71
Lubrication Areas on the Micromill 72
Lubrication of Leadscrews 73
Lubrication of Slideways 76
Lubrication of Thrust Bearings 79
Cleaning the Datum Microswitches 80
Gib Strip Adjustment 83
Spindle Drive Belt Adjustment 87

Section 14: Technical Support

Technical Support 88
Troubleshooting 89
Changing COM Ports 90
Electronics 91

Section 15: Appendix

Specification of Micromill CNC machine 94
What is a Part Program? 96
Composition of a Part Program 97
G Codes List 98
M Codes List 99
List of Program Address Characters 100
Denford Directives 101
EC Declaration of Conformity 103

Section 16: Glossary

Glossary 105

Section 17: Index

Index 109
Notes 112

Warning Notices

Warranty Disclaimer.

The Warranty on your Micromill CNC machine will be invalidated if any modifications, additional ancillary equipment is fitted, or any adjustments made to the controlling devices without prior notification from Denford Limited. Please refer to the information held in your separate Warranty pack, for specific details.
Do not carry out any portable appliance testing (PAT) on any of the supplied equipment.

Maintenance Disclaimer.

Always obtain permission from the person responsible for machinery in your establishment, before accessing the electrical panel at the rear of your Micromill CNC machine casing to carry out **any** maintenance work. All work must be carried out by personnel suitably qualified for each maintenance task, to avoid damage to both the machine systems and the maintenance personnel. Denford Limited **cannot accept responsibility** for any damage and/or loss that may occur through incorrect maintenance of your Micromill CNC machine.

Foreseen Use of Machine.

Your Micromill CNC machine is designed for milling synthetic materials such as wax, plastics, acrylics and non-hardened metals such as aluminium. In each case, the appropriate tooling, spindle speeds and feedrates should be used as recommended by the material supplier.
Only use water based soluble oil cutting fluids, do not use parafinic or potentially explosive cutting fluid.
Do not attempt to use your Micromill CNC machine for manual operations.
Never attempt to fit an abrasive wheel to the machine spindle.
If you have any doubts and/or questions regarding the specification, servicing, or features of your machine, please contact Denford Customer Services.
Denford Limited reserves the right to change the specification and/or operating features regarding this CNC machine without notice or documentation.

About this Manual

Using this manual	This manual provides information describing how to transport, site, setup and operate the basic functions of your Denford Micromill CNC machine, including any operational features of hardware specific to the Micromill series. A Routine Maintenance section is also included. More detailed information regarding the Virtual Reality CNC Milling software, used for controlling your Micromill, is contained in the separate "VR CNC Milling Software User's Manual", supplied with your machine. Please note that the Electrical Diagrams for your Micromill CNC machine are not included in this manual - they are delivered separately in the standard equipment box supplied with your Micromill CNC machine. If you have any doubts and/or questions regarding the specification, servicing, or features of your machine, please contact Denford Customer Services. Denford Limited reserves the right to change the specification and/or operating features regarding this CNC machine without notice or documentation.
Disclaimer	We take great pride in the accuracy of information given in this manual, but due to nature of hardware and software developments, be aware that specifications and features of this product can change without notice. No liability can be accepted by Denford Limited for loss, damage or injury caused by any errors in, or omissions from, the information supplied in this manual.
Screenshots	Please note that any screenshots are used for explanation purposes only. Any numbers, wording, window or button positions may be different for the configuration of the VR CNC Milling software being used to control your Micromill CNC machine.
Language	This manual is written using European English.
Contact	Any comments regarding this manual should be referred to the following e-mail address: customer_services@denford.co.uk
Updates	Any updates to this manual will be posted in the "Downloads" section of the Denford website: http://www.denford.co.uk

Conventions used in this Manual

Mouse Usage	When asked to left click on a menu tile or object, click the LEFT mouse button ONCE. When asked to right click on a menu tile or object, click the RIGHT mouse button ONCE. When asked to double click on an object, click the LEFT mouse button TWICE. When reference to either a left mouse button or right mouse button click command is omitted, always perform one click with the left mouse button.
Underlined text	This is used to show key words. The full definition of any terms are given in the Jargon Buster helpboxes. Similar helpboxes are also used to display any Important Notes or Tips to help you use your Micromill CNC machine.
"Quotation Marks"	Quotation marks are used to specify any software menu, title and window selections, e.g. click the "File" menu would mean click the left mouse button once, when the cursor is positioned over the File menu label. When a sequence of menu commands are requested, the menu and option names are separated by a vertical line, for example - Click "File | Open" would mean open the File menu, then click on the Open option.
Bold Text	Bold Text is used to show any characters, or text, that must be entered into the software, e.g. type **file1** would mean type the word file1 into the appropriate text entry box.
[Square Brackets]	Square brackets are used to show any on-screen software button selections, e.g. Click the [OK] button would mean click the left button of the mouse once, when the cursor is directly pointing over the button labelled OK.
[Bold Square Brackets]	Bold square brackets containing text show individual keys to press on your qwerty keyboard, e.g. press **[Enter]** would mean press the Enter key. If a number of keys must be pressed in sequence they are shown with plus signs outside any square brackets, e.g. press **[Alt]** + **[Enter]** would mean press the Alt key first followed by the Enter key second. If a number of keys must be pressed simultaneously they are shown with plus signs inside any square brackets, e.g. press **[Alt + Enter]** would mean press both the Alt key and Enter key together, at the same time.

1: Introducing your Micromill CNC machine

Congratulations on your purchase of a Micromill series CNC machine. In this manual you will learn how to setup and use your Micromill.

The Micromill is a full three axes CNC training machine. It has been designed with you in mind - making the processes involved both safe and easy to use.

Main Features:

- Designed specifically for Education and Training.
- Manufactured to industrial standards.
- Programming via International Standards Organisation format.
- CE approved for safety.
- Capable of cutting synthetic materials such as wax, plastics, acrylics and non-hardened metals such as aluminium.
- Links to various CAD/CAM software packages.
- Totally enclosed high visibility interlocked guard.
- Option of Including in FMS/CIM systems.

1: What is CNC?

CNC (Computer Numerical Control) is the general term used for a system which controls the functions of a machine using coded instructions, processed by a computer. CNC machines are a very important part of the modern manufacturing process. Indeed, many of the different types of products you use everyday have been made using some sort of CNC machine.

The CNC Manufacturing Process - Example.

The sequence shown below defines the main steps involved in producing a component using a CNC system.

1) A part program is written using G and M codes. This describes the sequence of operations that the machine must perform, in order to manufacture the component.
2) The part program is loaded into the machines computer, called the controller. At this stage, the program can still be edited or simulated using the machine controller.
3) The machine controller processes the part program and sends signals to the machine components. These direct the machine through the required sequence of operations necessary to manufacture the component.

What are the advantages of CNC?

CNC systems are automated and very accurate. Once programmed, a CNC machine will perform repeat tasks until instructed to stop. Each component produced will be exactly the same size and shape, saving money on designing any jigs and fixtures that might have otherwise been required.

Using CNC machines can reduce waste material, since a CNC machine is much less likely to make an error than a human operated machine. CNC machines can also run 24 hours a day, if necessary, with no signs of fatigue.

Companies can estimate the manufacturing costs for CNC production much more accurately, compared to a production line with conventional production machines.

Jargon Buster

CNC refers to Computer Numerical Control, the automatic system used to control a machine tool.
A Part Program is a list of coded instructions which describes how the designed component, or part, will be manufactured. The part program is also referred to as the CNC file, program, or G and M code program.
A G and M code is a series of letters and numbers that make up the language used by CNC machinery.

1: What is the VR CNC Milling software?

Virtual Reality (VR) CNC Milling is a Windows based software package allowing full editing and control of CNC files. The term Virtual Reality stems from the ability to control a Virtual CNC machine in the software. In other words, the computer can create an exact replica of the Micromill, which is fully controllable, just like the real machine. We call this software mode Offline, meaning we're working away from the production line. Similarly, Online means we're using the software to directly control the operation of a real Micromill CNC machine, connected to your computer.

Information is accessed and displayed using an interface similar to other popular Windows based software applications. The familiar dropdown menus, toolbars and software display windows can be configured to suit the level and requirements of each user.

Since the software supports full offline facilities, it allows many training tasks such as setting tool offsets, to be carried out away from the CNC machine itself. Options such as these allow groups of students to work simultaneously whilst helping to free valuable CNC machine resources.

The same interface is used online, allowing students to produce their designs without having to learn any new CNC machine control software.

Features available in the VR CNC Milling software package include:

- Full MDI CNC file editing.
- 2 Dimensional graphical simulation of CNC files.
- 3 Dimensional graphical simulation of CNC files.
- Comprehensive Tooling features.
- Full offline control of a CNC machine using Virtual Reality.
- Full online control of a CNC machine.
- Context sensitive online help, including help with G and M code Programming and CNC file structure.

Jargon Buster

Context Sensitive is when the type of input signal of an event automatically changes the output signal.
MDI refers to Manual Data Input, the entering of data by manual means rather than transferral by CD-ROM or floppy disk.
Virtual Reality is a fully interactive, three dimensional, computer based simulation of a real world object or event.

1: Before Installing...

Before beginning to setup your Micromill CNC machine, take a moment to check your separate order documentation, making sure that all items have been delivered to your establishment. Any missing or damaged items should be reported to Denford Customer Services as soon as possible.

Note

The standard eqiupment listed here is correct at the time of printing but is liable to change through continuous development of our products.

The following equipment is supplied as standard with your Micromill CNC machine:

- Micromill CNC machine. Note that the precise specification of your Micromill will depend on any options selected at the time of ordering (see below).
- Guard Door Interlock Switch - torx bit.
- RS 232 serial link cable.
- Set of imperial allen/hex keys (1/16", 5/64", 3/32", 1/8", 5/32", 3/16", 7/32", 1/4").
- Set of metric allen/hex keys (2mm, 2.5mm, 3mm, 4mm, 5mm, 6mm).
- Spindle Bar.
- Collet 1/4".
- End Mill 3/16".
- Drawbar with washer.
- Set of fuses (5x20mm , 6.3A, 10A, 3.15A)
- Micromill CNC machine warranty pack.
- Micromill CNC machine inspection certificate.
- Micromill CNC machine manual (this book).
- Virtual Reality CNC Milling Software CD-ROM.
- Mousemat.
- Virtual Reality CNC Milling Software manual.
- Machine Commissioning and Basic Instruction.
- 1 Day Training Course, for 2 persons, at Denford Limited (UK).

The following optional equipment may also be supplied with your Micromill CNC machine (please refer to your separate order documentation for confirmation):

- Workholding Vice.
- Work Holding Package (including Datum Plate, Miteebite Clamps and Tee-Nuts).
- Various Tooling Packages.
- CAD/CAM Software.
- Educational Project Books and Courseware.
- Project Material Packages.
- Additional and/or On-site Training Courses.

2: Safety Features Overview and Precautions

Safety Features Overview.

The following safety features are standard on your Micromill CNC machine:

- Emergency stop button.
- Totally enclosed guard door with interlock switch.
- Electronic shear key built into spindle controller.
- Option on control software to check CNC programs prior to machining.

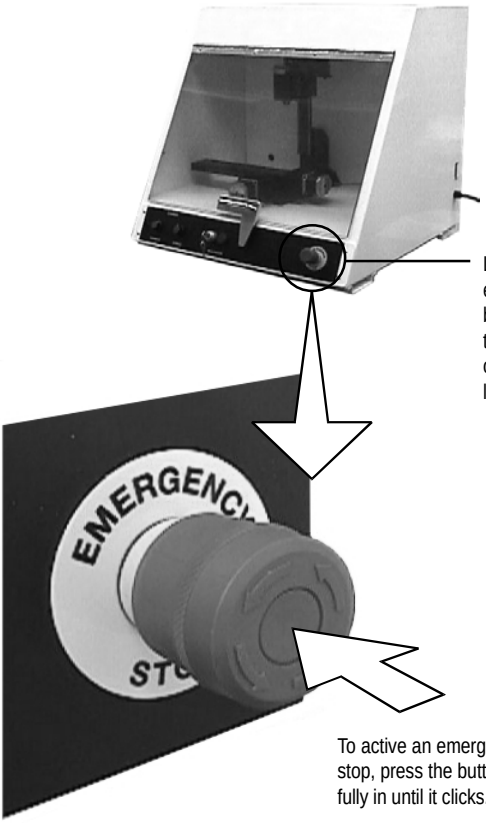
Safety Precautions.

Safety is very important when working with all forms of machinery but particularly when working with CNC equipment, due to the hazardous voltages, speeds and forces that exist in the hardware. Follow the rules below at all times, when using your Micromill CNC machine.

General Safety Precautions :

- Wear clothing suitable for machine operation and follow the safe working procedures in place at your establishment.
- Do not place any objects so that they interfere with the guards or the operation of the machine.
- Never try to clean the machine if any part of it is rotating or in motion.
- Always secure the work on the table or in a fixture or vice.
- Ensure that the correct cable for the power source is used.
- Ensure the mains power is switched off (and preferably unplugged) before starting any maintenance work on the machine. Post a notice informing others not to use the machine since it is undergoing maintenance.
- If power fails turn off the mains power switch immediately and unplug the machine from the mains power socket.
- Hazardous voltages can still exist immediately after switching the machine off. Always wait at least 5 minutes before removing the rear cover to access the electrical panel.
- Lubricate the required machine areas at the intervals specified in this manual, to prevent the axes from seizing (see the Maintenance section for further details).
- Observe caution when adding or removing machine tooling.
- When an emergency stop is required, press the circular red emergency stop button, located on the lower right front panel of the machine.

2: Safety Features - Emergency Stop Button



A circular, red emergency stop button is mounted on the righthand side of the Micromill CNC machine lower front panel. When pressed, it has the effect of stopping all axes and spindle movements immediately. The interlock switch will also close. When the safety guard door is in its closed position, this will prevent access to the working area of the machine.

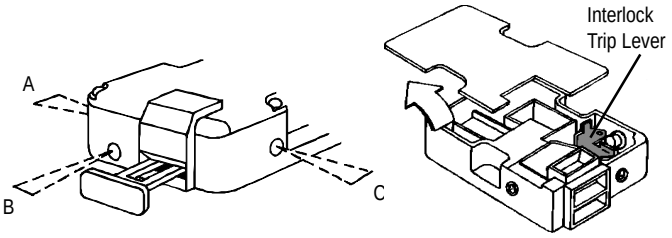
To active an emergency stop, press the button in until it clicks. The emergency stop button will continue to cut all power to the machine drives and continue to keep the interlock switch closed, until the release sequence is performed.

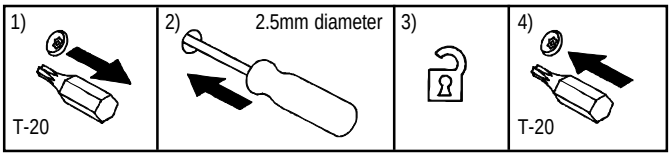
To release a closed emergency stop button, push in and turn the button clockwise until it springs back out.

2: Safety Features - Interlock Guard Switch

An interlock guard switch is fitted to the front machine door. The switch casing itself is attached behind the lower machine panel, accessible from beneath the front of the machine, with the guard switch lock fitted around the guard door handle. The switch must be manually released to enter the working area when the 24 volt circuit has failed and the door is clamped electrically. Note - For manual interlock release, the power supply must be switched off.

1) Remove security screw A, B or C (whichever is easiest to reach) using the supplied Torx T-20 adapter.
2) Insert a 2.5mm diameter tool (such as a small flat screwdriver) into the hole until it pushes the manual release lever.
3) Whilst holding the tool against this manual release lever, open the guard.
4) Remove the tool from the hole and replace the security screw.





3: Unpacking & Lifting your Micromill

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

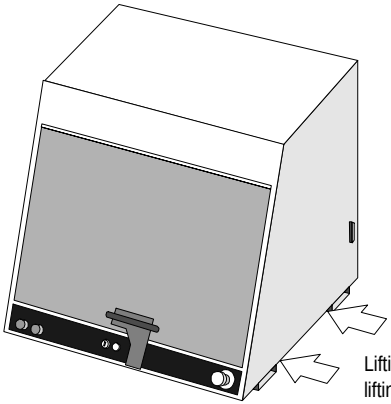
To obtain better access to the Micromill, remove all the sides from the delivery box.

Lift the Micromill from the packaging. Denford recommends that two or more persons are used to lift the Micromill, one at each side panel as illustrated in the diagram below.

Always use sensible lifting precautions in accordance with Health and Safety Regulations in your establishment.

Ensure that your Micromill is both secure and balanced before lifting. To transport your Micromill over longer distances, use a suitably sized trolley.

Data Panel
Lifting Data. Micromill CNC machine weight 70 KG (154 lb).



Lifting points (identical lifting points at opposite side panel).

3: Deciding on a Site for your Micromill

Remember when positioning the machine in the room, space will be required for opening of the electrical control box door. We recommend you leave a gap of at least 1ØØØmm. A 3ØØmm clearance must also be left at the rear of your Micromill. This space will be occupied by the safety guard when in its fully opened position.

Sufficient room should also be provided for effective maintenance to be carried out around the machine itself.

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

Ideally, the user will operate the machine when standing at its front, with a clear view of both the machine table (through the transparent guard window) and the computer being used as the controller unit (which should be angled towards the user), as shown in the diagram below.

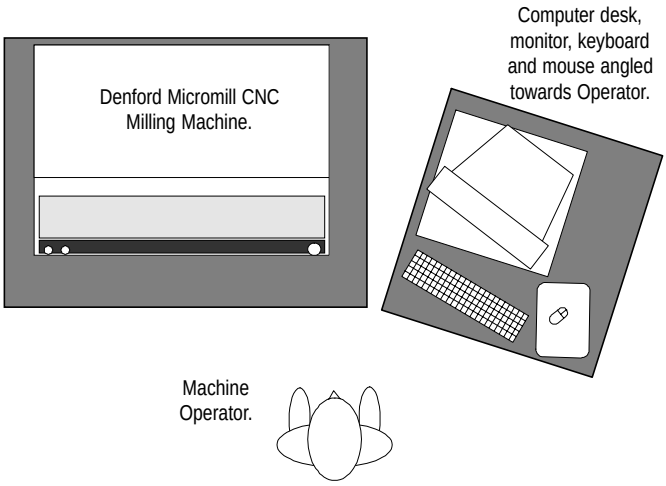
Data Panel - ❐ X

Dimensional Data.

Machine length 686mm (27”).

Machine depth 648mm (26 1/2”).

Machine height 686mm (27”).



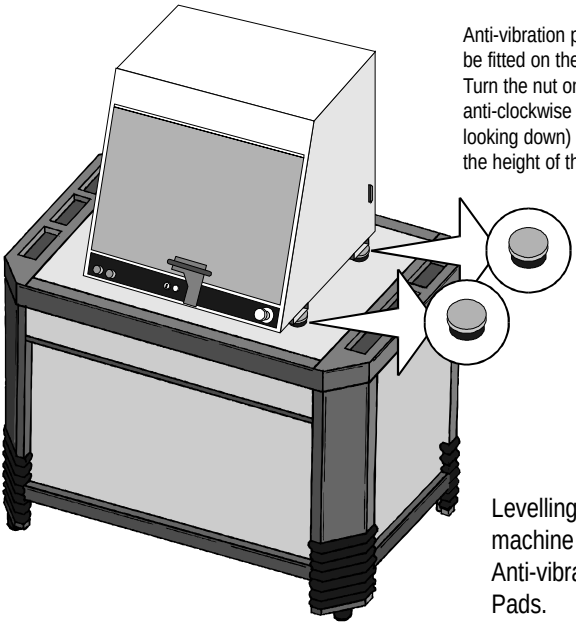
Plan View showing Ideal Machine Operating Positions.

3: Levelling your Micromill

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

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

It is important to remember that the pads are ONLY used to help stabilise the machine. The main weight of the machine should still be taken by the hollow sections (i.e. these sections should ALWAYS be in direct contact with the table surface).



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

Levelling the machine using Anti-vibration Pads.

3: Connecting your PC to the Micromill

Warning.

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

Note

Your PC must be equipped with hardware that allows it to:

1) run the Virtual Reality CNC Milling software.

2) be physically connect to the Micromill.

To check the PC specification required, please see page 23.

Your Micromill CNC machine is controlled using a standard IBM compatible PC (personal computer). In this role, the PC can be referred to as the machine controller computer. Ideally, the PC you intend to use should be placed next to your Micromill, with easy access to a mains power supply.

To connect your PC to the Micromill electrical control box:

1) Connect the elements of your PC together as described in your original PC manufacturers operating manual. At this stage, your PC should not be switched on.
2) The PC must be physically connected to the Micromill, using the supplied RS232 cable. This is the long, thin cable fitted with 25 pin female connector at one end and a 25 pin male connector at the opposite end.

Note

PC Terminology:

The COM ports on your PC may be labelled as Serial ports.

The Parallel port on your PC may be labelled as the Printer port.

Do NOT use a sole 9 to 25 pin adaptor cable in place of the RS 232 cable supplied with your machine.

The RS 232 cable supplied with your machine must always be used, since this cable features crossovers on some of the pin connections.

3) Connect the male end of the RS 232 cable to the 25 pin female port on the right side panel of your Micromill. This port is labelled **RS 232**.
4) Connect the remaining female end of the RS 232 cable to the 25 pin male **COM2** port on your PC. The COM ports are usually situated on the back panel of your PC. Note that some personal computers may be fitted with 9 pin male COM ports. If this is the case, a 25 pin to 9 pin adapter, available from most good computer/electrical retailers must be fitted to allow the cable to be connected to the PC. If you cannot identify any of the ports on your PC, please refer to your original PC manufacturers operating manual for further guidance.

   Do not confuse the 25 pin female parallel (printer) port on your PC with the 25 or 9 pin male COM ports.

A schematic diagram illustrating these component connections is shown on pages 22.

3: Connecting the Mains Supply

Data Panel

Electrical Connection Data.

Mains supply required: Single Phase, 220/240 Volts, 8 Amps, 50/60 Hz.

Cable required: 3 Core, 1.5mm^2 per core.

Spindle motor: 1.5 HP, 1.1Kw.

Axis stepper motors: 200 steps/rev.

The Mains Supply.

The electrical control box is delivered with standard mains specification cable connected directly into the isolator. The cable should be fitted with a standard 13 amp plug suitable for the mains power supply.

Warning.

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

All electrical connections should only be made by a suitably qualified electrical engineer.

Electrical Access Plate.

The Micromill electronics are located in the rear of the machine. To access the electronic hardware, remove the electrical panel access plate on the rear of the machine.

If the machine has previously been switched on, wait at least 5 minutes before attempting to open the access plate. Hazardous voltages can still exist immediately after switching off the power.

Electrical Drawings.

The Electrical Diagrams for your Micromill CNC machine are not included in this manual - they are delivered separately in the standard equipment box supplied with your Micromill CNC machine.

3: Removal of Protective Coatings

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

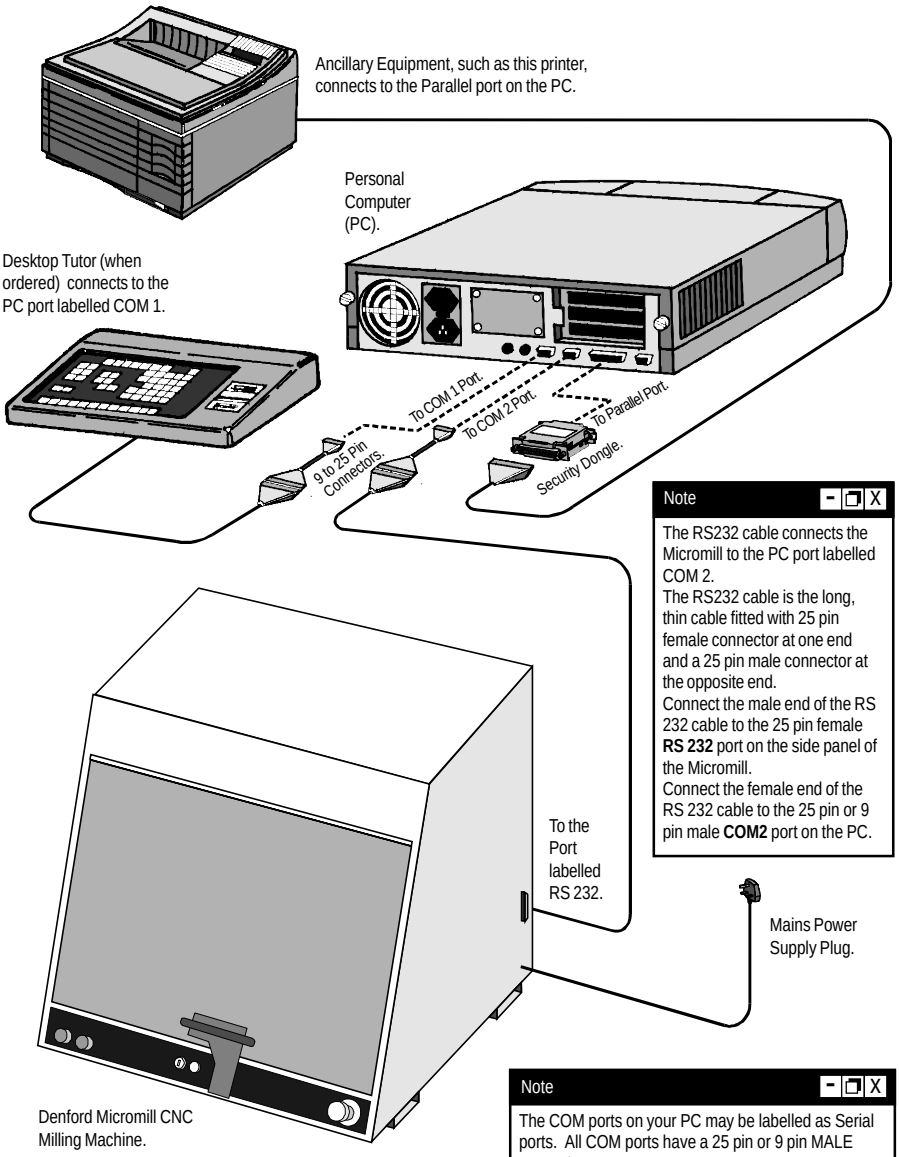
The protective coatings applied to the slideways and bright surfaces can be removed using a kerosene based solvent. The coatings must be removed from the slideways before any attempt to move them is made. Once these protective coatings have been removed, all untreated surfaces should be coated with a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

Warning.

Only use kerosene based solvents in accordance with the solvent manufacturers instructions and safety recommendations. Ensure that no naked flames are present.

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

3: Component Connection Schematic Diagram



Note

The RS232 cable connects the Micromill to the PC port labelled COM 2.
The RS232 cable is the long, thin cable fitted with 25 pin female connector at one end and a 25 pin male connector at the opposite end.
Connect the male end of the RS 232 cable to the 25 pin female **RS 232** port on the side panel of the Micromill.
Connect the female end of the RS 232 cable to the 25 pin or 9 pin male **COM2** port on the PC.

Note

The COM ports on your PC may be labelled as Serial ports. All COM ports have a 25 pin or 9 pin MALE connector.

The Parallel port on your PC may be labelled as the Printer port. The printer port has a 25 pin FEMALE connector.

3: PC Specifications for your Micromill Controller

Your Micromill CNC machine is controlled using a standard IBM compatible PC (personal computer). To run the VR CNC Milling software and connect to the Micromill electrical control box, your PC hardware must meet the minimum specification listed below.

Minimum Specification:

- IBM PC or 100 % compatible personal computer.
- Pentium 120MHz processor.
- 24Mb RAM.
- Windows 95 Operating System.
- Double speed CD-ROM drive.
- Microsoft 100% compatible mouse.
- 10Mb Free hard disk space.
- Colour Monitor running at 800 x 600 resolution with 16bit (High Colour) graphics.
- SVGA graphics card with 512KB VRAM.
- 1 free serial (COM) port.
- 1 free parallel (printer) port.

Recommended Specification:

- IBM PC or 100 % compatible computer.
- Pentium 166MHz MMX processor.
- 32 Mb RAM
- Windows 98 Operating System.
- Double speed CD-ROM drive.
- Microsoft 100% compatible mouse.
- 10Mb Free hard disk space.
- Colour Monitor running at 1024 x 768 resolution with 16bit (High Colour) graphics.
- 3D accelerator card with 4MB VRAM.
- Windows compatible soundcard.
- 2 free serial (COM) ports.
- 1 free parallel (printer) port.

3: VR CNC Milling Software Installation

Follow these instructions to install the VR CNC Milling software on your personal computer:

1) Switch on your PC and start Windows 95/98, if required.
2) Insert the VR CNC Milling CD-ROM into your CD-ROM drive. If your CD-ROM is set to autorun, the install program will start - move to section 5). If the install program does not automatically start, continue to section 3).
3) Double-click the left mouse button on the "My Computer" icon. In the "My Computer" window find your CD-ROM drive icon (usually labelled "D:" or "E:") and double-click the left mouse button on this icon.
4) The contents of the CD-ROM will be displayed in a new window. Double-click the left mouse button on the file named "Setup.exe" to start the installation program.
5) Click the square button next to the "Install VR CNC Milling" title and follow the on-screen instructions.
6) Select the Micromill as the CNC machine used for default software configuration. Click in the "Micromill" checkbox, so a "tickmark" is displayed.
7) Select the area of your hard-disk where the VR CNC Milling software can be installed, together with any program group names. We strongly recommend that you allow the Denford installer to create its own directory, if you have not used any Denford software previously.
8) Restart your computer before trying to run the VR CNC Milling software for the first time.

Note

It is recommended that you allow the Denford installation program to create its own directories and set up its default values. If you find these inconvenient, then feel free to alter them.

Note

Important -
Once the software has been installed, we recommend you place any software master copies in a safe dry location.

4: Using your Micromill - Overview

Several steps must be completed before the final manufacture of a part. The flowchart below lists the general steps that should be followed for CNC file creation, simulation and final part manufacture, in the recommended order. However, miscellaneous factors may warrant the user to complete the steps in a different order to that shown.

Note

For more detailed information regarding these steps please refer to your separate VR CNC Milling Software User's Manual.

Note

Steps F and G are not required when working with a Virtual Reality CNC machine.

Billet data is taken from Denford directives written in the CNC program - step B.

Tooling data is taken from the configuration of the software tooling - step C.

A: Switch on the Micromill and start the VR CNC Milling software
(see section 4: pages 26 to 27).

B: Load or create the CNC program
(see section 5: pages 31 to 32).

C: Configure the tooling in the VR CNC Milling software
(see section 6: pages 33 to 37).

D: Simulate the CNC program in 2D or 3D
(see section 7: pages 38 to 39).

E: Establish communications and home the CNC machine
(see section 8: pages 40 to 45).

F: Prepare any tooling hardware for the Micromill
(see section 9: pages 46 to 51).

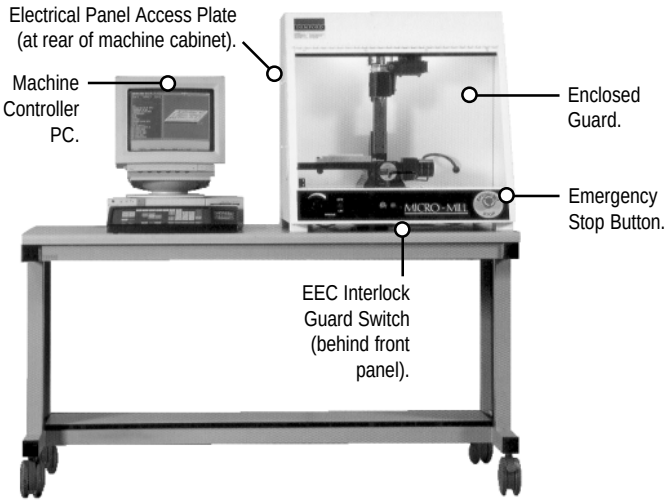
G: Load the billet onto the machine table
(see section 10: pages 52 to 58).

H: Configure the workpiece offset file and tool length offsets
(see section 11: pages 59 to 66).

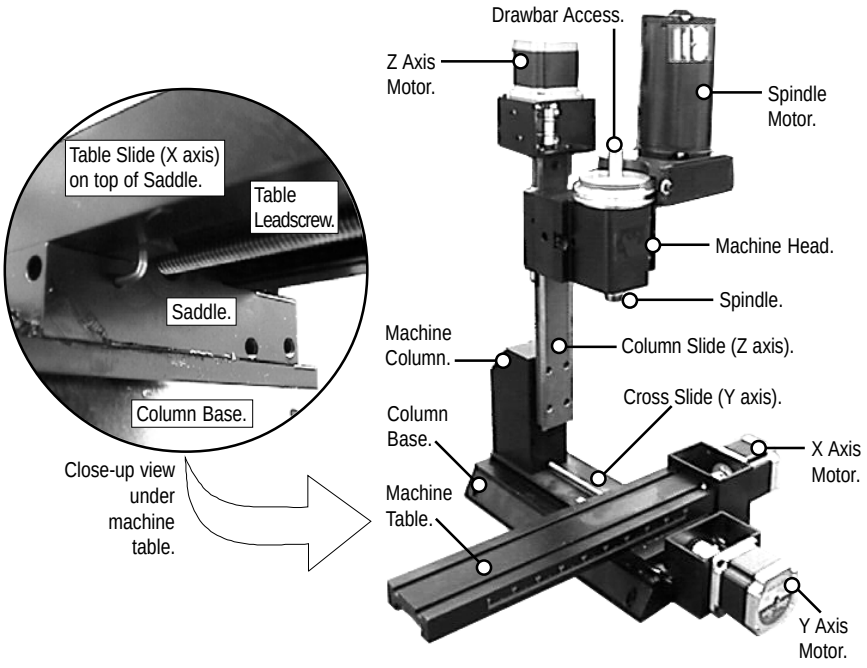
I: Manufacture the part
(see section 12: pages 67 to 70).

4: General Layout of Micromill Components

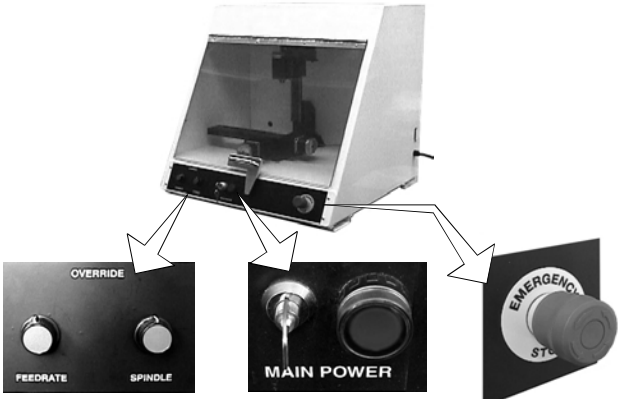
General Layout of Micromill CNC Machine (below).



Inside the Micromill CNC Machine Cabinet (below).



4: Micromill Front Panel Layout



Spindle Speed and Feedrate Override Potentiometer Controls.

Mains Power Keyswitch and Button.

Emergency Stop Button.

Spindle Speed and Feedrate Override Potentiometer Controls.

The spindle speed and feedrate of the Micromill can be manually overridden during a machining operation, using the potentiometer controls fitted on the lower front panel of the machine (illustrated above).

On machines not fitted with these controls, the software can be used to override both spindle speed and feedrate.

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

Mains Power Switch.

To activate power to the machine, turn the keyswitch one quarter clockwise, then push the circular red button. When power is being supplied to the machine, the circular red button will be illuminated.

Emergency Stop Button.

Pressing the emergency stop button has the effect of stopping all axes and spindle movements immediately.

To active an emergency stop, press the button in until it clicks. The emergency stop button will remain closed (continuing to cut all power to the machine drives) until the button is released. To release, push the button in and turn it clockwise until it springs back out.

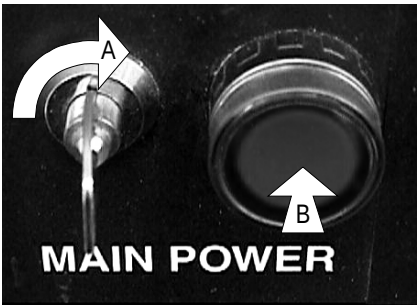
4: Switching your Micromill On and Off

Follow these instructions to switch on your Micromill CNC machine:

1) Ensure the RS232 lead is fitted securely between the PC being used to control the CNC machine and the RS232 socket at the back of the right end Micromill side panel.
2) Check the electrical panel access door (at the rear of the CNC machine) door is securely fastened.
3) Plug the mains supply cable from the CNC machine into an available power socket. Switch the power socket on.
4) Turn the keyswitch (mounted in the middle of the Micromill front panel) one quarter clockwise to its on position.
5) Press the circular red power button to the right of the keyswitch. The light inside the button will illuminate and you should hear the Micromill begin its power-up routine.

If the Micromill does not power-up, turn the keyswitch to its off position. When power is cut to the CNC machine, the light on the circular red power button should also disappear. Switch off the power at the mains socket. Check all connections and fuses to see that proper power and communication is established to the CNC machine.

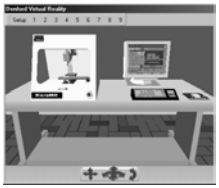
If you need to access the electrical panel at the rear of the CNC machine, wait at least 5 minutes before attempting to open the access plate. Hazardous voltages can still exist immediately after switching off the power.



To switch on the Micromill:
A: Turn the keyswitch, mounted in the middle of the front panel, one quarter turn clockwise.
B: Press the circular red power button.
In the diagram opposite, the keyswitch is shown in its off position.

4: Starting the Machine Controlling Software

The movements and operational features of your Micromill CNC machine are driven by the VR CNC Milling software.
Notice that the VR CNC Milling software operates in two different modes:



Virtual Reality Micromill.

i) Virtual Reality Mode.

This is where a computer generated 3D model of your Micromill can be controlled within the confines of your PC (see left). This is commonly referred to as Offline (ie. away from the production line involving real CNC machines).

For more information on how to use the software in this mode, please refer to the separate Virtual Reality CNC Milling Software Manual, or the Windows helpfile contained in the software.

Real Micromill CNC Machine.

ii) Machine Controller Mode.

This software mode is used when a real CNC machine (such as your Micromill hardware) is attached and controlled by the PC. In this mode, the Virtual Reality element of the software is not used. The PC is used purely as a machine controller. This mode is commonly referred to as Online (ie. directly involved with the operation of real CNC machines).

Follow these instructions to switch on the VR CNC Milling software:

1) Switch on your Micromill CNC machine, as described on the previous page.
2) Power-up the PC.
3) Start the VR CNC Milling software. You start and exit the VR CNC Milling software as you would any standard Windows application.

Startbar Icon.

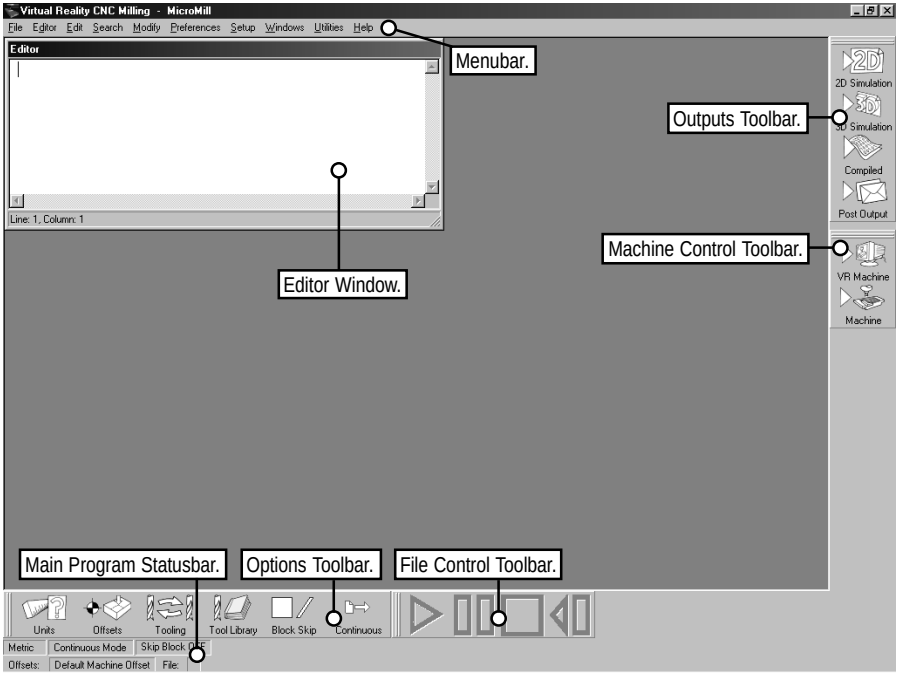
4) If VR CNC Milling has been installed using the recommended program groups, the software can be started from the Windows startbar menu in the following order, click "Start | Programs | Denford | VR Milling" (see icon shown on left).

Desktop Icon.

5) Alternatively, if you have setup a desktop shortcut to the VR CNC Milling software, double click this icon to start the software (see icon shown on left).
6) Due to the amount of information that can be shown by the software, we recommend a screen setting of at least 1024 x 768, in 16 bit High Colour.
7) To exit the VR CNC Milling software, click "File | Exit".

Important - Never exit the VR CNC Milling software when your Micromill is machining or processing any operational instructions.

4: General Layout of the Software



Note

Not all the VR CNC Milling software option windows are shown in the example screenshot.

The software will start (by default) with all toolbars docked to the edges of the main VR CNC Milling window. The toolbars contain buttons, which when clicked, open the various information windows or operate features in the software. These toolbars and windows can be freely moved around inside the main program window, to create your own screen layout.

The top titlebar of the software should read "Virtual Reality CNC Milling", followed by the name of the CNC machine that can be controlled by the software. If it does not read "Micromill", continue as outlined below.

Note

The default password to access the "Machine Properties" window is **denny**.

The name of the CNC machine that can be controlled by the VR CNC Milling software can be changed by clicking "Setup | Setup Machine Parameters". Note - a password may be required to access this part of the software. In the "Machine Properties Window", right click on the name of the CNC machine you want to control in future, then click the "Set As Active Machine" option. Refer to your separate VR CNC Milling Software User's Manual, or the helpfiles on the software CD-ROM for more detailed information.

5: Loading and Saving a CNC File

Loading a CNC File.

To load a previously saved CNC file, click "File | Open".

Select the directory used for storing the CNC file, using the "Look in:" panel.

Click on the name of the file required - its name will appear in the "File name:" dialogue box.

Click the [Open] button to load the CNC file into the "Editor" window.

Fast Loading of a known CNC file.

The "ReOpen" option can be used to gain fast access to CNC files that have been loaded in previous sessions.

Click "File | ReOpen|{choice of filename}", to reopen the required CNC file.

Saving a CNC File.

To save your CNC file, click "File | Save As".

Select the directory used for storing your CNC files, using the "Save in:" panel.

Enter the filename in the "File name:" dialogue box, using the file extension ".fnc", then click the [Save] button.

Creating a new CNC File.

Click "File | New" to begin creating a new CNC file.

A blank display will be created in the "Editor" window.

Jargon Buster

An fnc file is a FANUC milling file, containing G and M codes that describe the machining operations necessary for manufacture of the part.

5: The "Editor" window.

Note

Positioning the editing cursor:

- Position the cursor using the four computer **[Cursor]** arrow keys.
- Use **[Page Up]** to move to the top of the CNC file.
- Use **[Page Down]** to move to the bottom of the CNC file.
- Use **[Home]** to move to the beginning of the current CNC file line.
- Use **[End]** to move to the end of the current CNC file line.

Editing Shortcuts:

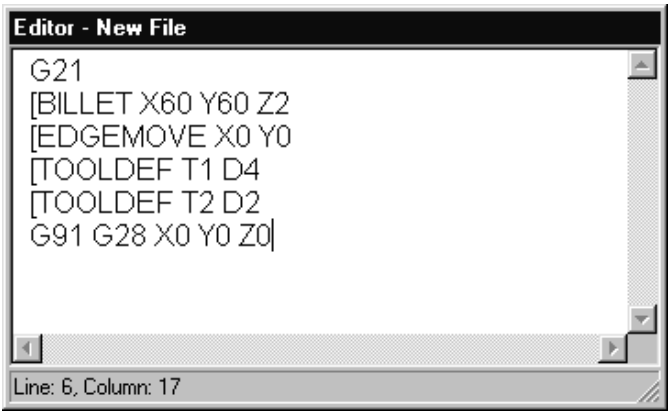
- **[CTRL + X]** will cut any highlighted text from the "Editor" window to the Windows clipboard.
- **[CTRL + C]** will copy any highlighted text from the "Editor" window to the Windows clipboard.
- **[CTRL + P]** will place any text held in the Windows clipboard to the current "Editor" window cursor position.

The "Editor" window displays the text content of your CNC file, describing the programmed sequence of commands and movements used to manufacture the part. Hence, CNC files are often referred to as Part Programs.

The Editor itself behaves in a similar way to a simple word processor, such as Windows Notepad.

Click the mouse cursor inside the "Editor" window, then begin typing in or editing the text from your CNC file.

When each line of text is completed, press the **[Enter/Return]** key to create a new program line.



CNC Programming Basics.

CNC files are constructed using G and M codes.

Each line of G and M codes is called a block, for example, "G91 G28 X0 Y0 Z0", from the part program shown above.

Each block is created from different program words, for example, "G91" is one word from the part program shown above.

Each program word is constructed from a letter, called the address, and a number. The address letter, together with its number describes the type of code used.

For more information about using G and M codes, click "Help | CNC Programming" to display the CNC Programming helpfile, containing sections on part program structure and illustrated descriptions explaining the use of each G and M code.

6: Using the Tool Library

Data relating to all the tools used with your CNC file must be entered into the "Tooling Library" and "Machine Tooling" windows, in order to perform any of the tasks outlined below:

- Simulating a CNC file in 2D or 3D.
- Running a CNC file on a Virtual Reality CNC machine.
- Running a CNC file on a real CNC machine.

Note

Upon initial installation, the default Tooling Library profiles available are as follows:

Metric Tooling:
Slot Drills, 1mm, 2mm, 3mm, 4mm, 6mm, 8mm, 10mm, 12mm;
End Mills, 1mm, 2mm, 3mm, 4mm, 6mm, 8mm, 10mm, 12mm;
Ball Noses, 1mm, 2mm, 3mm, 4mm, 6mm, 8mm, 10mm, 12mm.

Inch Tooling:
Slot Drills, 1/16" (0.0625"), 1/8" (0.125"), 3/16" (0.1875"), 1/4" (0.25"), 5/16" (0.3125"), 3/8" (0.375"), 1/2" (0.5");
End Mills, 1/16" (0.0625"), 1/8" (0.125"), 3/16" (0.1875"), 1/4" (0.25"), 5/16" (0.3125"), 3/8" (0.375"), 1/2" (0.5");
Ball Noses, 1/16" (0.0625"), 1/8" (0.125"), 3/16" (0.1875"), 1/4" (0.25"), 5/16" (0.3125"), 3/8" (0.375"), 1/2" (0.5").

The Tool Library contains the list of tool profiles that the VR CNC Milling software will let you use with your CNC machine. The data from each tool profile relates to a real cutting tool. The most common tool profiles are installed as standard, although you may need to edit some of their values to match the exact specifications of the real cutting tools you have available.

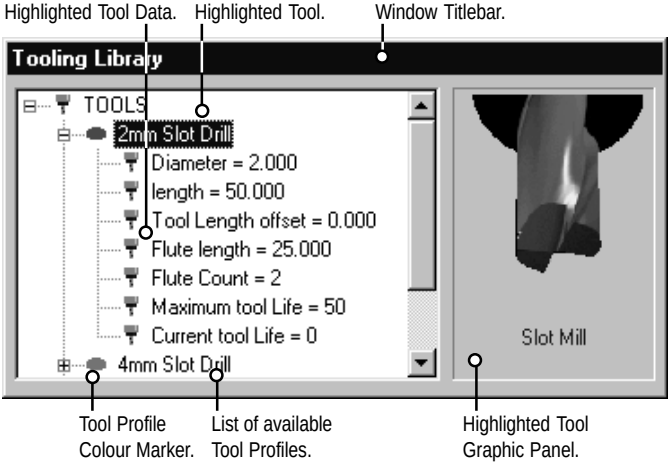
To display the "Tooling Library" window, click the [Tool Library] button, shown right, from the "Options" toolbar.

General Layout of the "Tooling Library" window.

Click on the [+] squares to expand the list of available tool profiles and data or the [-] squares to collapse any open lists.

To highlight a tool profile, click on its title. A graphic will be displayed in the right panel, relating to the type of tool selected.

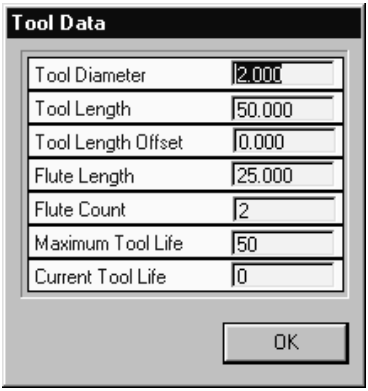
To close the "Tooling Library" window, click the [Tool Library] button, from the "Options" toolbar.



6: Viewing Tool Data

To display the data allocated to a specific tool, click the [+] square next to the text title of the tool required.

To edit a value in the tool data list, double-click the left mouse button on the data title required. The "Tool Data" window will be displayed, as shown on the right. Click the cursor in any of the yellow tool data fields, delete the old value and enter the new data.



Click the [OK] button to close the window and apply any changes made.

Note

Any data changes made are applied to identical tools from both the "Tool Library" window and the "Machine Tooling" window.

The options available in the "Tool Data" window are as follows:

Tool Diameter: The diameter of the tool, defined in mm or inches (see diagram below).

Tool Length: The length of the tool, measured from the end of the tool collet to the cutting tip of the tool, defined in mm or inches (see diagram below).

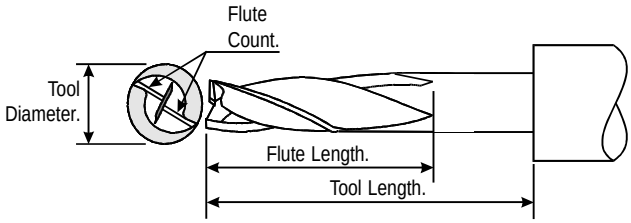
Tool Length Offset: The Z tool length offset value, defined in mm or inches. The objective of the Z tool length offsets is to allow different tool profiles to cut in the correct place on the billet, despite their obvious differences in length. Each tool is set against a common zero reference. The value indicated defines the position of this zero reference but only applies to the tool profile being viewed.

Flute Length: The length of the flute, measured from the beginning of the flute to the cutting tip of the tool, defined in mm or inches (see diagram below).

Flute Count: The number of flutes on the tool (see diagram below).

Maximum Tool Life: The working life of the tool, stated in hours.

Current Tool Life: The current life of the tool, stated in hours.



6: Adding New Tool Profiles

Creating a Personal List of Tool Profiles.

The Tool Library allows a personal list of tool profiles to be created, to match the real cutting tools used on your CNC machine.
Click the right mouse button on a highlighted tool title to display the "Tooling Library" window pop-up menu. Move the cursor down the list, highlight and click on the "Add Tool" option. A new tool will be created at the bottom of the current list of tools. Type a name for the new tool and press the [Enter] key.

Changing Tool Data Values.

To display the data allocated to the new tool, click the [+] square next to its text title. The new tool will inherit the data and graphic from the last tool highlighted in the library. To edit a value in the tool data list, double-click the left mouse button on the data title required. The "Tool Data" window will be displayed. Click the cursor in any of the yellow tool data fields, delete the old values and enter the new data. Click the [OK] button to close the window and apply any changes made.

Changing the Tool Graphic.

To change the graphic allocated to the new tool, displayed in the right panel of the "Tooling Library" window, click the right mouse button on the new tool title to display the pop-up menu. Move the cursor down the list, highlighting the "Set Tool Type" option, to display a secondary menu of possible graphic choices. Highlight and click on the title of the tool type to set the graphic in the right panel of the "Tooling Library" window.

Changing the Tool Colour.

To change the colour allocated to the new tool, click the right mouse button on the new tool title to display the pop-up menu. Move the cursor down the list, highlighting the "Set Tool Colour" option. The "Color" window will be displayed. Click one of the coloured squares in the "Basic colors" area, then click the [OK] button. The new tool colour is shown in the oval marker to the left of the new tool title in the "Tooling Library" window, as shown above. The same colour is also applied to the tool number, when the tool is being used in the "Machine Tooling" window.

6: Using the Machine Tooling window

Tool profiles from the "Tooling Library" are added to the "Machine Tooling" window, where they become ready for use with the CNC machine.

To display the "Machine Tooling" window, click the [Tooling] button, shown right, from the "Options" toolbar.

Note

When a tool profile is transferred from the "Tooling Library" window into the "Machine Tooling" window, it is assigned a tool number. Tool profiles must be transferred according to the tool numbers specified in your CNC file. In the case of ATC equipped CNC machines, this will also be the number of the carousel pocket into which the tool profile is placed.

General Layout of the "Machine Tooling" window.

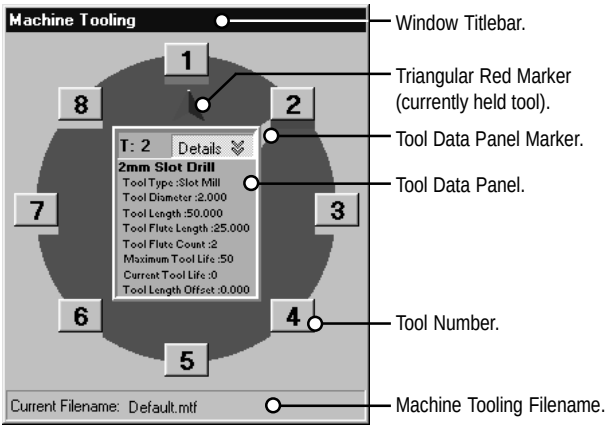
A black coloured number indicates the absence of any tool profile. When a tool is transferred from the "Tooling Library" window to the "Machine Tooling" window, the colour allocated to that tool profile is also transferred. These colours associated are applied to the tool numbers as they become filled.

Tools should be placed in numbers according to the number definitions defined in the CNC file being used. For example, if your CNC file defines T02 as a 2.0mm slot cutter, then a 2.0mm slot cutter must be transferred to number 2 in the "Machine Tooling" window.

The triangular red marker arrow indicates the tool currently held in the machine head. In the example below, tool number 1 is currently held in the machine head.

The data panel in the centre of the window relates to the tool number indicated by the grey pointer. In the example below, the data panel relates to tool number 2, a 2.0mm slot cutter.

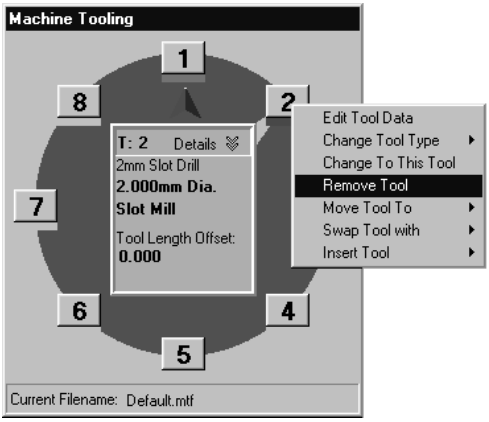
To close the "Machine Tooling" window, click the [Tooling] button, from the "Options" toolbar.



6: Transferring Tools into "Machine Tooling"

Deleting Tools from the "Machine Tooling" window.

Click the right mouse button on the number of the tool you want to remove, then highlight and click the "Remove Tool" option on the pop-up menu. The tool number will change to a black colour to indicate it is empty.

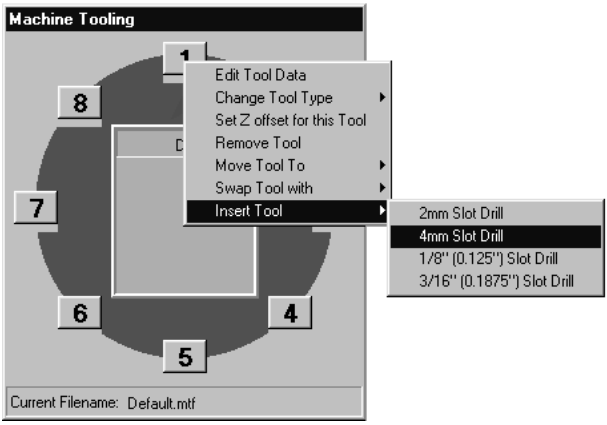


Note

Tools can also be dragged from the tooling lists in the "Tooling Library" window and dropped into the appropriate number slots in the "Machine Tooling" window.

Adding Tools into the "Machine Tooling" window.

Click the right mouse button on the empty (black) tool number in the "Machine Tooling" window. Highlight and click the "Insert Tool" option on the pop-up menu, then highlight and click on the title of the tool you want to add. The selected tool will then be allocated to the specified tool number.



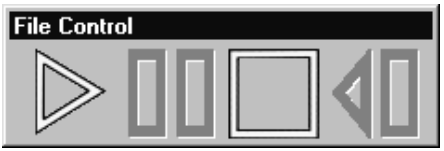
7: Running a 2D Simulation of a CNC File

The 2D Simulation window provides a plan view of the billet, together with a graphical representation of any machined parts.

To display the "2D Simulation" window, click the [2D Simulation] button, shown above, from the "Outputs" toolbar. To close the window, click the [2D Simulation] button again.

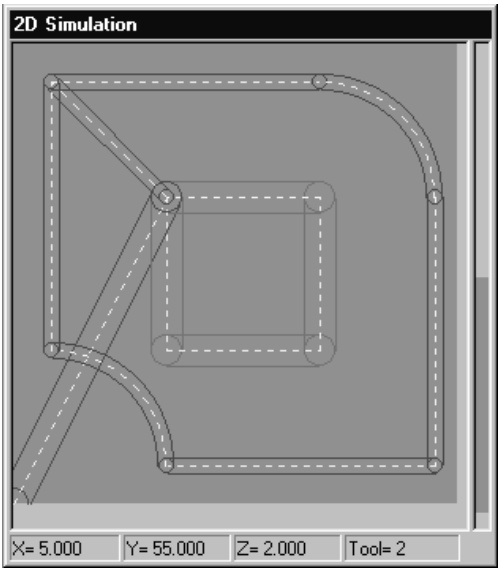
Click the "Use X,Y Offsets" option on the "2D Simulation" menu, so a tick mark is not shown next to the title. This will display the 2D simulation without using any simulated offsets.

To run the CNC file, ensure the "Editor" window cursor is positioned at the start of the first line of the CNC file.



Click the triangular [Play] button from the "File Control" toolbar, shown left.

The graphics in the "2D Simulation" window (shown below) are updated, according to the line being executed in the CNC file, until the end of the CNC file is reached.



The main area of the "2D Simulation" window depicts a full plan view of the billet.

The narrow righthand column of the "2D Simulation" window depicts the billet side view, used for indicating tool cutting depths.

Note

Before running the 2D simulation:

- Check that the units of measurement set for the VR CNC Milling software matches the units used in both the CNC file and any tooling profiles. The units of measurement setting for the VR CNC Milling software is configured using the [Units] button on the "Options" toolbar.
- Check that the tool numbers and tool profiles used in the "Machine Tooling" window match those used by your CNC file.

7: Running a 3D Simulation of a CNC File

The 3D Simulation window provides a three dimensional view of the billet, together with a graphical representation of any machined parts.

To display the "3D Simulation" window, click the [3D Simulation] button, shown above, from the "Outputs" toolbar. To close the window, click the [3D Simulation] button again.

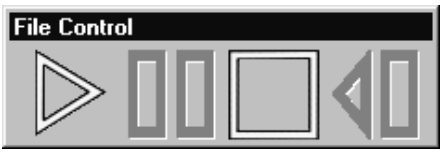
Click the "Use X,Y Offsets" option on the "3D Viewer" menu, so a tick mark is not shown next to the title. This will display the 3D simulation without using any simulated offsets.

Note

Before running the 3D simulation:

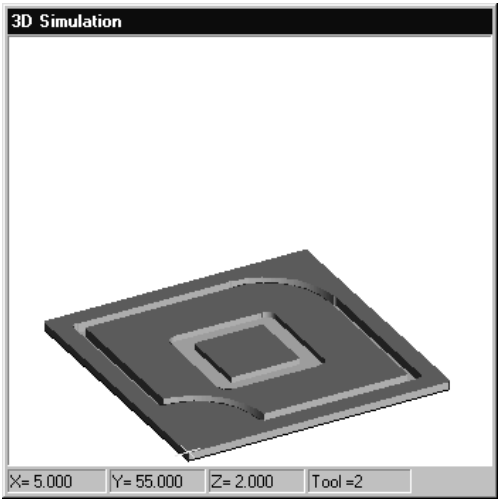
- Check that the units of measurement set for the VR CNC Milling software matches the units used in both the CNC file and any tooling profiles. The units of measurement setting for the VR CNC Milling software is configured using the [Units] button on the "Options" toolbar.
- Check that the tool numbers and tool profiles used in the "Machine Tooling" window match those used by your CNC file.

To run the CNC file, ensure the "Editor" window cursor is positioned at the start of the first line of the CNC file.



Click the triangular [Play] button from the "File Control" toolbar, shown left.

The graphics in the "3D Simulation" window (shown below) are updated, according to the line being executed in the CNC file, until the end of the CNC file is reached.



The "3D Simulation" window shows a three dimensional graphical view of the billet, which can be rotated into different viewing angles by the user.

8: Machine Communications

Follow these instructions to establish communications between the PC and your Micromill CNC machine:

1) Before attempting to connect to your Micromill CNC machine, check that the units of measurement set for the VR CNC Milling software matches the units used in both the CNC file and any tooling profiles you intend to use (ie. if your CNC file and tools are metric, select "Metric" units of measurement).

   The units of measurement setting for the VR CNC Milling software is configured using the [Units] button on the "Options" toolbar.

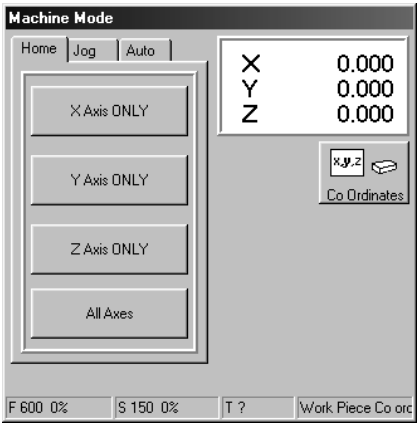
2) Click the [Machine] button, shown below, from the "Options" toolbar. This will establish a connection, via COM1 or COM2, allowing the software to control your Micromill CNC machine.

3) Once communication to your Micromill has been established, the "Machine Mode" window will appear. This window is used to control the movements of the CNC machine.

Note

If errors occur when attempting to connect to your Micromill CNC machine, please refer to section 14 : Technical Support for help.

8: Homing the CNC Machine - Home Mode



Note

The "Jog" and "Auto" tabs will not be displayed until the machine has been configured by homing all three machine axes.

When your Micromill CNC machine is first started, the "Machine Mode" window will be displayed with only the "Home" tab active, as shown above.

The "Home" tab is used for configuring the CNC machine before it can be fully used. This process is commonly referred to as homing the machine, or datuming each axis. Each of the three machine axes is sent to their fixed zero positions. This defines the three dimensional co-ordinate grid system (used for plotting tool movement positions) and the limits of movement used on the CNC machine.

After homing the machine, the zero position of the grid is referred to as the machine datum. You can find the position of the machine datum by switching the co-ordinate display in the “Machine Mode” window to read “Machine Co-ordinates”. The position of the machine datum is achieved when the X, Y and Z panels of the co-ordinate display all read zero (assuming no offsets are loaded).

Homing the CNC Machine Axes.

Note

In addition to homing the CNC machine after it has first been switched on, we also recommend homing the CNC machine after loading or configuring any offsets.

To home the machine X axis only, click the [X Axis ONLY] button. The X machine slide will move until it has found its limits of co-ordinate movement.

To home the machine Y axis only, click the [Y Axis ONLY] button. The Y machine slide will move until it has found its limits of co-ordinate movement.

To home the machine Z axis only, click the [Z Axis ONLY] button. The Z machine slide will move until it has found its limits of co-ordinate movement.

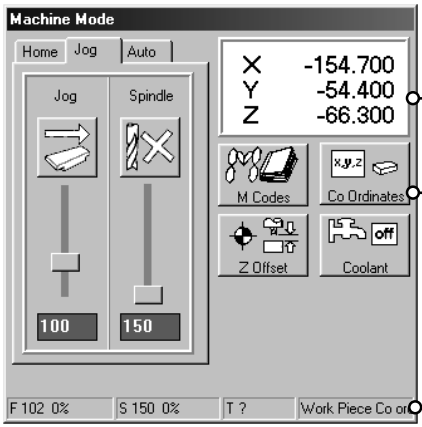
To home all three axes sequentially, click the [All Axes] button. All machine slides will move until their limits of co-ordinate movement have been found.

8: Co-ordinate System Display Modes

The [Co ordinates] button is used to switch between the two systems for displaying the co-ordinate positions: Work Piece or machine. The far right panel on the statusbar displays the current setting for this button.
When the [Units] of Measurement are set to "Inch" the co-ordinates are displayed using inches. When the [Units] of Measurement are set to "Metric" the co-ordinates are displayed using millimetres.

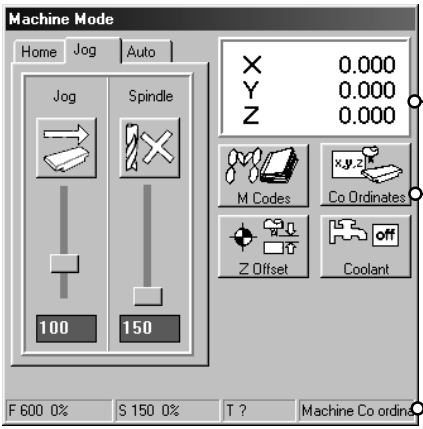
Work Piece Co-ordinates Display System

The Work Piece Co-ordinates system displays the co-ordinate position values relative to the moveable workpiece datum, defined through use of the machine offset facility.

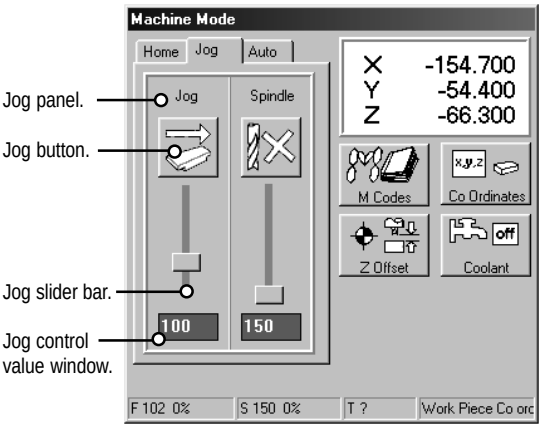


Machine Co-ordinates Display System

The Machine Co-ordinates system displays the co-ordinate position values relative to the fixed machine datum.



8: Jogging the Axes - Jog Mode



The "Jog" tab is used for manually moving the CNC machine axes within their co-ordinate working envelope.

Jog Control Modes.

Note

In order to move any of the machine axes, the "Jog panel" of the "Machine Mode" window must be active (ie, the titlebar is highlighted and the words "Jog" are highlighted in green).

The "Jog" panel displays the [Jog] button, a vertical slider bar and the jog control value window.

The machine table and head can be jogged, or moved, using two different methods, outlined on the next page. To change between these two methods, click the [Jog] button.

To change the jog control value, click and hold down the left mouse button on the slider bar, then drag the slider bar up or down to the new position.

When the [Units] of Measurement are set to "Inch" the rate of movement displayed in the jog control value window is measured using inches per minute. When the [Units] of Measurement are set to "Metric" the rate of movement displayed in the jog control value window is measured using millimetres per minute.

8: Jogging the Axes - Jog Mode

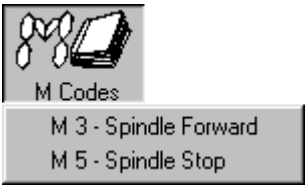
Jog Continuous.

In jog continuous mode, the selected machine axis will move at the speed displayed in the jog control value window, when one of the machine axis jog keys are pressed and held down. The selected machine axis will continue to move until the key is released. The slider bar can be moved to set jog speeds between 0 and 1000 units. When Jog Continuous is active, the [Jog] button graphic will be displayed as shown below.

Jog Step.

In jog step mode, the selected machine axis will move one increment (displayed in the jog control value window), each time the selected axis jog key is pressed. The slider bar can be moved to set jog increments of 0.01, 0.1, 0.5, 1, 5 and 10 units. When Jog Step is active, the [Jog] button graphic will be displayed as shown below.

8: Selecting M Codes



Note

If the M code you require is not displayed in the dropdown list, enter the M code in a blank "Editor" window. Run the single command line by clicking the "Auto" tab on the "Machine Mode" window, followed by the [Play] button on the "File Control" toolbar.

M codes are used for the following miscellaneous functions:

- M3 - Switch spindle on.
- M5 - Switch spindle off.

To access the list of M codes, click the "Jog" tab on the "Machine Mode" window.

Click the [M Codes] button to display the list of available miscellaneous machine functions. Highlight and click on the required M code.

For detailed information regarding M code Programming, click "Help | CNC Programming" to display the "Denford CNC Programming for Milling Machines" helpfile.

8: Jogging the Axes - Movement Keys

Moving the Axes.

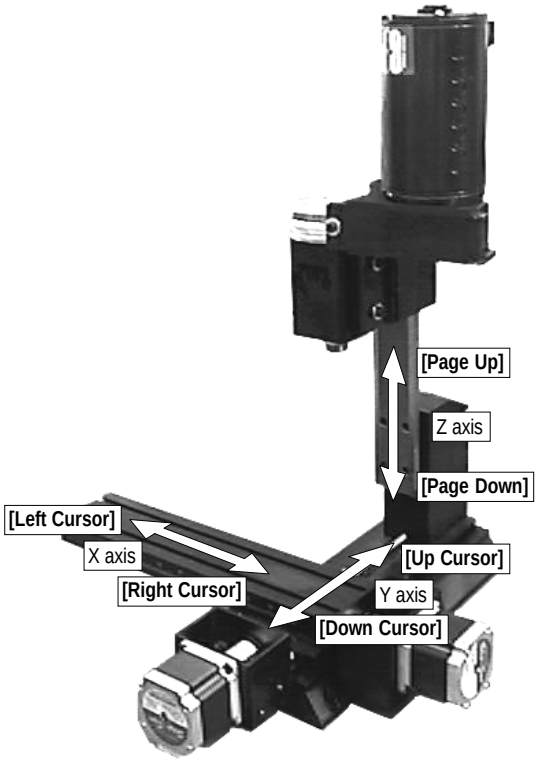
Six keys on your computer keyboard are used to manually jog (move) the CNC machine axes:

To move the X machine axis use the **[Left Cursor]** and **[Right Cursor]** arrow keys, with the "Machine Mode" window active.

To move the Y machine axis use the **[Up Cursor]** and **[Down Cursor]** arrow keys, with the "Machine Mode" window active.

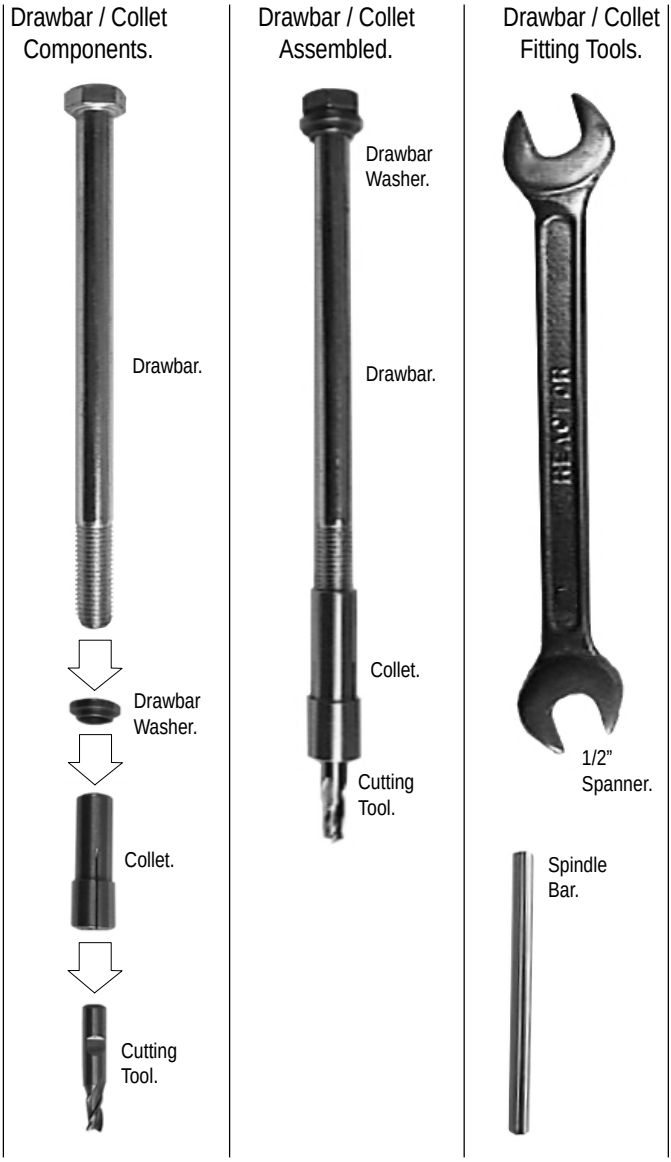
To move the Z machine axis use the **[Page Up]** and **[Page Down]** keys, with the "Machine Mode" window active.

In order to move any of the machine axes, the "Jog panel" of the "Machine Mode" window must be active (ie, the titlebar is highlighted and the words "Jog" are highlighted in green).



9: The Drawbar and Collet Tooling System

Diagram showing layout of the standard Drawbar and Collet Tooling System components.



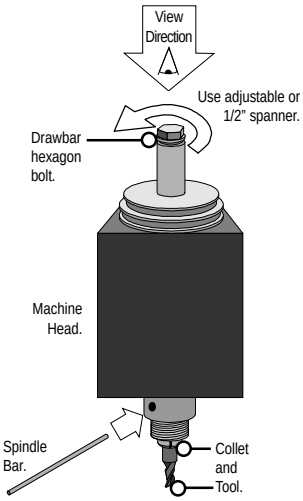
9: Removing a Tool from the Drawbar / Collet

Removing a Tool.

1) Place a soft cloth under the spindle, incase the tool unexpectedly falls out of the collet during the operation. Move the machine head approximately halfway down the Z axis, to allow easy access to both the top and bottom of the spindle head.
2) Fit the spindle bar (supplied) into one of the holes on the threaded spindle body under the machine head. Fit a 1/2" or adjustable spanner around the drawbar hexagon bolt on the top of the machine head.
3) Using the spindle bar to stop the spindle from rotating, loosen and turn the drawbar hexagon bolt anticlockwise three turns (see diagrams below). Take care not to over-loosen the drawbar hexagon bolt.

Note

If the collet falls out but the tool remains inside the collet, you will need to manually push the tool out.



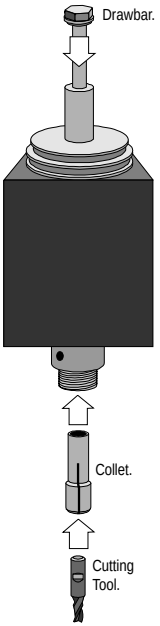
Safety First !

Never open the safety guard door and enter the working area when the spindle or machine axes are moving.

4) Using a hammer, gently tap the top of the drawbar hexagon bolt. The tool will loosen or completely break free from the collet.
5) If you would also like to remove the collet, continue to unscrew the drawbar until the collet is released.

9: Installing a Tool into the Drawbar / Collet

Installing a Tool.

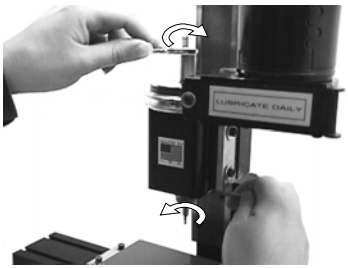


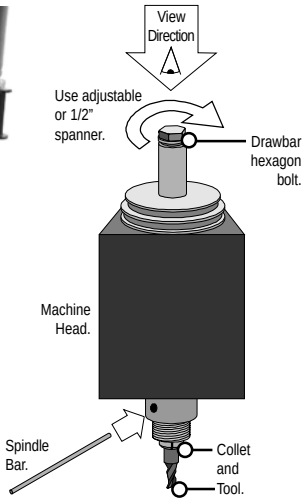
1) Move the machine head approximately halfway down the Z axis, to allow easy access to both the top and bottom of the spindle head.
2) If the collet is already in the spindle, proceed to step 3). If the collet needs to be installed, follow as outlined below.

   Feed the drawbar through the hole at the top of the machine head. Ensure the washer is fitted to the drawbar in the correct orientation. Reach under the machine head and insert the collet into the spindle. Begin to thread the collet onto the drawbar (see diagram left). Do not completely tighten the collet. At this stage, you only need to finger tighten the drawbar.
3) Place the desired tool into the collet. If necessary, tighten the drawbar by hand until the tool is held in the collet.
4) Fit the spindle bar (supplied) into one of the holes on the threaded spindle body under the machine head. Fit a 1/2" or adjustable spanner around the drawbar hexagon bolt on the top of the machine head.
5) Using the spindle bar to stop the spindle from rotating, tighten the drawbar hexagon bolt by turning it in a clockwise direction. Take care not to over-tighten the drawbar hexagon bolt (see diagrams below).

Safety First !

Never open the safety guard door and enter the working area when the spindle or machine axes are moving.

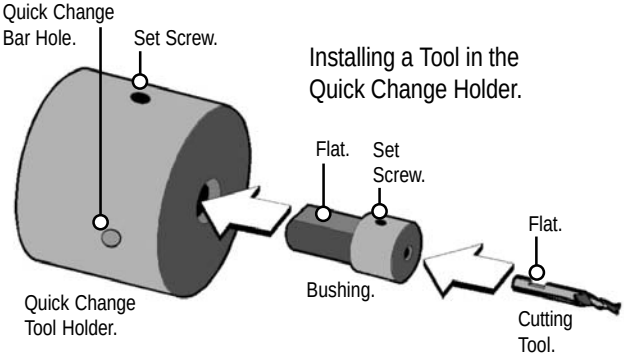




6) Check the tool is secure in the collet and the collet is secure in the spindle, before using the tool.

Installing a Tool in the Quick Change Holder.

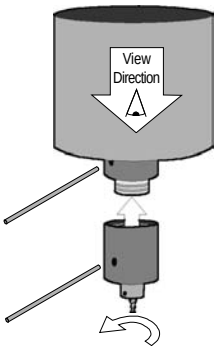
1) Remove the drawbar and collet, as outlined on page 47, before working with the quick change tooling.
2) Insert the cutting tool into the bushing, aligning the flat on the cutting tool with the position of the bushing set screw. Tighten the bushing set screw to lock the cutting tool in position.
3) Insert the bushing assembly into the quick change holder, aligning the flat on the bushing with the position of the quick change holder set screw. Tighten the quick change holder set screw to lock the bushing assembly in position.



Installing a Tool in the Quick Change Holder.

Installing a Quick Change Holder on the Spindle.

1) Thread the quick change tool holder onto the spindle until it is finger tight.
2) Fit the spindle bar into one of the holes on the spindle body under the machine head. Fit the quick change tool holder bar into one of the holes on the quick change holder body.
3) Using the spindle bar to stop the spindle from rotating, tighten the quick change holder using the quick change tool holder bar. Take care not to over-tighten the thread.
6) Check all components are secure before using the tool.



Installing a Quick Change Holder on the Spindle.

9: Manual Tool Changing with the Software

Note

Calling a Manual Tool Change.

Ensure the tool profiles and tool numbers have been set in the VR CNC Milling software to match the real tooling hardware and tool numbers used with your CNC machine. See the separate VR CNC Milling Software User's Manual or the helpfile for further information.

Note

Easier access to the tooling components may be obtained by moving the machine head halfway down the Z axis.

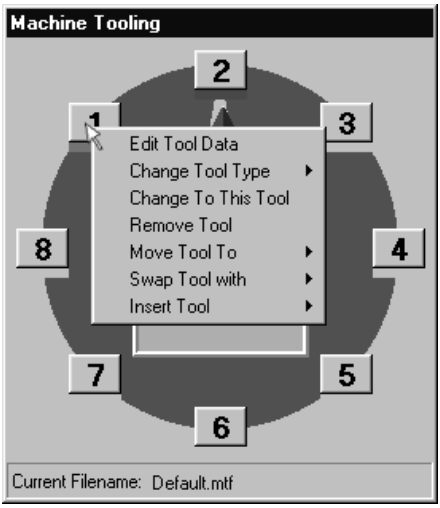
Safety First !

Never open the safety guard door and enter the working area when the spindle or machine axes are moving.

When manually moving (jogging) the axes.

Before beginning a manual tool change operation, we recommend you home all machine axes.

To home the machine axes, click the "Home" tab on the "Machine Mode" window.
Click the [Z Axis ONLY] button. The machine head will move up, along the Z slide, until it has reached its home position.
Click the [X Axis ONLY] button. The machine table will move along the X slide, until it has reached its home position.
Click the [Y Axis ONLY] button. The machine table will move along the Y slide, until it has reached its home position.



In the example screenshot, tool 2 is currently in the spindle. We want to change to tool 1, so we right click on the number 1 graphic, then highlight and click the "Change To This Tool" option.

Click the [Tooling] button on the "Options" toolbar to display the "Machine Tooling" window. Right click on the number of the new tool you want to use, then highlight and click the "Change To This Tool" option from the pop-up menu that appears.

An "Information" window will be displayed, prompting you to begin manually changing to the new tool number.

Always wait for the spindle and machine axes to stop moving, before attempting to open the safety guard door. After changing to the new tool number, close the safety guard door, then click the [OK] button on the "Information" window to update the tooling data used by the software.

9: Manual Tool Changing with the Software

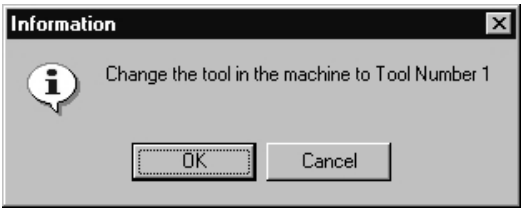
During the running of a CNC program.

On reading a tool change operation line in your CNC program, all three machine axes will move to their home positions, via an intermediate point, if programmed.

Note

Easier access to the tooling components may be obtained by moving the machine head halfway down the Z axis.

At this point, the software will pause the CNC program and a message window will be displayed (similar to the one shown below), prompting you to manually change tools.



Safety First !

Never open the safety guard door and enter the working area when the spindle or machine axes are moving.

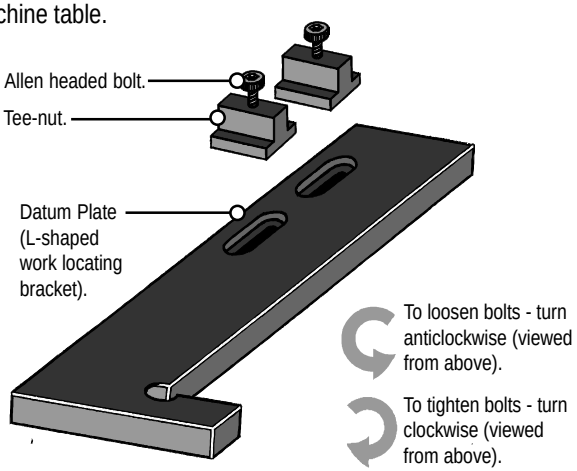
Always wait for the spindle and machine axes to stop moving, before attempting to open the safety guard door.

Replace the current tool number with the tool number specified in the message window (the tool profiles allocated to each tool number are listed at the beginning of your CNC program).

Close the safety guard door and click on the [OK] button to resume your machining.

10: The Datum Plate

The Datum Plate is an L-shaped bracket used for locating work to a precise position on the machine table. It is fixed in position using two Tee-nuts that are tightened using allen headed bolts. The tee-nuts are located in each of the two upside down T shaped channels that run horizontally (ie. parallel to the X axis) under the surface of the machine table.



Removal of Datum Plate.

To remove the datum plate, loosen the 2 allen headed bolts, by turning them in an anticlockwise direction. Slide the datum plate along the machine table, until the Tee-nuts are released from their channels, then withdraw the datum plate from the machine table.

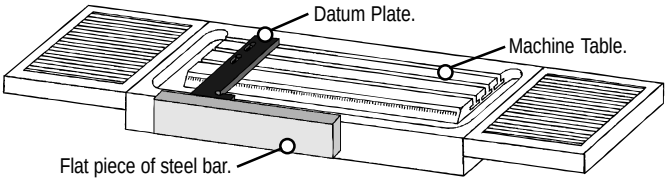
Fitting of Datum Plate.

To fit the datum plate in position on the machine table, place it at the end of the T shaped channels. Align each Tee-nut with its respective channel and slide it into position. Slide the datum plate along the machine table to the required position. Note that the datum plate can be slightly adjusted forwards and backwards (ie. parallel to the Y axis), if required. 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 with respect to the machine axes.

10: Setting the Datum Plate

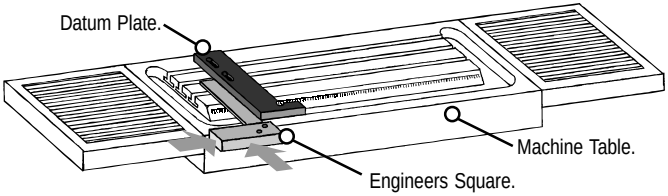
The following diagrams illustrate the various methods that can be used when positioning the datum plate square with respect to the machine table (ie. the edges of the datum plate run exactly parallel with the X and Y machine axes). Each method varies according to the level of position accuracy required.

Datum Plate Setting Method 1.



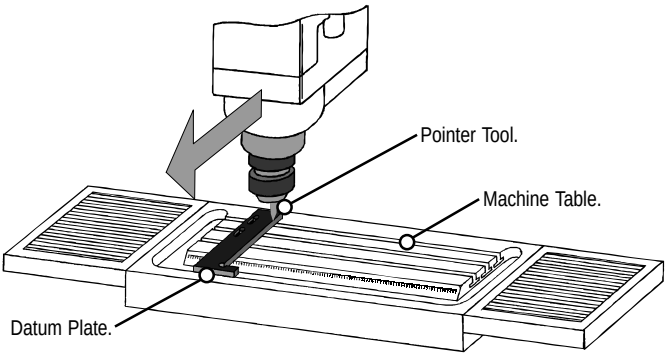
This method is useful if the front face of the datum plate can be positioned exactly level with the front edge of the machine table. Use the true flat face of a section of material, such as a piece of flat steel bar. Press the steel bar firmly against the front edge of the table and adjust the datum 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.

Datum Plate Setting Method 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 middle of the machine table.

Datum Plate Setting Method 3.

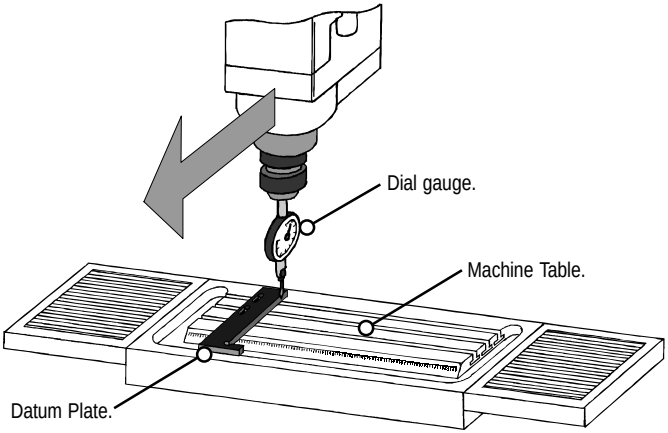


Set up the machine so a pointer is held in place of the cutting tool. Align the pointing tool so it is positioned slightly above one of the 2 edges of the datum plate, which run parallel with the Y axis.

Start with the pointer near the back of the datum plate edge you have chosen. 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 align, readjust the position of the datum plate. 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.

Datum Plate Setting Method 4.

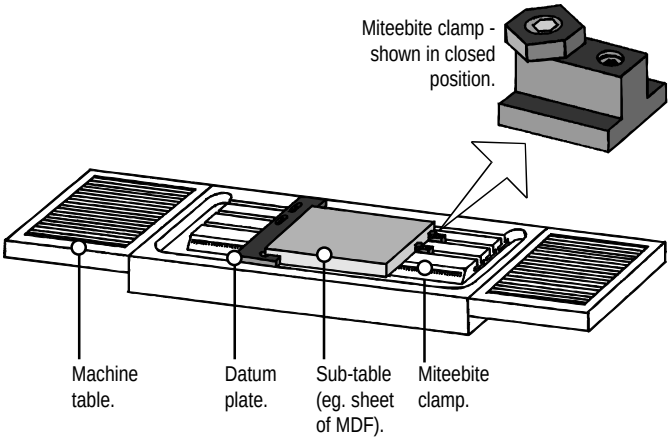


Set up the machine so a dial gauge is held in place of the cutting tool. Align the dial gauge so it is positioned along one of the 2 sides of the datum plate, which run parallel with the Y axis.

Start with the dial gauge near the back of the datum plate edge you have chosen. 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, readjust 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, until a suitable degree of accuracy has been obtained. Finally, tighten the allen headed bolts to fix the datum plate firmly in place.

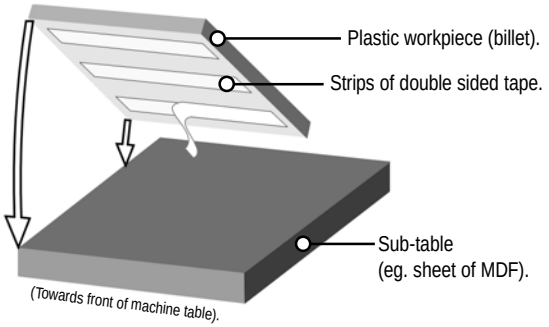
10: Miteebite Clamps

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



Loading the Billet.

The actual workpiece, such as a sheet of plastic, would be held in place on the sub-table using double sided tape. The billet is usually positioned with its front and lefthand edges aligned with the front and lefthand edges of the sub-table, as shown below.



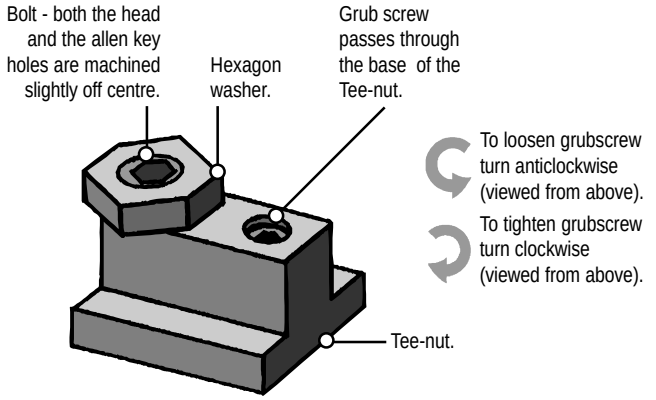
10: How does a Miteebite Clamp work?

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

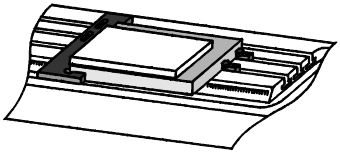
One of these threaded holes contains a grubscrew. When this is tightened, the base of the grubscrew pushes against the surface of the T channel in which it has been placed, securing the Tee-nut in position.

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 (ie. 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 (ie. 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.



10: Using Miteebite Clamps



The example used in the description below uses a plastic sheet billet, held on a temporary MDF sub-table using double sided tape.

Set the Datum Plate into position, then place the temporary MDF sub-table onto the machine table, so it is located correctly against the edges of the datum plate.

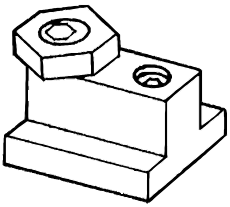
Next, position the miteebites into their respective T channels and slide them along until they touch the sub-table. Ensure that one of the six flat sides of the hexagon washers press against the sub-table, not one of the hexagon points. The hexagon washers should be positioned at this stage so they are open (ie. the off-centre allen key holes on the bolts should be facing away from the grubscrews).

Now tighten the grubscrews in each miteebite to lock them firmly in position. At this stage, it should still be possible to remove the sub-table. Remember, the grubscrews only lock the miteebites in position on the machine table - it is the hexagon washers which actually lock the sub-table in position.

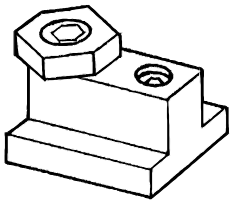
To lock the temporary MDF bed firmly in place, turn the bolts with the off-centre allen key holes 180 degrees so the hexagon washers are in the closed position (ie. the off-centre allen key holes on the bolts should now be facing towards the grubscrews).

Now that the miteebites have been set, the sub-table 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.

Hexagon washer set in open position.



Hexagon washer set in closed position.



11: Introducing Offsets

What are offsets?

Offsets are a collection of numerical values used to describe the location of the workpiece datum. Two types of offset file are used, in combination, to describe this location:

i) The workpiece offset file - This file allows global offset values to be set for the X, Y and Z axes. In other words, every tool profile will use the workpiece offset values.

ii) The tool length offset files - Every tool has its own individual tool length offset file, containing a single Z offset value. They are used to compensate for the differences in length between tools.

How is an offset calculated?

The X position of the workpiece datum is defined by the value entered into the X dialogue box of the workpiece offset file.

The Y position of the workpiece datum is defined by the value entered into the Y dialogue box of the workpiece offset file.

The Z position of the workpiece datum is defined by the combination of the value entered into the Z dialogue box of the workpiece offset file and the value entered into the dialogue box of the tool length offset file that belongs to the tool profile currently in use.

How is the workpiece datum used?

The machine controlling software uses the workpiece datum as the starting point (zero reference) for any co-ordinate movements it receives. These co-ordinate movements are read from our loaded CNC file. In other words, the position of the workpiece datum will determine the place on the CNC machine where our part is manufactured.

Jargon Buster

The moveable workpiece datum defines the zero point on our workpiece (the material we want to machine) - the starting point for any cutting co-ordinates supplied by the machine controller.

The fixed machine datum defines the zero point for the three dimensional co-ordinate grid system used by the machine.

What actually happens when I program my workpiece datum position?

Configuring the workpiece datum position shifts, or offsets, the entire three dimensional co-ordinate grid system used by the CNC machine. The workpiece datum will now be read by the CNC machine as its zero position, rather than the machine datum (to find out more about the machine datum see page 41).

Where should I position the workpiece datum on my billet?

This depends on the position of the part datum set in your CNC program. The part datum is the zero reference, or starting point, used when plotting all the co-ordinates that describe the shape of your design.

The part datum could have been set by the programmer, when manually writing the CNC program from a traditional engineering drawing, or automatically set by a CAD/CAM software package.

For example, if you used the CAD/CAM software package, Denford MillCAM Designer, your design would have been drawn within a fixed area, representing the size of the billet you intend to use. The software would then have generated the CNC program, automatically setting the front, left upper corner of this imaginary billet as the part datum. In this case, you would need to position the workpiece datum in the front, left upper corner of the real billet on the machine table.

What happens if I don't use any offsets with my CNC file?

If no offset is programmed, the machine controlling software will use the machine datum as the starting point (zero reference) for any co-ordinate movements it receives. Since it is unlikely that the position of the machine datum is the place where you want any machining to begin, your CNC machine will attempt to manufacture your design in the wrong place in its working area. Offsets are very important because without them, the CNC machine will not know where to begin cutting on your billet. Offsets must always be configured before manufacturing the part.

Are standard offset files supplied?

No, you must set your own. We DO NOT supply any standard offset files with the machine software. However, once you have configured and saved your offset files, the same files may be used over and over again, so long as the following holds true:

- The same cutting tools are used.
- The billet size does not change.
- The fixture that holds the billet does not move position on the machine table.

Jargon Buster

The part datum defines the zero point in our CNC program - the starting position from which all co-ordinates that describe the shape of our design are plotted.

11: Configuring a Workpiece Offset

The workpiece offset file contains three values, used to describe the location of your workpiece datum. They determine how much you want to shift the zero reference position of the CNC machine along the X, Y and Z axes.

However, if your CNC file uses two or more tool profiles, the workpiece offset file will not account for the difference in length between the tools. To achieve this, you must also configure a tool offset value for each tool profile you intend to use (see page 64).

Before you can begin entering the workpiece offset values, you must position the tool over your workpiece datum.

Move the Tool over the Workpiece Datum.

Highlight the "Machine Mode" window and select the "Jog" tab. To move the tool, use the six movement keys:

X Axis: **[Cursor Right]** or **[Cursor Left]**

Y Axis: **[Cursor Up]** or **[Cursor Down]**

Z Axis: **[Page Up]** or **[Page Down]**

Remember, the tool and machine table can only be moved when the "Machine Mode" window is active.

Move the tool so its cutting tip just touches your chosen workpiece datum position. Take care not to damage the cutting tip, when manoeuvring the tool.

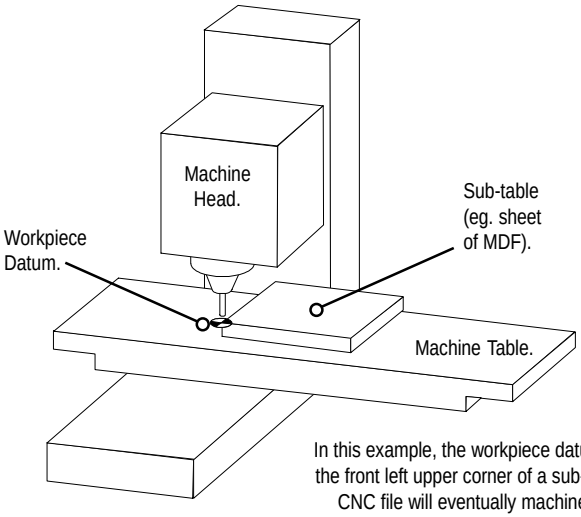
Note

Movement Modes:

To switch between the two modes of movement, click the [Jog] button in the "Machine Mode" window.

Use "Jog Step Continuous" for continuous movement, when a movement key is pressed and held down. The slider bar can be moved to set jog speeds between 0 and 1000 units. Use this mode for rough positioning.

Use "Jog Step Mode" for a single increment movement, each time a movement key is pressed. The slider bar can be moved to set jog increments of 0.01, 0.1, 0.5, 1, 5 and 10 units. Use this mode for precise positioning.

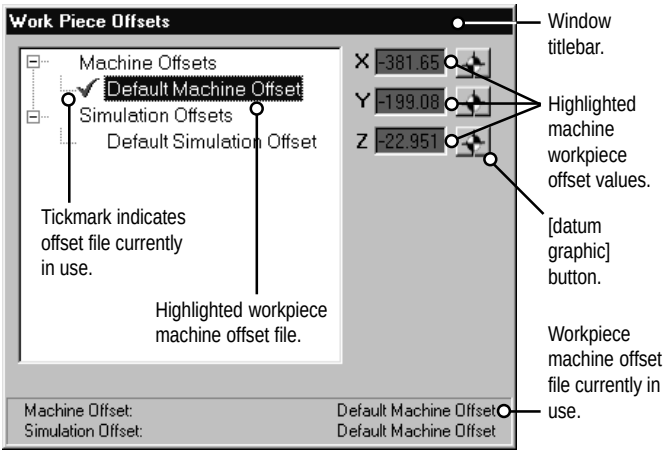


In this example, the workpiece datum is set at the front left upper corner of a sub-table. The CNC file will eventually machine a piece of plastic that will be taped to the sub-table.

11: Creating a new Workpiece Offset File

To display the "Work Piece Offsets" window, click the [Offsets] button, shown left, from the "Options" toolbar. To close the window, click the [Offsets] button again.

General Layout of the "Work Piece Offsets" window.



Click on the [+] squares to expand the "Machine Offsets" list or the [-] squares to collapse an open list. To highlight a machine offset, click on its title.

The co-ordinate display panel, to the right of the "Machine Offsets" list, shows the X, Y and Z co-ordinates assigned to the highlighted machine offset.

A red tick mark is used to indicate the active (currently used) machine offset, also shown in the statusbar, positioned at the bottom of the "Work Piece Offsets" window.

Note

Notice there are two types of workpiece offset file used in the VR CNC Milling software:

i) The workpiece **machine** offset files - these values are used by any VR or real CNC machines. It is these files that are referred to in this section.

ii) The workpiece **simulation** offset files - these values are only used when simulating an offset with the 2D or 3D graphics.

To create a new Workpiece Machine Offset file:

1) Highlight the current machine offset, then click the right mouse button on its title to display a pop-up menu. Highlight and click the "Add Offset" option from the pop-up menu.
2) The new offset is always added to the bottom of the machine offsets list, with all co-ordinate values set to zero. Highlight the "New Offset", then click the right mouse button on its title to display a pop-up menu. Highlight and click the "Make Current" option from the pop-up menu. This configures the "New Offset" as the currently active machine offset, ie, the offset file used by the CNC machine.

To transfer the X, Y or Z co-ordinate values into the Workpiece Machine Offset file:

1) Transfer the Co-ordinate Value.

   Check the "Machine Mode" window is configured to display workpiece co-ordinates, by clicking the [Co-ordinates] button so that "Work Piece Co-ordinates" is displayed in the statusbar. Check that the required machine offset file is highlighted in the "Work Piece Offsets" window and its X, Y and Z values are set to zero. Click one of the three [datum graphic] buttons to the right of either the X, Y or Z offset value display box - the specific [datum graphic] button clicked will depend on which offset value is being configured.

2) Specify the Cutter Orientation.

   Before transferring the X, Y or Z co-ordinate value, the "Set Offset" window is displayed. The settings are used to account for the cutter orientation method used. For example, if the centre of the tool is aligned over the workpiece datum, select the graphic with crosshairs over the centre of the tool and check the "Use Cutter Radius" option. Click the green tickmarks or red crosses to change the options. Click the [OK] button to confirm the settings.

3) Check the Co-ordinate Value has registered correctly.

   The original X, Y or Z co-ordinate value from the "Machine Mode" window should have been transferred into the appropriate offset value display box.

   The X, Y or Z co-ordinate value in the "Machine Mode" window should read zero, indicating that the selected component of the workpiece datum has been registered correctly.

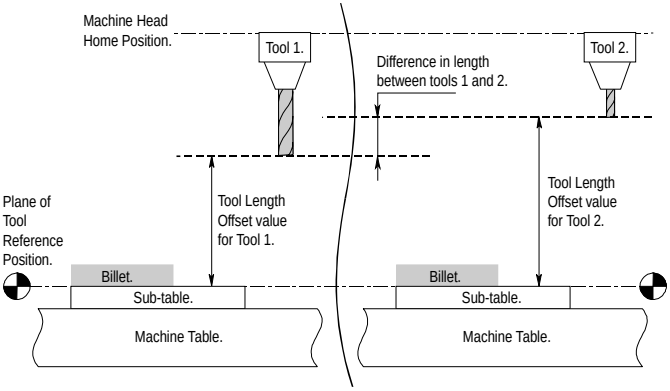
Note

More detailed information regarding the theory and practice of configuring Workpiece Offset files can be found in your separate VR CNC Milling Software User's Manual.

11: Configuring the Tool Length Offset

The tool length offset files each contain a single Z value. A separate tool length offset file must be configured for every tool we want to use. They allow us to establish a common workpiece datum position, no matter what length of tool is used with the CNC file.

Select a point on your machine table, sub-table, or billet that can be reached by all the tools you intend to use. All tool length offsets are configured against this common tool offset reference point. When values are entered into each individual Z length tool offset file, each tool will use this reference point as their zero co-ordinate along the Z axis. It is this figure that compensates for the differences in length when various tools are used together on the same job.



Before you can begin entering the machine offset values, you must position the tool over your workpiece datum.

11: Configuring the Tool Length Offset

Move the Tool over the chosen Tool Offset Reference Position.

Note

Movement Modes:

To switch between the two modes of movement, click the [Jog] button in the "Machine Mode" window.

Use "Jog Step Continuous" for continuous movement, when a movement key is pressed and held down. The slider bar can be moved to set jog speeds between 0 and 1000 units. Use this mode for rough positioning.

Use "Jog Step Mode" for a single increment movement, each time a movement key is pressed. The slider bar can be moved to set jog increments of 0.01, 0.1, 0.5, 1, 5 and 10 units. Use this mode for precise positioning.

Move the Cutter to the chosen Tool Offset Reference position.

Highlight the "Machine Mode" window and select the "Jog" tab. To move the tool, use the six movement keys:

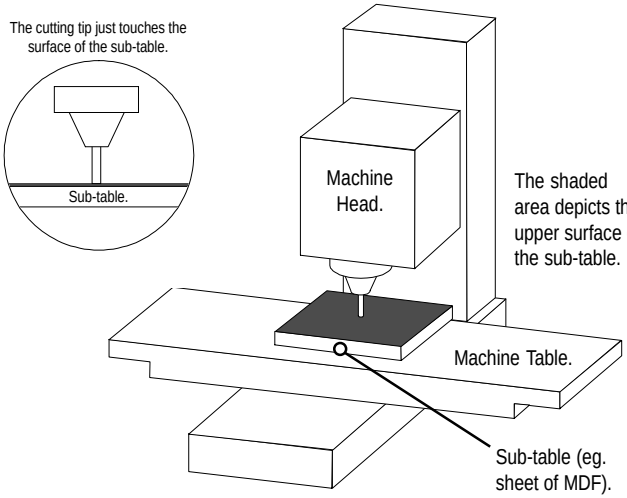
X Axis: **[Cursor Right]** or **[Cursor Left]**

Y Axis: **[Cursor Up]** or **[Cursor Down]**

Z Axis: **[Page Up]** or **[Page Down]**

Remember, the tool and machine table can only be moved when the "Machine Mode" window is active.

Move the tool so its cutting tip just touches the tool offset reference position. Take care not to damage the cutting tip, when manoeuvring the tool.



In this example, the tool reference position has been chosen as the upper surface of a sub-table. This position can be easily reached by all the different tool profiles we intend to use with the CNC file.

11: Configuring the Tool Length Offset

To transfer the Z co-ordinate value into the Tool Length Offset:

Note

The Z Tool Length Offset value is saved to the tool profile itself, viewed in the "Machine Tooling" window, not the workpiece machine offset file viewed in the "Work Piece Offsets" window.

1) Specify the Tool Length Offset Value.

   From the "Machine Mode" window, click the [Z Offset] button. The "Set Offset" window, used for entering the tool length offset value, is displayed. The Z co-ordinate currently shown in the "Machine Mode" window is the value (including any sign) that must be entered into the dialogue box. Note that the software will automatically display this value in the dialogue box, by default, but feel free to change it if necessary.

2) Confirm and set the Tool Length Offset Value.

   Click the [OK] button to confirm the value and close the "Set Offset" window. This completes the tool length offset process for the profile currently held in the machine head.

3) Repeat the Tool Length Offset process for the next tool profile.

   Place the next tool profile in the machine head. Align this tool against the tool offset reference position, then specify and confirm the tool length offset values as described in 1) and 2). Repeat the tool length offset process for all remaining tool profiles.

   Note - It is important that all tools are configured against the same tool offset reference position.

Note

More detailed information regarding the theory and practice of configuring Tool Length Offset files can be found in your separate VR CNC Milling Software User's Manual.

12: Manufacturing your Part - Auto Mode

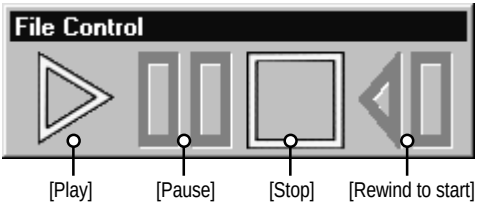
Before beginning to manufacture your part, check to see that the following tasks have been completed:

- Billet mounted and secure.
- Tools prepared and numbered ready for use (according to your CNC file).
- Safety guard door closed and machine switched on.
- CNC file loaded and checked via simulation.
- Offset files configured or loaded.
- Machine homed (datumed).

To manufacture your part:

1) Check the cursor in the "Editor" window is positioned at the beginning of your CNC file.
2) Click the "Auto" tab in the "Machine Mode" window.
3) Click the triangular [Play] button on the "File Control Toolbar", to begin the machining process.

File Control Toolbar Buttons.



Safety First !

Never open the safety guard door during the machining process.

4) A message window will be displayed whenever a tool change is required. Wait for all machine movement to stop before opening the safety guard door, then change to the new tool number requested. Close the safety guard door and click the [OK] button in the message window to resume machining.

Note

The spindle **will not** switch off when the [Stop] button on the "File Control" toolbar is clicked.

The spindle **will** switch off when the emergency stop button is pressed.

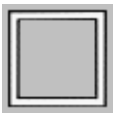
To stop machining, click the square [Stop] button on the "File Control" toolbar or press the emergency stop button located on the righthand side of the Micromill front panel. Read the next page for important information regarding stopping the machining process.

12: Stopping the Machining Process

Machining will need to be restarted if the part manufacture process is halted. Follow the instruction below to restart machining:

If the emergency stop button is used to stop the machining process:

1) Release the emergency stop button by pressing the button in whilst turning it in a clockwise direction. The button will spring back out.
2) A machine error window may be displayed. Click the [OK] button to continue.
3) Click the [Stop] button on the "File Control" toolbar.
4) If required, click the [Rewind to start] button to move the cursor in the "Editor" window back to the beginning of the CNC file.
5) Click the "Home" tab in the "Machine Mode" window, then home all three the machine axes.
6) Click the "Auto" tab in the "Machine Mode" window, then click the [Play] button on the "File Control" toolbar, to restart the machining process.



If the [Stop] button on the "File Control" toolbar is used to stop the machining process:

1) The spindle will not switch off when using the [Stop] button. To switch off the spindle, click the "Jog" tab in the "Machine Mode" window, then click the [M Codes] button. Highlight and click the "Spindle Stop" option from the pop-up menu.
2) If required, click the [Rewind to start] button to move the cursor in the "Editor" window back to the beginning of the CNC file.
3) Click the "Home" tab in the "Machine Mode" window, then home all three the machine axes.
4) Click the "Auto" tab in the "Machine Mode" window, then click the [Play] button on the "File Control" toolbar, to restart the machining process.

12: Overriding Feedrates and Spindle Speeds

Using Potentiometer Controls.

The left side of the front panel, when viewed from the front of the machine, contains the spindle speed and feedrate override potentiometer controls.

Note

In Auto Mode, feedrate and/or spindle speed changes will only be registered when an actual feedrate or spindle speed is being applied by the controller.

The spindle speed and feedrate of the Micromill can be manually overridden during a machining operation using the potentiometer controls fitted on the lower front panel of the machine (illustrated above).

On machines not fitted with these controls, the software can be used to override both spindle speed and feedrate (see the next page).

The spindle speed can be overridden between 50% and 120%.

The feedrate can be overridden between 0% and 150%.

To increase the spindle speed or feedrate, rotate the appropriate control clockwise.

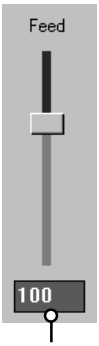
To decrease the spindle speed or feedrate, rotate the appropriate control anticlockwise.

12: Overriding Feedrates and Spindle Speeds

On CNC machines not fitted with potentiometer controls, both the feedrate and spindle speed can be changed using the VR CNC Milling software slider bars, shown below.

Software Feedrate Override.

> **Note**
>
> In Auto Mode, feedrate and/or spindle speed changes will only be registered when an actual feedrate or spindle speed is being applied by the controller.



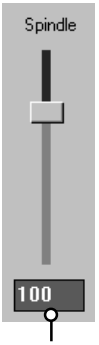
Percentage readout.

The "Feed" panel displays a vertical slider bar and the current feedrate override, displayed as a percentage. Override values between 1-100% are set in the lower grey portion of the slider bar. Override values between 100-150% are set in the upper red portion of the slider bar.

To change the feedrate value, click and hold down the left mouse button on the slider bar, then drag the slider bar up or down to the new position.

When the [Units] of Measurement are set to "Inch" the feedrate is measured using inches per minute. When the [Units] of Measurement are set to "Metric" the feedrate is measured using millimetres per minute.

Software Spindle Speed Override.



Percentage readout.

The "Spindle" panel displays a vertical slider bar and the current spindle speed override, displayed as a percentage. Override values between 1-100% are set in the lower grey portion of the slider bar. Override values between 100-120% are set in the upper red portion of the slider bar.

To change the spindle speed value, click and hold down the left mouse button on the slider bar, then drag the slider bar up or down to the new position.

The spindle speed is measured using revolutions per minute.

13: Maintenance Schedule and Charts

Maintenance Schedule.

Daily	• Clean and remove any swarf. • Check that all slides are lubricated. • Clean tooling and tool holders.
Weekly	• Clean the machine. • Check all exposed screws and nuts for tightness. • Oil all slides, leadscrews and thrust bearings. • Oil optional vice slideways.
Bi-annually	• Check the condition of any electrical connections. • Check and clean tooling collets. • Check all cables for kinks and breaks. • Check and adjust spindle drive belt. • Clean the 3 axis datum microswitches.
Annually	• Check the slides for wear. • Check and adjust gib strips.

Note

If your CNC machine is used intensively, we recommend that the maintenance tasks listed in the above schedule are performed on a more regular basis.

Lubrication Chart.

Lubrication Point	Lubricating System	Frequency	Recommended Oil	Quantity
Slide ways, Leadscrews & Thrust Bearings.	Pump-action oil can	As required	Light Machine Oil, such as “3 in 1” or “WD40”	As required

Note

The bearings in the machine head are lifetime lubricated bearings, therefore require no periodic lubrication.

13: Lubrication Areas on the Micromill

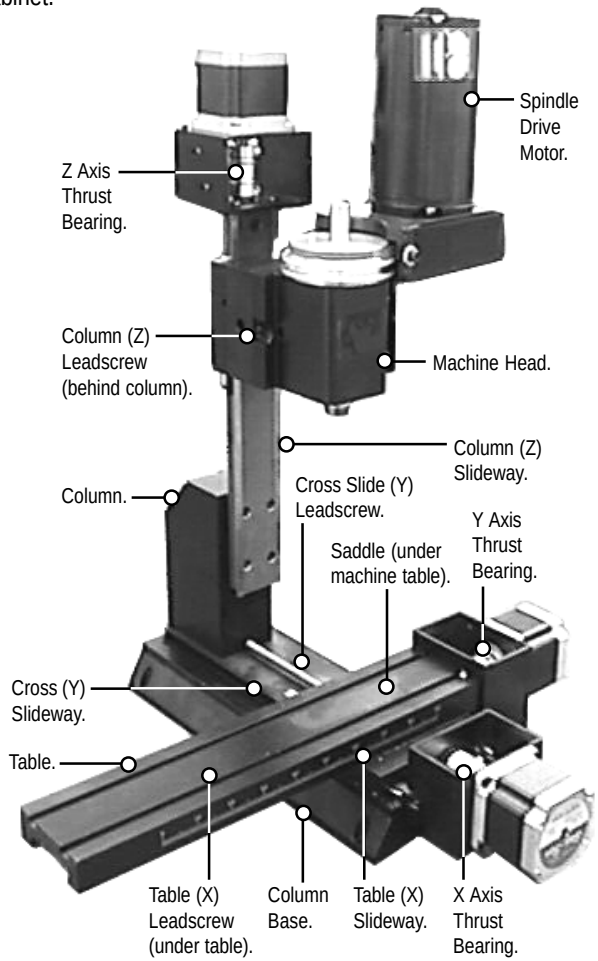
The following items on the machine should be oiled daily, using a light machine oil:

A) Slideways.

B) Leadscrews.

C) Thrust bearings.

The photo below labels each area which requires daily lubrication. Your mill is encased in a cabinet. For illustration purposes, this photo shows a view of the mill column and slideways without the cabinet.



13: Lubrication of Leadscrews

Table (X Axis) Leadscrew Lubrication.

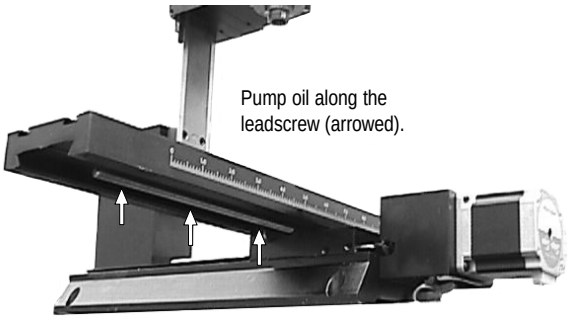
Run the machine table fully to the right, to gain access to the table slide leadscrew. The leadscrew is located under the machine table. Pump oil directly onto the exposed surface of the leadscrew.

Run the saddle fully to the left, to gain access to the opposite end of the leadscrew. Pump oil directly onto the remaining surface of the leadscrew.

Finally, run the table left and right along the X axis, to distribute the oil along the full length of the leadscrew.

Location

Look under table.



13: Lubrication of Leadscrews

Cross Slide (Y Axis) Leadscrew Lubrication.

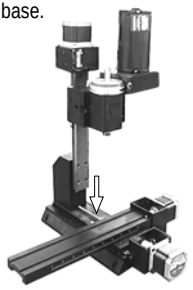
Run the saddle back towards the machine column, to gain access to the front section of the cross slide leadscrew. Pump oil directly onto the exposed surface of the leadscrew.

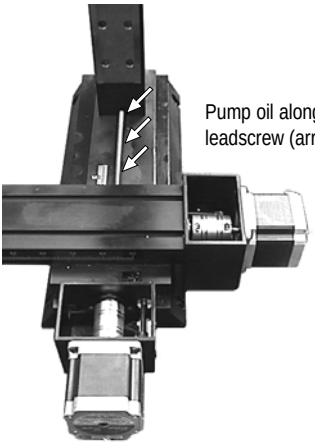
Run the saddle forwards, to gain access to the opposite end of the cross slide leadscrew. Pump oil directly onto the remaining surface of the leadscrew.

Finally, run the saddle forwards and backwards along the Y axis, to distribute the oil along the full length of the leadscrew.

Location

Look along column base.





Pump oil along the leadscrew (arrowed).

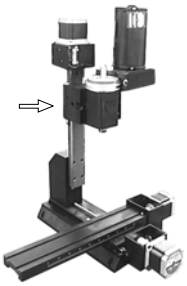
13: Lubrication of Leadscrews

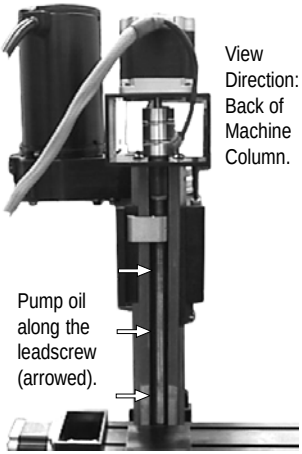
Column (Z Axis) Leadscrew Lubrication.

Using a pump action oilgun, reach around the back of the column, then pump oil directly onto the surface of the leadscrew. Run the head up and down the column to distribute the oil.

Location

Look behind column.





Lubrication of Optional Vice.

Clean, then lubricate directly onto the vice slideways each week, using a pump action oilcan.

13: Lubrication of Slideways

Table (X Axis) Slideways Lubrication.

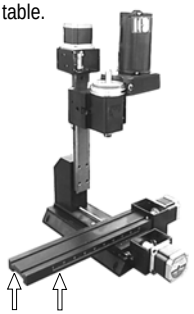
Run the table to the extreme left of its travel, to expose the right side of the table slideways. Use a pump oilcan (with a flexible nozzle) or an aerosol to lubricate the slideways. Reaching under the table, directly lubricate the exposed surfaces of the two table slideways.

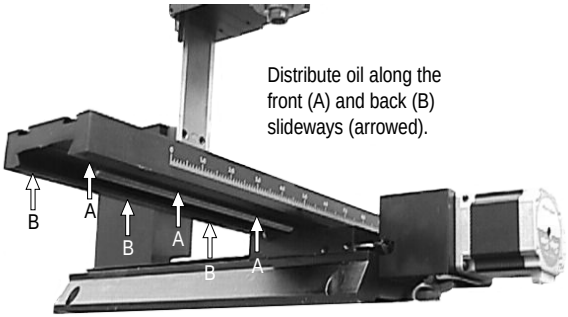
Run the table to the extreme right of its travel, to expose the left side of the table slideways. Reaching under the table, directly lubricate the remaining exposed surfaces of the two table slideways.

Run the table fully along the X axis to distribute the oil along the slideways.

Location

Look under machine table.





Distribute oil along the front (A) and back (B) slideways (arrowed).

13: Lubrication of Slideways

Cross (Y Axis) Slideways Lubrication.

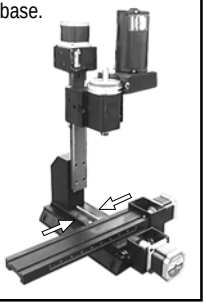
Run the saddle fully forward, to expose the back section of the column base slideways. Using a pump oilcan (with a flexible nozzle) or an aerosol, directly lubricate the exposed surfaces of the two table slideways.

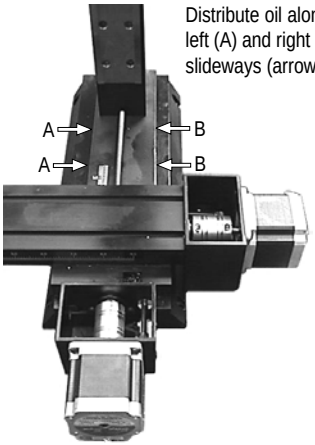
Run the saddle fully back against the machine column, to expose the front of the column base slideways. Directly lubricate the remaining exposed surfaces of the two table slideways.

Run the saddle fully along the Y axis to distribute the oil along the slideways.

Location

Look at sides of column base.





Distribute oil along the left (A) and right (B) slideways (arrowed).

13: Lubrication of Slideways

Column (Z Axis) Slideways Lubrication.

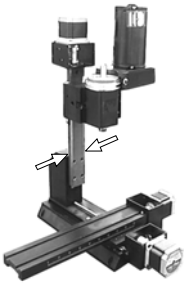
Run the machine head fully up, to expose the bottom of the table slideways. Using a pump oilcan (with a flexible nozzle) or an aerosol, directly lubricate the exposed surfaces of the two table slideways.

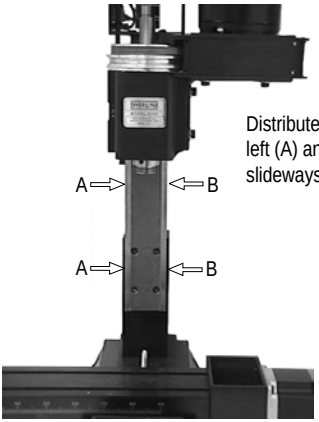
Run the machine head fully down, to expose the top of the table slideways. Directly lubricate the remaining exposed surfaces of the two table slideways.

Run the machine head fully along the Z axis to distribute the oil along the slideways.

Location

Look at sides of column.





Distribute oil along the left (A) and right (B) slideways (arrowed).

13: Lubrication of Thrust Bearings

Location

Look at right end of table.

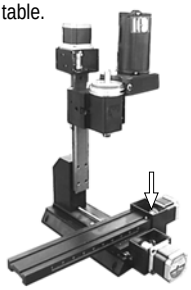
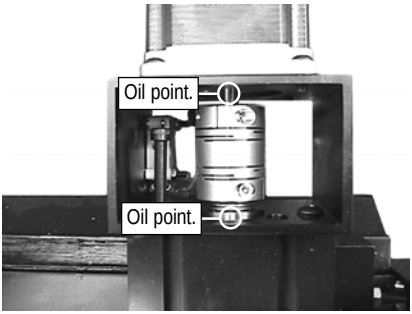


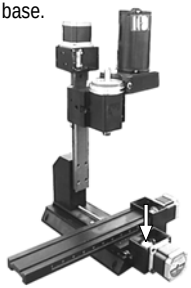
Table (X Axis) Thrust Bearing Lubrication.



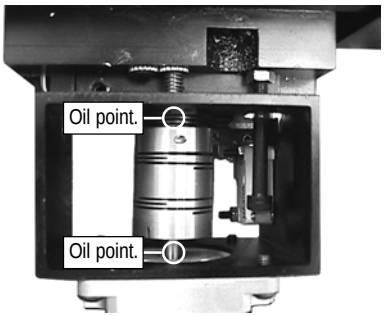
Using a pump action oilcan, pump oil directly onto the two exposed shafts of the thrust bearing (shown left).

Location

Look at front of column base.



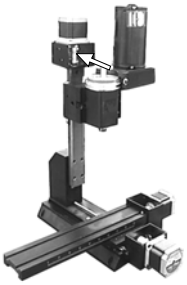
Cross Slide (Y Axis) Thrust Bearing Lubrication.



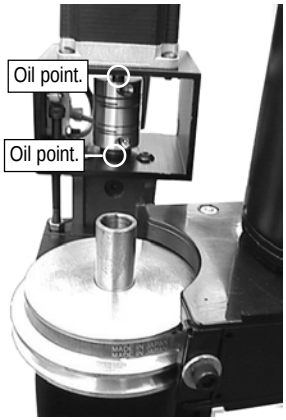
Using a pump action oilcan, pump oil directly onto the two exposed shafts of the thrust bearing (shown left).

Location

Look at top of column.



Column (Z Axis) Thrust Bearing Lubrication.



Using a pump action oilcan, pump oil directly onto the two exposed shafts of the thrust bearing (shown left).

13: Cleaning the Datum Microswitches

Table (X Axis) Datum Microswitch.

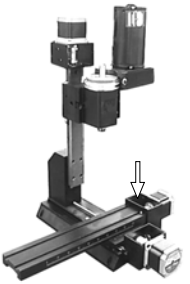
The microswitch defining the X axis datum is positioned inside the rectangular thrust bearing casing, on right end of the machine table, when viewed from the front of the machine.

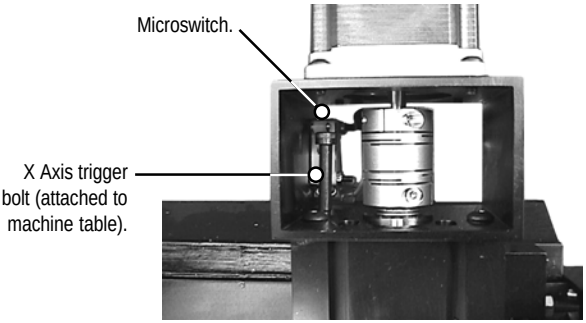
Using a soft bristled brush, carefully clean dust and debris away from the microswitch.

If the microswitch is being cleaned with the X axis at its datum position (as shown in the photo below), take care not to disturb the trigger bolt attached to the end of the machine table casting.

Location

Look at right end of machine table.





13: Cleaning the Datum Microswitches

Cross Slide (Y Axis) Datum Microswitch.

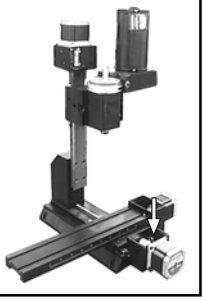
The microswitch defining the Y axis datum is positioned inside the rectangular thrust bearing casing, on the front of the machine column base, when viewed from the front of the machine.

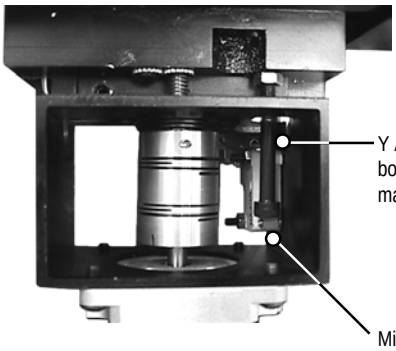
Using a soft bristled brush, carefully clean dust and debris away from the microswitch.

If the microswitch is being cleaned with the Y axis at its datum position (as shown in the photo below), take care not to disturb the trigger bolt attached to the end of the cross slide casting.

Location

Look at end of machine column base.





13: Cleaning the Datum Microswitches

Column (Z Axis) Datum Microswitch.

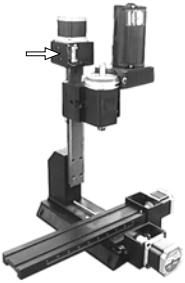
The microswitch defining the Z axis datum is positioned inside the rectangular thrust bearing casing, on the top of the machine column, when viewed from the front of the machine.

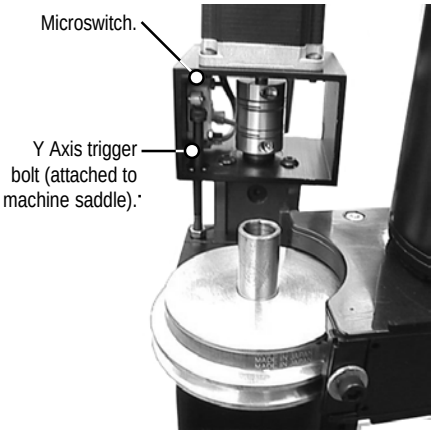
Using a soft bristled brush, carefully clean dust and debris away from the microswitch.

If the microswitch is being cleaned with the Z axis at its datum position (as shown in the photo below), take care not to disturb the trigger bolt attached to the machine head.

Location

Look at top of machine column.

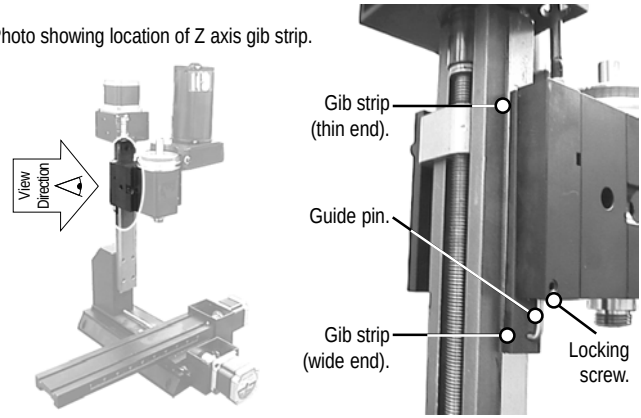




13: Gib Strip Adjustment

A gib strip is an angled section of metal or plastic, tapered along its length, which is fitted to one side of a machine slide. One gib strip is fitted to each of the three machine slides on your Micromill. Essentially, driving one of these wedges further along a slide will gradually decrease the amount of free play in the axis. The photo below shows a gib strip fitted to the Z axis of a Micromill machine column.

Photo showing location of Z axis gib strip.



During the day-to-day use of your Micromill, wear will occur in each of the gib strips. This wear is normal and if checked regularly will not affect the performance of your machine.

Denford recommends that the gib strips be checked annually. Any slack present in the slides should be rectified, by adjusting the gib strips, as described on the following pages.

Checking the Condition of the Gib Strips.

As the gib strips wear down, varying degrees of free play will become noticeable in each of the three machine axes.

You can quickly judge the slack available in each slide, by holding the table, saddle, and machine head at their extreme ends. Try to move, or wobble them from side to side, across the width of the slide. Any excessive movement will indicate that the slide in question requires adjustment.

To accurately assess the condition of each slide, Denford recommends the use of a dial gauge.

Note that a slight degree of movement will always be present in every slide. If there is no free movement whatsoever, then the slides would be too tight to move.

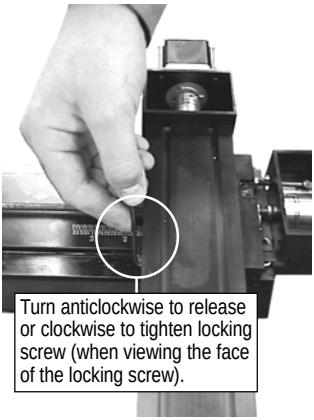
13: Gib Strip Adjustment

X Axis Gib Strip Adjustment Method:

1) Release the gib strip locking screw, positioned on the back of the machine saddle (shown below).

Location

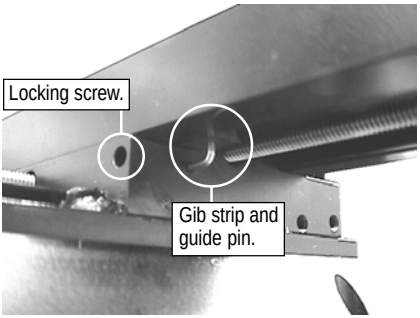
Look at back of machine saddle.



2) Push the wider end of the gib strip further into its channel to eliminate the free play in the slideway. The gib strip is accessed by reached under the machine table (see below). Remember that a slight degree of movement must be left in the slide to allow the slide to move.

Location

Look under machine table.

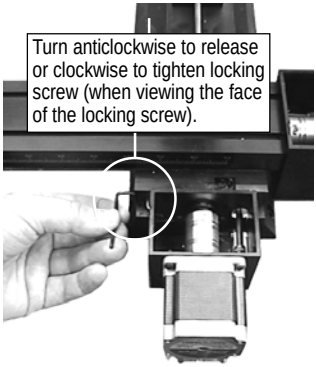


3) Tighten the gib strip locking screw.
4) Move the machine table left and right across the saddle, to ensure that the movement is smooth.
5) If the movement is not smooth, adjust the gib strip by repeating steps 1) to 3). Run the machine table again to assess the smoothness of movement.

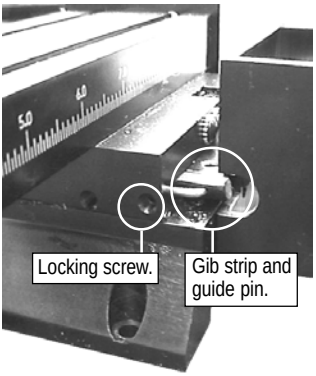
13: Gib Strip Adjustment

Y Axis Gib Strip Adjustment Method:

1) Release the gib strip locking screw, positioned on the left side of the machine saddle (shown below).



2) Push the wider end of the gib strip further into its channel to eliminate the free play in the slideway. The gib strip is accessed from the front of the machine table (shown below). Remember that a slight degree of movement must be left in the slide to allow the slide to move.



3) Tighten the gib strip locking screw.
4) Move the machine table left and right across the saddle, to ensure that the movement is smooth.
5) If the movement is not smooth, adjust the gib strip by repeating steps 1) to 3). Run the machine table again to assess the smoothness of movement.

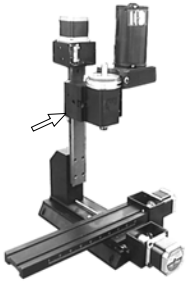
13: Gib Strip Adjustment

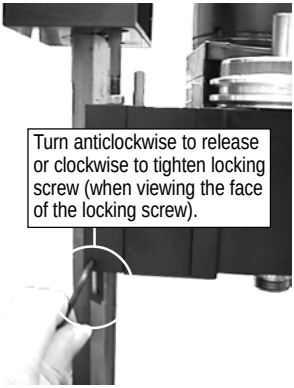
Z Axis Gib Strip Adjustment Method:

1) Release the gib strip locking screw, positioned on the left side of the machine head (shown below).

Location

Look at left side of machine head.

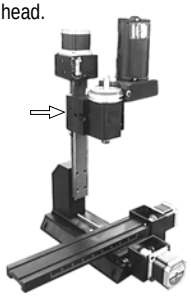


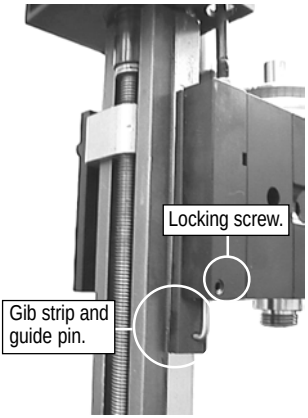


2) Push the wider end of the gib strip further up into its channel to eliminate the free play in the slideway. The gib strip is best accessed by reaching around the back of the machine head (shown below). Remember that a slight degree of movement must be left in the slide to allow the slide to move.

Location

Look at back of machine head.



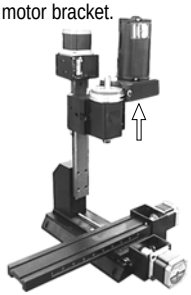


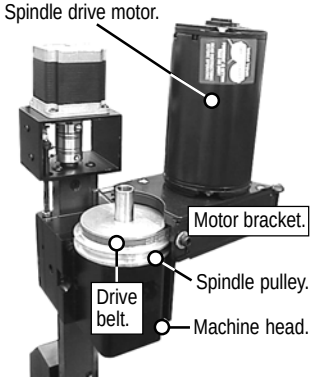
3) Tighten the gib strip locking screw.
4) Move the machine table left and right across the saddle, to ensure that the movement is smooth.
5) If the movement is not smooth, adjust the gib strip by repeating steps 1) to 3). Run the machine table again to assess the smoothness of movement.

13: Spindle Drive Belt Adjustment

Location

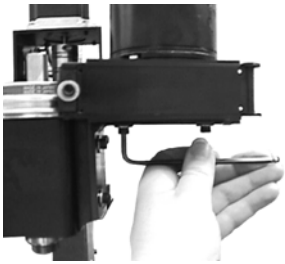
Look under spindle motor bracket.





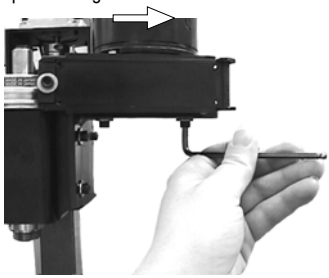
View of two motor adjusting bolts under motor bracket.

The drive belt from the motor to the machine spindle may become slack from time to time, due to constant use. To adjust the tension, release the two bolts holding the motor. Slide the motor further away from the spindle until slack in the drive belt has been eliminated, then tighten the two bolts holding the motor.



1) Release motor bolt 1.

3) Slide motor further away from spindle to tighten drive belt.



2) Release motor bolt 2.

Turn anticlockwise to release or clockwise to tighten the bolts holding the spindle drive motor in position (when directly viewing the face of the bolt from beneath the bracket).

14: Technical Support

Denford Limited provides unlimited telephone Technical Support on this CNC machine to registered users. On-site visits by our engineers may be chargeable. Please refer to the information held in your separate Warranty pack, for specific details.

Before contacting Denford for support, please read your hardware and software manuals and check the FAQ section on our website.

When you request support, please be at your CNC machine, with your hardware and software documentation to hand. To minimise delay, please be prepared to provide the following information:

- CNC Machine Serial Number.
- Registered user's name / company name.
- The controller software name and version number (found in the "Help | About" menu).
- The wording of any error messages that appear on your computer screen, if applicable.
- A list of the steps that were taken to lead up to the problem.
- A list of any maintenance work that has been carried out on the CNC machine.

Contact Details:

Denford Limited,

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

Telephone: 01484 712264

Fax: 01484 722160

ISDN: 01484401157:01484401161

E-mail: service@denford.co.uk

Technical Support: Monday to Friday 8.30am - 4.30pm GMT

For USA please contact:

Denford Inc.

815 West Liberty Street, Medina, Ohio 44256, USA.

Telephone: 330 7253497

Fax: 330 7253297

E-mail: service@denford.com

Technical Support: Monday to Friday 8.30am - 4.30pm Eastern

Internet

http://www.denford.com

COM Port Set-up (Machine Communication) :
Note: Your screen may display the message "Error 50 Mint 3.28 Disconnect" (or similar, depending on the hardware fitted).
If the VR CNC Milling software cannot connect to your Micromill CNC machine, check the following:

1) Power is reaching your Micromill and the electrical control box is switched on.
2) The RS 232 cable is connected to a valid serial port. If so, identify whether it is COM1 or COM2.
3) The VR CNC Milling software is set to read the correct COM port number. Follow the instructions on page 110 to change the COM port numbers in the software.
4) The COM port is functioning correctly and it is active in your computers BIOS (assessable on computers during initial boot-up). The method used for accessing this information will depend on the hardware fitted in your computer - check your computer documentation for details on how to access your BIOS.
5) The LED display status on your Micromill control card. Refer to the instructions on page 112. An L or 8 should be displayed. If the LED is not lit, check the control board fuse.

The part is being cut at an incorrect depth :
Check the validity of the following:

1) The Z value entered in the tool length offset.
2) The Z value entered in the workpiece offset file.
3) The number (size) used for defining the depth of cut used in your CNC program.
4) The sign (+ or -) used for defining the depth of cut used in your CNC program. If your workpiece datum is aligned with the upper surface of your billet, any Z values cutting into this billet will have a minus sign.

The machine begins cutting the part at the wrong location :
Check the X and Y values entered in the workpiece offset file.

When using double sided tape, the billet keeps lifting from the sub-table (sheet of MDF) :
Check the surface of the sub-table is clean and smooth before attempting to add the billet. Routinely clean tape adhesive residue from the sub-table. Check tape adequately covers all parts of the billet. Check the sequence in which the various parts of your design are machined. For example, machine any small or etched surfaces before cutting pieces completely out from the billet. If you keep the largest solid area of the billet attached to the sub-table for as long as possible, you reduce the likelihood of the billet moving during machining.

14: Changing COM Ports

Note

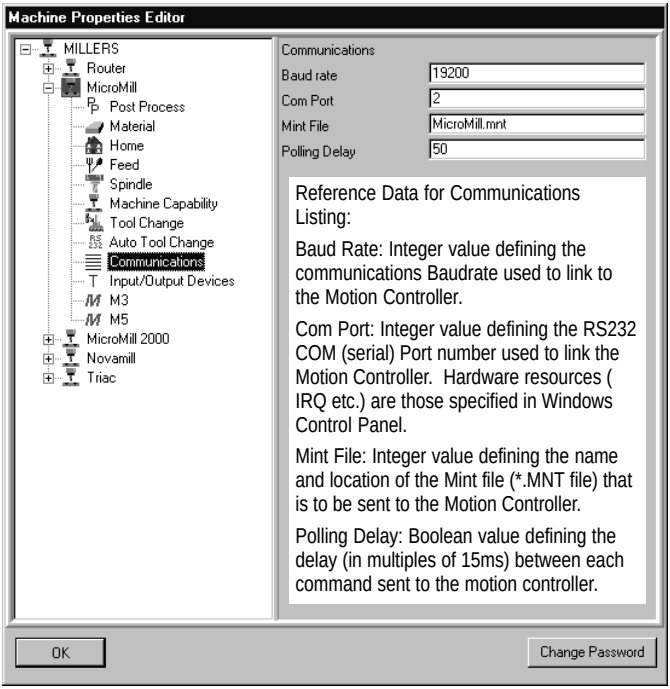
The password used to access the "Machine Properties" window can be changed by the user.

Remember that the default password listed here will not be recognised if you have changed it.

If you change any passwords, we recommend you make a note of them in the Notes section at the back of this manual.

The COM port setting in the VR CNC Milling software must match the COM port being used by your Micromill CNC machine. Follow these steps to change the COM port assignment in the software:

1) Start the VR CNC Milling software.
2) Click "Setup | Setup Machine Parameters".
3) You may be required to enter a password. The default password is **denny**. Type the password and click [OK].
4) The "Machine Properties" window will open. Double click the "Micromill" listing.
5) Click the "Communications" listing.
6) Change the "COM Port" setting to match the number of the COM port being used by your CNC machine RS 232 cable, then click [OK].

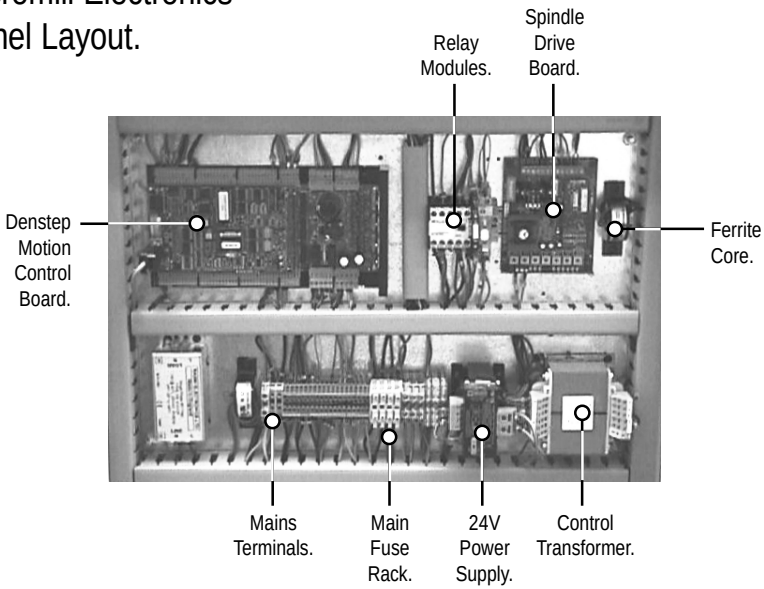


14: Electronics

The photo below labels all important areas of your Micromill CNC machine's electronics, contained in the electrical control box.

Please note that the configuration for your electrical control box may differ from the photo, depending on components and options fitted to your Micromill.

Micromill Electronics Panel Layout.



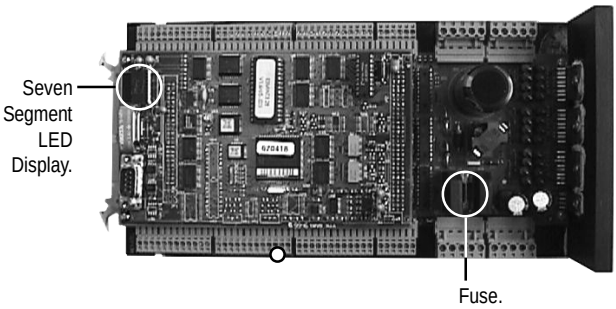
Note

The Electrical Diagrams for your Micromill CNC machine are not included in this manual. They are delivered separately in the standard equipment box supplied with your Micromill CNC machine. Further electrical schematics are available on request.

14: Electronics

The Denstep Motion Control Board.

The denstep motion control board is mounted in the top lefthand corner of the electrical panel. It controls the motors that drive the three CNC machine axes.



LED Status and Fault Display.

The LED display is mounted on the lefthand side of the denstep motion control board.

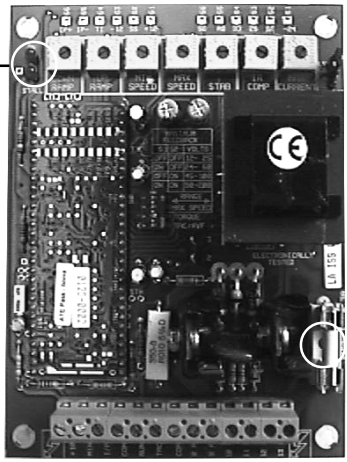
Display.	Meaning.
0	No comms board address fault.
-	Servo power off.
8	Servo power up and idle.
c	Cam profiling.
c	Cam table (superscript).
C	Circular interpolation.
3	Encoder following mode.
F	Flying shear (static symbol).
H	Homing (datuming).
J	Jogging (velocity) mode.
o	Offset mode.
P	Linear positional mode.
q	Torque control mode.
S	Stop asserted.
U	Pulse following mode.

Errors are all shown with a flashing dot.

Display.	Meaning.
E	External error.
E	Software abort or interpreter error.
F	Maximum servo following error exceeded.
L	Limit switch open.

The Spindle Drive Board.

The spindle drive board, mounted in the top righthand corner of the electrical panel, controls the motor for the programmable spindle.



Spindle Drive LED Status.

The two spindle drive status LEDs are mounted in the top lefthand corner of the spindle drive board.

Display.	**Meaning.**
ON LED	Spindle drive board operational. If this is not lit, check the spindle drive board fuse (shown in the diagram above) then the fuse labelled F1 on the main fuse rack (see the diagram on page 111).
STALL LED	Motor stall. This indicates a faulty motor or component on the spindle drive board.

15: Specification of Micromill CNC machine

Standard Equipment:
Micromill CNC machine.
VR CNC Milling operating software.
Tailored courseware and project books.
CNC machine and controller software manuals.
Set of maintenance tools and spare parts list.
Machine commissioning and basic instruction.
Training (UK Only).

Extra Equipment:
Various tooling packages.
Additional CAD/CAM software.
Additional (offline) VR CNC Milling software licenses.
Machine work bench.
Video conferencing system.
Training.
PC & PC workstation.
Workholding vice.
Miteebite clamps.

Safety Features:
Manual operation, totally enclosed, interlocked, safety guard door.
Emergency stop button.
2D and 3D simulation graphics to verify part programs prior to machining.
Automatic tool retraction & spindle stop for tool changing.
Key operated power switch.
Program protect keyswitch, preventing student tampering.

15: Specification of Micromill CNC machine

Mechanical Details:
Table size 70mm x 330mm (2 3/4" x 13").
2 tee slots 6mm width, 38mm centres.
Travel X axis, Standard and Micromill 2000: 229mm (9").
Travel Y axis, Standard: 76mm (3"), Micromill 2000: 127mm (5 1/8").
Travel Z axis, Standard and Micromill 2000: 165mm (6 1/2").
Leadscrews 6.35mm diameter x 1mm pitch.
Spindle to table 200mm (8").
Spindle to column 55mm (2 1/4").
Throat 87mm (3 1/2").
Spindle bore 10mm (1/2").
Spindle nose thread 3/4" 16 TPI.
Spindle taper #1 Morse.
Spindle speed 0 - 3000 rpm, programmable.
Machine resolution 0.01mm (0.0004").

Dimensions:
Machine length 686mm (27").
Machine depth 648mm (26 1/2").
Machine height 686mm (27").

Weights:
Machine weight 70 KG (154 lb).

Electrical Details:
Mains supply required: 1 Phase, 220/240 Volts, 8 Amps, 50/60 Hz.
Spindle motor: 1.5 HP, 1.1Kw.
Axis stepper motors: 200 steps/rev.
AC servo max. torque: 11Nm.
LoVo light: 110 Volts AC.

15: 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 International Standards Organisation code, which includes the G and M code groups.

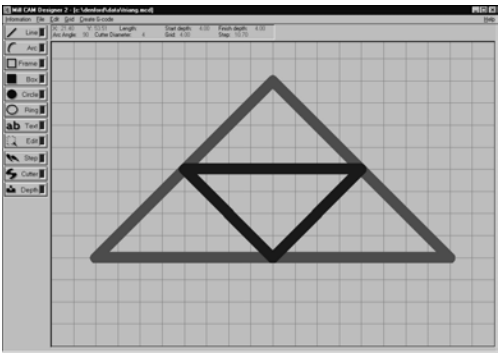
Each program word is composed from a letter, called the address, along with a number.

These terms are illustrated on the following pages.

15: Composition of a Part Program

A component is designed using a CAD/ CAM software package, such as Mill CAM Designer.

The CAD/CAM software package automatically generates the part program, including all the G and M codes required to manufacture the component.



Part Program example -

(Mill CAM Designer - triang.MCD)
(3/3/1997)
(Novamill (metric))
(Post fanucm:1.2Ø 24 June 1994)
G21
[BILLET X8Ø Y55 Z1Ø
[EDGEMOVE XØ YØ
[TOOLDEF T1 D2
NØØ1Ø G91G28XØYØZØ;
NØØ2Ø M6T1;
NØØ3Ø G43H1;
NØØ4Ø M3S15ØØ;
NØØ5Ø G9ØGØX4ØY48;
NØØ6Ø Z2;
NØØ7Ø G1Z-Ø.5F1ØØ;
NØØ8Ø X72Y16F15Ø;
NØØ9Ø X8;
NØ1ØØ X4ØY48;
NØ11Ø GØZ2;
NØ12Ø X24Y32;
NØ13Ø G1Z-1F1ØØ;
NØ14Ø X56F15Ø;
NØ15Ø X4ØY16;
NØ16Ø X24Y32;
NØ17Ø GØZ2;
NØ18Ø M5;
NØ19Ø G91G28XØYØZØ;
NØ2ØØ M3Ø;

Denford Directive Example - [BILLET

Address example - G

Word example - G1

Block example - NØ13Ø G1Z-1F1ØØ;

15: G Codes List

Note - Not all G codes may apply to your CNC 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.

Note

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.

15: M Codes List

Note - Not all M codes may apply to your CNC 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	Guard Door Open
M39	Guard Door Close
M62	Auxiliary Output 1 On
M63	Auxiliary Output 2 On
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.

Note

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).

15: List of Program 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 - Incremental 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.

15: Denford Directives

Directives are program terms defined by Denford Limited.

They are used to help generate the 2D and 3D 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.

continued...

[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.

EC Declaration of Conformity

The responsible person ..... Mr N.J.H.Crowther.

Business Name ..... Denford Limited.

Address ..... Birds Royd,
Brighouse,
West Yorkshire,
HD6 1NB,
UK.

Declares that the machinery described:

Manufacturer ..... Denford Limited.

Model Name ..... Micromill Series CNC Milling Machine.

Serial Number ..... (please refer to warranty card and/or machine casing).

conforms to the following directives: .... EC Machinery directive 89/392/EEC as amended by directive 91/368 EEC and directive 93/44/EEC, CE marking directive 93/68/EEC and low voltage directive 73/23/EEC.

and the following standards ..... BS EN 6Ø2Ø4 - 1 : 1993
and complies with the relevant health and safety requirements.

Signature .....

Position within company ..... Director / General Manager.

Signed at ..... Denford Limited,
Birds Royd,
Brighouse,
West Yorkshire,
HD6 1NB,
UK.

16: Glossary

ABSOLUTE ..... In absolute programming, zero is the point from which all other dimensions are described.

ALLEN HEAD ..... A hexagon shaped hole on the head (top) of a set screw. These are tightened and loosened using allen keys/wrenches.

ARC ..... A portion of a circle.

ATC ..... Automatic Tool Changer.

AUTOMATIC CYCLE ..... A mode of control operation that continuously runs a cycle or stored program until a program stop or end of program word is read by the controller.

AUXILIARY FUNCTION ..... The function of the CNC machine (ie, F, S, T, M codes etc.), other than co-ordinate based commands.

AXIS (AXES) ..... The planes of movement for the cutting tool, usually referred to as X (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. Axes are also referred to as slides or slideways.

BILLET ..... The actual material being machined, sometimes referred to as the "workpiece" or "stock".

BLOCK ..... A collection of program words that collectively describe a machining operation. A single program line in the CNC file.

CHARACTER ..... A number, letter or symbol as entered into a CNC program.

CIRCULAR INTERPOLATION G-code term for a programmed arc movement.

COMMAND ..... A signal (or group of signals) instructing one step / operation to be carried out.

CONTEXT SENSITIVE ..... When the type of input signal of an event automatically changes the output signal.

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.

CNC ..... Computer Numerical Control.

CNC FILE ..... The sequence of commands describing the manufacture of a part on a CNC machine, written using G and M codes, also called the CNC program.

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.

16: Glossary

DIRECTORY ..... An area of a disk containing the names and locations of the files it currently holds.

DISK ..... A computer information storage device, examples, C: (drive) is usually the computers hard (internal) disk and A: (drive) is usually the floppy (portable 3.5" diskette) disk.

DRIVE ..... The controller unit for a disk system.

DRY RUN ..... An operation used to test how a CNC program will function without driving the machine itself.

DWELL ..... A programmed time delay.

EDIT ..... The mode used for altering the content of a CNC program via the Desktop Tutor or qwerty keyboard.

END OF BLOCK SIGNAL ... The symbol or indicator (;)that defines the end of a block of data. The equivalent of the pc **[return]** key.

ERROR ..... The deviation of an attained value from a desired value.

G-CODE ..... The programming language understood by the machine controller.

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.

FORMAT ..... The pattern or way that data is organised.

FNC ..... FANUC Miller file, extension ".fnc". Contains G and M codes describing the machine and cutting operations.

G CODE ..... A preparatory code function in a CNC program that determines the control mode.

HARDWARE ..... Equipment such as the machine tool, the controller, or the computer.

HOME ..... Operation to send the axes of the CNC machine to their extreme limits of movement. Defines the co-ordinate based grid system of the CNC machine. Commonly referred to as homing the machine, or sending the machine to its home position.

INCREMENTAL ..... Incremental programming uses co-ordinate movements that are related from the previous programmed position. Signs are used to indicate the direction of movement.

INPUT ..... The transfer of external information (data) into a control system.

INTERFACE ..... The medium through which the control/computer directs the machine tool.

JOG CONTROL ..... Manual movement mode for the machine axes, using very small pre-defined movements, called jog steps. One stepped movement is applied per movement key/ button press.

M CODE ..... A miscellaneous code function in a CNC program used to indicate an auxiliary function (ie, coolant on, tool change etc.).

MACHINE CODE ..... The code obeyed by a computer or microprocessor system with no need for further translation.

16: Glossary

MACHINE DATUM ..... A fixed zero reference point set by the machine manufacturer. The machine datum is used to define the co-ordinate based grid system of the CNC machine. All machining co-ordinates originate from this point. However, this point can be temporarily moved using the machine offset facility.

MACHINE OFFSET ..... The workpiece offset file used with VR and real CNC machines.

MACHINE TABLE ..... See table.

MDI ..... Manual Data Input - A method used for manually inserting data into the control system (ie, Desktop Tutor, qwerty keyboard etc.).

MITEEBITE CLAMP ..... Method of securing work to the machine table, using the series of machine table T channels.

MODAL ..... Modal codes entered into the controller by a CNC program are retained until changed by a code from the same modal group or cancelled.

NC ..... Numerical control.

OFFSET ..... Combination of two types of file, the workpiece offset and the tool offset. Used to describe the workpiece datum, a zero reference used on the CNC machine to ensure machining occurs in the correct place on the billet. Offsets are used to shift parts of the three dimensional co-ordinate based grid system, used by the CNC machine.

PART DATUM ..... Used as a zero reference point in a CNC file. All machining co-ordinates originate from this point.

PC ..... Personal computer.

PRJ ..... Denford CNC Project file, extension ".prj". Project files contain global information about user defined settings in the VR CNC Milling software, such as toolpost set-up, tooling library, offsets, toolbar and window positions etc.

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 used in setting the limits of movement for its slides (axes).

RPM ..... Revolutions per minute (rev/min) - a measure of spindle speed.

SIMULATION OFFSET ..... The workpiece offset file only used with VR CNC Milling software 2D and 3D graphics.

SLIDES ..... The 3 machine axes - see axis.

SPINDLE SPEED ..... The rate of rotation (velocity) of the machine head / cutting tool, measured in RPM.

SUB-TABLE ..... A secondary table that is 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. For example, a sheet of MDF. Sometimes referred to as a temporary table or platen.

16: Glossary

SOFTWARE ..... Programs, tool lists, sequence of instructions etc...

TABLE ..... The horizontal platform upon which work is secured, sometimes referred to as the machine table.

T CHANNEL ..... There are three slots, or channels (upside down 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.

TEE-NUT ..... An upside down 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, so they can all be used by one CNC file. See OFFSET.

TOOLPOST ..... The holder for the various cutting tools.

TRAVERSE ..... Movement of the cutting tool through the 3 machine axes between cutting settings.

TXT ..... Standard Windows text only file, extension ".txt".

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 or stock.

WORKPIECE DATUM ..... Used as a zero reference point on the real billet. All machining co-ordinates originate from this point, when offset files are used.

WORKPIECE OFFSET ..... A file containing X, Y and Z values that can shift the entire three dimensional co-ordinate based grid system, used by the CNC machine. See OFFSET.

WORD ..... A combination of a letter address and digits, used in a CNC program (ie, G42, M04 etc.).

VIRTUAL REALITY ..... A fully interactive, three dimensional, computer based simulation of a real world object or event.

XNC ..... Denford Compiled CNC file, extension ".xnc". A compiled file is a FANUC Miller file that is formatted to allow 3D elements such as the 3D Viewer to run as quickly as possible. XNC files can also be used to drive an attached CNC machine when run through the VR CNC Milling software.

Z TOOL OFFSET ..... See Tool Offset

17: Index

A

Address characters list 100
Advantages of CNC 10
Auto mode 67

C

Changing COM ports 90
Changing tools (software procedure) 50
Clamping work (miteebites) 56
CNC (definition and overview) 10
CNC file
- 2D simulation 38
- 3D simulation 39
- editing 32
- loading 31
- saving 31

Co-ordinate display modes 42
COM port set-up 90
Connecting
- schematic diagram showing connections 22
- the mains supply 20
- the PC to the Micromill 19

Contact information 2
Contents 3
Conventions (used in manual) 8

D

Datum microswitches (cleaning) 81
Datum plate
- removal and fitting 52
- setting methods 53

Denford directives list 101
Denstep motion control board 92
Disclaimer 7
Drawbar tooling system 46

E

EC declaration of conformity 103
Editor window 32
Electronics
- denstep motion control board 92
- inside the control box 91
- spindle drive board 93

Emergency stop button 14

F

FAQs 89
Feedrate override 69

G

G codes list 98
Gib strip
- adjustment on X axis 84
- adjustment on Y axis 85
- adjustment on Z axis 86
- general 83

Glossary 105

H

Help (technical support) 88
Home mode 41
Homing the machine 41

I

Installation
- before you start 12
- VR CNC milling software 24

Installing a tool (drawbar system) 48
Interlock guard switch 15
Introduction
- Micromill 9
- VR CNC milling software 11
- what is CNC? 10

17: Index

J

Jog
- continuous 44
- step 44

Jog mode 43
Jogging the axes 45

L

Layout of Micromill components 26
Leadscrews (lubrication) 73
Levelling your Micromill 18
Lifting your Micromill 16
Linking to the Micromill 40
Loading a CNC file 31
Lubrication
- areas 72
- column leadscrew 75
- column slideways 78
- cross slide leadscrew 74
- cross slideways 77
- table leadscrew 73
- table slideways 76
- thrust bearings 79

Lubrication chart 71

M

M codes
- list 99
- selecting 44

Machine datum 41
Machine tooling window 36
Maintenance
- datum microswitches 80
- gib strips 83
- leadscrews 73
- lubrication areas 72
- lubrication chart 71
- schedule 71
- slideways 76
- spindle drive belt 87
- thrust bearings 79

Maintenance schedule 71
Manufacturing a part
- checklist 67
- feedrate and spindle speed override 69
- overview 25
- running the CNC file 67
- stopping the machining process 68

Micromill
- front panel layout 27
- introduction 9
- jogging the axes 45
- machine communications 40
- parts identification 26
- specification 94
- starting the software 29
- switching on and off 28

Microswitches
- column 82
- cross slide 81
- table 80

Miteebite clamps 56
Moving the axes 45

O

Offsets
- configure tool length offset 64
- configure workpiece offset 61
- create new workpiece offset file 62
- move to tool offset reference 65
- move to workpiece datum 61
- overview 59

17: Index

P

Part program
- composition example ... 97
- what is a part program? ... 96

PC connections to your Micromill ... 19
PC specifications required ... 23
Potentiometer controls ... 69
Protective coatings (removal) ... 21

Q

Quick change tooling ... 49

R

Removal of protective coatings ... 21
Removing a tool (drawbar system) ... 47
Running a CNC file (auto mode) ... 67

S

Safety
- emergency stop button ... 14
- features ... 13
- interlock guard switch ... 15
- precautions ... 13

Saving a CNC file ... 31
Simulation
- 2D ... 38
- 3D ... 39

Siting your Micromill ... 17
Slideways (lubrication) ... 76
Software
- auto mode ... 67
- changing COM ports ... 90
- co-ordinate display modes ... 42
- general layout ... 30
- home mode ... 41
- introduction ... 11
- jog mode ... 43
- override feedrates and spindle speeds ... 70
- starting ... 29

Specification of Micromill ... 94
Spindle drive belt adjustment ... 87
Spindle drive board ... 93
Spindle speed override ... 69
Standard equipment ... 12
Starting the software ... 29
Sub-tables ... 56
Switching your Micromill on and off ... 28

T

Technical support ... 88
Thrust bearings (lubrication) ... 79
Tool data panel ... 34
Tool length offset ... 64
Tool library window ... 33
Tooling
- adding new tool profiles ... 35
- drawbar system components ... 46
- installing a tool (drawbar system) ... 48
- machine tooling window ... 36
- quick change tooling system ... 49
- removing a tool (drawbar system) ... 47
- software procedure for tool changing ... 50
- tool data panel ... 34
- tool library window ... 33

Troubleshooting ... 89

U

Unpacking your Micromill ... 16
Using your Micromill (overview) ... 25

W

Warning notices ... 6
What is CNC? ... 10
Workpiece datum ... 61
Workpiece offsets ... 61

17: Notes

Use this page to make a note of any parts of the software you have changed or configured, for example, common tooling set-ups, machine parameters, changes to installation paths or passwords etc.