DENFORD

Total Commitment to Education and Training WorldWide.

Virtual Reality
CNC Milling
for Windows
User's Manual.

Contents

Contents

About this Manual

Using this manual	This manual provides a basic introduction to the features available in the VR CNC Milling software, using a tutorial based format. The same tutorials are also included in the VR CNC Milling software helpfile.

The manual applies to the following VR CNC Milling software users:

Offline

Offline meaning no physical Denford CNC machine is attached to your computer.

CNC machine operations must be carried out using the 3D CNC machine models available in the "Denford Virtual Reality" window.

Online

Online meaning a physical Denford CNC machine is attached to your computer.

CNC machine operations can be carried out offline using the 3D CNC machine models available in the "Denford Virtual Reality" window.

CNC machine operations can be carried out online using the attached physical Denford CNC machine. When the VR CNC Milling software is being used online, this manual should be used in conjunction with your separate Denford CNC Machine Manual.

More Detailed Information

Detailed information about specific software features, not covered in this manual, is available in the VR CNC Milling software helpfile.

Disclaimer	We take great pride in the accuracy of information given in this manual, but due to nature of software developments, be aware that software 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 you are using.
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.
<u>Underlined text</u>	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 the program.
"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, 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: Introduction

Congratulations on your purchase of Denford Virtual Reality CNC Milling for Windows software. VR CNC Milling is a Windows based software package allowing full editing and control of CNC files, either offline (away from the CNC machine) or online (controlling the operation of a CNC machine).

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:

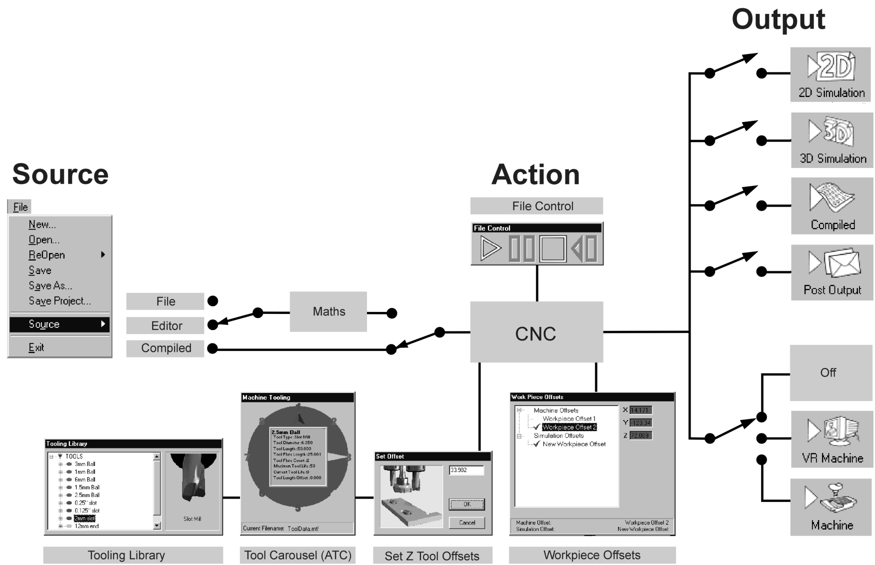
- Full MDI CNC file editing.
- 2 Dimensional graphical simulation of CNC files.
- 3 Dimensional graphical simulation of CNC files.
- Comprehensive Tooling features.
- Full offline control of a CNC machine using Virtual Reality.
- Full online control of a CNC machine.
- Context sensitive online help, 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.
CNC refers to Computer Numerical Control, the automatic system used to control a machine tool.
A G and M code is a series of letters and numbers that make up the language used by CNC machinery.
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: VR CNC Milling Software in Use

The diagram below shows how all the different elements of the VR CNC Milling software work together, from CNC file source through to a software or hardware outcome...



2: Minimum System Requirements

The following hardware is required to run VR CNC Milling software.

System Requirements:

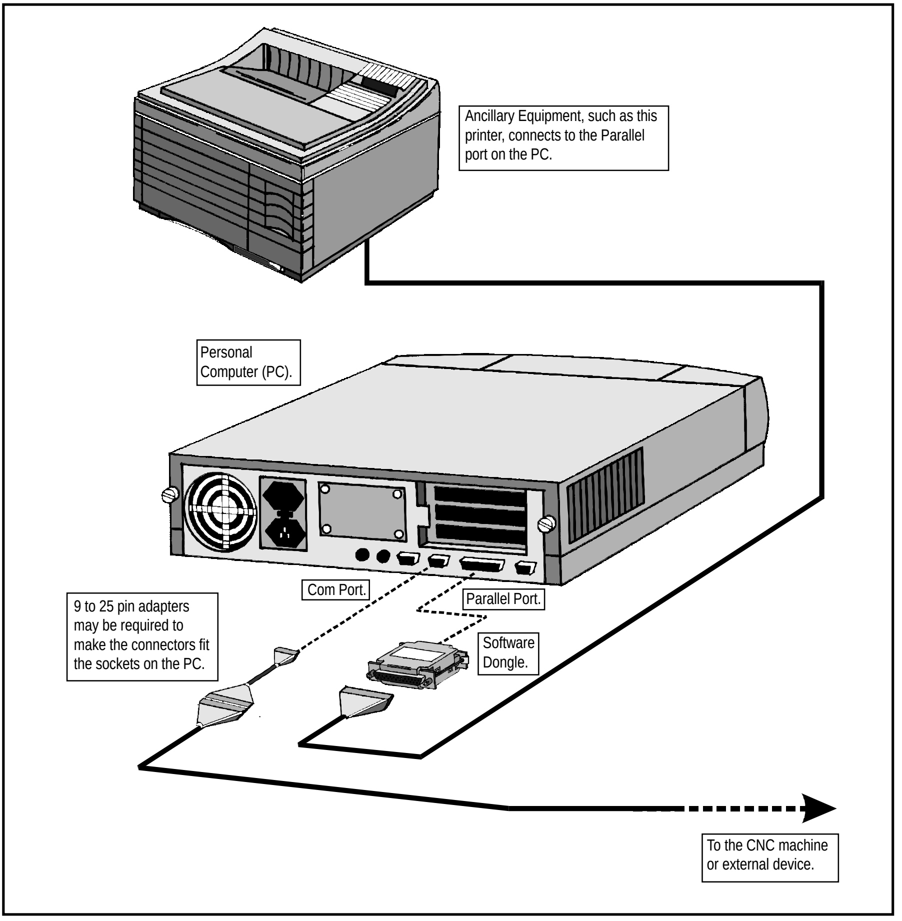
Minimum Specification:

- IBM PC or 100 % compatible computers.
- 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 port.
- 1 free parallel port.

Recommended Specification:

- IBM PC or 100 % compatible computers.
- 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 ports.
- 1 free parallel port.

2: Hardware Connections



Note

Hardware connections are shown for offline use of the VR CNC Milling software. Users wishing to connect a physical Denford CNC machine to their computers should refer to their separate Denford CNC Machine Manual.

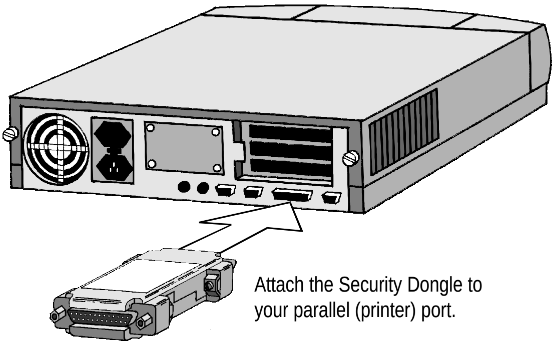
2: Fitting the Security Dongle

The VR CNC Milling software will not run without the Denford Security Dongle.

Fit the security dongle onto the parallel port of your computer.

The parallel port is usually positioned on the back panel of your computer - a long, thin 25 pin male connector plug. Note that your parallel port may be labelled as the printer port.

The security dongle has a pass-through feature, allowing data to be sent to other external devices, such as printers, scanners or zip drives. This feature is only operational when the VR CNC Milling software is not being used. Simply plug the new parallel port cable, supplied with your additional external device, directly into the male plug at the back of the security dongle, which remains attached to your computer.



Attach the Security Dongle to your parallel (printer) port.

2: Installation Procedure

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

1) Switch on your computer 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 title of the CNC machine used for default configuration by the VR CNC Milling software, by clicking in one of the checkboxes. The selected CNC machine is shown using a tick mark.
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. Ensure that the security dongle is fitted to the parallel port of your computer (see previous page).

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.

2: Technical Support

Denford Limited provides unlimited Technical Support on this software.

Technical Support is only available to registered users. If your software was not registered with Denford Limited at the point of sale, e-mail or fax your registration details to Denford Limited, or your authorised Denford shipping agent as soon as possible.

When you request Technical Support, please be at your computer, with your computer and software documentation to hand. To minimise delay, please be prepared to provide the following information:

- Dongle Serial Number.
- Registered user's name / company name.
- Software name and version number (found in the "Help | About" menu).
- The wording of any error messages that appear on your computer screen.
- A list of the steps that were taken to lead up to the problem.

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

3: Using the Tutorials - Overview

> **Note**
>
> The numbered sections in this manual are arranged to form a complete tutorial, introducing you to the various features available in the VR CNC Milling software. Use the sample CNC files listed on pages 14 to 21. Sections can be studied individually, or followed chronologically, as outlined in the flowchart.
>
> If you are new to using the VR CNC Milling software, we recommend that you follow the entire course from start to finish.
>
> The full course should take around 1 hour to complete.

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.

A: Start the VR CNC Milling software
(see section 4: pages 22 to 23).

↓

B: Load or create the CNC program
(see section 5: pages 29 to 38).

↓

C: Configure the tooling in the VR CNC Milling software
(see section 6: pages 39 to 44 and 47 to 50).

↓

D: Simulate the CNC program in 2D or 3D
(see section 7: pages 52 to 55).

↓

E: Start and home the CNC machine
(see section 8: pages 66 to 67).

↓

F: Prepare any tooling hardware for the CNC machine
(see your separate CNC machine manual).

↓

G: Load the billet onto the machine table
(see your separate CNC machine manual).

↓

H: Configure the workpiece offset file and tool length offsets
(see section 9: pages 76 to 88).

↓

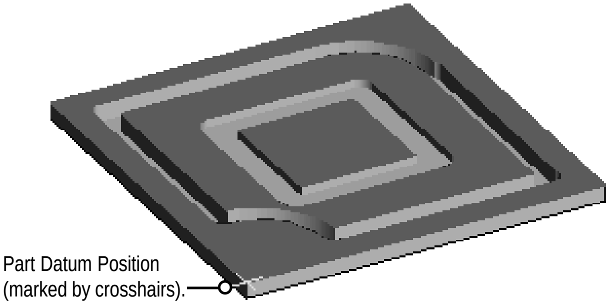
I: Manufacture the part
(see section 12: pages 108 to 109).

> **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.

3: Sample CNC File - Metric

The CNC file shown on pages 15 to 17 can be used throughout the series of tutorials:

Below: 3D simulation of the metric (millimetres) sample CNC file.



Billet: High Density Polystyrene.

Part Datum Position: Top lefthand front corner of proposed billet.

Dimensions: X (length) 60mm, Y (width) 60mm, Z (height) 2mm

Tools required:

- 2mm slot cutter (cutting 2mm deep)
- 4mm slot cutter (cutting 1mm deep)

3: Sample CNC File - Metric

Metric (millimetres) sample CNC file with no formatting:

Check that the units of measurement are set to "Metric". The units of measurement are configured using the [Units] button on the "Options" toolbar.

Copy and paste the CNC file listed below into a new (blank) "Editor" window. Save the CNC file as "Metric.fnc".

```
G21
[BILLET X60 Y60 Z10
[EDGEMOVE X0 Y0
[TOOLDEF T1 D4
[TOOLDEF T2 D2
G91G28X0Y0Z0
M6T1
G43H1
M3S1500
G90G0X20Y40
Z2
G1Z-1F100
Y20F150
X40
Y40
X20
G0Z2
M5
G91G28X0Y0Z0
M6T2
G43H2
M3S1500
G90G0X5Y55
Z2
G1Z-2F100
X40F150
G2X55Y40J-15
G1Y5
X20
G3X5Y20I-15
G1Y55
G0Z2
M5
G91G28X0Y0Z0
M30
```

3: Sample CNC File - Metric

Explanation of metric (millimetres) sample CNC file:

N 001 G21 ; (G20 defines the units of measurement being used as metric - millimetres)

N 011 [BILLET X60 Y60 Z2 (BILLET defines the size of the material being machined, called the billet, with X length 60mm, Y width 60mm, Z height 2mm)

N 021 [EDGEMOVE X0 Y0 (EDGEMOVE defines work datum shift for the program, X0, Y0 means no shift is applied)

N 031 [TOOLDEF T1 D4 (TOOLDEF defines tool number 1 with a cutting diameter of 4mm)

N 041 [TOOLDEF T2 D2 (TOOLDEF defines tool number 2 with a cutting diameter of 2mm)

N 051 G91 G28 X0 Y0 Z0 ; (G91 instructs the machine to follow incremental movements until told otherwise (incremental means all movements are described relative to the co-ordinate position achieved in the last program line). G28X0Y0Z0 moves the cutter to the machine datum via the intermediate point indicated)

N 061 M6 T1 ; (M6 instructs the machine to perform a tool change, if required. Change to tool number 1)

N 071 G43 H1 ; (G43 instructs the machine to use tool length compensation for tool number 1)

N 081 M3 S1500 ; (M3 instructs the machine to switch the spindle on clockwise, with a speed of 1500 rpm)

N 091 G90 G0 X20 Y40 ; (G90 instructs the machine to follow absolute movements until told otherwise (absolute means all movements are described relative to the work datum point). G0 instructs the machine to fast traverse to position X20, Y40)

N 101 Z2 ; (instructs the machine to move the cutter until it is 2mm above the surface of the billet. The last G code given was G0 (on the previous line) so the machine will fast traverse to this position)

N 111 G1 Z-1 F100 ; (G1 instructs the machine to cut a straight line from point to point. Z-1 instructs the machine to cut 1mm into the material, since Z0 is the surface of the material. F100 instructs the machine to use a feedrate of 100mm per minute)

N 121 Y20 F150 ; (the last G code issued was G1, on line N111. The cutter will move to position Y20, cutting a slot, with a feedrate of 150mm per minute)

N 131 X40 ; (the last G code issued was G1, on line N111. The cutter will move to position X40, cutting a slot, continuing with a feedrate of 150mm per minute)

N 141 Y40 ; (the last G code issued was G1, on line N111. The cutter will move to position Y40, cutting a slot, continuing with a feedrate of 150mm per minute)

N 151 X20 ; (the last G code issued was G1, on line N111. The cutter will move to position X20, cutting a slot, continuing with a feedrate of 150mm per minute. This finishes the square shape etched into the centre of the billet)

N 161 G0 Z2 ; (G0 instructs the machine to fast traverse to position Z2, moving the cutter 2mm above the surface of the billet)

N 171 M5 ; (M5 instructs the machine to switch off the spindle)

N 181 G91 G28 X0 Y0 Z0 ; (G91 instructs the machine to follow incremental movements until told otherwise (incremental means all movements are described relative to the co-ordinate position achieved in the last program line). G28X0Y0Z0 moves the cutter to the machine datum via the intermediate point indicated)

N 191 M6 T2 ; (M6 instructs the machine to perform a tool change. Change to tool number 2)

N 201 G43 H2 ; (G43 instructs the machine to use tool length compensation for tool number 2)

N 211 M3 S1500 ; (M3 instructs the machine to switch the spindle on clockwise, with a speed of 1500 rpm)

N 221 G90 G0 X5 Y55 ; (G90 instructs the machine to follow absolute movements until told otherwise (absolute means all movements are described relative to the work datum point). G0 instructs the machine to fast traverse to position X5, Y55)

N 231 Z2 ; (instructs the machine to move the cutter until it is 2mm above the surface of the billet. The last G code given was G0 (on the previous line) so the machine will fast traverse to this position)

N 241 G1 Z-2 F100 ; (G1 instructs the machine to cut a straight line from point to point. Z-2 instructs the machine to cut 2mm into the material (cutting completely through the billet) since Z0 is the surface of the material. F100 instructs the machine to use a feedrate of 100mm per minute)

N 251 X40 F150 ; (the last G code issued was G1, on line N241. The cutter will move to position X40, cutting a slot, with a feedrate of 150mm per minute)

N 261 G2 X55 Y40 J-15 ; (G2 instructs the machine to cut a clockwise arc. The arc is cut from the position reached in the previous program line to the position X55, Y40. The centre point of the arc is defined by J-15, where J indicates the X axis. The centre point is -15mm along the X axis from the start position of the arc. The arc is cut with a feedrate of 150mm per minute)

N 271 G1 Y5 ; (G1 instructs the machine to cut a straight line from point to point. The cutter will move to position Y5, cutting a slot, continuing with a feedrate of 150mm per minute)

N 281 X20 ; (the last G code issued was G1, on line N271. The cutter will move to position X20, cutting a slot, continuing with a feedrate of 150mm per minute)

N 291 G3 X5 Y20 I-15 ; (G3 instructs the machine to cut an anti-clockwise arc. The arc is cut from the position reached in the previous program line to the position X5, Y20. The centre point of the arc is defined by I-15, where I indicates the Y axis. The centre point is -15mm along the Y axis from the start position of the arc. The arc is cut with a feedrate of 150mm per minute)

N 301 G1 Y55 ; (G1 instructs the machine to cut a straight line from point to point. The cutter will move to position Y55, cutting a slot, continuing with a feedrate of 150mm per minute)

N 311 G0 Z2 ; (G0 instructs the machine to fast traverse to position Z2, moving the cutter 2mm above the surface of the billet)

N 321 M5 ; (M5 instructs the machine to switch off the spindle)

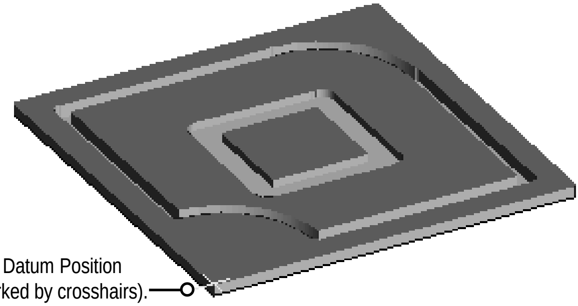
N 331 G91 G28 X0 Y0 Z0 ; (G91 instructs the machine to follow incremental movements until told otherwise (incremental means all movements are described relative to the co-ordinate position achieved in the last program line). G28X0Y0Z0 moves the cutter to the machine datum via the intermediate point indicated)

N 341 M30 ; (M30 defines the end of the program and rewinds back to the start of the CNC file)

3: Sample CNC File - Inch

The CNC file shown on pages 22 to 25 can be used throughout this series of tutorials:

Below: 3D simulation of the imperial (inch) sample CNC file.



Part Datum Position (marked by crosshairs).

Billet: High Density Polystyrene.

Part Datum Position: Top lefthand front corner of proposed billet.

Dimensions: X (length) 2.5", Y (width) 2.5", Z (height) 1/8" (0.125)

Tools required:

- 1/8" (0.125) slot cutter (cutting 1/8" (0.125) deep)
- 3/16" (0.1875) slot cutter (cutting 1/16" (0.0625) deep)

3: Sample CNC File - Inch

Imperial (inches) sample CNC file with no formatting:

Check that the units of measurement are set to "Inch". The units of measurement are configured using the [Units] button on the "Options" toolbar.

Copy and paste the CNC file listed below into a new (blank) "Editor" window. Save the CNC file as "Inch.fnc".

```
G20
[BILLET X2.5 Y2.5 Z0.5
[EDGEMOVE X0 Y0
[TOOLDEF T1 D0.1875
[TOOLDEF T2 D0.125
G91G28X0Y0Z0
M6T1
G43H1
M3S1500
G90G0X0.75Y1.75
Z0.08
G1Z-0.0625F3.9
Y0.75F5.9
X1.75
Y1.75
X0.75
G0Z0.08
M5
G91G28X0Y0Z0
M6T2
G43H2
M3S1500
G90G0X0.25Y2.25
Z0.08
G1Z-0.125F3.9
X1.75F5.9
G2X2.25Y1.75J-0.5
G1Y0.25
X0.75
G3X0.25Y0.75I-0.5
G1Y2.25
G0Z0.08
M5
G91G28X0Y0Z0
M30
```

3: Sample CNC File - Inch

Explanation of imperial (inches) sample CNC file:

N 001 G20 ; (G20 defines the units of measurement being used as imperial - inch)

N 011 [BILLET X2.5 Y2.5 Z0.125 (BILLET defines the size of the material being machined, called the billet, with X length 2.5", Y width 2.5", Z height 0.125")

N 021 [EDGEMOVE X0 Y0 (EDGEMOVE defines work datum shift for the program, X0, Y0 means no shift is applied)

N 031 [TOOLDEF T1 D0.1875 (TOOLDEF defines tool number 1 with a cutting diameter of 0.1875")

N 041 [TOOLDEF T2 D0.125 (TOOLDEF defines tool number 2 with a cutting diameter of 0.125")

N 051 G91 G28 X0 Y0 Z0 ; (G91 instructs the machine to follow incremental movements until told otherwise (incremental means all movements are described relative to the co-ordinate position achieved in the last program line). G28X0Y0Z0 moves the cutter to the machine datum via the intermediate point indicated)

N 061 M6 T1 ; (M6 instructs the machine to perform a tool change, if required. Change to tool number 1)

N 071 G43 H1 ; (G43 instructs the machine to use tool length compensation for tool number 1)

N 081 M3 S1500 ; (M3 instructs the machine to switch the spindle on clockwise, with a speed of 1500 rpm)

N 091 G90 G0 X0.75 Y1.75 ; (G90 instructs the machine to follow absolute movements until told otherwise (absolute means all movements are described relative to the work datum point). G0 instructs the machine to fast traverse to position X0.75, Y1.75)

N 101 Z0.08 ; (instructs the machine to move the cutter until it is 0.08" above the surface of the billet. The last G code given was G0 (on the previous line) so the machine will fast traverse to this position)

N 111 G1 Z-0.0625 F3.9 ; (G1 instructs the machine to cut a straight line from point to point. Z-0.06 instructs the machine to cut 0.06" into the material, since Z0 is the surface of the material. F3.9 instructs the machine to use a feedrate of 3.9" per minute)

N 121 Y0.75 F5.9 ; (the last G code issued wasG1, on line N111. The cutter will move to position Y0.75, cutting a slot, with a feedrate of 5.9" per minute)

N 131 X1.75 ; (the last G code issued wasG1, on line N111. The cutter will move to position X1.75, cutting a slot, continuing with a feedrate of 5.9" per minute)

N 141 Y1.75 ; (the last G code issued wasG1, on line N111. The cutter will move to position Y1.75, cutting a slot, continuing with a feedrate of 5.9" per minute)

N 151 X0.75 ; (the last G code issued wasG1, on line N111. The cutter will move to position X0.75, cutting a slot, continuing with a feedrate of 5.9" per minute. This finishes the square shape etched into the centre of the billet)

N 161 G0 Z0.08 ; (G0 instructs the machine to fast traverse to position Z0.08, moving the cutter 0.08" above the surface of the billet)

N 171 M5 ; (M5 instructs the machine to switch off the spindle)

N 181 G91 G28 X0 Y0 Z0 ; (G91 instructs the machine to follow incremental movements until told otherwise (incremental means all movements are described relative to the co-ordinate position achieved in the last program line). G28X0Y0Z0 moves the cutter to the machine datum via the intermediate point indicated)

N 191 M6 T2 ; (M6 instructs the machine to perform a tool change. Change to tool number 2)

N 201 G43 H2 ; (G43 instructs the machine to use tool length compensation for tool number 2)

N 211 M3 S1500 ; (M3 instructs the machine to switch the spindle on clockwise, with a speed of 1500 rpm)

N 221 G90 G0 X0.25 Y2.25 ; (G90 instructs the machine to follow absolute movements until told otherwise (absolute means all movements are described relative to the work datum point). G0 instructs the machine to fast traverse to position X0.25, Y2.25)

N 231 Z0.08 ; (instructs the machine to move the cutter until it is 0.08" above the surface of the billet. The last G code given was G0 (on the previous line) so the machine will fast traverse to this position)

N 241 G1 Z-0.125 F3.9 ; (G1 instructs the machine to cut a straight line from point to point. Z-0.125 instructs the machine to cut 0.125" into the material (cutting completely through the billet) since Z0 is the surface of the material. F3.9 instructs the machine to use a feedrate of 3.9" per minute)

N 251 X1.75 F5.9 ; (the last G code issued was G1, on line N241. The cutter will move to position X1.75, cutting a slot, with a feedrate of 5.9" per minute)

N 261 G2 X2.25 Y1.75 J-0.5 ; (G2 instructs the machine to cut a clockwise arc. The arc is cut from the position reached in the previous program line to the position X2.25, Y1.75. The centre point of the arc is defined by J-0.5, where J indicates the X axis. The centre point is -0.5" along the X axis from the start position of the arc. The arc is cut with a feedrate of 5.9" per minute)

N 271 G1 Y0.25 ; (G1 instructs the machine to cut a straight line from point to point. The cutter will move to position Y0.25, cutting a slot, continuing with a feedrate of 5.9" per minute)

N 281 X0.75 ; (the last G code issued was G1, on line N271. The cutter will move to position X0.75, cutting a slot, continuing with a feedrate of 5.9" per minute)

N 291 G3 X0.25 Y0.75 I-0.5 ; (G3 instructs the machine to cut an anti-clockwise arc. The arc is cut from the position reached in the previous program line to the position X0.25, Y0.75. The centre point of the arc is defined by I-0.5, where I indicates the Y axis. The centre point is -0.5" along the Y axis from the start position of the arc. The arc is cut with a feedrate of 5.9" per minute)

N 301 G1 Y2.25 ; (G1 instructs the machine to cut a straight line from point to point. The cutter will move to position Y2.25, cutting a slot, continuing with a feedrate of 5.9" per minute)

N 311 G0 Z0.08 ; (G0 instructs the machine to fast traverse to position Z0.08, moving the cutter 0.08" above the surface of the billet)

N 321 M5 ; (M5 instructs the machine to switch off the spindle)

N 331 G91 G28 X0 Y0 Z0 ; (G91 instructs the machine to follow incremental movements until told otherwise (incremental means all movements are described relative to the co-ordinate position achieved in the last program line). G28X0Y0Z0 moves the cutter to the machine datum via the intermediate point indicated)

N 341 M30 ; (M30 defines the end of the program and rewinds back to the start of the CNC file)

4: Starting the VR CNC Milling Software

Startbar Icon.

Desktop Icon.

Follow these instructions to start the VR CNC Milling software:

1) Power-up the PC.
2) Start the VR CNC Milling software (see note below). You start and exit the VR CNC Milling software as you would any standard Windows application.
3) 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).
4) 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).
5) 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.
6) To exit the VR CNC Milling software, click "File | Exit".

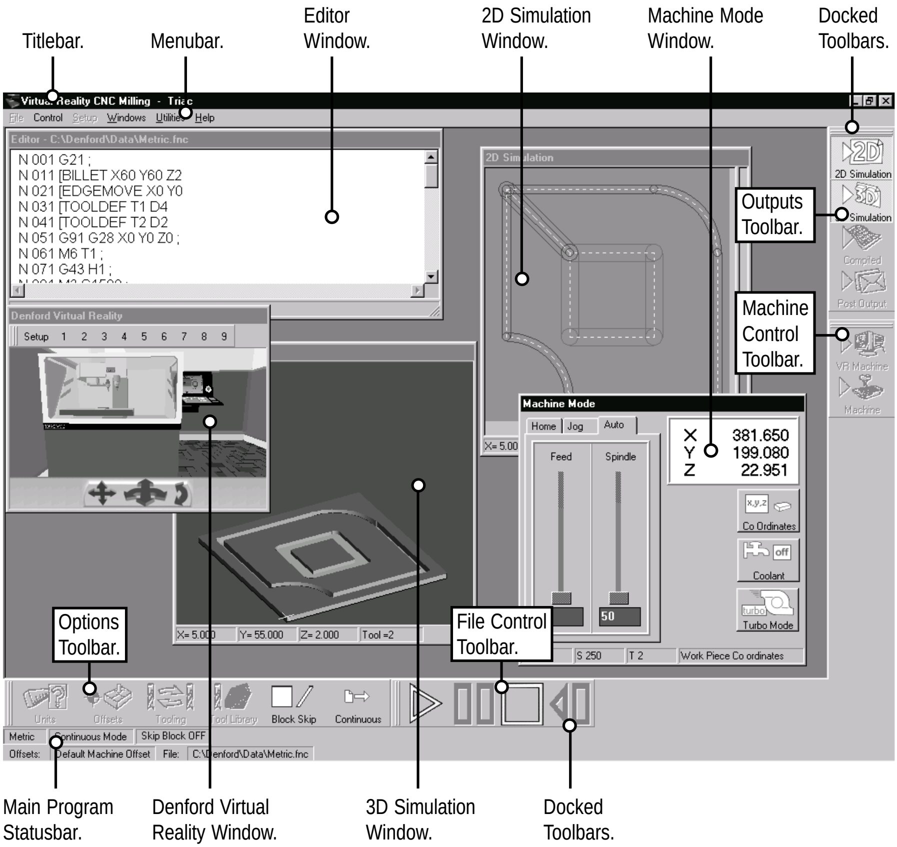
   Important - Never exit the VR CNC Milling software when your Novamill is machining or processing any operational instructions.

Note

When using the VR CNC Milling software to drive a real CNC machine, attached to your computer.
The real CNC machine MUST be switched on BEFORE you start the VR CNC Milling software.

4: General Layout of the Software

The example screenshot, below, shows the general layout of the different elements in the VR CNC Milling software.



Note

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

4: Using the Toolbars

The various toolbars in the software can be repositioned to form different screen layouts, as required.

Note - Only toolbars can be docked and undocked. Any windows appearing through use of the toolbar buttons can only be displayed in the main software window.

If you are unsure about the function of any toolbar button, hover your mouse cursor over the button to display a pop-up hint caption.

Docked Toolbar Example:

A docked toolbar can be positioned anywhere on the main software window docking bars. Docking bars are provided at the grey border edges of the main software window. To move a docked toolbar click and hold your left mouse button on the two grey lines at the end of the toolbar, highlighted by the grey ellipse in the screenshot above. Drag the toolbar to the new position and release the mouse button. To undock a toolbar, drag it off the window docking bar into the main software window, then release the mouse button.

Undocked Toolbar Example:

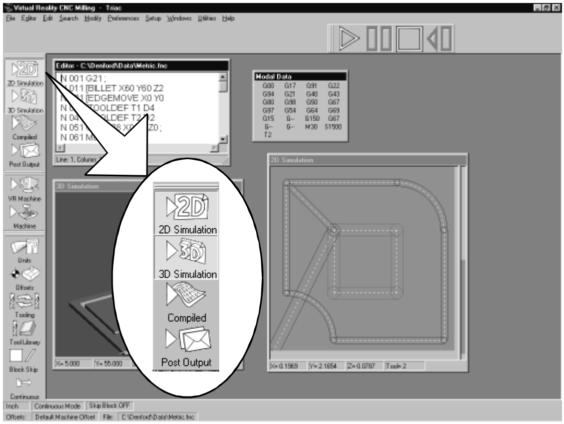
An undocked toolbar can be positioned anywhere in the main software window. To move an undocked toolbar click and hold your left mouse button on the toolbar titlebar, highlighted by the grey ellipse in the screenshot above. Drag the toolbar to the new position and release the mouse button. To dock a toolbar drag and position it over one of the grey border edges of the main software window.

4: Using the Toolbars

Toolbar buttons can be displayed in three different ways, according to the settings specified in the "Setup | Toolbars" menu.

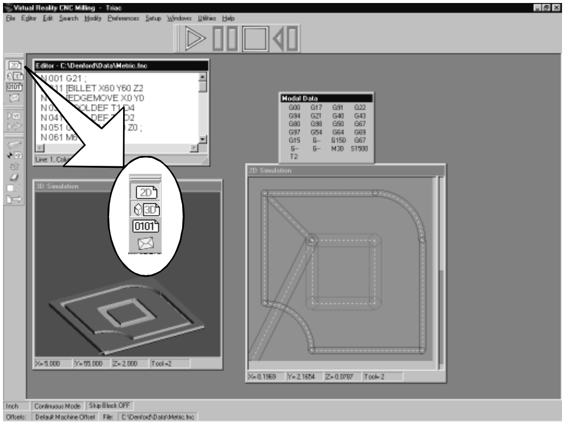
Default 1

Click "Setup | Toolbars | Load Level 1 Defaults" to format the toolbars with large picture buttons (graphics and text titles), as shown right.



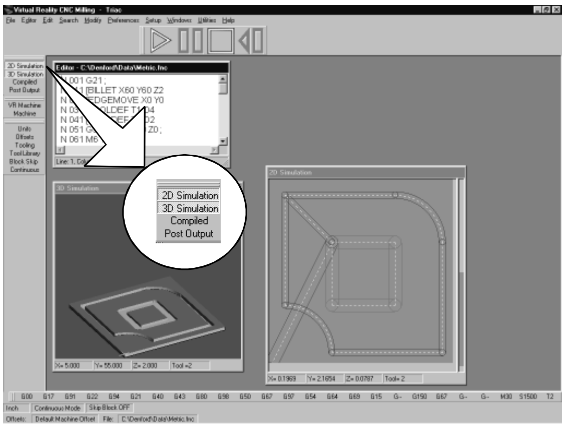
Default 2

Click "Setup | Toolbars | Load Level 2 Defaults" to format the toolbars with small picture buttons (graphics only), as shown right.



Default 3

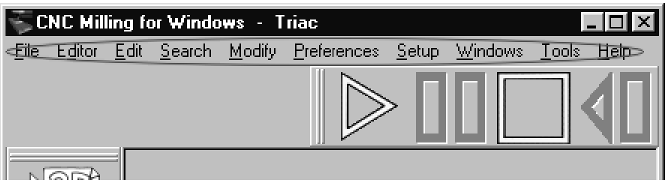
Click "Setup | Toolbars | Load Level 3 Defaults" to format the toolbars with large text buttons (text titles only), as shown right.



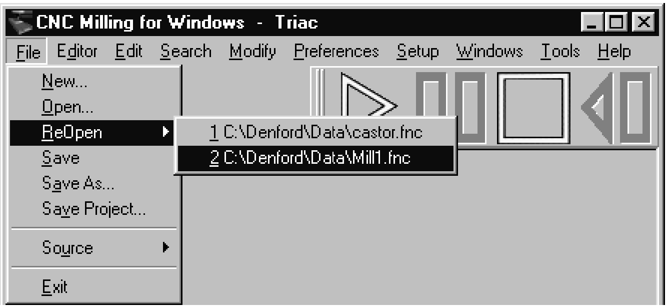
Note

The File Control Toolbar is always displayed using large graphics with no text.

4: Using the Menubars



The VR CNC Milling software menubar, highlighted by the grey ellipse in the screenshot above, is located under the main software title. It contains text captions identifying each individual menu.



To display the options available in each menu, click the menu text title to display its dropdown list, as shown above, then move the mouse cursor down the list to highlight the options. Click the highlighted option to select it or display its sub-menu, when available.

Context Sensitive Menus

The menus available will change according to the windows that are active in the software.

When the software is first started, the following menus are available:

- File, Editor, Edit, Search, Modify, Preferences, Setup, Windows, Tools, Help.

Additional Menus become available on selection of various software options:

- Machine Tooling on selection of the [Tooling] button.
- Tooling Library on selection of the [Tool Library] button.
- 2D Simulation on selection of the [2D Simulation] button.
- 3D Viewer on selection of the [3D Simulation] button.

4: Using the Helpfiles

What is Context Sensitive Help?

At the press of a key, context sensitive help automatically guides you to the appropriate sections of helpfiles, whenever you need help with various parts of the software. Context sensitive help is available for the following items:

- **Menu**: To obtain context sensitive help on a software menu, click the menu title to display its dropdown list of options, then press the **[F1]** key.
- **Window**: To obtain context sensitive help on a software window, press the **[F1]** key when the required window is active (ie, the required window titlebar is highlighted).
- **G and M codes**: To obtain context sensitive help on an individual G or M code, position the "Editor" window cursor in the middle of the text for the code required, then press the **[Ctrl + F1]** keys.

Available Helpfiles

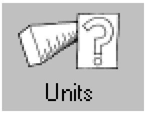
The VR CNC Milling software contains two separate helpfiles, both available from the "Help" menu title.

- **VR CNC Milling**: The VR CNC Milling for Windows software helpfile. This helpfile contains VR CNC Milling tutorials, detailed information about the various features of the VR CNC Milling software and troubleshooting guides.
- **CNC Programming**: The CNC Milling Programming helpfile. This helpfile contains detailed information about individual G and M codes and structure of CNC files.

4: Setting the Units of Measurement

The units of measurement used by the VR CNC Milling software must be set to match the units of measurement used by your CNC file and any tool profiles used.

For example, if you set the VR CNC Milling software to run in Metric Mode, you must use a metric compatible CNC file and metric tooling.



Click the [Units] button (shown above) from the "Options" toolbar, to change the units of measurement mode between:

"Metric" Mode: Metric - millimetre units.

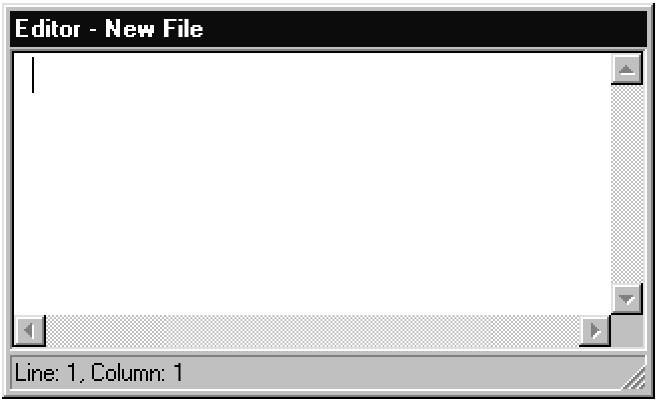
"Inch" Mode: Imperial - inch units.

The current setting of the option is displayed in the main program status bar, positioned in the bottom left corner of the main program window. The first information box on the upper line of this status bar (highlighted white in the screenshot above) indicates the units of measurement currently in use.

The units of measurement can also be configured using the "Setup | Units" menu option.

5: Creating a New CNC File

Click "File | New" to create a new CNC file, as shown above.



The blank "Editor" window will be displayed, as shown above.

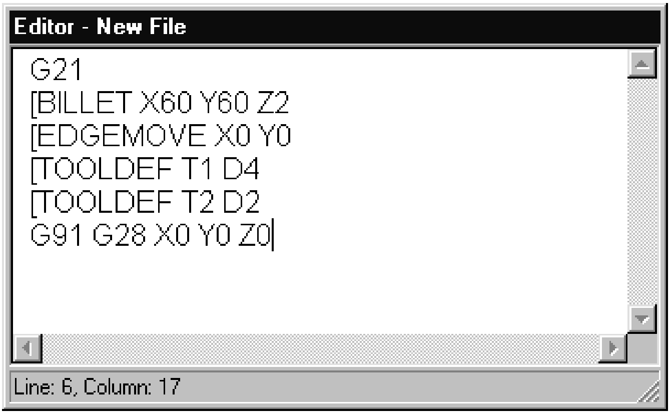
The "Editor" window behaves in a similar way to a simple word processor, such as Windows Notepad.

5: Entering Data into the "Editor" window

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

Your CNC file describes the program of commands and movements used to manufacture the part, hence CNC files are often referred to as Part Programs.

If you are following our tutorials, use the sample CNC files listed on pages 14 to 21.



As text characters are typed, they will appear on the appropriate line of the "Editor" window, as shown above.

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.

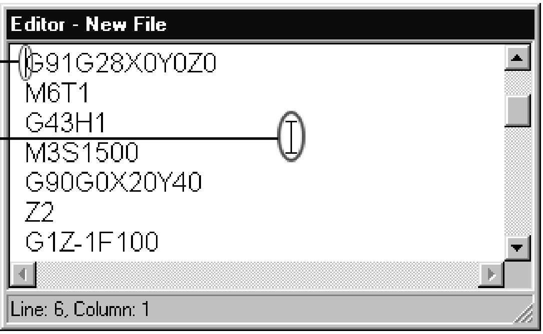
Each program word is constructed from a letter, called the address, and a number. The address letter, together with its number describes the type of code used.

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.

5: Positioning the "Editor" window cursor

"Editor" window cursor.

Mouse positioning cursor.



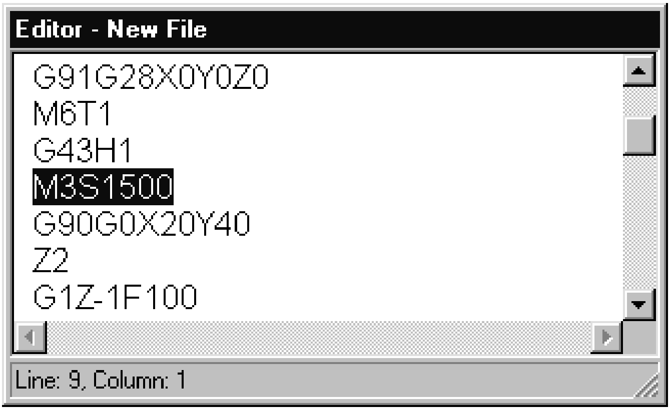
The "Editor" window cursor is a flashing vertical black line, highlighted in the above screenshot. This cursor shows where characters can currently be inserted, removed or highlighted. To remove characters directly behind the "Editor" window cursor, press the **[Delete]** key. To create a new CNC file line, press the **[Enter/ Return]** key.

The mouse positioning cursor is a vertical black line with bars at its top and bottom, highlighted in the above screenshot. This cursor is used to move the "Editor" window cursor to new positions in the CNC file.

To reposition the cursor in the "Editor" window:

- Position the mouse positioning cursor in the required area, then click the left mouse button to move.
- Use the four computer **[Cursor]** arrow keys to move the "Editor" window cursor to the required position.
- Use the **[Page Up]** key to move to the top of the CNC file.
- Use the **[Page Down]** key to move to the bottom of the CNC file.
- Use the **[Home]** key to move to the beginning of the current CNC file line.
- Use the **[End]** key to move to the end of the current CNC file line.

5: Selecting and editing areas of text



To select areas of text in the "Editor" window, position the "Editor" window cursor (the vertical black line) at the start or end of the text required, then click and hold down the left mouse button. Drag over the required characters to highlight them, as shown above.

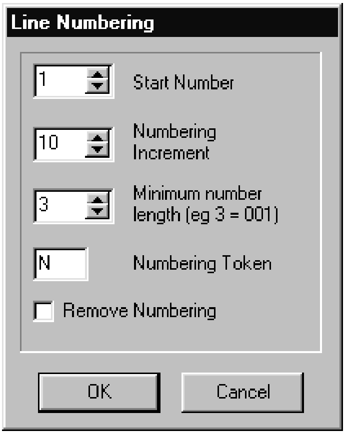
To select all the text in the "Editor" window, click "Edit | Select All".

The highlighted characters can be edited using the following commands:

- Select the "Cut" option from the "Edit" menu to cut any highlighted text from the "Editor" window to the Windows clipboard. Computer keyboard shortcut: **[CTRL + X]**
- Select the "Copy" option from the "Edit" menu to copy any highlighted text from the "Editor" window to the Windows clipboard. Computer keyboard shortcut: **[CTRL + C]**
- Select the "Paste" option from the "Edit" menu to place any text held in the Windows clipboard to the current "Editor" window cursor position. Computer keyboard shortcut: **[CTRL + P]**
- Select the "Undo" option from the "Edit" menu to undo the last command performed in the "Editor" window.
- Select the "Redo" option from the "Edit" menu to repeat the last command performed in the "Editor" window.

5: Adding Program Line Numbering

To add program line numbers to your finished CNC file, click "Modify | Line Numbering..." to display the "Line Numbering" window.



Enter the number you want to use as first line of the program in the "Start Number" dialogue box. In the example left, 1 has been specified.

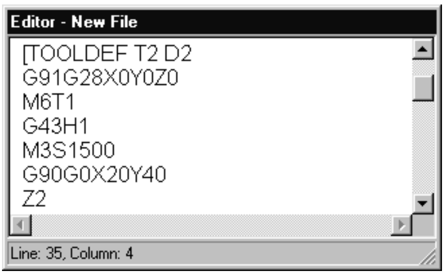
The "Numbering Increment" dialogue box is used to set the numerical gap between each program line number. In the example left, 10 has been specified, so the program line numbers will follow the sequence 1, 11, 21, 31, 41, 51 etc.

The "Minimum number length" dialogue box is used to set the amount of characters used to display each program line number. In the example above, 3 has been specified, so the program line numbers will follow the sequence 001, 011, 021, 041, 051 etc.

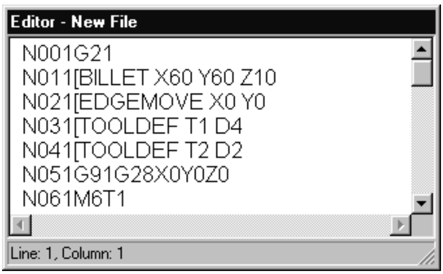
The "Numbering Token" dialogue box is used to add an address character to start of each program line number. The standard numbering token used is N. In the example above, the program line numbers will follow the sequence N 001, N 011, N 021, N 041, N 051 etc.

Click the [OK] button to apply program line numbering settings to the CNC file.

An example of a modified CNC file is shown below.



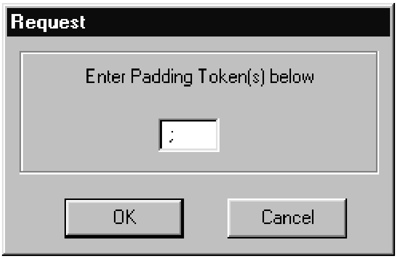
Before modifications.



After modifications (add program line numbering).

5: Adding End of Block symbols

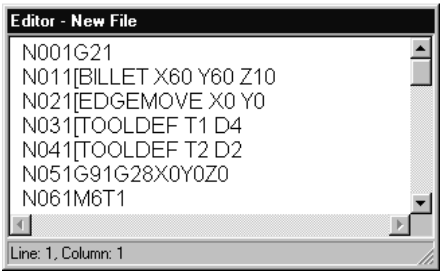
To add end of block symbols to your finished CNC file, click "Modify | Append Line End Token..." to display the "Request" window.



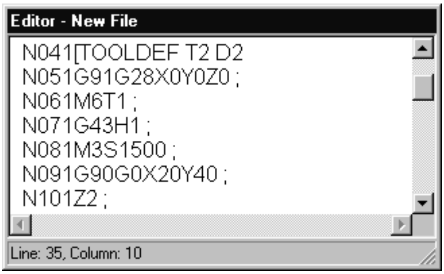
If you want to add end of block symbols to only part of the CNC file, drag across the required program lines to highlight them. If no program lines are highlighted, end of block symbols will be applied to the whole of the CNC file.

In the "Request" window dialogue box enter the character/s to be used for denoting the end of program lines, then click the [OK] button. The standard symbol used when CNC programming is the [semicolon] character ;.

An example of a modified CNC file is shown below.



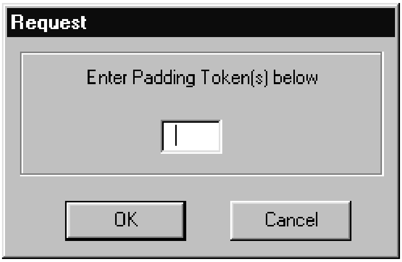
Before modifications.



After modifications (add ; symbol).

5: Adding Program Line Spacing

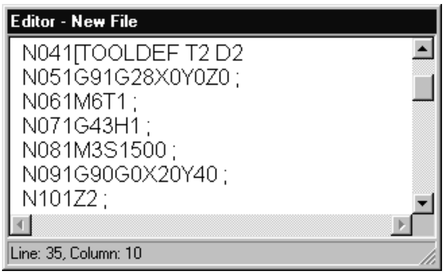
To add padding spaces between program words in your finished CNC file, click "Modify | Add Padding Token..." to display the "Request" window.



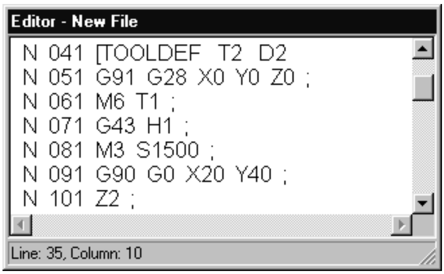
If you want to add padding spaces to only part of the CNC file, drag across the required program lines to highlight them. If no program lines are highlighted, padding spaces will be applied to the whole of the CNC file.

In the "Request" window dialogue box enter the number of spaces required between each program word, then click the [OK] button. In the example above, 2 spaces have been specified.

An example of a modified CNC file is shown below.



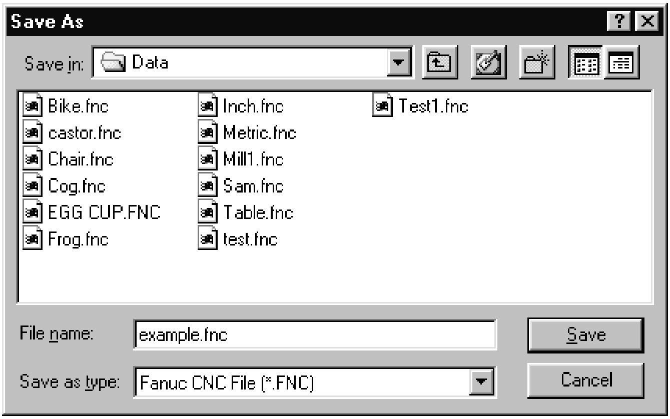
Before modifications.



After modifications (add 2 padding spaces).

5: 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.

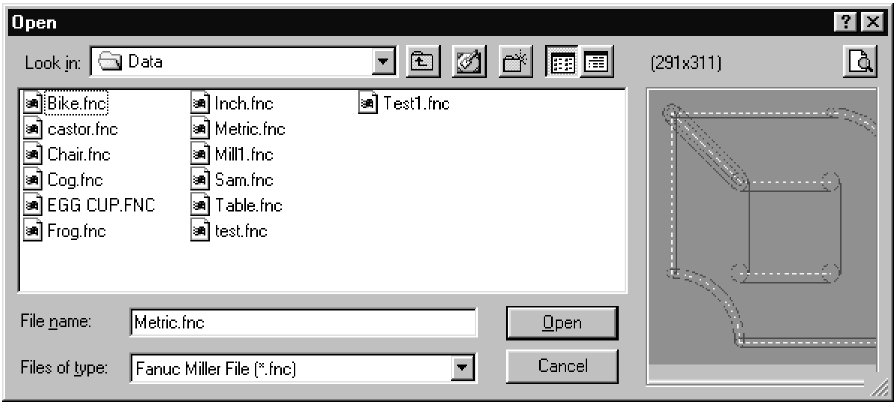
Enter the filename in the "File name:" dialogue box, using the file extension ".fnc", as shown above, then click the [Save] button.

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: Loading a CNC File

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



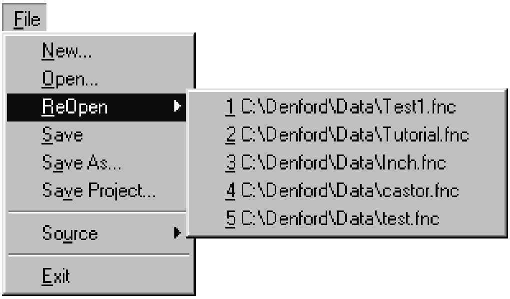
Select the directory used for storing the CNC file, using the "Look in:" panel.

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

Graphic bitmaps of the CNC file are also displayed in the righthand panel, when previously saved, as shown above.

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

5: 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.

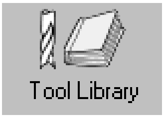
Click "File | ReOpen|{choiceof filename}", to reopen the required CNC file, as shown above.

6: Using the Tool Library

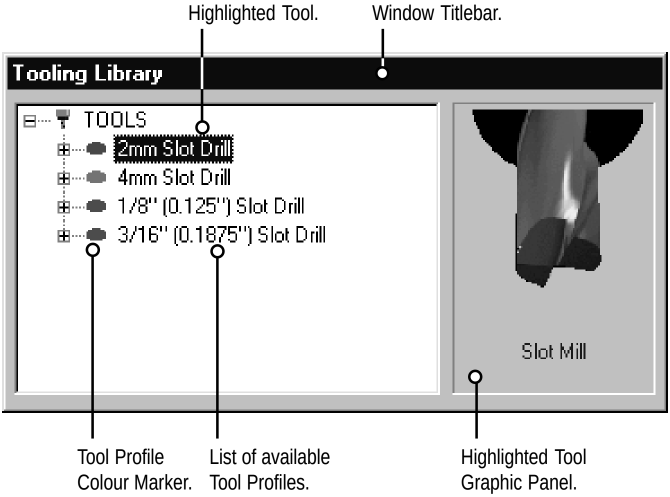
> **Note**
>
> Upon initial installation, the default Tooling Library profiles available are as follows:
>
> **Metric Tooling**:
> Slot Drills, 1mm, 2mm, 3mm, 4mm, 6mm, 8mm, 10mm, 12mm;
> End Mills, 1mm, 2mm, 3mm, 4mm, 6mm, 8mm, 10mm, 12mm;
> Ball Noses, 1mm, 2mm, 3mm, 4mm, 6mm, 8mm, 10mm, 12mm.
>
> **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").

Displaying the "Tooling Library" window

The Tooling Library contains the list of tool profiles that care available for use with a CNC machine. Tools from this list can be added to the "Machine Tooling" window, where they are assigned a tool number, ready for use with any CNC files. A selection of the most common tool profiles are automatically created upon initial installation of the VR CNC Milling software.



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



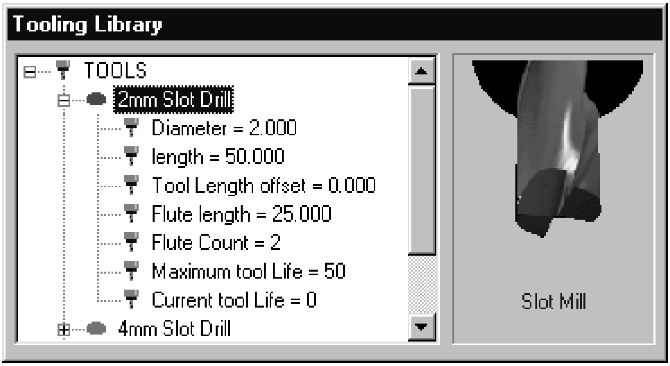
General Layout

Click on the [+] squares to expand the list of available tool profiles or the [-] squares to collapse an open list.

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. Highlighted tool profiles are shown using white title text on a blue background.

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

6: Viewing and Editing Tool Profile Data



To display the data allocated to a specific tool, click the [+] square next to the text title of the tool required. In the example above, the red "2mm Slot Drill" has been expanded to show its tool data list.

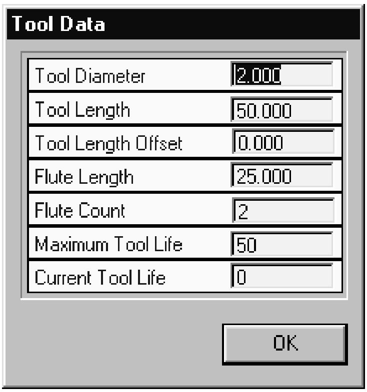
To edit a value in the tool data list, double-click the left mouse button on the data title required.

Before editing any tool profile data:

- Check that the units of measurement set for the VR CNC Milling software matches the units used by any tooling profiles. The units of measurement setting for the VR CNC Milling software is configured using the [Units] button on the "Options" toolbar.

Note

Any data changes made are applied to identical tool profiles used in the "Tooling Library" and "Machine Tooling" windows.



For example, if you are editing a metric tool, the VR CNC Milling software units of measurement must be set to metric.

The "Tool Data" window will be displayed, as shown on the left. 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.

6: Using the "Tool Data" window

> **Note**
>
> Any data changes made are applied to identical tool profiles used in the "Tooling Library" and "Machine Tooling" windows.

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).

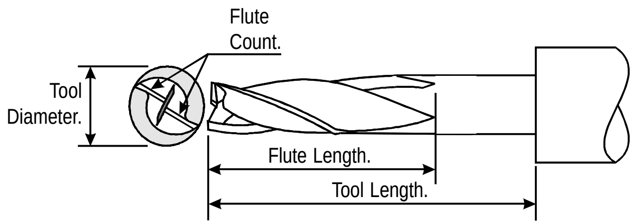
Tool Length Offset: The Z tool length offset value, defined in mm or inches. The objective of the Z tool length offsets is to allow different tool profiles to cut in the correct place on the billet, despite their obvious differences in length. Each tool is set against a common zero reference. The value 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).

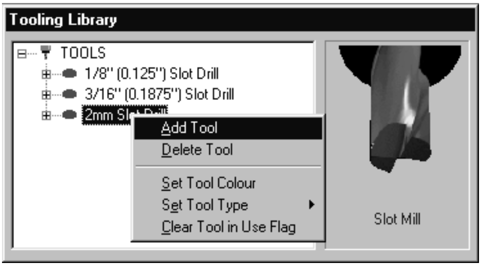
Maximum Tool Life: The working life of the tool, stated in hours.

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

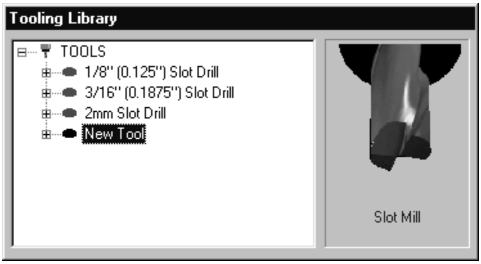


6: Creating a New Tool Profile

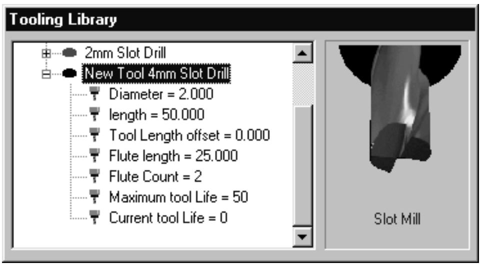
Check that the units of measurement set for the VR CNC Milling software matches the units used by any tooling profiles you intend to create. The units of measurement setting for the VR CNC Milling software is configured using the [Units] button on the "Options" toolbar.



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, as shown above.

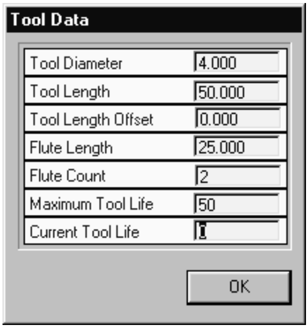


A new tool will be created at the bottom of the current list of tools, as shown above. Type a name for the new tool and press the [Enter] key.

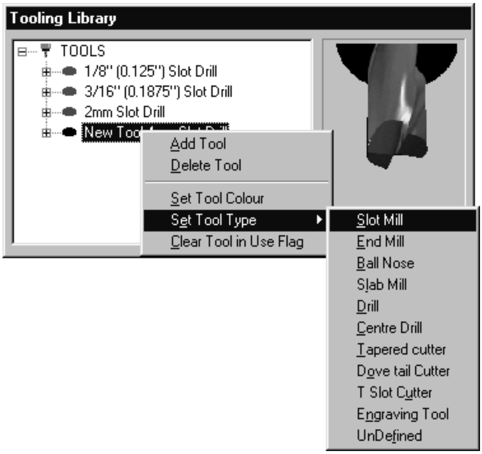


To display the data allocated to the new tool, click the [+] square next to its text title, as shown above. The new tool will inherit the data and graphic from the last tool highlighted in the library. To edit a value in the tool data list, double-click the left mouse button on the data title required.

6: Creating a New Tool Profile

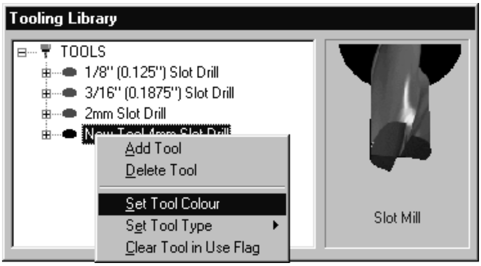


The "Tool Data" window will be displayed, as shown above. Click the cursor in any of the yellow tool data fields, delete the old values and enter the new data. Click the [OK] button to close the window and apply any changes made.

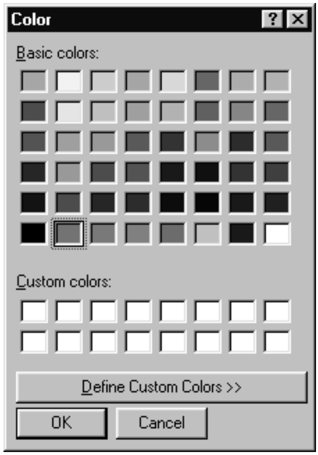


To change the graphic allocated to the new tool, displayed in the right panel of the "Tooling Library" window, click the right mouse button on the new tool title to display the pop-up menu. Move the cursor down the list, highlighting the "Set Tool Type" option, to display a secondary menu of possible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

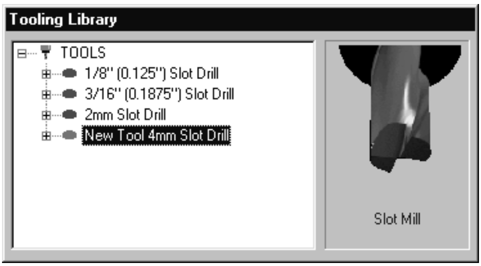
6: Creating a New Tool Profile



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, as shown above.



The "Color" window will be displayed, as shown right. 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 transferred to the "Machine Tooling" window.

6: Tool Profiles for the sample Metric CNC file

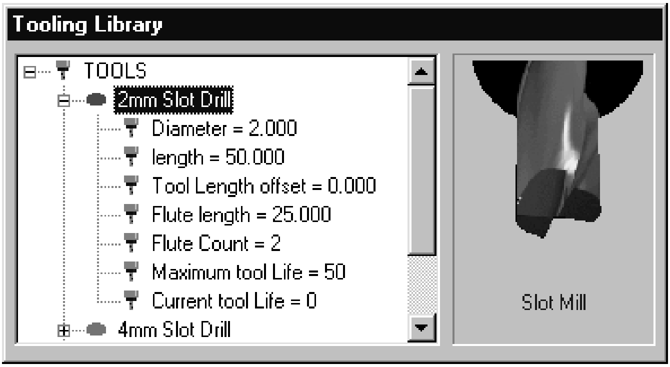
> **Note**
>
> When using a real ATC, ensure that tools are added to the numbered carousel pockets matching their tool number definitions.

This page explains how to configure the Tool Profiles for use with the sample Metric CNC file, listed on pages 14 to 17.

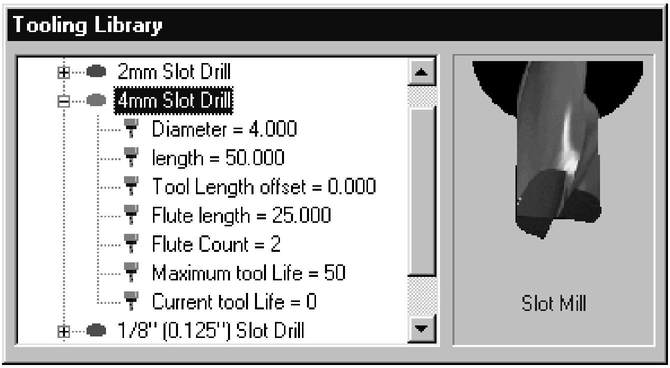
Check that the units of measurement are set to "Metric". The units of measurement setting for the VR CNC Milling software is configured using the [Units] button on the "Options" toolbar.

The Metric version of the sample CNC file uses two tool profiles:

2mm slot cutter: Used to cut the 2mm deep perimeter of the design. This profile should be assigned tool number 2. Default data for this tool profile is shown below:



4mm slot cutter: Used to etch the 1mm depth square design in the centre of the billet. This profile should be assigned tool number 1. Default data for this tool profile is shown below:



6: Tool Profiles for the sample Inch CNC file

Note

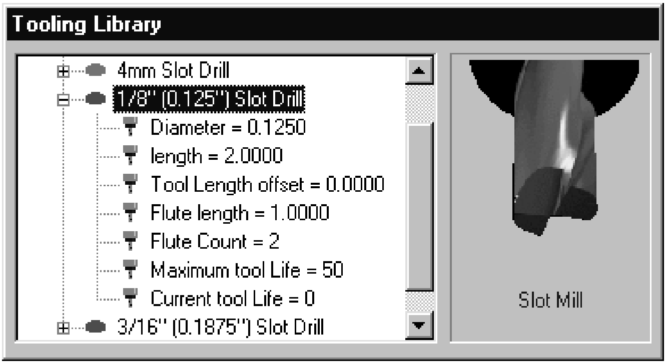
When using a real ATC, ensure that tools are added to the numbered carousel pockets matching their tool number definitions.

This page explains how to configure the Tool Profiles for use with the sample Inch CNC file, listed on pages 18 to 21.

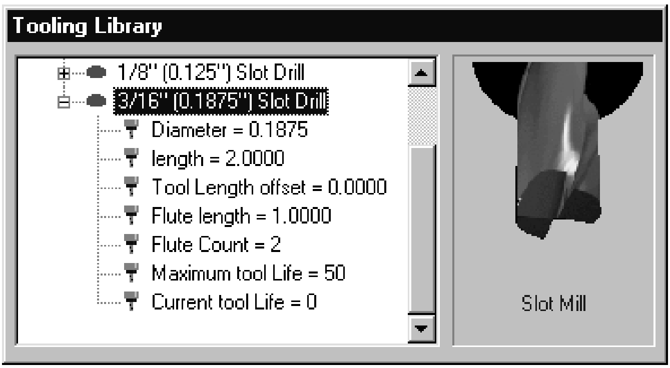
Check that the units of measurement are set to "Inch". The units of measurement setting for the VR CNC Milling software is configured using the [Units] button on the "Options" toolbar.

The Inch version of the sample CNC file uses two tool profiles:

1/8" (0.125) slot cutter: Used to cut the 1/8" (0.125) deep perimeter of the design. This profile should be assigned tool number 2. Default data for this tool profile is shown below:



3/16" (0.1875) slot cutter: Used to etch the 1/16" (0.0625) depth square design in the centre of the billet. This profile should be assigned tool number 1. Default data for this tool profile is shown below:

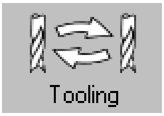


6: Using the "Machine Tooling" window

Displaying the "Machine Tooling" window

Individual tool profiles from the "Tooling Library" window are added to the "Machine Tooling" window, where they become ready for use with any CNC files.

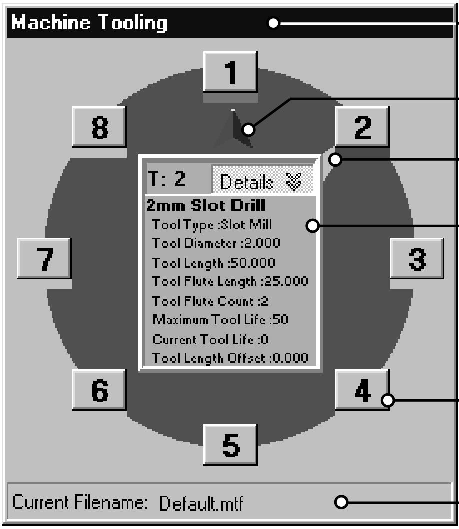
This procedure programs the VR CNC Milling software, so it knows what type of tool profile is associated with each tool number. The information is also used when generating any 2D/3D graphical simulations and saving Tool Length Offset files.

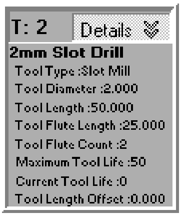


To display the "Machine Tooling" window, click the [Tooling] button, shown above, from the "Options" toolbar.

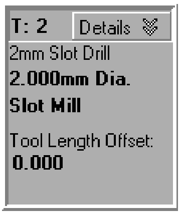
6: Using the "Machine Tooling" window

General Layout





Maximum information tool data panel.



Minimum information tool data panel.

Each coloured number refers to a tool number (defined in your CNC file). A black coloured number indicates the absence of any tool profile. When a tool is transferred into the "Machine Tooling" window, the colour allocated to that tool in the "Tooling Library" window is also transferred. These colours are applied to the different tool numbers as they become allocated.

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

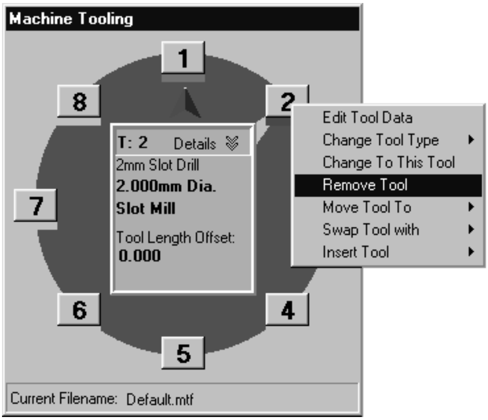
The triangular red marker arrow indicates the tool currently held in the machine head. In the example above, tool number 1 is currently held in the machine head.

The data panel in the centre of the window relates to the tool number indicated by the grey pointer. Click the [Details] button to change the amount of information displayed (see left). In the example above, 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.

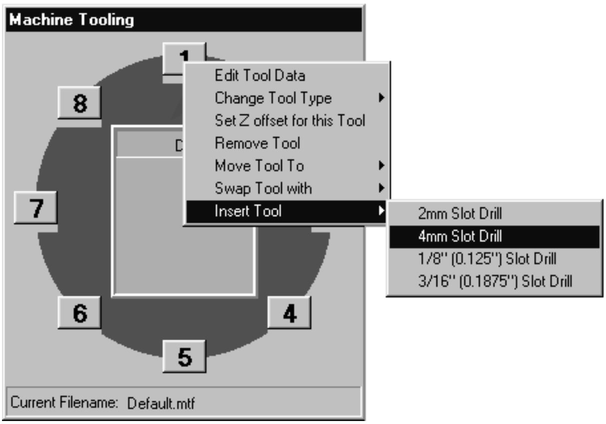
6: Transferring Tools into "Machine Tooling"

Deleting Tools from the "Machine Tooling" window



Click the right mouse button on the number relating to the tool profile 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.

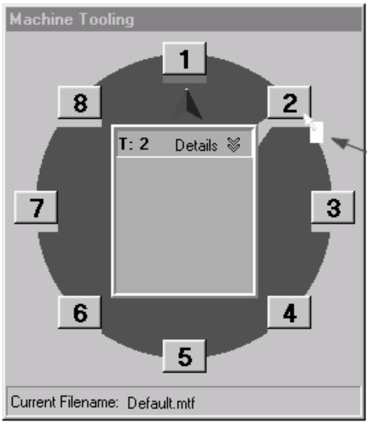
Adding Tools into the "Machine Tooling" window

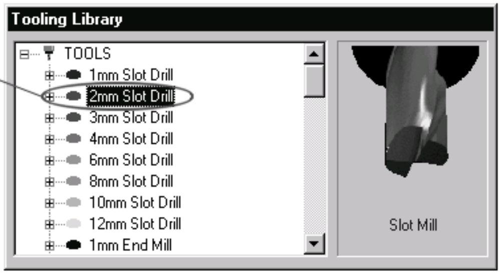


Click the right mouse button on a tool number in the "Machine Tooling" window. Highlight and click the "Insert Tool" option on the pop-up menu, then highlight and click on the title of the tool profile you want to add. The selected tool will then be assigned the specified tool number. Any new tool profiles added will overwrite any old data assigned to the chosen tool number.

6: Transferring Tools into "Machine Tooling"

Drag and Dropping Tools into the "Machine Tooling" window





In the "Tooling Library" window, click and hold down the left mouse button on the title of the tool profile you want to add to the "Machine Tooling" window. Whilst continuing to hold down the left mouse button, drag the tool out from the "Tooling Library" window and into the "Machine Tooling" window. Position the cursor over the required tool number and release the left mouse button. The selected tool will then be assigned with the specified tool number.

6: "Machine Tooling" with the sample CNC files

If you are following our tutorials, using the sample CNC files listed on pages 14 to 21, this page describes how the "Machine Tooling" window should be set.

Configuring the "Machine Tooling" for use with the sample Metric CNC file:

The Metric version of the sample CNC file uses two tool profiles:

- 2mm slot cutter: Used to cut the 2mm deep perimeter of the design. This tool profile should be assigned tool number 2.
- 4mm slot cutter: Used to etch the 1mm depth square design in the centre of the billet. This tool profile should be assigned tool number 1.

Configuring the "Machine Tooling" for use with the sample Inch CNC file:

The Inch version of the sample CNC file uses two tool profiles:

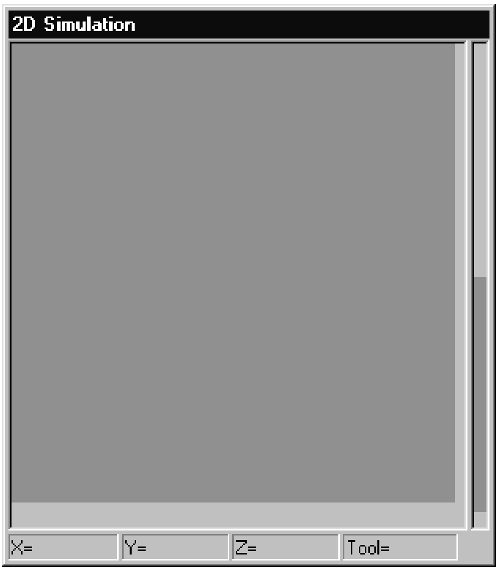
- 1/8" (0.125) slot cutter: Used to cut the 1/8" (0.125) deep perimeter of the design. This tool profile should be assigned tool number 2.
- 3/16" (0.1875) slot cutter: Used to etch the 1/16" (0.0625) depth square design in the centre of the billet. This tool profile should be assigned tool number 1.

7: Displaying the 2D Simulation window

The 2D Simulation window provides a plan view of the billet, together with any machined parts when the CNC file is executed.

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

Click the "Use X,Y Offsets" option on the "2D Simulation" menu, so a tick mark is not shown next to the title, as shown above. This will display the 2D simulation without using any simulated offsets. The "2D Simulation" window will show a fullsize view of the billet, as shown below. The narrow righthand column shows a side view of the billet, used for indicating tool cutting depths.



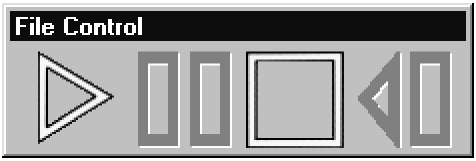
To close the "2D Simulation" window, click the [2D Simulation] button, shown above, from the "Outputs" toolbar.

7: Running a 2D Simulation of a CNC file

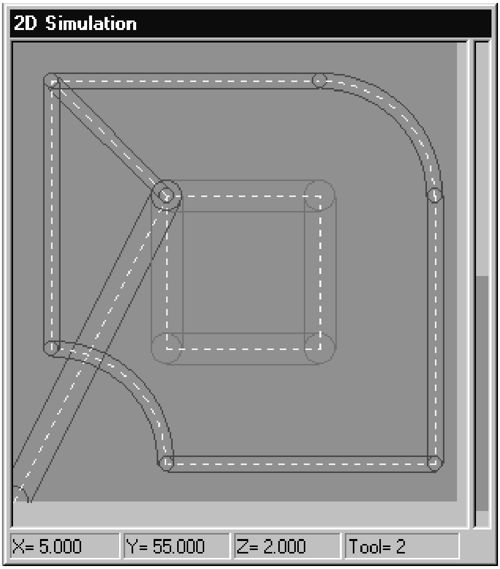
Before running the 2D simulation:

- 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.

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

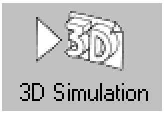


Click the triangular [Play] button from the "File Control" toolbar, shown above.



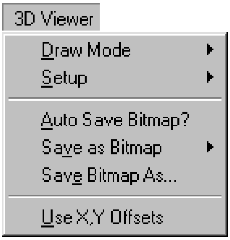
The 2D Simulation window will update according to the line being executed in the CNC file, until the end of the CNC file is reached.

7: Displaying the 3D Simulation window

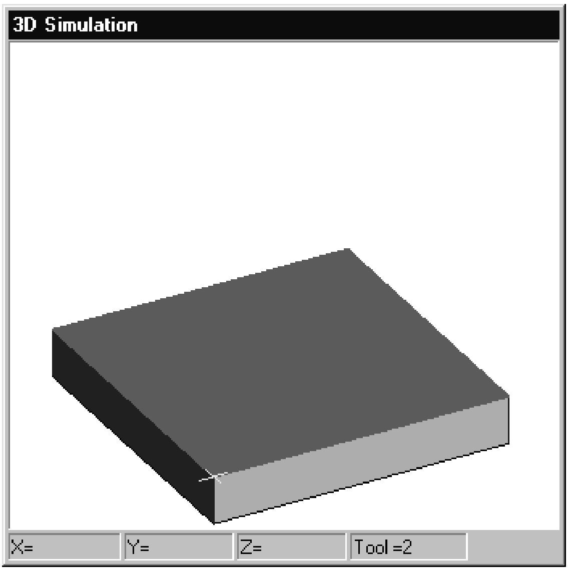


The 3D Simulation window provides a three dimensional view of the billet, together with any machined parts when the CNC file is executed.

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



Click the "Use X,Y Offsets" option on the "3D Viewer" menu, so a tick mark is not shown next to the title, as shown above. This will display the 3D simulation without using any simulated offsets. The "3D Simulation" window will show a fullsize view of the billet, as shown below.



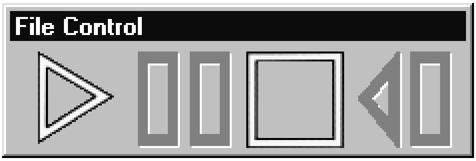
To close the "3D Simulation" window, click the [3D Simulation] button, shown above, from the "Outputs" toolbar.

7: Running a 3D Simulation of a CNC file

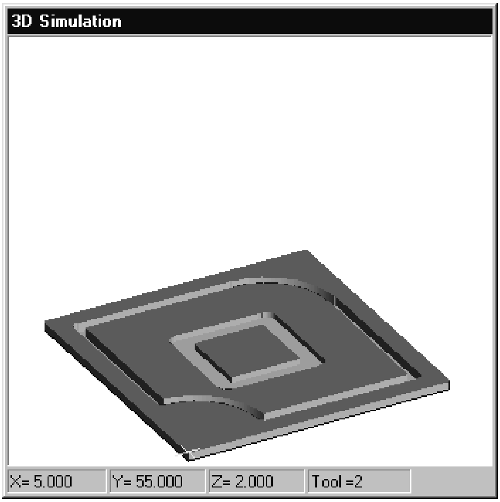
Before running the 3D simulation:

- 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 above.



The 3D Simulation window will update according to the line being executed in the CNC file, until the end of the CNC file is reached.

7: Why use Simulated Offsets?

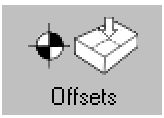
The theory behind offsets are described in greater detail in section 13 - CNC Theory.

Offsets are used to describe the position of the workpiece datum. This is the place where you want any machining co-ordinates to begin. The VR CNC Milling software allows a workpiece offset file to be created for the CNC machine, or solely for the use of the 2D and 3D simulations:

- Workpiece Machine Offset Files: Workpiece machine offsets are used to configure the position of the workpiece datum on the Virtual Reality and more importantly, any real CNC machines. Reconfiguring these offsets for viewing the simulation windows could disturb any values previously saved for safe machining operations on the real CNC machine. When machining finally takes place, an incorrect offset file could cause damage to occur to the CNC hardware.
- Workpiece Simulation Offset Files: Workpiece simulation offsets provide a computer generated representation of what will happen when an offset is used but avoids disturbing any values set for use with the VR or real CNC machines.

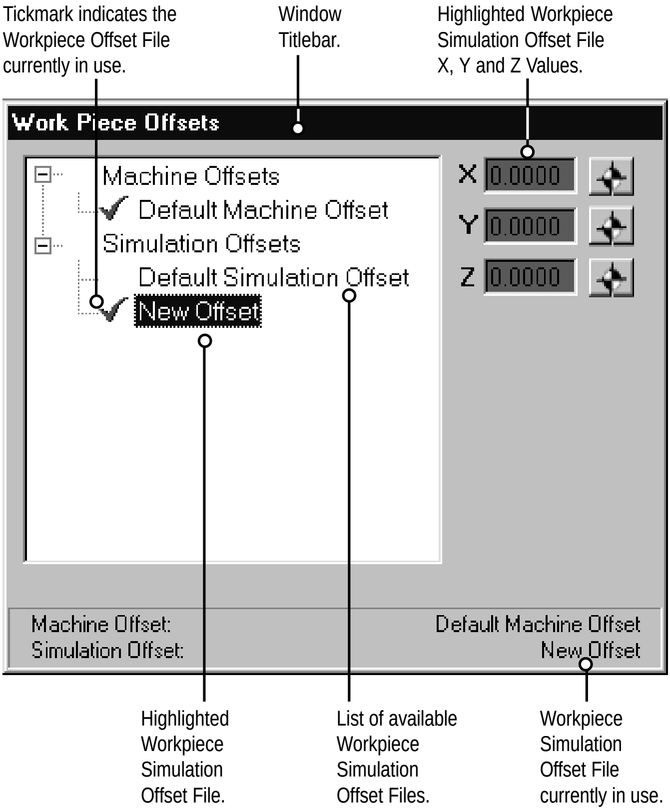
Using the "Work Piece Offsets" Window

The "Work Piece Offsets" window displays the various lists of workpiece offsets available. The "Simulation Offsets" are used to simulate workpiece offsets with the 2D and 3D simulation graphics windows.



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

7: The Work Piece Offsets window



Click on the [+] squares to expand the "Simulation Offsets" list or the [-] squares to collapse an open list.

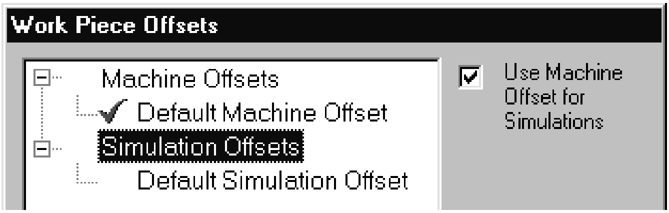
To highlight a simulation offset, click on its title. Highlighted simulation offsets are shown using white title text on a blue background.

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

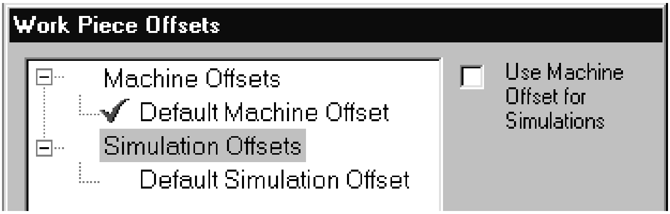
A red tick mark is used to indicate the active (currently used) simulation offset, also shown in the statusbar, positioned at the bottom of the "Work Piece Offsets" window.

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

7: Creating a Simulated Offset

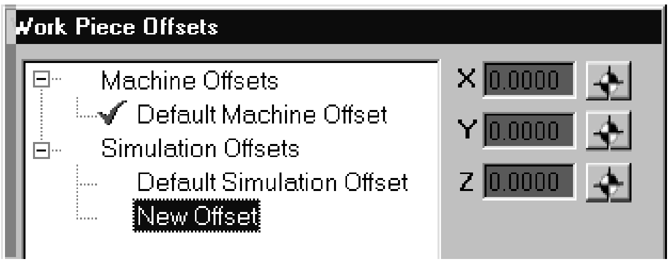


Highlight "Simulation Offsets" by clicking its title, as shown above.



Click the "Use Machine Offset for Simulations" checkbox so that a tick mark is not displayed, as shown above. This allows simulated offset values to be configured and used.

Right click the "Simulation Offsets" title then left click the "Add Offset" option from the pop-up menu, as shown above.

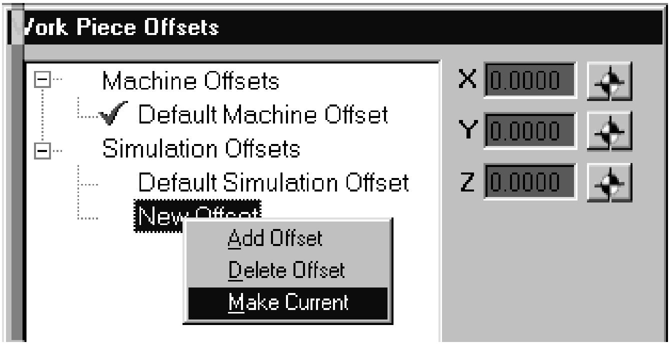


A "New Offset" is added to the list of "Simulation Offsets", as shown above.

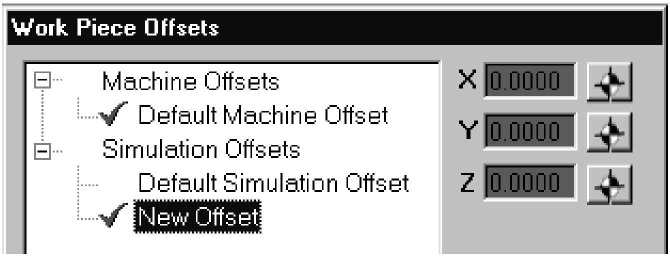
7: Creating a Simulated Offset

If some or all of the simulation offset values are already known, enter their values into the X, Y and Z co-ordinate dialogue boxes. Otherwise, click the cursor inside the X, Y and Z co-ordinate dialogue boxes, entering values of zero in each, as shown below.

To rename this simulation with a title of your choice, click the cursor at the end of the "New Offset" title, delete the text, then enter a new title and press the **[Enter/Return]** key.



Right click the "Simulation Offsets" title then highlight and left click the "Make Current" option from the pop-up menu, as shown above.



A red tick mark is placed next to the title of the currently active simulation offset, as shown above. Notice that there is a separate machine offset, also highlighted with a red tick mark. This machine offset is used to configure the workpiece datum position on the Virtual Reality and any real CNC machines attached to the computer.

7: Displaying a 2D Simulation using X,Y Offsets

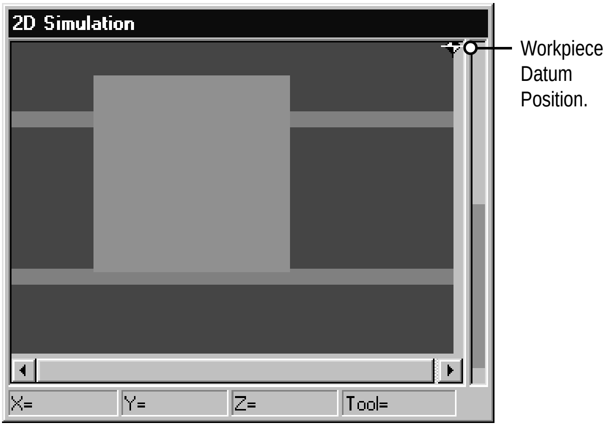
Check that the required "Simulation Offset" is highlighted and configured in the "Work Piece Offsets" window, as described on pages 58-59.

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

Click the "Use X,Y Offsets" option on the "2D Simulation" menu, so a tick mark is shown next to the title, as shown below.

This will display the 2D simulation using the "Simulation Offset" highlighted in the "Work Piece Offsets" window.

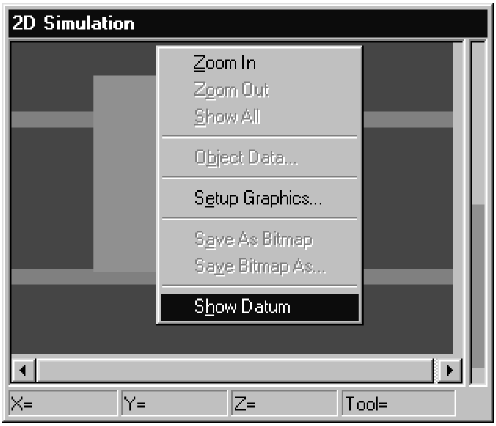
The "2D Simulation" window will show a fullsize view of the billet on the machine table, as shown below.



In the example shown above, the workpiece datum is currently positioned in the top righthand corner of the machine table. The narrow righthand column shows a side view of the billet, used for indicating tool cutting depths.

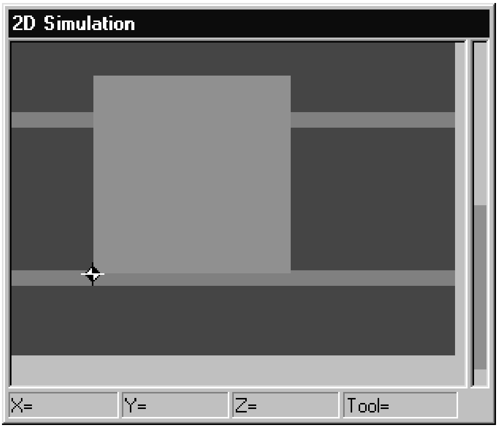
To close the "2D Simulation" window, click the [2D Simulation] button, shown above, from the "Outputs" toolbar.

7: Moving the Workpiece Datum Point in 2D



To display and move the datum symbol, right click in the "2D Simulation" window, then highlight and left click the "Show Datum" option from the pop-up menu, as shown above.

The workpiece datum symbol looks like this: 



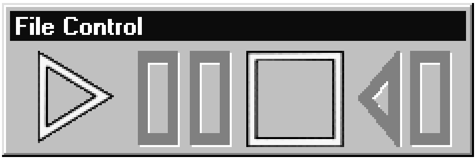
To move the datum, left click the mouse button on the datum symbol to lift it from the plan view. Move the datum to the new position, then left click the mouse button to fix the datum in its new position, as shown above.

7: Running a 2D Simulation using X,Y Offsets

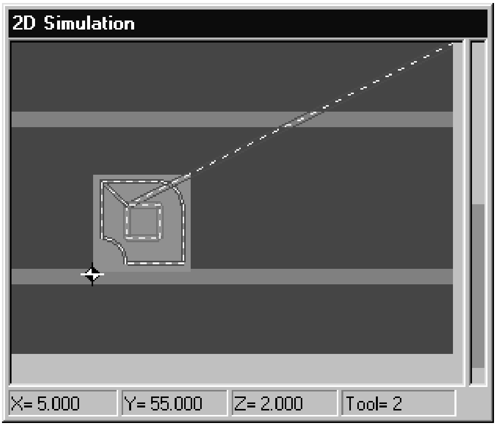
Before running the 2D simulation:

- 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.

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



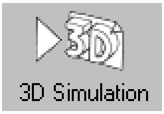
Click the triangular [Play] button from the "File Control" toolbar, shown above.



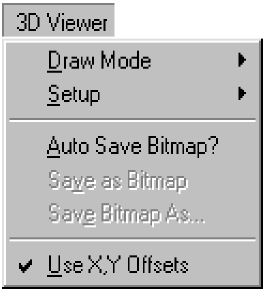
The simulation window will show the billet being cut on the machine table, using the configured simulation tool offset, as shown above.

7: Displaying a 3D Simulation using X,Y Offsets

Check that the required "Simulation Offset" is highlighted and configured in the "Work Piece Offsets" window, as described on pages 58-59.

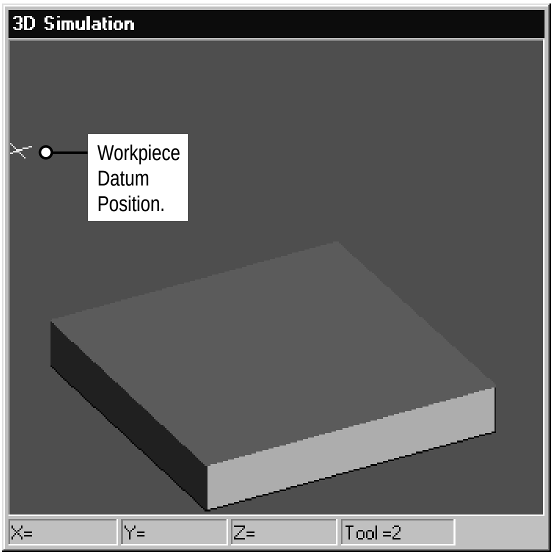


To display the "3D Simulation" window, click the [3D Simulation] button, shown above, from the "Outputs" toolbar.



Click the "Use X,Y Offsets" option on the "3D Viewer" menu, so a tick mark is shown next to the title, as shown left.

This will display the 3D simulation using the "Simulation Offset" highlighted in the "Work Piece Offsets" window.

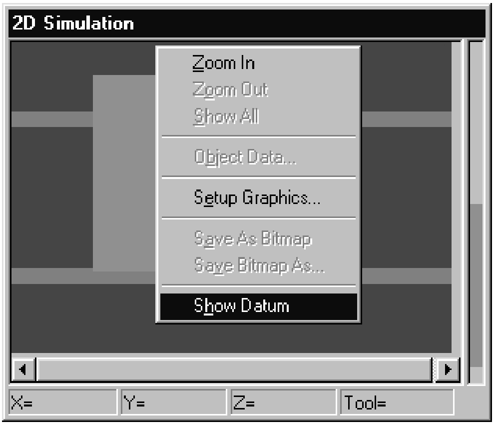


The position of the workpiece datum is shown by the yellow crosshairs. For the tutorials included in this manual, the workpiece datum would need to be set at the top left front corner of the billet.

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

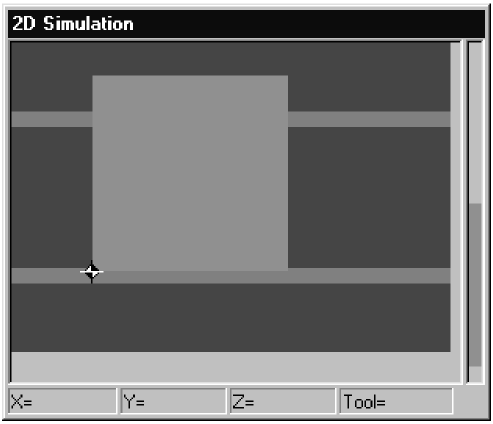
7: Moving the Workpiece Datum Point in 3D

The position of the 3D simulation workpiece datum is set using the "2D Simulation" window.



To display and move the datum symbol, right click in the "2D Simulation" window, then highlight and left click the "Show Datum" option from the pop-up menu, as shown above.

The workpiece datum symbol looks like this: 



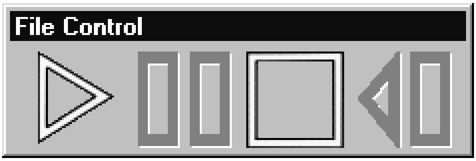
To move the datum, left click the mouse button on the datum symbol to lift it from the plan view. Move the datum to the new position, then left click the mouse button to fix the datum in its new position, as shown above. The datum crosshairs in the "3D Simulation" window will be updated to this new position.

7: Running a 3D Simulation using X,Y Offsets

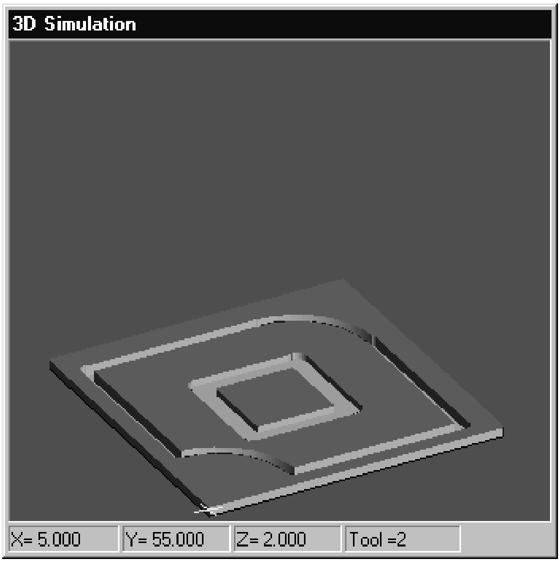
Before running the 3D simulation:

- 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 above.



The "3D Simulation" window will show the billet being cut, using the configured simulation tool offset, as shown above.

8: Starting a CNC Machine

Before starting any 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. 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.

Note

Although the procedure for entering offset data for both real and VR CNC machines is always the same, the actual offset positions used can differ between real and VR CNC machines, even when using the same CNC file.

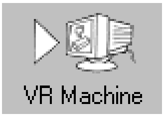
When using a real CNC machine and a CNC file that includes sections that will be completely cutaway (like the samples used in these tutorials), a sub-table must always be fitted. This prevents the machine table from being damaged when the tool machines completely through the thickness of the billet. Any offset positions are configured with this sub-table in position.

Sub-tables are not used on a VR CNC machine, since no damage can occur to the hardware.

Starting a VR CNC Machine.

1) To start the Virtual Reality CNC Machine, click the [VR Machine] button, shown below, from the "Machine Control" toolbar.
2) The "Denford Virtual Reality" window will open. This window is used for viewing the 3D model of the VR CNC machine. Any tools present in the "Machine Tooling" window (the Automatic Tool Changer) will also be loaded, when applicable.

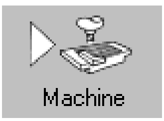
   The "Machine Mode" window will also open. This window is used for controlling the movements of the VR CNC machine.



Starting a real CNC Machine.

1) Ensure the RS232 lead is fitted securely between the computer and the CNC machine.
2) Switch on the CNC machine.
3) Power up the computer and start the VR CNC Milling software.
4) Establish communications to the real CNC Machine by clicking the [Machine] button, shown below, from the "Machine Control" toolbar.

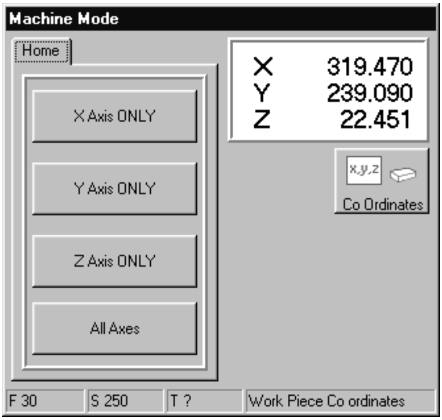
   The "Machine Mode" window will open. This window is used for controlling the movements of the real CNC machine.



8: Homing a CNC Machine - Home Mode

Note

The numerical figures depicted on any screenshots will differ according to the CNC machine type, the units of measurement setting for the VR CNC Milling software and any offsets being used on your computer system.



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

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.

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 (this assumes that no offsets are loaded).

Note

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

Homing the CNC Machine Axes

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.

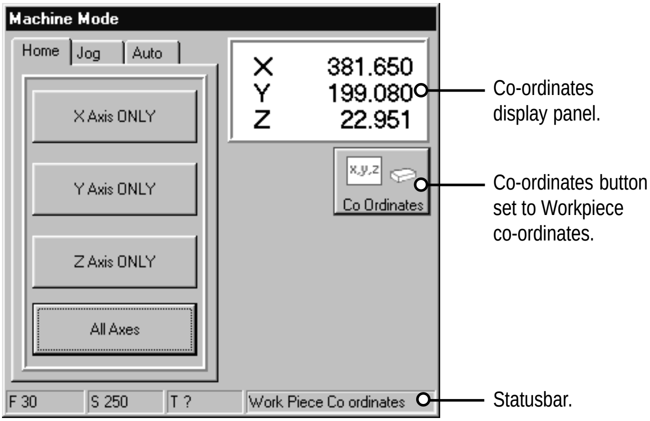
8: Co-ordinate System Display Modes

The [Co ordinates] button is used to switch between the two systems for displaying the co-ordinates positions. 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.

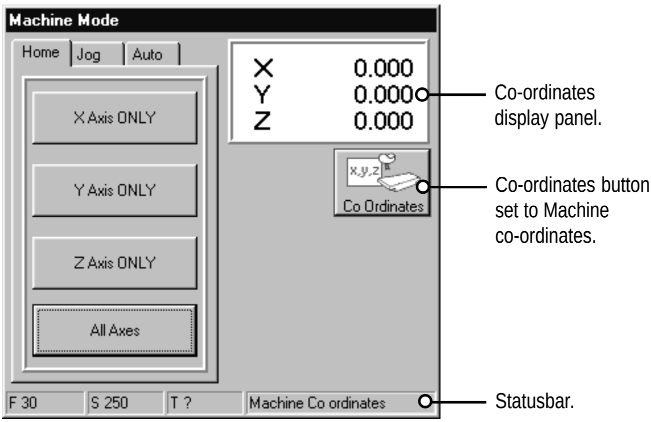
Work Piece Co-ordinates Display System

The Work Piece Co-ordinates system displays the co-ordinate position values relative to the moveable workpiece datum.



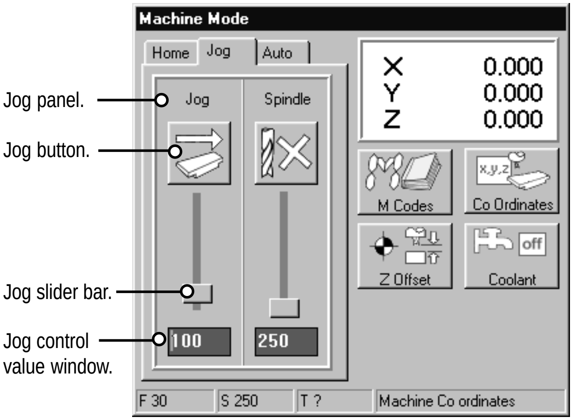
Machine Co-ordinates Display System

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



> **Note**
>
> The numerical figures depicted on any screenshots will differ according to the CNC machine type, the units of measurement setting for the VR CNC Milling software and any offsets being used on your computer system.

8: Moving 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.

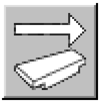
To change the jog control value, click and hold down the left mouse button on the slider bar, then drag the slider bar up or down to the new position.

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.

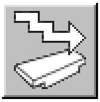
8: Moving the Axes - Jog Mode

Jog Control Modes.

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 movement 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 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. When Jog Step is active, the [Jog] button graphic will be displayed as shown below.



Movement (Jog) Keys.

There are six Jog Control movement keys:

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).

8: Selecting M Codes

Selecting M Codes.

M codes are used for miscellaneous functions, such as switching the spindle on / off and opening / closing the CNC machine guard.

Click the [M Codes] button to display the list of M codes, shown above. Move the cursor down the list to highlight the options. Click the highlighted option to select it.

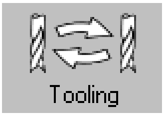
For detailed information regarding M code Programming, click "Help | CNC Programming" to display the "Denford CNC Programming for Milling Machines" helpfile.

Note

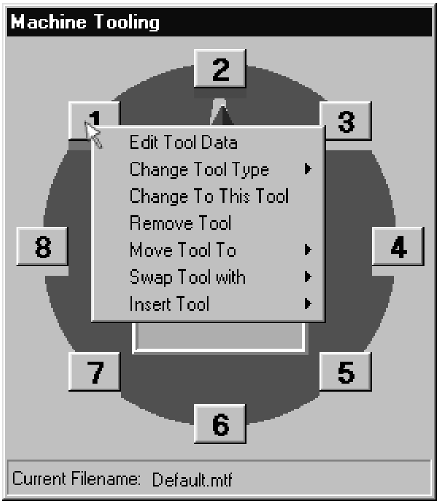
If the M code you require is not displayed in the dropdown list, enter the M code in a blank "Editor" window. Run the single command line by clicking the "Auto" tab on the "Machine Mode" window, followed by the [Play] button on the "File Control" toolbar.

8: Changing Tools Manually

The "Machine Tooling" window is used to change tools in the VR CNC Milling software.

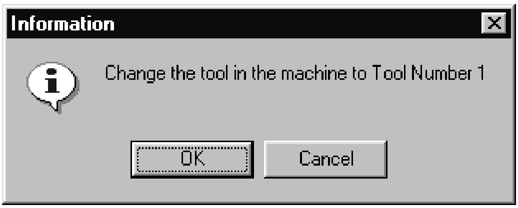


To display the "Machine Tooling" window, click the [Tooling] button, shown left, from the "Options" toolbar.



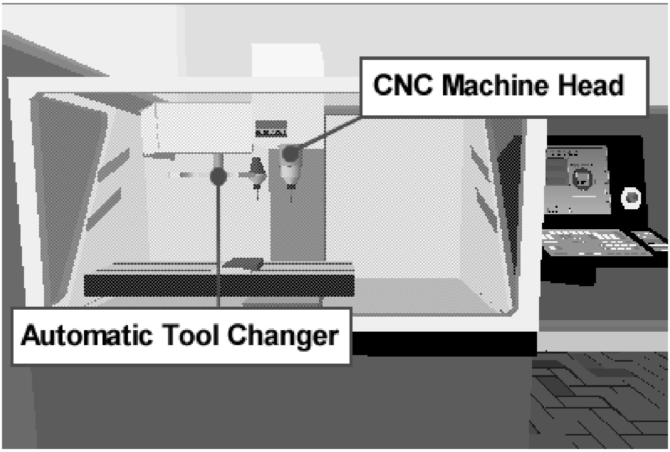
Click the right mouse button when the cursor is positioned over the number of the tool you want to place into the machine head. Highlight and click the "Change To This Tool" option, from the pop-up menu that is displayed, as shown above.

> **Note**
>
> For detailed information regarding operation of the tool change system hardware fitted to your CNC machine, please refer to your separate CNC Machine User's Manual.

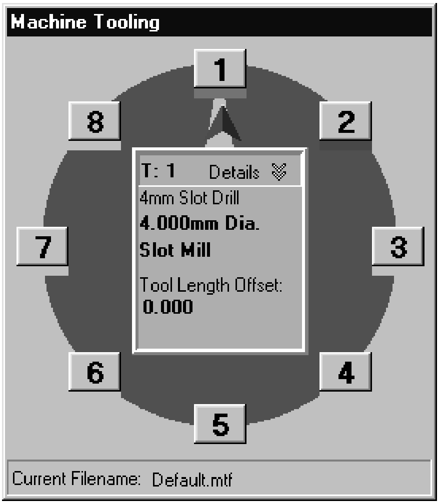


On CNC machines fitted with manual tool change systems, an "Information" window is displayed, as shown above. If you are using a real CNC machine, wait for the spindle and axes to stop moving, then open the safety guard door. Manually replace the tool holder with the tool number indicated in the "Information" window, then close the safety guard door. Confirm that the tool change operation has been completed by clicking the [OK] button.

8: Changing Tools Manually



On CNC machines fitted with an Automatic Tool Changer system, the tool change operation will be performed automatically.



Following completion of the tool change operation, the new tool profile number held in the machine head is shown by the triangular red arrow at the top of the "Machine Tooling" window, as shown above.

9: What are 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.

9: What are Offsets?

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.

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.

Note

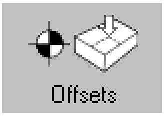
For further information about Offsets and CNC terminology, please see section 13 - CNC Theory.

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.

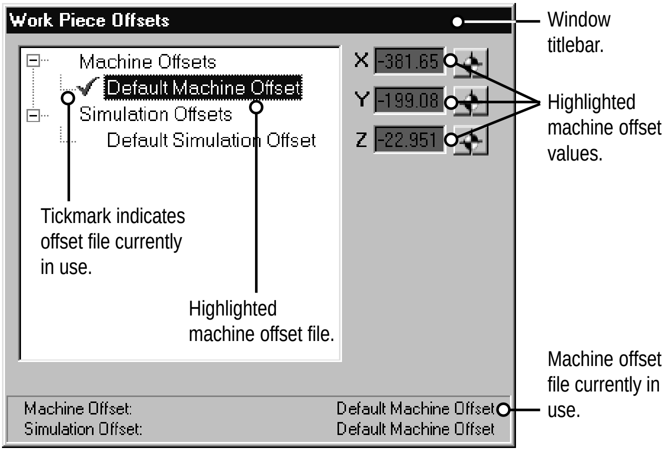
9: Creating a new Workpiece Offset file



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

The "Work Piece Offsets" window displays the various lists of offsets available. The Workpiece "Machine Offsets" are used to configure the position of the workpiece datum on the Virtual Reality and attached real CNC machines.

General Layout.



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. Highlighted machine offsets are shown using white title text on a blue background.

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.

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

9: Creating a new Workpiece Offset file

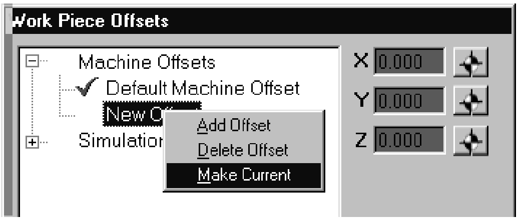
Creating a new Workpiece "Machine Offset" file.

A) Add a New Machine Offset to the list.

Highlight the current machine offset, then click the right mouse button on its title.

Highlight and click the "Add Offset" option, from the pop-up menu that is displayed, as shown above.

B) Set the New Machine Offset as the current Offset File.

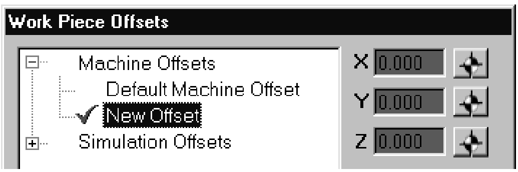


The new offset is always added to the bottom of the machine offsets list, with all co-ordinate values set to zero.

Highlight the "New Offset", then click the right mouse button on its title.

Highlight and click the "Make Current" option, from the pop-up menu that is displayed, as shown above.

This configures the "New Offset" as the currently active machine offset, ie, the machine offset file used by the CNC machine.



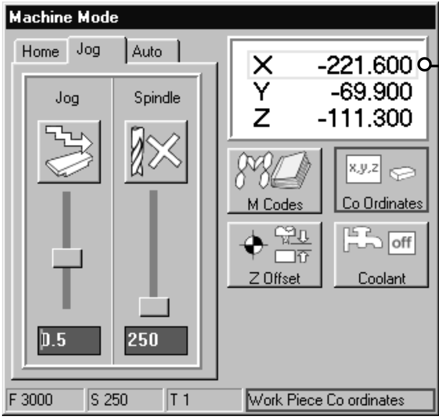
The currently active machine offset is indicated using a red tick mark, as shown above.

9: Configuring the X Offset Value

> **Note**
>
> The tool must be moved to align with the position of your workpiece datum before attempting to enter any X co-ordinate values. See section 10 - Using a Virtual Reality CNC Machine and section 11 - Using a real CNC Machine for further information.

If you are following our tutorials, using the sample CNC files listed on pages 14 to 21, the following 2 pages describe how to transfer the X co-ordinate value of the X datum edge into the new offset file.

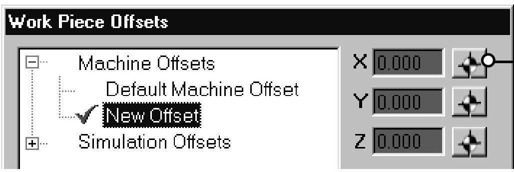
A) Transfer the X Co-ordinate Value into the Offset file.



This is the X co-ordinate value that will be transferred into the offset file.

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, as shown below.



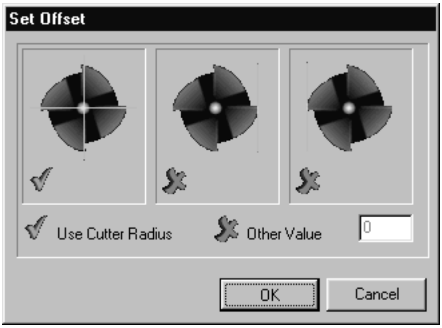
X co-ordinate transfer [datum] button.

Using the "Work Piece Offsets" window, click the [datum] button to the right of the X offset value display box, as shown in the above screenshot.

B) Specify the Cutter Orientation.

Before transferring the X co-ordinate value, the "Set Offset" window is displayed. The settings are used to account for the cutter orientation method used. Click the green tickmarks or red crosses to change the options.

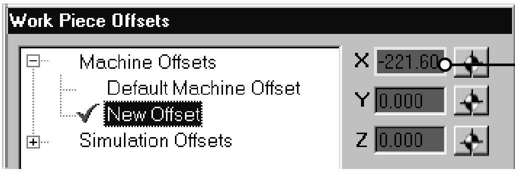
9: Configuring the X Offset Value



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 [OK] button to confirm the settings.

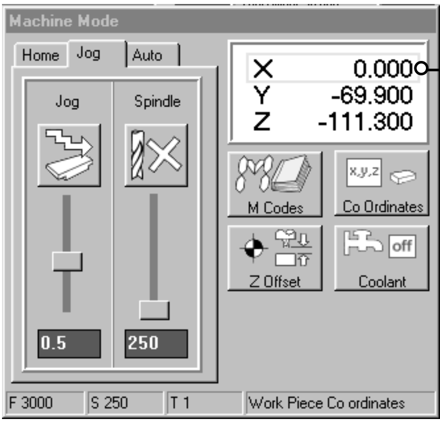
C) Check the X Co-ordinate Value has registered correctly.



The X co-ordinate value is transferred into the offset file.

The original X co-ordinate value from the "Machine Mode" window is transferred into the X offset value display box.

> **Note**
>
> The X workpiece datum position need only be set once, since workpiece offset values are global. They will be used with any tool profile, irrespective of tool length.



The X co-ordinate value changes to zero.

The X co-ordinate value in the "Machine Mode" window, shown in the above screenshot, will read zero, indicating that the X component of the workpiece datum has now been set.

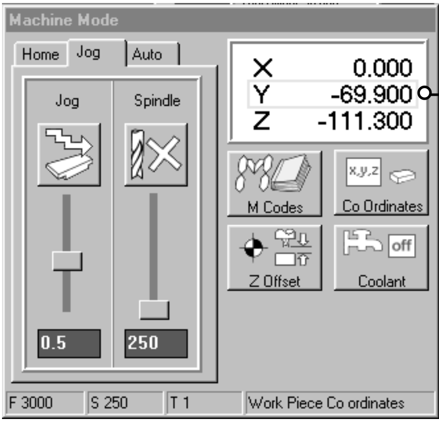
9: Configuring the Y Offset Value

Note

The tool must be moved to align with the position of your workpiece datum before attempting to enter any Y co-ordinate values. See section 10 - Using a Virtual Reality CNC Machine and section 11 - Using a real CNC Machine for further information.

If you are following our tutorials, using the sample CNC files listed on pages 14 to 21, the following 2 pages describe how to transfer the Y co-ordinate value of the Y datum edge into the new offset file.

A) Transfer the Y Co-ordinate Value into the Offset file.



This is the Y co-ordinate value that will be transferred into the offset file.

The Y co-ordinate value that will be transferred into the offset file is shown under the X co-ordinate value.

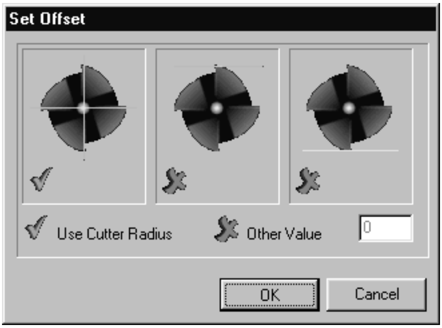
Y co-ordinate transfer [datum] button.

Using the "Work Piece Offsets" window, click the [datum] button to the right of the Y offset value display box, as shown in the above screenshot.

B) Specify the Cutter Orientation.

Before transferring the Y co-ordinate value, the "Set Offset" window is displayed. The settings are used to account for the cutter orientation method used. Click the green tickmarks or red crosses to change the options.

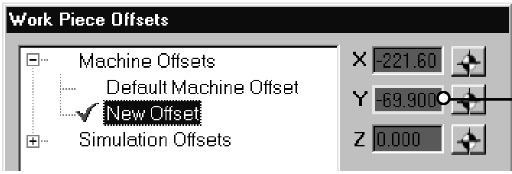
9: Configuring the Y Offset Value



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 [OK] button to confirm the settings.

C) Check the Y Co-ordinate Value has registered correctly.

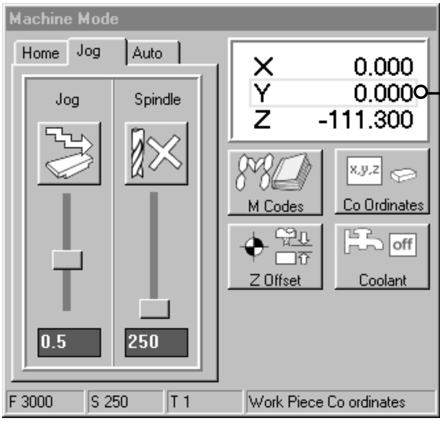


The Y co-ordinate value is transferred into the offset file.

The original Y co-ordinate value from the "Machine Mode" window is transferred into the Y offset value display box.

Note

The Y workpiece datum position need only be set once, since workpiece offset values are global. They will be used with any tool profile, irrespective of tool length.



The Y co-ordinate value changes to zero.

The Y co-ordinate value in the "Machine Mode" window, shown in the above screenshot, will read zero, indicating that the Y component of the workpiece datum has now been set.

9: Configuring the Tool Length Offset Value

If you are following our tutorials, using the sample CNC files listed on pages 14 to 21, the following 5 pages describe how to transfer the Z co-ordinate value into the tool length offset file.

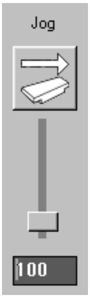
The tool length offset facility is used to compensate for the differences in length between all the tools used with the CNC file.

Note that the procedure must be repeated for all tools you intend to use.

A) Move the tool so the cutting tip just touches the chosen tool length offset reference point - see the diagrams on the opposite page.

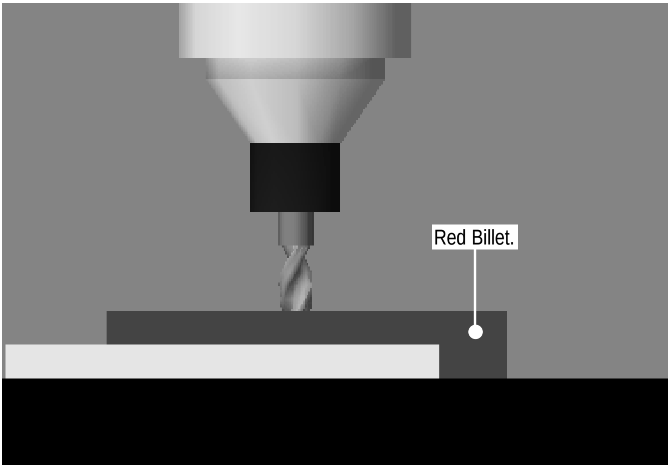
All tool length offsets are configured against a common tool length offset reference point. When values are entered into each individual tool length offset file, each tool will use this reference point as their zero co-ordinate along the Z axis. It is this figure that compensates for the differences in length when various tools are used together on the same job.

Check the machine axes are set in "Jog Continuous Mode" by clicking the [Jog] button in the "Machine Mode" window, so the button displays a straight arrow graphic.

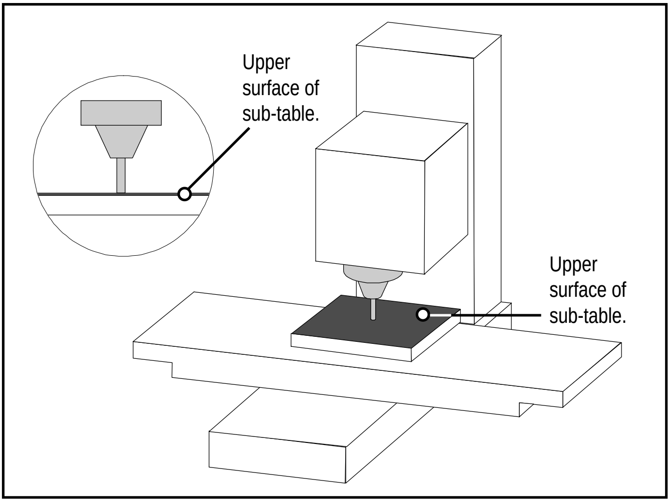


In "Jog Continuous Mode", the selected machine axis will move at the indicated speed when one of the machine axis movement keys are pressed and held down. The selected machine axis will continue to move until the key is released. The slider bar can be moved to set jog speeds between 0 and 1000 units. When the [Units] of Measurement are set to "Inch" the rate of movement is measured using inches per minute. When the [Units] of Measurement are set to "Metric" the rate of movement is measured using millimetres per minute.

9: Configuring the Tool Length Offset Value



If you are using a Virtual Reality CNC machine, the position of the tool length offset reference point fro each tool is anywhere on the upper surface of the red billet, as shown above. The Z value in the workpiece machine offset file value can remain zero.



If you are using a real CNC machine, the position of the tool length offset reference point for each tool is anywhere on the upper surface of the sub-table, as shown above. If your workpiece datum is the top surface of the billet, a global Z offset value must also be added into the workpiece machine offset file, to account for the billet thickness.

9: Configuring the Tool Length Offset Value

Move the cutter down towards the tool length offset reference point, using the six movement keys:

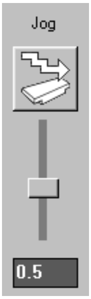
X Axis: **[Cursor Right]** or **[Cursor Left]**

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

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

Try to position the tip of the tool about 10mm (1/2") above the chosen tool length offset reference point.

Switch to "Jog Step Mode" by clicking the [Jog] button, so the button displays a stepped arrow graphic.



In "Jog Step Mode", the selected machine axis will move one indicated increment, each time the selected axis 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. When the [Units] of Measurement are set to "Inch" the rate of movement is measured using inches per minute. When the [Units] of Measurement are set to "Metric" the rate of movement is measured using millimetres per minute.

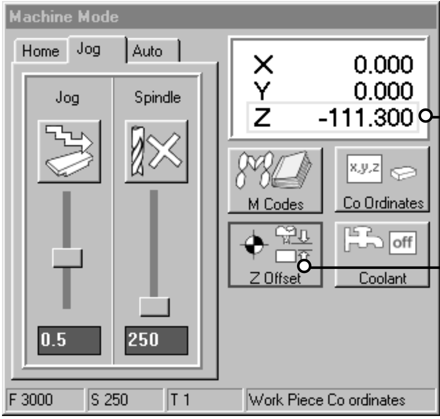
Use the [Page Up] or [Page Down] keys to position the cutter, so that the tip of the tool is just touching the chosen tool length offset reference point.

9: Configuring the Tool Length Offset Value

B) Transfer the Z Co-ordinate Value into the Z Tool Length Offset File.

Note

The tool length offset value you must enter will not be the same as the value depicted in our example screenshot. Remember that all tool length offset values will be different, since tools differ in length.



This is the Z co-ordinate value that will be transferred into the tool length offset file.

Click the [Z Offset] button to display the "Set Offset" window.

The Z co-ordinate value that will be transferred into the tool length offset file is shown in the above screenshot.

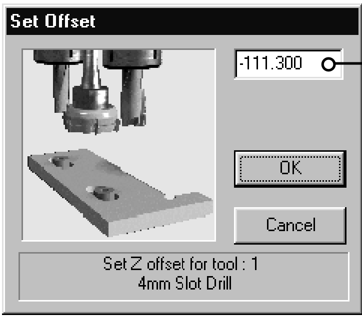
Note that this value will only apply only to the tool currently held in the machine head. This value will be different when you configure the Z tool length offset using another tool.

Remember, Z tool length offset files are individual to each tool profile, unlike workpiece offset files which are global.

Click the [Z Offset] button.

Note

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



Enter the tool length offset value here.

C) Specify the Tool Length Value.

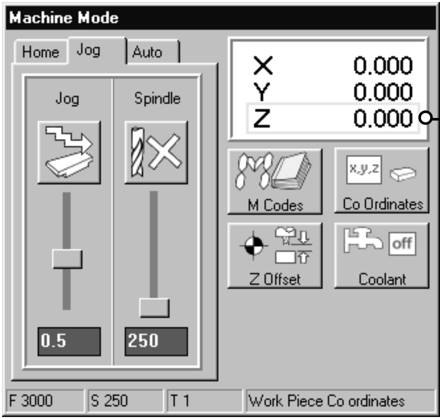
The "Set Offset" window for Z tool length offset values is displayed. Enter the Z tool length offset value into the dialogue box, if required.

In the tutorial example, the Z co-ordinate value from the "Machine Mode" window will automatically be entered into the dialogue box.

Click the [OK] button to confirm the value.

9: Configuring the Tool Length Offset Value

D) Check the Z Co-ordinate Value has registered correctly, for the tool currently held in the machine head.



The Z co-ordinate value changes to zero.

The Z co-ordinate value in the "Machine Mode" window, shown in the above screenshot, will read zero, indicating that the Z component of the workpiece datum has been set, but only for the particular tool profile you are using.

E) Repeat the Configure Z Tool Length Offsets process for all remaining Tool Profiles.

The Z Tool Length Offsets process must be repeated for all the other tool profiles that will be used with the CNC File.

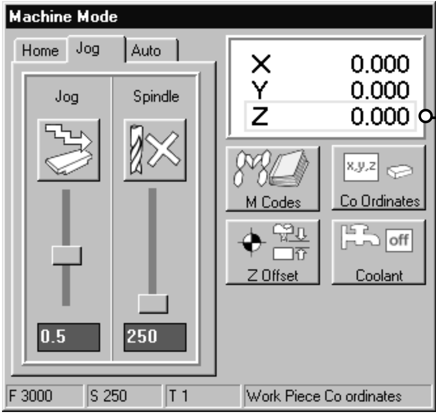
If you are following our tutorials, the X and Y offsets do not require configuring again, since the X and Y values are common to both tools used. Only the Z value needs to be configured since both tools differ in length.

Tutorial VR CNC Machine Users: Note that a value of zero is left in the Z dialogue box of the workpiece offset file. This is because the workpiece datum and tool length offset reference points lie along the same position in the Z axis (the tool length offset reference point was the surface of the billet).

Tutorial Real CNC Machine Users: Note that a global Z offset value must be entered into the "Work Piece Offsets" window, to account for billet thickness (the tool length offset reference point was the surface of the sub-table, rather than the surface of the billet). See the next two pages for further details.

9: Configuring a global Z Offset Value

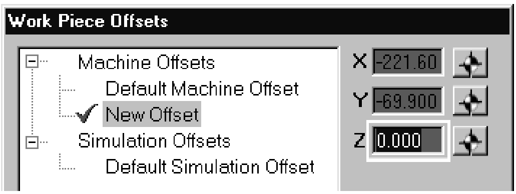
If you are following our tutorials on a real CNC machine, using the sample CNC files listed on pages 14 to 21, the following 2 pages describe how to enter a global Z value into the offset file, to account for billet thickness.



The zero value is indicated when the tool is touching the surface of the sub-table.

A Z value of zero is indicated when the tool is touching the surface of the sub-table. In order for the tool to cut at the correct depth, this Z value must read zero when the tool is touching the surface of the billet. A global Z value is entered to account for the thickness of the billet that will be used.

A) Position the cursor inside the Z Offset Value display box.



Using the "Work Piece Offsets" window, click the cursor inside the Z offset value display box, highlighted by the white square in the above screenshot.

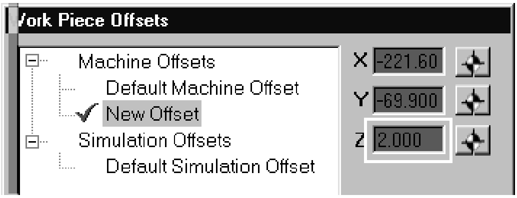
Drag the cursor back over the current offset figures to highlight them, shown using white numbers on a blue background.

9: Configuring a global Z Offset Value

Note

The value entered into the offset file is sign sensitive. For our sample CNC files, you must enter a positive value, in order to shift the workpiece datum up the Z axis.

B) Enter the new Z Offset Value.



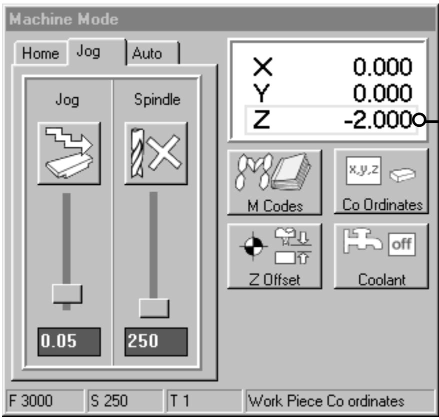
Enter the new Z offset value. The Z offset value entered should be the thickness of the billet. If you are using the CNC sample files, this value will be either 2mm (shown highlighted by the white square in the above screenshot) or 1/8" (0.125").

Deselect the "Machine Mode" window, so that it no longer active, by clicking on any other open VR CNC Milling window (for example, the "Editor" window).

C) Check the global Z Co-ordinate Value has registered correctly.

Note

The Z value accounting for billet thickness need only be set once, since values in the work piece offset file are global. They will be used with any tool profile, irrespective of tool length.



The Z co-ordinate value changes to indicate the workpiece datum has shifted.

The Z co-ordinate value in the "Machine Mode" window, shown in the above screenshot, will read the same value as the billet thickness but with a negative sign, indicating that the cutter is now below the workpiece datum point, ie, it would be cutting completely through the billet material.

Finally, home the CNC machine, by clicking the "Home" tab in the "Machine Mode" window, then clicking the [All Axes] button.

Important !

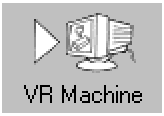
When you fit the plastic billet to the temporary MDF machine bed, prior to machining, the front bottom lefthand corner of the plastic billet must be exactly aligned with the front upper lefthand corner of the temporary MDF machine bed. This is because the front and lefthand edges of the temporary MDF machine bed are used as global X and Y datum edges.

10: Starting a VR CNC Machine

Before starting a Virtual Reality 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. 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.



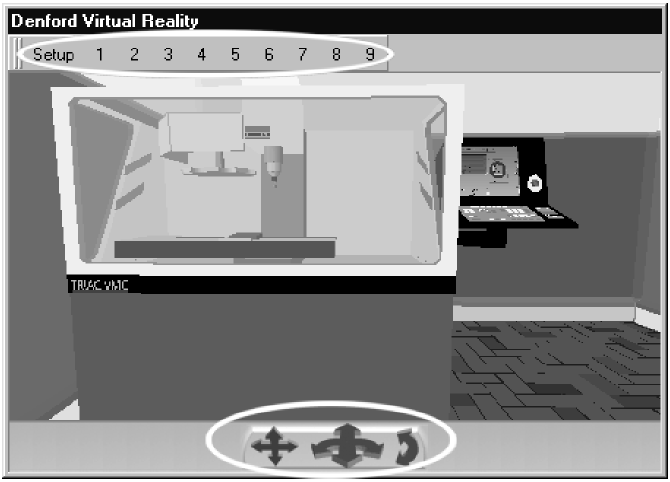
To start the Virtual Reality CNC Machine, click the [VR Machine] button, shown above, from the "Machine Control" toolbar.

The "Denford Virtual Reality" window will open. This window is used for viewing the 3D model of the VR CNC machine. Any tools present in the "Machine Tooling" window (the Automatic Tool Changer) will also be loaded, when applicable.

The "Machine Mode" window will also open. This window is used for controlling the movements of the VR CNC machine.

10: Virtual Reality Window - General Layout

The "Denford Virtual Reality" window is used to display a three dimensional representation of the CNC machine. This VR CNC machine is driven and responds in exactly the same way as its real-life counterpart, making the VR Machine Mode ideal for offline CNC training. In the example below, a Denford Triac VMC CNC Milling Machine is being controlled through virtual reality.



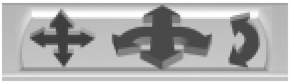
The "Denford Virtual Reality" window, shown above, is split into three basic areas:

The Viewbar (top circled) is the area of the window where preset viewpoints can be applied and the virtual reality world configured.

The Main Viewing Area, the largest area of the window, is where the objects and devices in the virtual worlds can be seen.

The Movebar (bottom circled) is the area of the window containing the controls for moving around the virtual world.

10: Moving around the Virtual Reality World

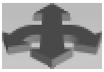


The movebar tools allow you to freely 'fly' around the virtual world, using your mouse. Try to think of the display in the Main Viewing Area as being the view from a floating camera head, which you can control using the three movement icons. Each icon is used to control a different type of movement in the virtual world.

Using the Movebar to move around the Virtual Reality World.

You can change the position of your viewpoint using the icons in the Movebar.

 : Moves the viewpoint in the vertical plane.

 : Moves the viewpoint in the horizontal plane, forwards, backwards and turning.

 : Tilts the viewpoint up and down.

Click and hold the left mouse button on one of the three movement icons, then drag the mouse in the required direction. As you drag the mouse the icon indicates the direction you are moving, and the viewpoint moves in the corresponding direction. The further you move the cursor from the icon, the faster you will move. Release the mouse button to stop the movement.

Interactive Objects

Some objects on the VR CNC machines are interactive:

Right click on the guard frame to open the machine guard.

Left click on the guard frame to close the machine guard.

Right click on the red and yellow power switch, positioned on the electrical control cabinet, to turn the CNC machine on and off.

10: Using the Predefined Viewpoints

> **Note**
>
> Many of the assigned viewpoints are designed to help during configuration of offsets.

Viewpoints are particularly useful for navigating quickly, or regaining position when lost in the virtual world. To select a viewpoint, click on the number button of the viewpoint required. Hovering the mouse cursor over a viewpoint number button will display a pop-up description of the assigned view, as shown above. A number of different viewpoints are available for each CNC machine.

For example, the Triac CNC milling machine has the following nine viewpoints:

Button [1]: Front view of the Milling Machine.

Button [2]: Right view of the cutter.

Button [3]: Front view of the cutter.

Button [4]: Left view of the cutter.

Button [5]: Front view, attached to the cutter.

Button [6]: Front view of the billet.

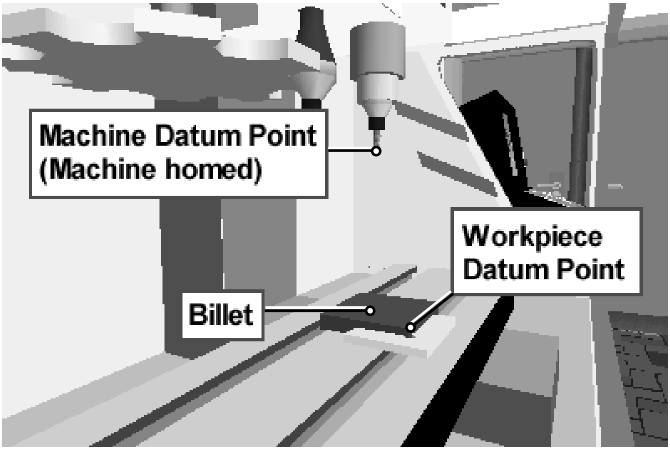
Button [7]: Left side view of the billet.

Button [8]: Plan view of the billet.

Button [9]: View of the Milling Machine power switch.

10: Using Offsets with the sample CNC Files

If you are following our tutorials, using the sample CNC files listed on pages 14 to 21, the following 6 pages describe how to locate the position of the workpiece datum and tool length offset reference point, so co-ordinate vales can be entered into the VR Milling software.



The screenshot above shows the locations of the billet, machine datum and programmed workpiece datum points on the VR CNC machine. The tool length offset point is the upper surface of the billet.

The billet in the "Denford Virtual Reality" window is shown in red, clamped next to the L-shaped yellow datum plate on the machine table.

All screenshot examples are shown using the Denford Triac VMC CNC Milling Machine as the default VR CNC machine.

Note

We do not use a temporary machine sub-table in Virtual Reality, despite the fact the sample CNC file include sections where the tool cuts completely through the billet. Naturally, if we attempted this on a real CNC machine, we would cause considerable damage to the machine table!

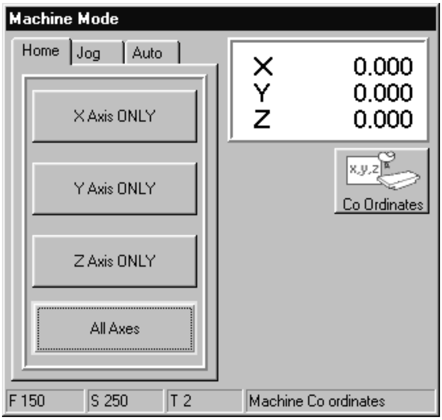
10: X and Y Workpiece Datum Alignment

The example screenshots show the process of aligning the tool with the workpiece datum position on a VR CNC machine, for the sample CNC files used in our tutorials.

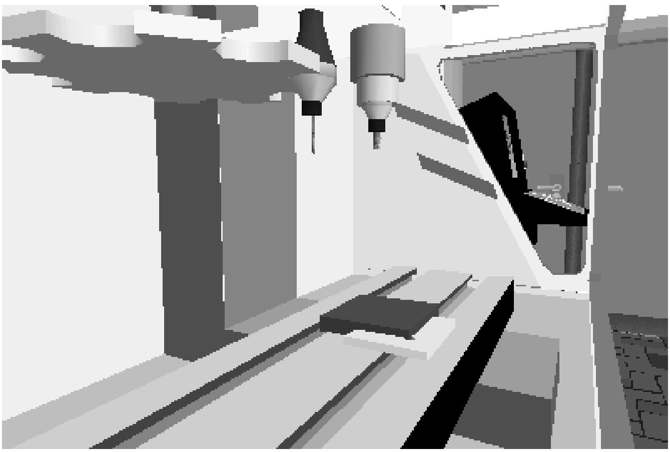
A) Home the VR CNC Machine (if required).

Note

Although the positions of workpiece datums can vary, the principles shown here apply to all cases.



Click the "Home" tab of the "Machine Mode" window. Home the VR CNC machine axes by clicking the [All Axes] button.



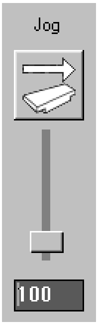
The VR CNC machine head will move to its machine datum position, as shown in the above screenshot.

10: X and Y Workpiece Datum Alignment

B) Move to the approximate workpiece datum position in the Z axis.

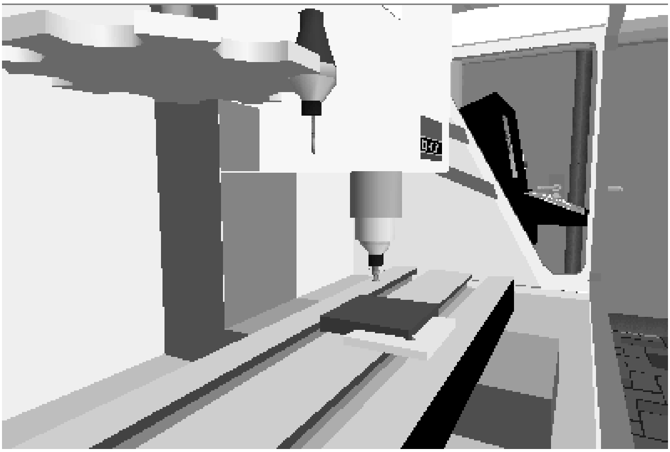
Click the "Jog" tab of the "Machine Mode" window.

Check the machine axes are set in "Jog Continuous Mode" by clicking the [Jog] button, so the button displays a straight arrow graphic, as shown below.



In "Jog Continuous Mode", the selected machine axis will move at the indicated speed when one of the machine axis movement 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 the [Units] of Measurement are set to "Inch" the rate of movement is measured using inches per minute. When the [Units] of Measurement are set to "Metric" the rate of movement is measured using millimetres per minute.

Move the VR CNC machine head down towards the red billet, using the **[Page Down]** or **[Page Up]** keys.

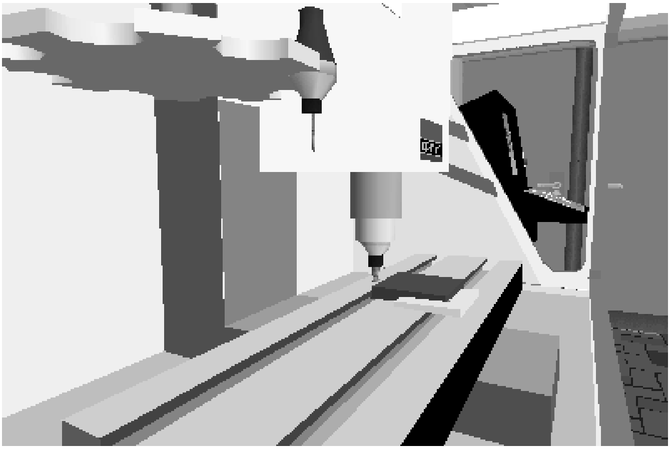


Try to position the tip of the tool about 10mm (1/2") above the surface of the billet, as shown in the screenshot above.

10: X and Y Workpiece Datum Alignment

C) Move to the approximate workpiece datum position in the X axis.

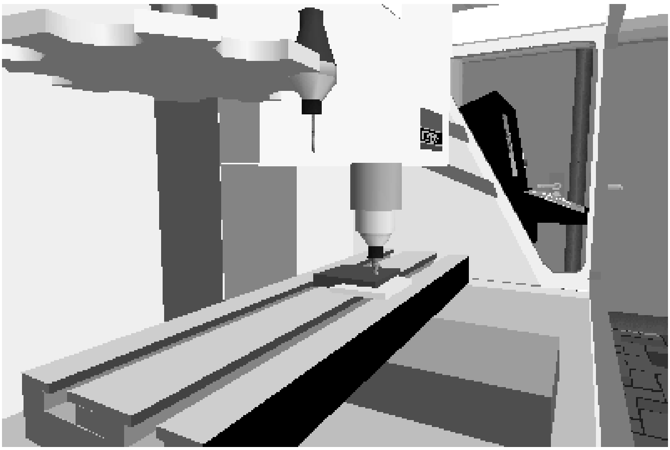
Move the tip of the tool towards the edge of the billet, using the **[Cursor Right]** or **[Cursor Left]** keys.



Try to position the tip of the tool over the X datum edge of the billet, as shown in the screenshot above.

D) Move to the approximate workpiece datum position in the Y axis.

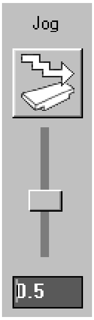
Move the tip of the tool towards the corner of the billet, using the **[Cursor Up]** or **[Cursor Down]** keys.



Try to position the tip of the tool over the Y datum edge of the billet, as shown in the screenshot above.

10: X and Y Workpiece Datum Alignment

E) Move to the exact intersection of the X and Y workpiece datum edges.



Check the machine axes are set in "Jog Step Mode" by clicking the [Jog] button, so the button displays a stepped arrow graphic, as shown left.

In "Jog Step Mode", the selected machine axis will move one indicated increment, each time the selected axis 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 (set as 0.5 units in the example left). When the [Units] of Measurement are set to "Inch" the rate of movement is measured using inches per minute. When the [Units] of Measurement are set to "Metric" the rate of movement is measured using millimetres per minute.

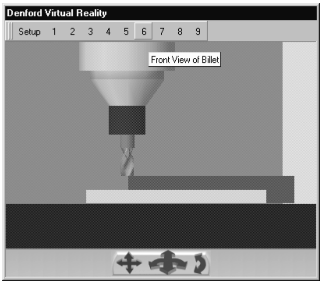
Move the tip of the tool so the centre of the cutter is positioned exactly over the intersection of the X and Y workpiece datum edges, using all six axis movement keys:

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

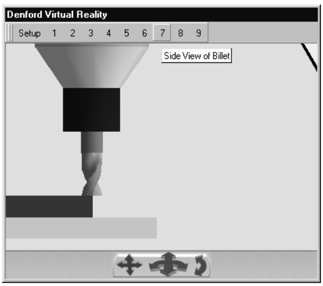
Y Axis: **[Cursor Up]** or **[Cursor Down]**

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

Use viewpoint [6] - the front view of the billet and viewpoint [7] - the side view of the billet, to help align the tool. The tool does not need to be positioned so it is touching the billet surface.



Left: Aligning the centre of the tool with the Y workpiece datum edge (ie, front view of billet shown).



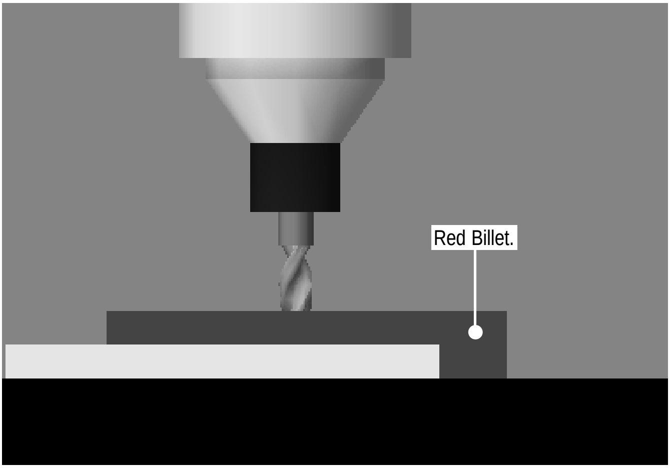
Left: Aligning the centre of the tool with the X workpiece datum edge (ie, side view of billet shown).

Now transfer the X and Y values into the Work Piece Offsets file.

10: Tool Length Offset Reference Point Alignment

> **Note**
>
> Remember, every tool must be manually aligned against the same tool length offset reference point, then its individual values transferred to its own tool length offset file.

This page shows the process of aligning the tool with the tool length reference point on a VR CNC machine, for the sample CNC files used in our tutorials.



> **Note**
>
> If you are running through the tutorials using the sample CNC files on a VR CNC machine:
>
> Pages 78 to 79 show the procedure for entering the single X co-ordinate offset value.
>
> Pages 80 to 81 show the procedure for entering the single Y co-ordinate offset value.
>
> Pages 82 to 86 show the procedure for entering the two different Z co-ordinate tool length offset values.

The position of the tool length offset reference point is anywhere on the upper surface of the red billet, as shown above. This position reveals the Z value required for the tool length offset file, but only for the tool being held in the machine head.

Move the CNC machine head down towards the surface of the billet, using the **[Page Down]** or **[Page Up]** keys. Use "Jog Continuous Mode" by clicking the [Jog] button, so the button displays a straight arrow graphic.

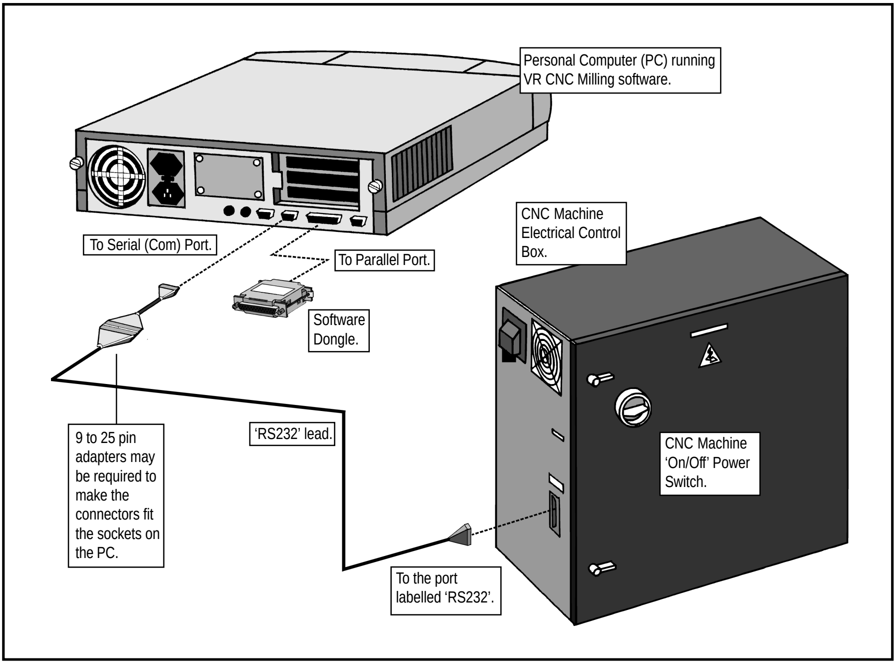
When the tip of the tool is close to the surface of the billet, switch to "Jog Step Mode" by clicking the [Jog] button, so the button displays a stepped arrow graphic. This will allow the tool to be controlled with greater accuracy.

In "Jog Step Mode", the selected machine axis will move one indicated increment, each time the selected axis 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. When the [Units] of Measurement are set to "Inch" the rate of movement is measured using inches per minute. When the [Units] of Measurement are set to "Metric" the rate of movement is measured using millimetres per minute.

Move the tool so the cutting tip is just making contact with the surface of the billet, as shown in the screenshot above.

Now enter the value into the Z tool length offset file, then repeat the process for all remaining tools to be used with the CNC file.

11: Connecting to a real CNC Machine



Data is sent and received between your computer and your CNC machine using an RS232 lead.

One end of the RS232 lead fits into the serial (COM) port of the computer running the VR CNC Milling for Windows software. Computer serial ports are the small plugs, each containing two rows of 5 and 4 pins, usually positioned on the back panel of your computer.

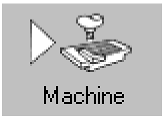
The opposite end of the RS232 lead fits into the serial (COM) port on your CNC machine casing or electrical control box.

Note

More detailed information regarding CNC hardware connections are provided in your separate CNC machine manual.

11: Starting a real CNC Machine

1) Ensure the RS232 lead is fitted securely between the computer and the CNC machine.
2) Switch on the CNC machine.
3) Power up the computer and start the VR CNC Milling software.
4) Before connecting to a real 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. 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.
    - Check that the position of any tools present in a real Automatic Tool Changer, when fitted, match the exact pocket numbers of the tools configured in the "MachineTooling" window.
5) To start the real CNC Machine, click the [Machine] button, shown below, from the "Machine Control" toolbar.



The "Machine Mode" window will open. This window is used for controlling the movements of the real CNC machine.

11: Using the sample CNC files

If you are running through the tutorials using the sample CNC files, this page describes what equipment will be required in order to manufacture the sample designs.

Running the CNC file: Metric.fnc

Tooling required: 2mm slot cutter set as tool number 2

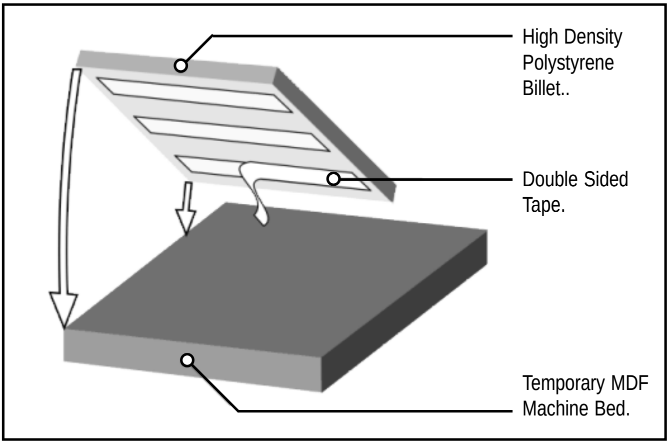
4mm slot cutter set as tool number 1

Billet Material: High Density Polystyrene

Billet Dimensions: X (length) 60mm, Y (width) 60mm, Z (height) 2mm

Temporary machine sub-table: MDF (medium Density Fibreboard) or similar. Approx. 80mm x 80mm x 10mm

The 2mm slot cutter will cut completely through the billet, so the billet must be mounted on a temporary machine sub-table, using double sided tape, as shown in the diagram below.



Running the CNC file: Inch.fnc

Tooling required: 1/8" (0.125) slot cutter set as tool number 2

3/16" (0.1875) slot cutter set as tool number 1

Billet Material: High Density Polystyrene

Billet Dimensions: X (length) 2.5", Y (width) 2.5", Z (height) 1/8" (0.125)

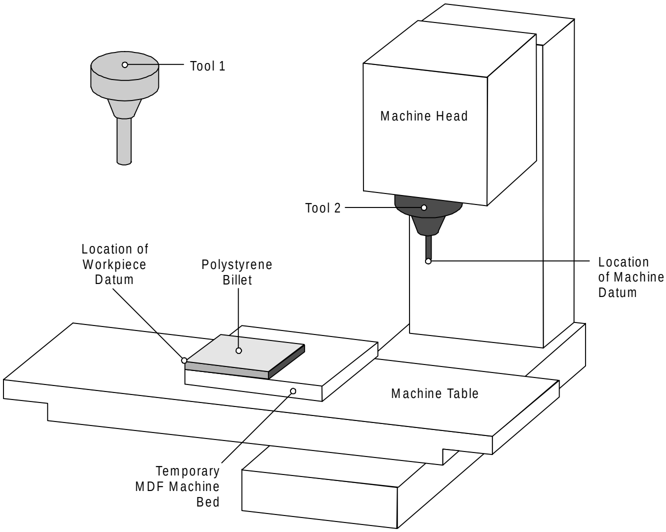
Temporary machine sub-table: MDF (medium Density Fibreboard) or similar. Approx. 3.5" x 3.5" x 0.5"

The 1/8" (0.125) slot cutter will cut completely through the billet, so the billet must be mounted on a temporary machine sub-table, using double sided tape, as shown in the diagram above.

11: Using Offsets with the sample CNC Files

If you are following our tutorials, using the sample CNC files listed on pages 14 to 21, the following 6 pages describe how to locate the position of the workpiece datum and tool length offset reference point, so co-ordinate vales can be entered into the VR Milling software.

Note - In the diagram below, all axes have been homed.



Important !

Ensure that a temporary machine sub-table is used when running the sample CNC files with a real CNC machine.

You must use a temporary machine sub-table because the sample CNC files include sections where the tool cuts completely through the billet. Naturally, if we attempted to use the sample CNC file without this hardware, we would cause considerable damage to the actual machine table! We recommend that you use MDF (medium density fibreboard) or a similar smooth flat material for the temporary machine sub-table. The plastic billet should be held in position using double sided tape, as shown on page 101.

11: X and Y Workpiece Datum Alignment

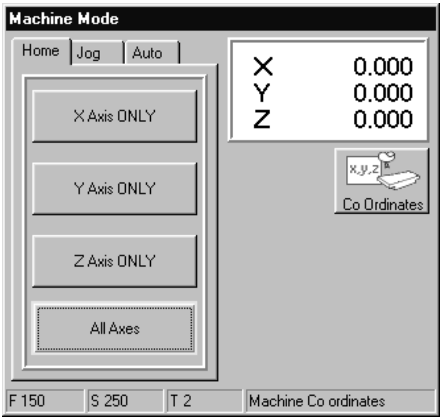
The example diagrams show the process of aligning the tool with the workpiece datum position on a real CNC machine, for the sample CNC files used in our tutorials.

> **Note**
>
> Although the positions of workpiece datums can vary, the principles shown here apply to all cases.

At this stage only the temporary MDF machine bed needs to be fitted to the CNC machine table, not the plastic billet.

Try to use smallest cutting tool for configuring the offsets. You will find it much easier to align the centre of the smallest tool with the workpiece datum edges. If you are using our sample CNC files, this will be tool number 2.

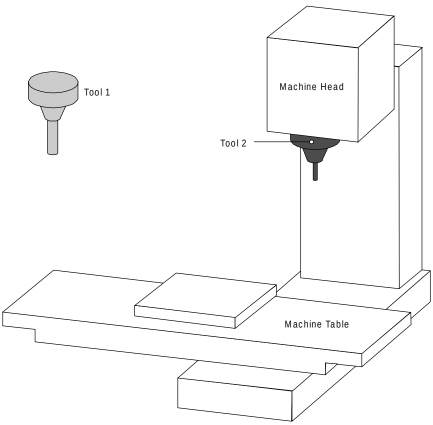
A) Home the CNC Machine (if required).



Click the "Home" tab of the "Machine Mode" window.

Home the CNC machine axes by clicking the [All Axes] button.

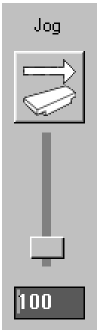
The CNC machine head will move to its machine datum position, as shown below.



B) Move to the approximate workpiece datum position in the Z axis.

Click the "Jog" tab of the "Machine Mode" window.

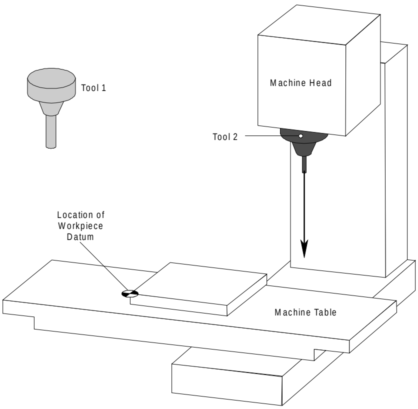
Check the machine axes are set in "Jog Continuous Mode" by clicking the [Jog] button, so the button displays a straight arrow graphic, as shown below.



In "Jog Continuous Mode", the selected machine axis will move at the indicated speed when one of the machine axis movement 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 the [Units] of Measurement are set to "Inch" the rate of movement is measured using inches per minute. When the [Units] of Measurement are set to "Metric" the rate of movement is measured using millimetres per minute.

Move the CNC machine head down towards the temporary MDF machine bed, using the **[Page Down]** or **[Page Up]** keys.

Try to position the tip of the tool about 10mm (1/2") above the surface the temporary MDF machine bed.

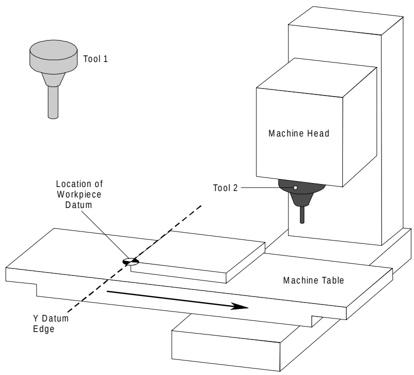


11: X and Y Workpiece Datum Alignment

C) Move to the approximate position of the Y datum edge.

Move the tip of the tool towards the Y datum edge of the temporary MDF machine bed, using the **[Cursor Right]** or **[Cursor Left]** keys.

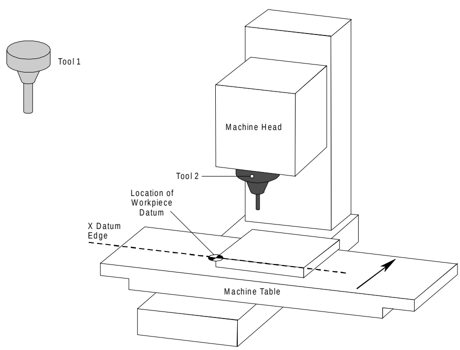
Try to position the tip of the tool as near as possible over the Y datum edge of the temporary MDF machine bed.



D) Move to the approximate position of the X datum edge.

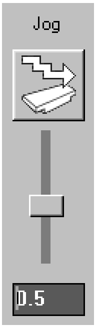
Move the tip of the tool towards the X datum edge of the temporary MDF machine bed, using the **[Cursor Up]** or **[Cursor Down]** keys.

Try to position the tip of the tool as near as possible over the X datum edge of the temporary MDF machine bed.



11: X and Y Workpiece Datum Alignment

E) Move to the exact intersection of the X and Y workpiece datum edges.



Check the machine axes are set in "Jog Step Mode" by clicking the [Jog] button, so the button displays a stepped arrow graphic, as shown left.

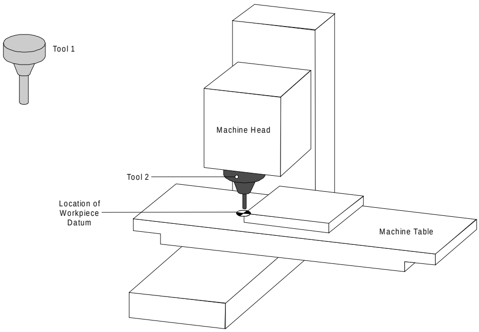
In "Jog Step Mode", the selected machine axis will move one indicated increment, each time the selected axis movement key is pressed. The slider bar can be moved to set jog increments of 0.01, 0.1, 0.5, 1, 5 and 10 units (set as 0.5 units in the example left). When the [Units] of Measurement are set to "Inch" the rate of movement is measured using inches per minute. When the [Units] of Measurement are set to "Metric" the rate of movement is measured using millimetres per minute.

Move the tip of the tool so the centre of the cutter is positioned exactly over the intersection of the X and Y workpiece datum edges, using all six axis movement keys:

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

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

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



Note

The tool does not need to be positioned so it is touching the billet surface.

F) Now transfer the X and Y values into the Work Piece Offsets file.

11: Tool Length Offset Reference Point Alignment

Note

Remember, every tool must be manually aligned against the same tool length offset reference point, then its individual values transferred to its own tool length offset file.

Note

If you are running through the tutorials using the sample CNC files on a real CNC machine:

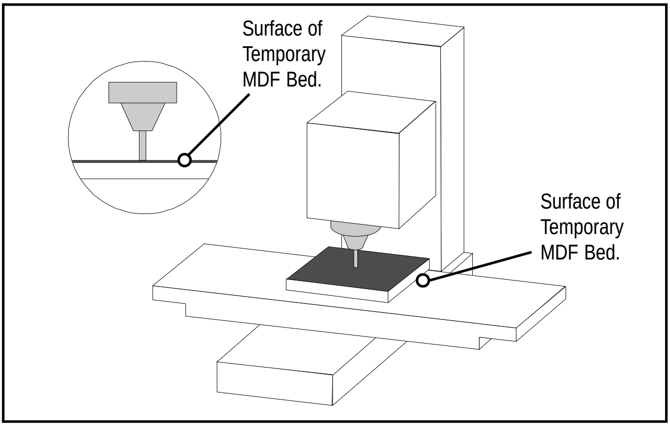
Pages 78 to 79 show the procedure for entering the single X co-ordinate offset value.

Pages 80 to 81 show the procedure for entering the single Y co-ordinate offset value.

Pages 82 to 86 show the procedure for entering the two different Z co-ordinate tool length offset values.

Pages 87 to 88 shows the procedure for entering the single Z co-ordinate value, used as a global Z offset to account for billet thickness. This is because the surface of the temporary MDF bed was used as the tool length offset reference point, rather than the surface of the billet.

This page shows the process of aligning the tool with the tool length reference point on a VR CNC machine, for the sample CNC files used in our tutorials.



The position of the tool length offset reference point is anywhere on the upper surface of the temporary MDF machine bed. This position reveals the Z value required for the tool length offset file but only for the tool being held in the machine head. A global Z offset value must also be added into the workpiece offset file to account for the billet thickness.

Move the CNC machine head down towards the temporary MDF machine bed, using the **[Page Down]** or **[Page Up]** keys. Use "Jog Continuous Mode" by clicking the [Jog] button, so the button displays a straight arrow graphic.

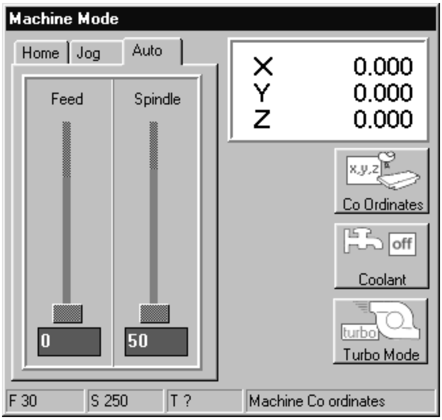
When the tip of the tool is close to the surface of the temporary MDF machine bed, switch to "Jog Step Mode" by clicking the [Jog] button, so the button displays a stepped arrow graphic. This will allow the tool to be controlled with greater accuracy.

In "Jog Step Mode", the selected machine axis will move one indicated increment, each time the selected axis 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. When the [Units] of Measurement are set to "Inch" the rate of movement is measured using inches per minute. When the [Units] of Measurement are set to "Metric" the rate of movement is measured using millimetres per minute.

Move the tool so the cutting tip is just making contact with the surface of the temporary MDF bed. Now enter the value into the Z tool length offset file, then repeat the process for all remaining tools to be used with the CNC file.

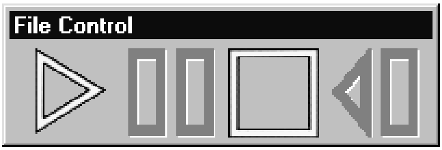
12: Running a CNC file on a CNC Machine.

Auto Mode.



The "Auto" tab is used for controlling the CNC machine when running a CNC file.

When potentiometer controls are fitted to the CNC machine, allowing direct manual override of both the feedrate and spindle speed, both the software "Feed" and "Spindle" control panels will be disabled as shown above.



Note

The numerical figures depicted on any screenshots will differ according to the CNC machine type, the units of measurement setting for the CNC Milling software and any offsets being used on your computer system.

To start 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 above.

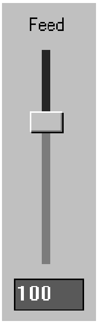
12: Feedrate & Spindle Speed Overrides.

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 (when available).

Note

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



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.

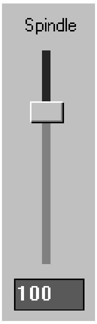
To change the feedrate value, click and hold down the left mouse button on the slider bar, then drag the slider bar up or down to the new position.

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 (when available).

Note

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



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 100-150% 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: Homing the Machine

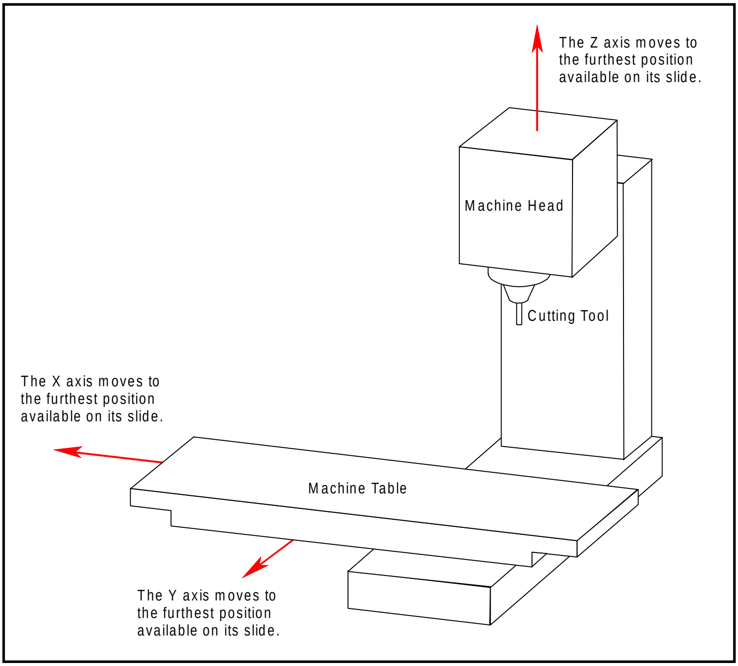
Immediately after being switched on, all three axes of the CNC machine must be homed.

When you home the CNC machine, all three axes will move to the furthest positions available on their slides, as shown in the diagram below.

Homing the CNC machine defines:

- The co-ordinate based system used for plotting any programmed movements - this gives us a working envelope for the cnc machine.
- The machine datum - the zero reference point for the CNC machine.

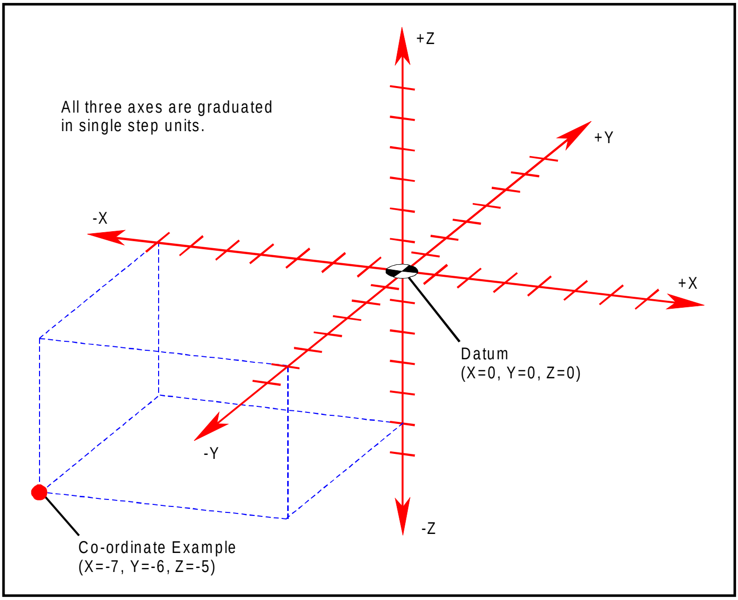
In addition to homing the CNC machine after it has been switched on, it is also recommended that the CNC machine is homed after loading or configuring offsets.



13: The Co-ordinate based Grid System

Precise points are plotted on the CNC machine using the positions of the X, Y and Z axes. These X, Y and Z values relate to a three dimensional grid, as shown in the example below.

The zero point of this grid is called the datum. The graduated gridlines represent the directions of the three CNC machine axes.

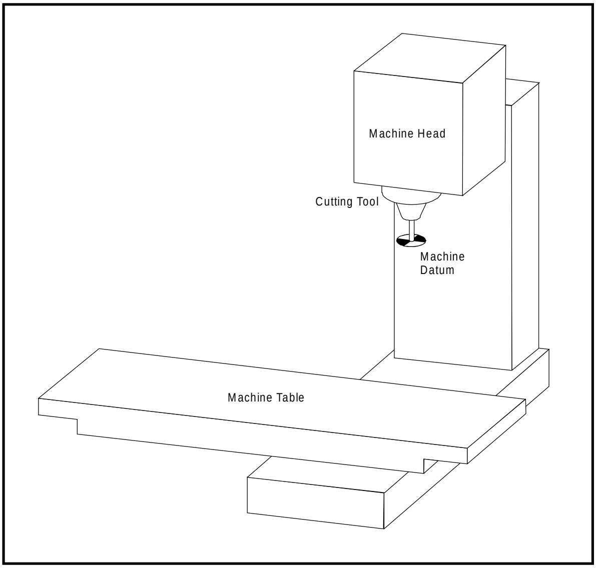


13: The Machine Datum

The machine datum, or home position, is the zero reference point of the CNC machine. It's the point from which all co-ordinates we load or program are calculated. If there are no offsets loaded and we begin to run a CNC program, the machine datum is the location from which all machining co-ordinates are taken.

The position of the machine datum is set by your CNC machine manufacturer and can never be moved, since it defines the physical movement capability of the machine.

If we place a tool in the machine head, then home all three axes, we can describe the machine datum as being the centre of the cutting tool tip. The position of the machine datum, when using machine co-ordinates, will be X=0, Y=0 and Z=0, as shown in the diagram below.

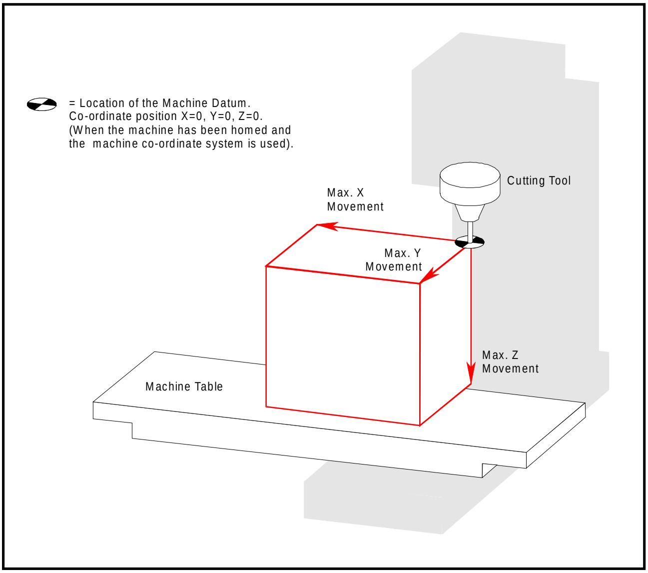


13: The CNC Machine Working Envelope

The diagram below shows the home position of the CNC machine.

The block represents the maximum working envelope of the machine, effectively the full length of movement in each of the three axes. This is the largest possible size of workpiece that the CNC machine could manage to cut, with the tool currently held in the machine head.

However, workpieces larger than this size can be accommodated, so long as they don't foul the insides of the machine cabinet when the machine table moves.



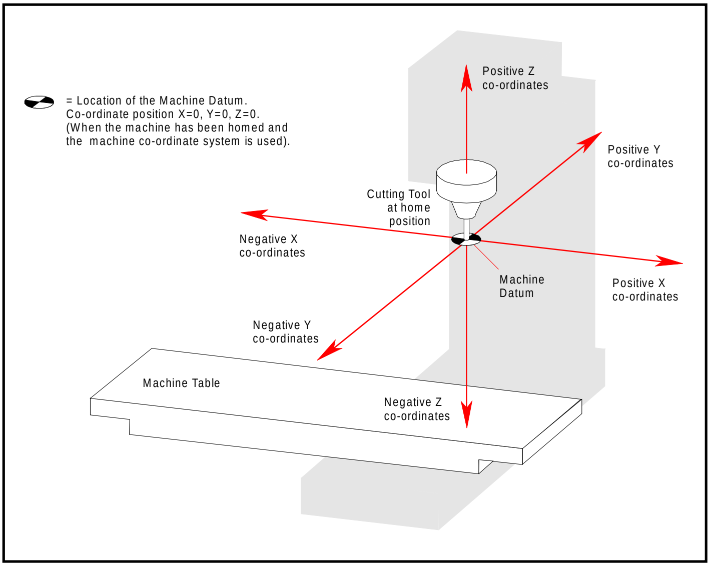
13: Machine Co-ordinates Display Mode

Co-ordinate System Display Modes

The CNC machine display co-ordinate movements using two different modes:

1) **Machine Co-ordinates**: Any co-ordinate values shown relate to the fixed machine datum. The co-ordinate display always shows the true position of the machine. The machine datum position is set by your CNC machine manufacturer and can never be moved, since it defines the physical movement capability of the machine.

When running in Machine Co-ordinates Mode, the Machine Datum is defined with the position X=0, Y=0, Z=0.

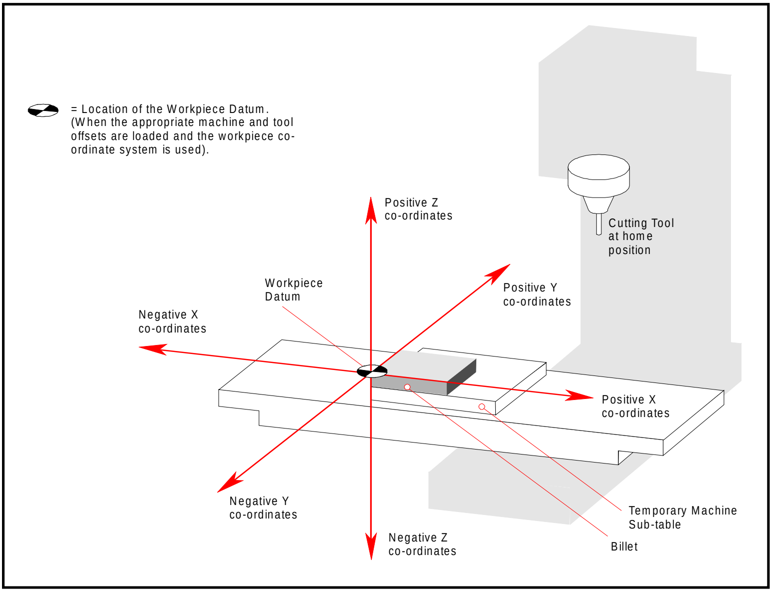


13: Workpiece Co-ordinates Display Mode

Co-ordinate System Display Modes

2) **Workpiece Co-ordinates**: Any co-ordinate values shown relate to the programmed workpiece datum, described through use of the offset facility. The workpiece datum is set by the operator as the location from which all machining co-ordinates will be taken. Offsets temporarily shift the entire co-ordinate based grid system of the machine, as shown below.

When running in Workpiece Co-ordinates Mode, the Workpiece Datum is defined with the position X=0, Y=0, Z=0.

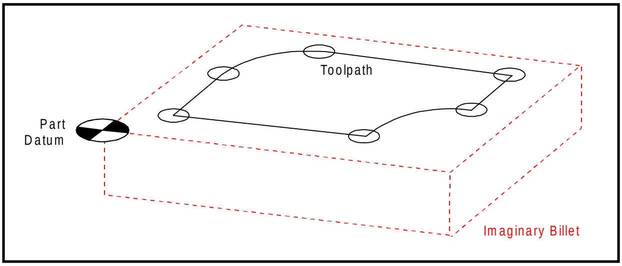


13: What are Offsets?

When we write a CNC program, all co-ordinates used for describing the shape of the part are stated relative to a zero reference, called the part datum.

The part datum should be positioned in a convenient location with respect to the actual size of the billet you intend to use, as shown below. This position will need to be identified later on the real billet.

We recommend that the Z co-ordinate of the part datum is set on the upper surface of the billet you intend to use. In doing so, any negative Z values programmed will indicate that the tool is cutting into the billet, any positive Z values programmed will indicate that the tool is clear from the billet.



However, the part datum can be positioned anywhere. It could be positioned in one of the corners of a part design drawing. On a largely circular design it could be positioned in the centre of one of its circles. In most CAD/CAM software packages it may be set automatically when the CNC program is generated. For example, Denford MillCAM Designer positions the part datum in the front upper lefthand corner of the billet

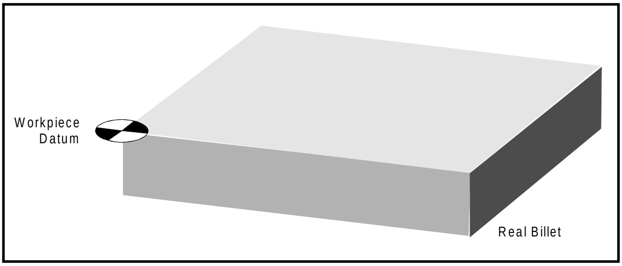
The CNC machine also has a zero reference, called the machine datum. If no offsets are loaded, our CNC program will use this position as the start location from which all machining co-ordinates are taken.

Offsets are used to establish the location of the workpiece datum on the real billet. The workpiece datum is the position where we want any physical machining co-ordinates to be taken from. Using the Offsets facility, we can temporarily shift the entire co-ordinate based grid system of the CNC machine. We must move the three dimensional grid, so the position of the workpiece datum registers as zero, rather than the position of the machine datum.

It is important to note that the physical position of the machine datum does not move, since it is permanently fixed. Remember, it's the co-ordinate based grid system of the CNC machine that temporarily moves, giving the illusion that the machine datum itself has moved.

13: What are Offsets?

Note that the workpiece datum must be positioned on the real billet in the same place as the part datum was positioned with respect to the imaginary billet. Compare the position of the workpiece datum in the diagram below with the position of the part datum in the diagram on page 116 - they are identical. If these datums were not identical, the part would be machined in the wrong place on the real billet.



However, that there may be occasions when you want to set the workpiece datum in different positions. For example, if you have a number of different CNC files that combine to make a part. If these CNC files don't use a common part datum between them, you would have to set workpiece datums individually so each CNC file machines in the correct area of the real billet. Or you might want to machine the same CNC file a number of times on the same billet to produce identical parts. Then you would need to set separate workpiece datums for the position of each part.

Offsets are very important because without them, the CNC machine will not know where to begin cutting on the billet. Offsets must always be configured before manufacturing our part. However, once you configure and save an offset file, the same file may be used over and over again, as 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.

Bearing this in mind, offset files can be created using titles for specific projects or student groups to save setup time during lectures.

13: Types of Offset

Offsets are configured in two distinct stages:

1) **Workpiece Offsets**

A workpiece offsets file contains three values relating to any co-ordinate shift required along the direction of the X, Y and Z axes.

Without any workpiece offsets file loaded, the zero co-ordinate of the CNC machine is the location of the machine datum.

Workpiece offsets are used to temporarily shift the entire co-ordinate based grid system of the machine, setting a new location for the zero co-ordinate of the CNC machine.

Any values entered into a workpiece offsets file are global, ie, they are used by all tool profiles, irrespective of their different lengths.

2) **Tool Length Offsets**

Tool Length Offsets apply along the direction of the Z axis only.

They allow a variety of tool profiles to be used together on the same CNC program, by offsetting their differences in length against a common fixed reference point.

Each tool has its own individual tool length offset value that is only applied when that particular tool is used on the CNC machine.

When tool length offsets are used, their values are combined with any Z value saved in the loaded workpiece offsets file, to calculate the Z co-ordinate describing the location of the workpiece datum.

13: Configuring Offsets for more than One Tool

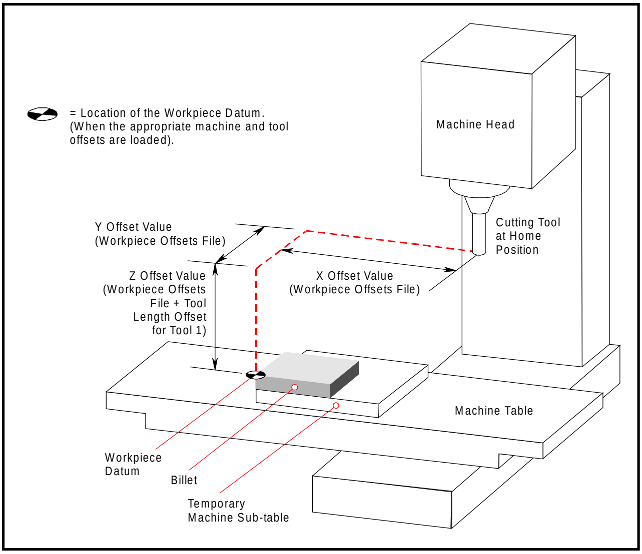
Configuring Offsets when using two or more Tools.

This sequence briefly explains the theory behind configuring the offsets, when using two or more tools with a CNC program.

The position of the workpiece datum is defined by the values in the workpiece offsets file and the appropriate tool length offset values relating to each of the tools you want to use.

In this example, tool profile 1 has a greater length compared with tool profile 2.

Configuring the position of the workpiece datum for tool 1.



1) **Defining the X position of the workpiece datum for tool 1.**

When the tool 1 is at the home position, the X offset position is defined as the distance between the centre of the tool and the workpiece datum, parallel to the X axis. The X offset is the value entered into the X component of the workpiece offsets file, this will be a negative value. Since workpiece offsets are global, this value is applied to every tool profile.

2) **Defining the Y position of the workpiece datum for tool 1.**

When the tool 1 is at the home position, the Y offset position is defined as the distance between the centre of the tool and the workpiece datum, parallel to the Y axis. The Y offset is the value entered into the Y component of the workpiece offsets file, this will be a negative value. Since workpiece offsets are global, this value is applied to every tool profile.

3) **Defining the Z position of the workpiece datum for tool 1.**

When the tool 1 is at the home position, the Z offset position is defined as the distance between the tip of the tool and the workpiece datum, parallel to the Z axis. The Z offset is a combination of the tool length offset value for tool profile 1 and the value entered into the Z component of the workpiece offsets file.

If the tool length offset reference point is selected as the surface of the billet, then:

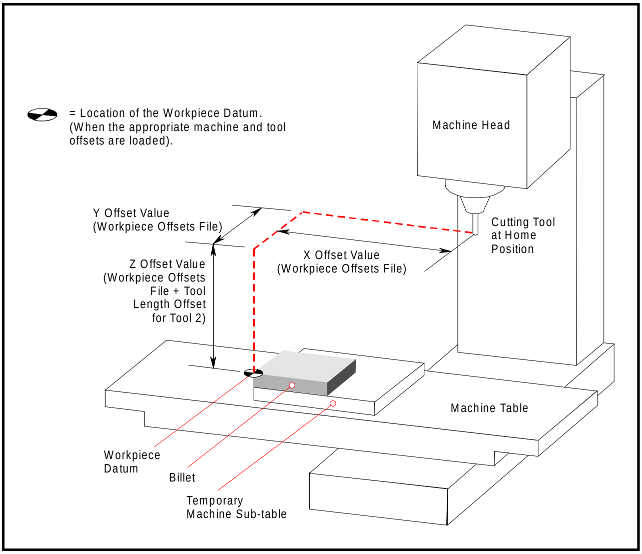
(workpiece datum position) = (Z workpiece offset value, this will be zero) + (tool length offset value for tool 1, this will be a negative value)

Notice that in the above equation, the Z workpiece offset value will be zero because the tool length offset reference point and the workpiece datum point share the same Z axis co-ordinate position.

If the tool length offset reference point is selected as the top surface of the temporary machine sub-table, then:

(workpiece datum position) = (Z workpiece offset value, this will be the thickness of the billet, a positive value) + (tool length offset value for tool 1, this will be a negative value).

Configuring the position of the workpiece datum for tool 2.



13: Configuring Offsets for more than One Tool

Remember, in this example, tool profile 1 has a greater length compared with tool profile 2.

1) **Defining the X position of the workpiece datum for tool 2.**

When the tool 2 is at the home position, the X offset position is defined as the distance between the centre of the tool and the workpiece datum, parallel to the X axis. We do not need to set any X offset value for tool 2, since this distance is exactly the same as the X offset for tool 1. This is why the X offset value for tool 1 was entered into the workpiece offsets file - since it is global it will be used by both tools 1 and 2.

2) **Defining the Y position of the workpiece datum for tool 2.**

When the tool 1 is at the home position, the Y offset position is defined as the distance between the centre of the tool and the workpiece datum, parallel to the Y axis. We do not need to set any Y offset value for tool 2, since this distance is exactly the same as the Y offset for tool 1. This is why the Y offset value for tool 1 was entered into the workpiece offsets file - since it is global it will be used by both tools 1 and 2.

3) **Defining the Z position of the workpiece datum for tool 2.**

When the tool 2 is at the home position, the Z offset position is defined as the distance between the tip of the tool and the workpiece datum, parallel to the Z axis. The Z offset is a combination of the tool length offset value for tool profile 2 and the value entered into the Z component of the workpiece offsets file.

If the tool length offset reference point is selected as the surface of the billet, then:

(workpiece datum position) = (Z workpiece offset value, this will be zero) + (tool length offset value for tool 2, this will be a negative value)

Notice that in the above equation, the Z workpiece offset value will be zero because the Z length tool offset reference point and the workpiece datum point share the same Z axis co-ordinate position.

If the tool length offset reference point is selected as the top surface of the temporary machine sub-table, then:

(workpiece datum position) = (Z workpiece offset value, this will be the thickness of the billet, a positive value) + (tool length offset value for tool 2, this will be a negative value)

Notice that the numerical value of the tool length offset for tool profile 2 will be larger than the numerical value of the tool length offset for tool profile 1. This compensates for the fact that tool profile 1 is longer than tool profile 2.

13: Configuring Tool Length Offsets

Configuring Tool Length Offsets.

When a tool is called into use on a CNC machine, the Z component of the workpiece datum is a numerical combination of the tool length offset value for tool profile being used and the value entered into the Z component of the workpiece offsets file.

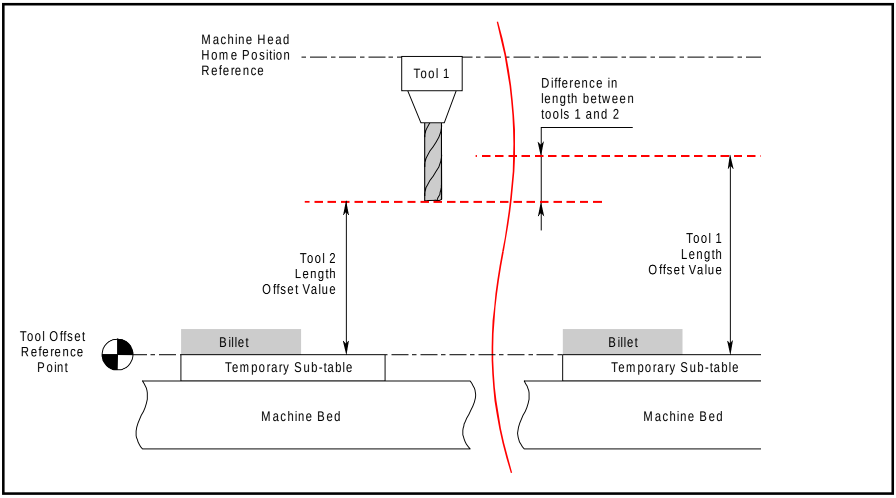
The tool length offset facility is used to compensate for the differences in length between all the tools used with the CNC program.

How are Tool Length Offsets used?

This sequence briefly explains the theory behind configuring the tool length offsets.

The Z position of the workpiece datum is defined by the Z value in the workpiece offsets file and the appropriate tool length offset values relating to each of the tools you want to use.

In this example, tool profile 1 has a greater length compared with tool profile 2.



Define a Tool Offset Reference Point.

All tool length offsets are configured against a common reference point. When values are entered into each individual tool length offset file, each tool will use this reference point as their zero co-ordinate along the Z axis.

Choose a tool offset reference point that's easy to reach with all the tool profiles you intend to use. The tool offset reference point could be the top surface of the billet or the top surface of a temporary machine sub-table. In the above example, we have chosen the top surface of the temporary machine sub-table as the tool offset reference point.

13: Configuring Tool Length Offsets

Define the tool length offset value for tool profile 1.

When the tool 1 is at the home position, the tool length offset value will be the distance between the tip of tool 1 and the chosen tool offset reference point. In our example, we have chosen the upper surface of the temporary machine sub-table as the tool offset reference point.

Define the tool length offset value for tool profile 2.

When the tool 2 is at the home position, the tool length offset value will be the distance between the tip of tool 2 and the same tool offset reference point used for tool 1.

Notice that the numerical value of the tool length offset for tool profile 2 will be larger than the numerical value of the tool length offset for tool profile 1. This compensates for the fact that tool profile 1 is longer than tool profile 2. This compensation allows the machine to cut in the correct place on the billet, irrespective of whether it is using tool 1 or 2.

To configure the tool length offset for a tool profile:

1) Move the cutting tool so its tip is just touching the chosen tool offset reference point.
2) Click the [Z Offset] button in the "Jog Mode" panel of the "Machine Mode" window, to display the "Set Offset" window.
3) Enter the tool length offset value into the "Set Offset" window dialogue box, if required. The VR CNC Milling software will automatically suggest a value for the tool currently held in the machine head - this suggested value will set the position of the tool tip as the zero reference point.
4) Click the [OK] button to confirm the tool length offset value.
5) Repeat steps 1 through 4 for all remaining tools, ensuring that the same tool offset reference point is used from step 1. Note that the data in the "Set Offset" window will only be applicable to the tool profile currently held in the machine head.

How is the Tool Length Offset value used?

Remember that the Z position of the workpiece datum is a numerical combination of the tool length offset value for tool profile being used and the value entered into the Z component of the workpiece offsets file.

If the Z value in the workpiece offset file was left as zero, the Z component of the workpiece datum would be in the same position as the tool offset reference point.

In our example, if the Z component of the workpiece datum needed to be the upper surface of the billet, a value equal to the thickness of the billet would be manually entered into the Z dialogue box of the workpiece offset file. This would shift the workpiece datum up to the correct position.

Note that the value manually entered into the Z dialogue box of the workpiece offset file will be sign sensitive. A positive value will move the workpiece datum up the Z axis, whilst a negative value will move the workpiece datum down.

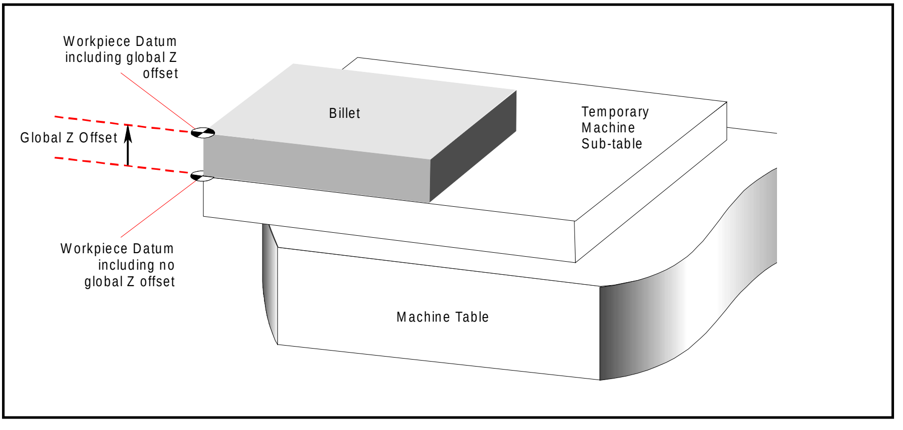
13: Configuring a Z Offset for Material Thickness

Using Workpiece Offset Z Values as Material Thicknesses.

If you find that you always seem to place the real billet and its temporary machine sub-table in exactly the same position on the machine table, then this sequence can be used so offsets need only be configured once, so long as the following holds true:

- The position of your temporary machine sub-table on the machine table never changes.
- The X and Y workpiece datum position never changes, ie, the X and Y edges of the real billet always align with the X and Y edges of the temporary machine sub-table.
- The thickness of the temporary machine sub-table never changes.
- The part datum, defined in your CNC files, is always positioned in the top lefthand corner of the proposed billet (note that this position can be automatically chosen as the part datum in CAD/CAM software packages, such as Denford MillCAM Designer). We recommend this position since any negative Z values in the CNC file will show that the tool is cutting into the billet.
- All available tools have their tool length offsets configured against the same tool length offset reference point.

This sequence effectively sets the workpiece datum at the top lefthand corner of the temporary machine sub-table. Then, by entering the thickness of the billet into the Z component of the workpiece offsets file, you can move the workpiece datum to the top lefthand corner of the real billet.



13: Configuring a Z Offset for Material Thickness

Remember that when using two or more tool profiles with a CNC file, the Z position of the workpiece datum is a numerical combination of the tool length offset value for tool profile being used and the value entered into the Z component of the workpiece offsets file. The workpiece datum is the position where any machining co-ordinates are read from.

Set the X and Y components of the workpiece datum against the top lefthand corner of the temporary machine sub-table. The X and Y offset values are entered in the workpiece offset file. You only need to do this using one tool profile. Remember, values in a workpiece offset file are global - they will be used by all other tool profiles.

Now, set all your individual tool length offsets against a common reference point. This will be the top surface of the temporary machine sub-table.

You have now set the workpiece datum in the top lefthand corner of the temporary machine sub-table, for all your tool profiles. We call this position the 'workpiece datum including no global Z offset' in our example.

Remember that the part datum, defined in your CNC files, is always positioned in the top lefthand corner of the proposed billet. So when we place the real billet on the temporary machine sub-table, you must shift the workpiece datum further up the Z axis so it aligns with the top lefthand corner of the real billet. You do this by manually entering a value equal to the thickness of the billet into the Z dialogue box of the workpiece offsets file. Note that this value will be sign sensitive - a positive value will move the workpiece datum up, whilst a negative value will move the workpiece datum down. We call this value the 'global Z offset', since it will be used by all tool profiles.

The workpiece datum is now in the correct position, ready for machining to take place. We call this position the 'workpiece datum including global Z offset' in our example.

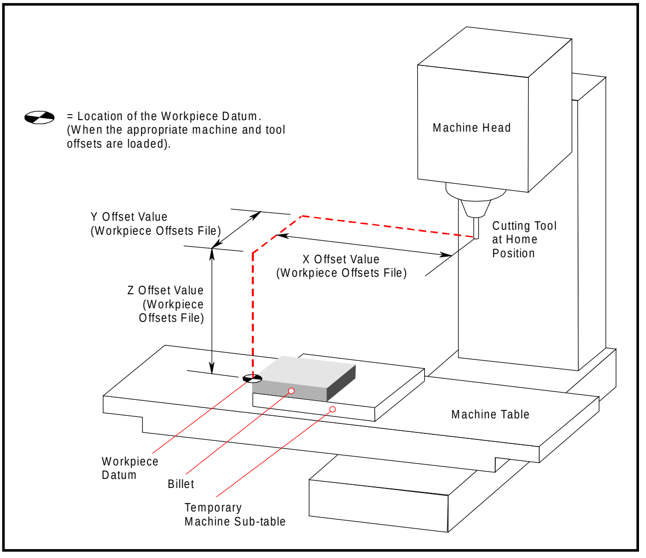
In the future, if you load a CNC file that uses a different thickness of billet, all you have to do is change the Z value in the workpiece offsets file. This new value should be the thickness of the new billet. No other changes are necessary to the workpiece offsets file or tool length offsets.

13: Configuring Offsets for a Single Tool

Configuring Offsets when using only one Tool.

This sequence briefly explains the theory behind configuring the offsets, when using only one tool with a CNC program.

This sequence can save considerable time when configuring the offsets, when using only one tool.



1) **Defining the X position of the workpiece datum.**

When the tool is at the home position, the X offset position is defined as the distance between the centre of the tool and the workpiece datum, parallel to the X axis. The X offset is the value entered into the X component of the workpiece offsets file, this will be a negative value.

2) **Defining the Y position of the workpiece datum.**

When the tool is at the home position, the Y offset position is defined as the distance between the centre of the tool and the workpiece datum, parallel to the Y axis. The Y offset is the value entered into the Y component of the workpiece offsets file, this will be a negative value.

3) **Defining the Z position of the workpiece datum.**

When the tool is at the home position, the Z offset position is defined as the distance between the centre of the tool and the workpiece datum, parallel to the Z axis. The Z offset is the value entered into the Z component of the workpiece offsets file, this will be a negative value.

Note that the Z position of the workpiece datum is defined by the Z value in the workpiece offsets file only. Since workpiece offset file values are global, no other tool profile can be used because the Z offset would always be set for the length of the original tool.

14: Glossary

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

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.

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.

DESKTOP TUTOR .... The input control keypad for the machine. Keypad overlays are interchangeable according to the type of controller required.

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

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

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

14: Glossary

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.

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 machine datum, also called the 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.

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

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

MACHINE DATUM .... A fixed zero reference point, used to define the co-ordinate based grid system of the CNC machine. All machining co-ordinates originate from this point.

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

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

NC .... Numerical control.

14: Glossary

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 shift the entire co-ordinate based grid system of 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.

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

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

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.

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 a "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.

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

15: Index

15: Index

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.