DENFORD

Total Commitment to Education and Training WorldWide.

Virtual Reality CNC Turning for Windows User's Manual.

Contents

Contents

Contents

About this Manual

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. First printed July, 2001.

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 Turning 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 title or object, click the LEFT mouse button ONCE. When asked to right click on a menu title 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 click 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: Introducing the VR CNC Turning Software

Congratulations on your purchase of Denford Virtual Reality CNC Turning for Windows software. VR CNC Turning 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 configuring 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 manufacture their designs without having to learn any new CNC machine control software.

Features available in the VR CNC Turning software package include:

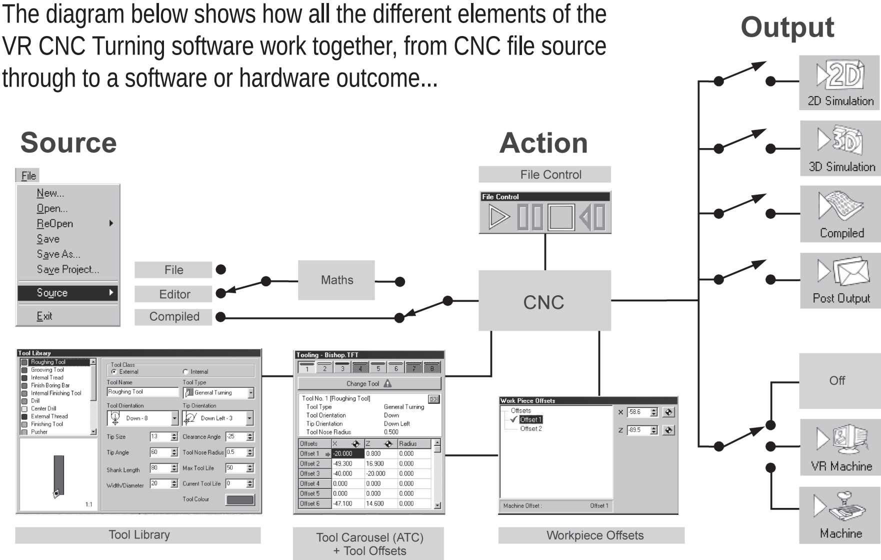
- Full MDI CNC file editing.
- 2 Dimensional graphical simulation of CNC files.
- 3 Dimensional graphical simulation of CNC files.
- Comprehensive Tooling features.
- Full offline control of a CNC machine using Virtual Reality.
- Full online control of a CNC machine.
- Context sensitive online help,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

Jargon Buster

When shortcut keys are used to call a Context Sensitive helpfile, the pages displayed relate directly to the current active elements within the software.
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 Turning Software in Use

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



1: Using the VR CNC Turning Software Manual

This manual provides a basic introduction to the features available in the VR CNC Turning software, using a tutorial based format. The manual applies to the following VR CNC Turning 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 Turning 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 Turning software helpfile.

2: Minimum System Requirements

> **Note**
>
> Flash Screen Versions of the VR CNC Turning software DO NOT require a security dongle.

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

System Requirements:

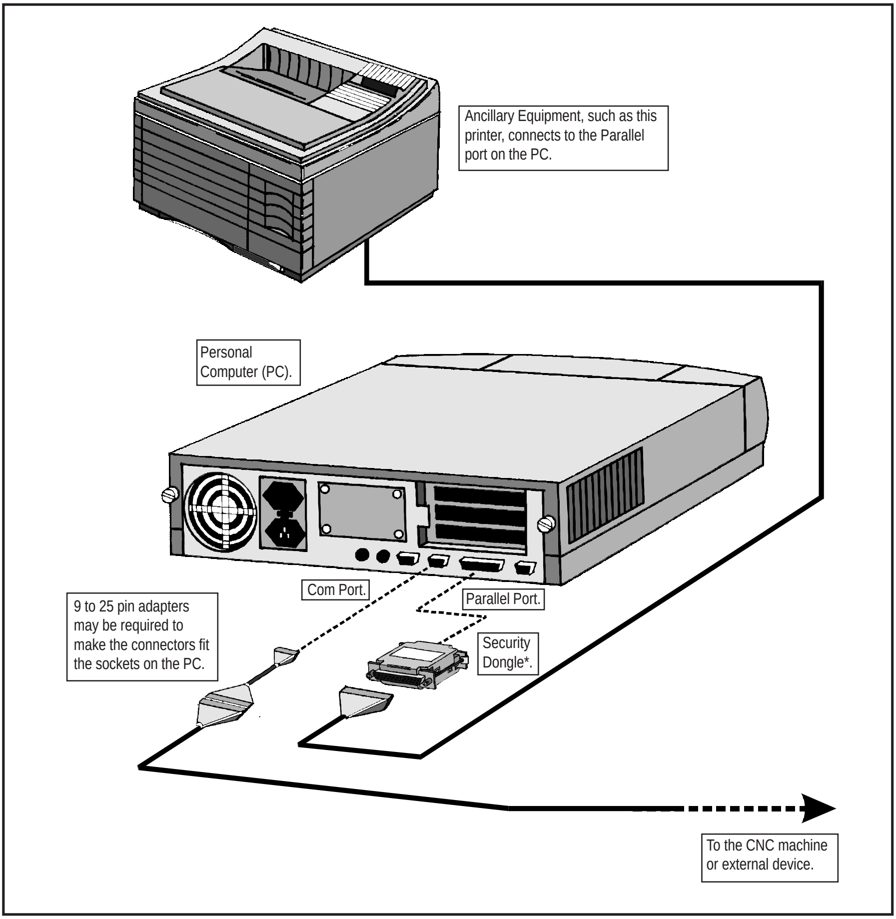
Minimum Specification:

- 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 (for real machine connection).
- 1 free parallel port (for security dongle / printer connection).

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 for real machine connection, 1 for DeskTop Tutor keyboard, if required).
- 1 free parallel port (for security dongle / printer connection).

2: Hardware Connections



Note*

Flash Screen Versions of the VR CNC Turning software DO NOT require a security dongle.

Note

Hardware connections are shown for offline use of the VR CNC Turning 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

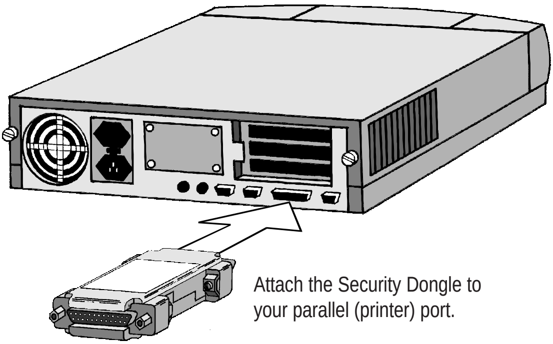
> **Note**
>
> Flash Screen Versions of the VR CNC Turning software DO NOT require a security dongle.

Security keyed versions of the VR CNC Turning 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 Turning 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

> **Note**
>
> This page explains the most common installation routines for a stand alone computer:
> 1) Flash Screen Version - you must install the flash registration software (from the supplied floppy disk) once the main VR CNC Turning software has been installed.
> 2) Security Keyed Version - you must always fit the security dongle to the parallel port of your computer.

> **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.

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

1) Switch on your computer and start Windows 95/98, if required.
2) Insert the VR CNC Turning 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 Turning" title and follow the on-screen instructions.
6) Select the title of the CNC machine used for default configuration by the VR CNC Turning software. The selected CNC machine is shown using a tick mark. If you intend to control a real CNC lathe with the VR CNC Turning software, select the name of the CNC lathe you want to control. This will also be the default CNC lathe displayed in any "Denford Virtual Reality" window.
7) Select the area of your hard-disk where the VR CNC Turning 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 Turning software for the first time.
9) If you are using a security keyed version of VR CNC Turning, ensure that the security dongle is fitted to the parallel port of your computer.

   If you are using a flash screen version of VR CNC Turning, you must install the Flash Registration software. Insert the separate "Registration Floppy Disk" in your floppy disk drive and run the "Setup.exe" file to register your software.

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:

- Security Dongle or Flash Registration 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 with a Virtual Reality CNC machine, introducing you to the various features available in the VR CNC Turning software. Use the sample CNC file listed on pages 14 to 17. Sections can be studied individually, or followed sequentially, as outlined in the flowchart.
>
> If you are new to using the VR CNC Turning software, we recommend that you follow the entire course from start to finish.
>
> The full course should take around 1 hour to complete.

> **Note**
>
> Steps A, H and I are not required when working with a Virtual Reality CNC machine.

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 real CNC machine.
(see section 8: pages 62 to 64 and section 10: pages 78 to 79).

↓

B: Start the VR CNC Turning software.
(see section 4: page 18).

↓

C: Load or create the CNC program.
(see section 5: pages 26 to 35).

↓

D: Configure the tooling in the VR CNC Turning software.
(see section 6: pages 36 to 48).

↓

E: Simulate the CNC program in 2D or 3D.
(see section 7: pages 49 to 61).

↓

F: Start the VR CNC machine.
(see section 8: pages 62 to 64 and section 9: pages 73 to 77).

↓

G: Home the CNC machine.
(see section 8: pages 65).

↓

H: Prepare any tooling hardware when using a real CNC machine.
(see your separate CNC machine manual).

↓

I: Load the billet into the chuck on a real CNC machine.
(see your separate CNC machine manual).

↓

J: Configure the workpiece offset file.
(see section 11: pages 80 to 95).

↓

K: Configure the tool offsets.
(see section 12: pages 96 to 109).

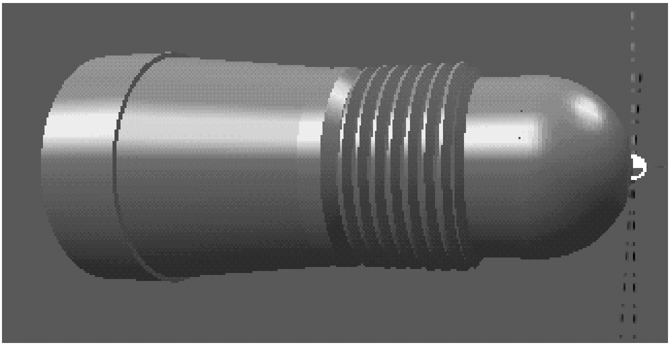
↓

L: Manufacture the part.
(see section 13: pages 110 to 111).

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: Brass or Steel

Part Datum Position: Centre line of part and opposite the chuck end.

Dimensions: X (Diameter) 25mm, Z (Length) 55mm

Tools required:

- Tool 1: General Roughing/Finishing Turning tool.
- Tool 3: Threading tool.

3: Sample CNC File - Metric

Metric (millimetres) sample CNC file with no formatting:

This program is written in Diameter programming. Ensure the VR CNC Turning software is configured for diameter programming before proceeding. See page 25 for information on how to configure this option.
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 the CNC file listed below into a new (blank) "Editor" window.
Save the CNC file as "dalek.fnl".

! Warning ! Do not renumber this program as block N1 is used to define the start of the profile.

```
[BILLET X25 Z55
O0001
G21
G99
M06 T0101
G97 S1500 M03
G00 X25 Z0
G01 X-1 F.05
G00 X25 Z2
G71 U1.5 R.25
G71 P1 Q2 U1.0 W.1 F.07
N1 G00 X0
G01 Z0
G03 X17 Z-8.5 R8.5
G01 Z-13.5
X20 Z-14.36
Z-28.5
X24 Z-46.5
X26
N2 G00 X26 Z2
G70 P1 Q2
G28 U0 W0
M06 T0303
G97 S350 M03
G00 X20.5 Z-10
G76 P050060 Q035 R.05
G76 X18.00 Z-26.5 P1000 Q070 F1.5
G28 U0 W0
M30
```

3: Sample CNC File - Metric

Explanation of metric (millimetres) sample CNC file:

[BILLET X25 Z55	(BILLET defines the size of the material being machined, called the billet, with X diameter 25mm and Z length 55mm)
O0001	(O defines the program name as 0001)
G21	(G21 defines the units of measurement being used as metric - millimetres)
G99	(G99 defines the feedrate command in units defined moved per revolution)
M06 T0101	(M06 instructs the machine to perform a tool change, if required. Change to tool number 01 and load/activate offset file number 01)
G97 S1500 M03	(G97 defines the spindle command in revolutions per minute, set at 1500 revs/min. M03 instructs the spindle to rotate in a clockwise [forwards] direction. The direction is determined by viewing from the back of the machine headstock)
G00 X25 Z0	(G00 instructs the machine to fast traverse to position X25, Z0)
G01 X-1 F.05	(G01 instructs the machine to cut in a straight line to position X-5, using a feedrate of 0.05mm per minute)
G00 X25 Z2	(G00 instructs the machine to fast traverse to position X25, Z2)
G71 U1.5 R.25	(G71 instructs the machine to perform stock removal in the X axis, with U1.5 referring to the depth of cut in the X axis and R.25 referring to the retraction [escape] amount)
G71 P1 Q2 U1.0 W.1 F.07	(G71 instructs the machine to perform stock removal in the X axis, with P1 referring to the sequence number of the first block of the finished shape [N1 on the first line directly below], Q2 referring to the sequence number of the last block of the finished shape [N2 on the ninth line directly below], U1.0 referring to the depth of cut in the X axis, W.1 referring to the distance and direction of the finishing allowance in the Z axis and F.07 referring to the roughing feedrate)
N1 G00 X0	(part of the G71 canned cycle, the first block of the finished shape, instructing the machine to rapid traverse to position X0)
G01 Z0	(part of the G71 canned cycle, instructing the machine to cut in a straight line to position X0)
G03 X17 Z-8.5 R8.5	(part of the G71 canned cycle, instructing the machine to cut an anticlockwise circular path, to position X17, Z-8.5, with a radius of R8.5)
G01 Z-13.5	(part of the G71 canned cycle, instructing the machine to cut in a straight line to position Z-13.5)

3: Sample CNC File - Metric

X20 Z-14.36	(part of the G71 canned cycle, instructing the machine to cut in a straight line to position X20, Z-14.36)
Z-28.5	(part of the G71 canned cycle, instructing the machine to cut in a straight line to position Z-28.5)
X24 Z-46.5	(part of the G71 canned cycle, instructing the machine to cut in a straight line to position Z-46.5)
X26	(part of the G71 canned cycle, instructing the machine to cut in a straight line to position X26)
N2 G00 X26 Z2	(part of the G71 canned cycle, the last block of the finished shape, instructing the machine to rapid traverse to position X26, Z2)
G70 P1 Q2	(G70 instructs the machine to perform a finishing cycle for the stock removal in the X axis operation above, with P1 referring to the sequence number of the first block of the operation and Q2 referring to the sequence number of the last number of the operation)
G28 U0 W0	(G28 instructs the machine to return to the reference point, using U0 and W0 [incremental] as an intermediate pass through point)
M06 T0303	(M06 instructs the machine to perform a tool change, if required. Change to tool number 03 and load/activate offset file number 03)
G97 S350 M03	(G97 defines the spindle command in revolutions per minute, set at 350 revs/min. M03 instructs the spindle to rotate in a clockwise [forwards] direction. The direction is determined by viewing from the back of the machine headstock)
G00 X20.5 Z-10	(G00 instructs the machine to fast traverse to position X20, Z-10)
G76 P050060 Q035 R.05	(G76 instructs the machine to perform a multiple thread cutting pass, with P050060 referring to 05 thread finishing passes, 00 amount of chamfer and a 60 degree tool tip angle, Q035 referring to the minimum cutting depth and R.05 referring to the finishing allowance)
G76 X18.00 Z-26.5 P1000 Q070 F1.5	(part of the G76 canned cycle, X18.00 referring to the X axis end position of the thread, Z-26.5 referring to the Z axis end position of the thread, P1000 referring to the depth of the thread, Q070 referring to the depth of the first pass and F1.5 referring to the size of the thread pitch)
G28 U0 W0	(G28 instructs the machine to return to the reference point, using U0 and W0 [incremental] as an intermediate pass through point)
M30	(M30 defines the end of the program and rewinds back to the start of the CNC file)

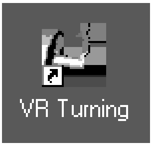
4: Starting the VR CNC Turning Software

Follow these instructions to start the VR CNC Turning software:

1) Power-up the PC.
2) Start the VR CNC Turning software (see note below). You start and exit the VR CNC Turning software as you would any standard Windows application.

Startbar Icon.

3) If VR CNC Turning 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 Turning" (see icon shown on left).



Desktop Icon.

4) Alternatively, if you have setup a desktop shortcut to the VR CNC Turning 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 Turning software, click "File | Exit".

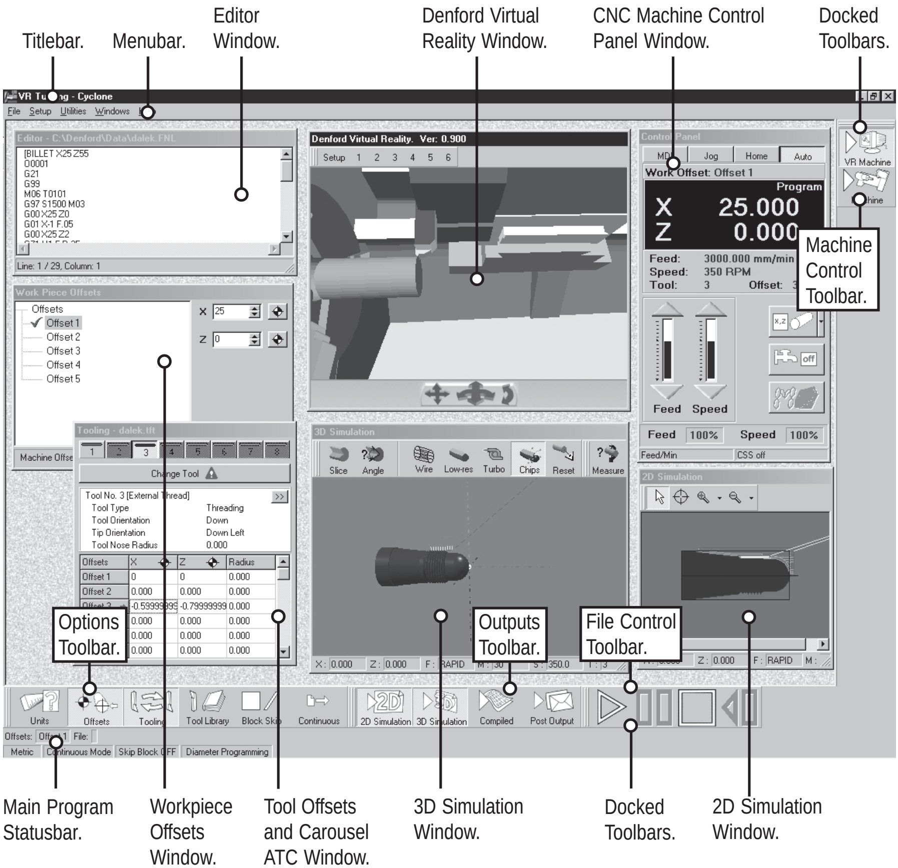
   Important - Never exit the VR CNC Turning software when machining operations are in progress on a real CNC machine.

Note

When using the VR CNC Turning 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 Turning software.

4: General Layout of the Software

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



> **Note**
>
> Not all the VR CNC Turning software option windows are shown in the example screenshot.

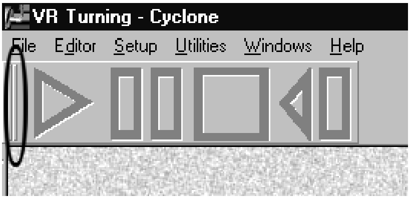
4: Using the Toolbars

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.

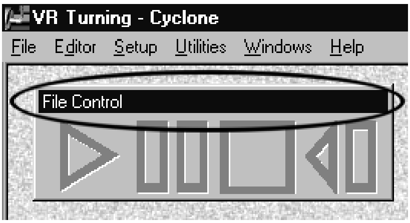
If you are unsure about the function of any toolbar button, hover your mouse cursor over the button to display a pop-up hint caption.

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 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 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, then release the mouse button.

4: Using the Toolbars

The buttons on each toolbar can be configured between a number of different display settings.

Right click on a toolbar button to display the pop-up menu (shown below):

Note

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

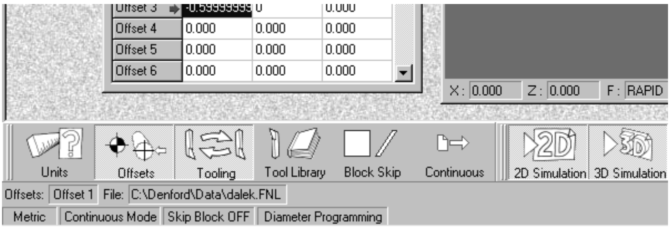
The options available in the Toolbar pop-up menu are as follows:

ToolBar Icons: Click "Small" to display small pictures on the buttons. Click "Large" to display large pictures on the buttons. Click "None" to display no pictures on the buttons. The current selection is indicated using a black bullet marker.

ToolBar Captions: When a "tickmark" is displayed, text captions will be used when pictures are applied to the buttons. When a "tickmark" is not displayed, text captions will not be used when pictures are applied to the buttons. Click the title to toggle between the two settings.

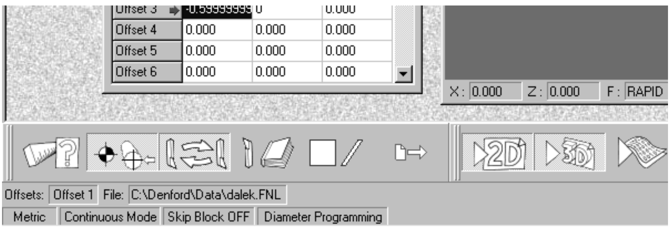
Setting 1: Large Graphics with Captions (shown below).

This is the default toolbar display setting. To change to this setting, select "ToolBar Icons | Large", with "ToolBar Captions" on.



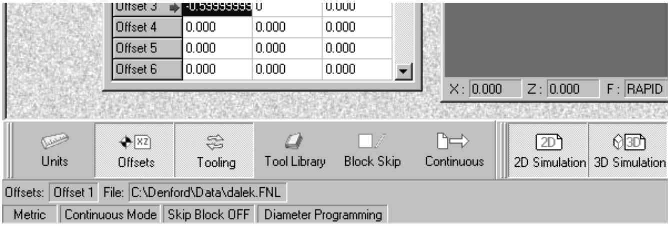
Setting 2: Large Graphics with no Captions (shown below).

To change to this setting, select "ToolBar Icons | Large", with "ToolBar Captions" off.



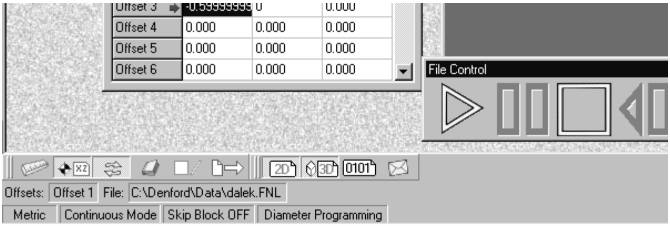
Setting 3: Small Graphics with Captions (shown below).

To change to this setting, select "ToolBar Icons | Small", with "ToolBar Captions" on.



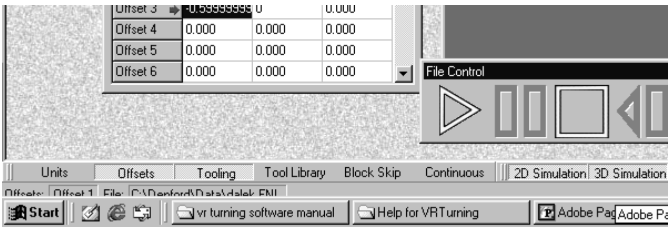
Setting 4: Small Graphics with no Captions (shown below).

To change to this setting, select "ToolBar Icons | Small", with "ToolBar Captions" off.



Toolbar Display Setting 5: No Graphics (shown below).

To change to this setting, select "ToolBar Icons | None". The "ToolBar Captions" option dos not function in this mode.



4: Using the Menubars

The VR CNC Turning software menubar, highlighted by the ellipse in the screenshot left, 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. Note that the following menu titles are common to all the available options in the software:

- "File", "Setup", "Utilities", "Windows", "Help".

The following menus become available on selection of various software options (each menu title listed appears after "File" in the menubar):

- "Editor" on selection of the "Editor" window (always visible).
- "Tooling" on selection of the "Tooling" window (accessed by clicking the [Tooling] button).
- "Tool Library" on selection of the "Tool Library" window (accessed by clicking the [Tool Library] button).
- "Offsets" on selection of the "Offsets" window (accessed by clicking the [Offsets] button).
- "2D" Simulate on selection of the "2D Simulation" window (accessed by clicking the [2D Simulation] button).
- "3D" Simulate on selection of the "3D Simulation" window (accessed by clicking 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 Turning software contains two separate helpfiles, both available from the "Help" menu title.

- **VR CNC Turning**: The VR CNC Turning for Windows software helpfile. This helpfile contains VR CNC Turning tutorials, detailed information about the various features of the VR CNC Turning software and troubleshooting guides.

  To access this helpfile, click "Help | Contents".
- **CNC Programming**: The CNC Turning Programming helpfile. This helpfile contains detailed information about individual G and M codes and structure of CNC files.

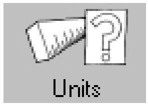
  To access this helpfile, click "Help | CNC Programming".

4: Setting the Units of Measurement

The units of measurement used by the VR CNC Turning 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 Turning software to run in Metric Mode, you must use a metric compatible CNC file and metric tooling.

Click the [Units] button (shown right) 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 lower line of this status bar indicates the units of measurement currently in use (shown as "Metric" in the screenshot below).

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

4: Setting Diameter or Radius Programming

Note - ❐ X

The VR CNC Turning software must be restarted if you reconfigure the diameter / radius programming mode setting.

Note - ❐ X

For more information regarding Diameter/ Radius Programming and CNC terminology, refer to page 123.

The VR CNC Turning software is configurable for either radius or diameter programming. The current setting of the option is displayed in the main program status bar, on the bottom row, in the last data box.

To change between radius and diameter programming modes, click "Setup | Machine Parameters" to display the "Setup Parameters" window (the default password for access is **denny**).

Select the "Machine Capability" option from the parameter list of your required lathe. Scroll down the list of parameters in the right panel until "Radius Programming" is visible. Click in the checkbox to display or remove the tickmark. A visible tickmark indicates radius programming mode will be active. Click the [OK] button to confirm and apply any changes.

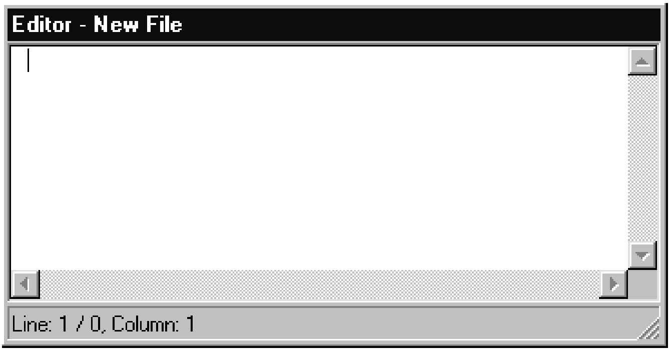
5: Creating a New CNC File

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

Note

If the "Editor" window is not displayed, the CNC file source must be changed to run in Editor mode.

From the menubar, click "File | Source | Editor".



The blank "Editor" window will be displayed with the title "New File", as shown above.

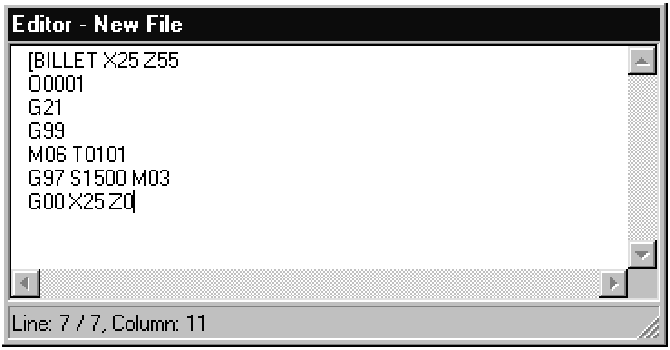
The "Editor" window behaves in a similar way to a simple word processor, such as Windows Notepad.

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 file listed on pages 14 to 17.



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.

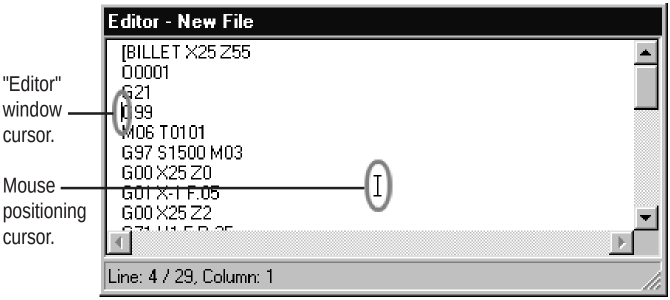
Each line of G and M codes is called a block, for example, "G97 S1500 M03", from the part program shown above.

Each block is created from different program words, for example, "G97" 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



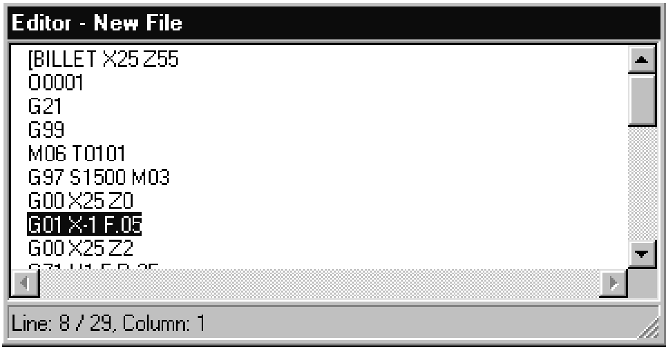
The "Editor" window cursor is a flashing vertical black line, highlighted in the above screenshot. This cursor shows where characters can currently be inserted, removed or highlighted. To remove characters directly behind the "Editor" window cursor, press the **[Delete]** key. To create a new CNC file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

- 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



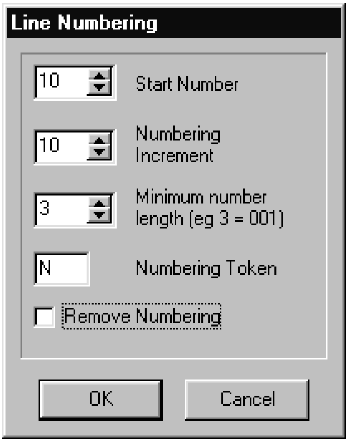
To select areas of text in the "Editor" window, position the "Editor" window cursor (the vertical black line) at the start or end of the text required, then click and hold down the left mouse button. Drag over the required characters to highlight them, as shown above.

The highlighted characters can be edited using the following commands:

- Select the "Undo" option from the "Editor | Edit" menu to undo the last command performed in the "Editor" window.
- Select the "Redo" option from the "Editor | Edit" menu to repeat the last command performed in the "Editor" window.
- Select the "Cut" option from the "Editor | 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 "Editor | 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 "Editor | 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 "Select All" option from the "Editor | Edit" menu to select all the text in the "Editor" window. Computer keyboard shortcut: **[CTRL + A]**

5: Adding Program Line Numbering

To add program line numbers to your finished CNC file, click "Editor | 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, 10 has been specified.

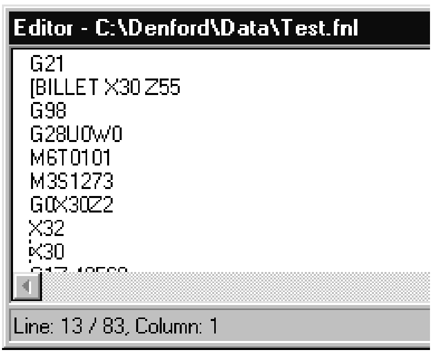
The "Numbering Increment" dialogue box is used to set the numerical gap between each program line number. In the example left, 10 has been specified, so the program line numbers will follow the sequence 10, 20, 30, 40, 50 etc.

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 010, 020, 030, 040, 050 etc.

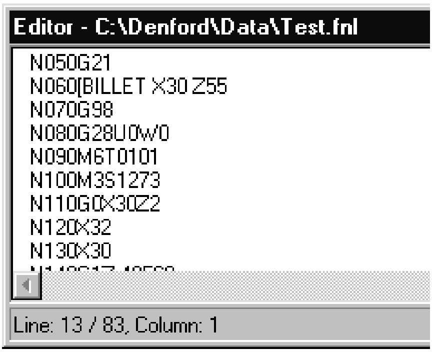
The "Numbering Token" dialogue box is used to add an address character to start of each program line number. The standard numbering token used is N. In the example above, the program line numbers will follow the sequence N010, N020, N030, N040, N050 etc.

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

An example of a modified CNC file is shown below.



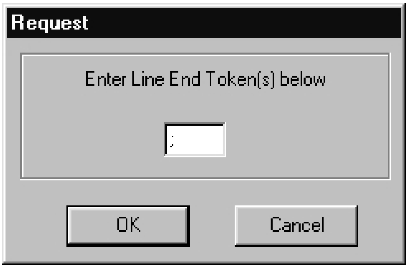
Before modifications.



After modifications (add program line numbering).

5: Adding End of Block Symbols

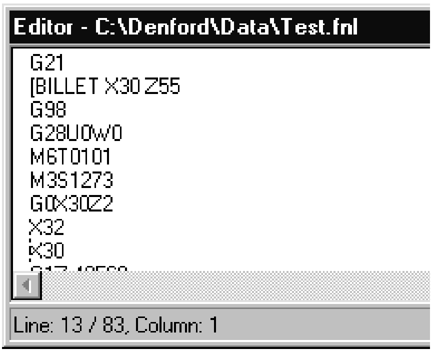
To add end of block symbols to your finished CNC file, click "Editor | Modify | Append Line End Token..." to display the "Request" window.



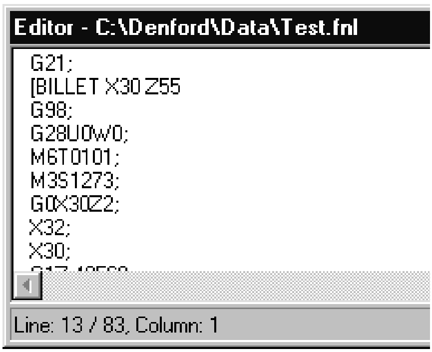
If you want to add end of block symbols to only part of the CNC file, drag across the required program lines to highlight them. If no program lines are highlighted, end of block symbols will be applied to the whole of the CNC file.

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.



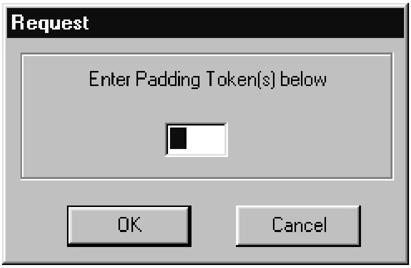
Before modifications.



After modifications
(add ; symbol).

5: Adding Program Line Spacing

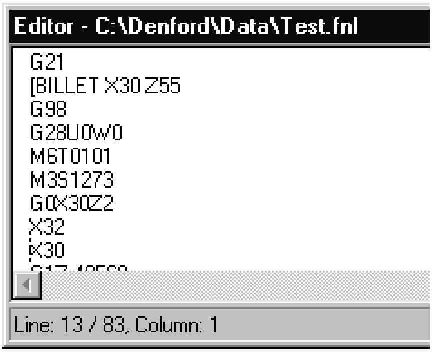
To add padding spaces between program words in your finished CNC file, click "Editor | Modify | Add Padding Token..." to display the "Request" window.



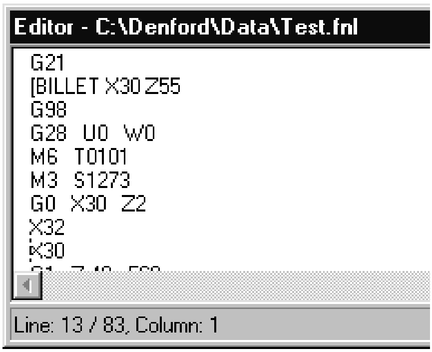
If you want to add padding spaces to only part of the CNC file, drag across the required program lines to highlight them. If no program lines are highlighted, padding spaces will be applied to the whole of the CNC file.

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, 3 spaces have been specified.

An example of a modified CNC file is shown below.



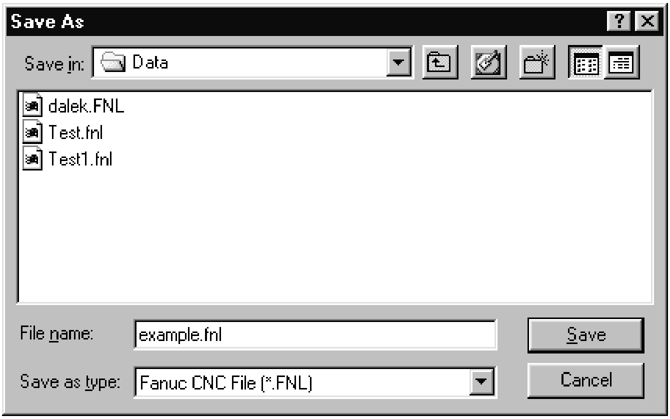
Before modifications.



After modifications (add 3 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 ".fnl", as shown above, then click the [Save] button.

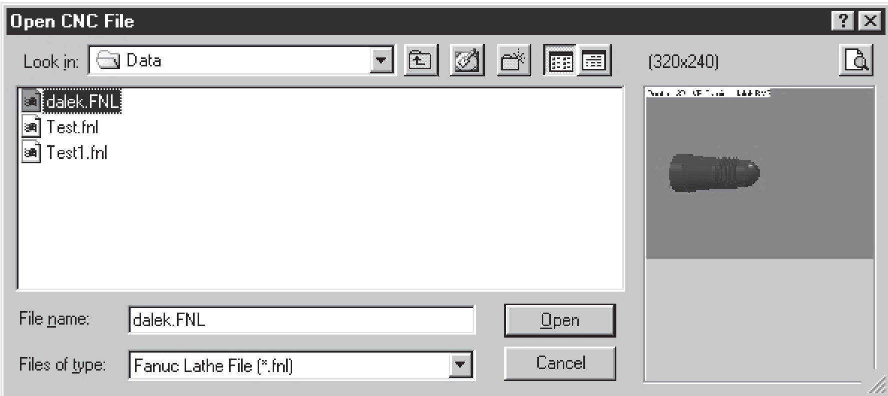
If you are following our tutorials, save your file as "dalek.fnl".

Jargon Buster

An fnl file is a FANUC Turning 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".



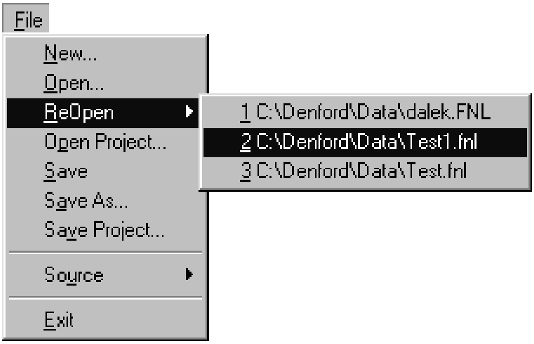
Select the directory used for storing the CNC file, using the "Look in:" panel.

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 right-hand 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 | {choice of filename}", to reopen the required CNC file, as shown above.

6: Displaying the Tool Library Window

Note

Upon initial installation, the default Tool Library profiles available are as follows:
(All values shown are for Metric tools but would be displayed in Inches if Imperial units were selected)

Roughing Tool:
- Tip Size=12
- Length=60
- Width Diameter=12

Grooving Tool:
- Tip Size=6
- Length=60
- Width Diameter=12

Internal Thread:
- Tip Size=9
- Length=60
- Width Diameter=12

Boring Bar:
- Tip Size=12
- Length=60
- Width Diameter=12

Center Drill:
- Tip Size=10
- Length=80
- Width Diameter=19

External Thread:
- Tip Size=12
- Length=60
- Width Diameter=12

Finishing Tool:
- Tip Size=12
- Length=60
- Width Diameter=12

Drill:
- Tip Size=15
- Length=80
- Width Diameter=12

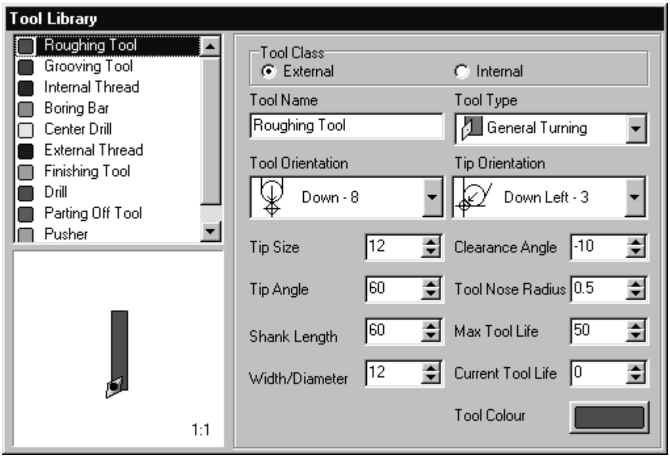
Parting Off Tool:
- Tip Size=4
- Length=60
- Width Diameter=12

Pusher:
- Length=60
- Width Diameter=12
- TipIndex=18

Displaying the "Tool Library" window

The Tool Library contains the list of tool profiles that are available for use with a CNC machine. Tools from this list can be added to the "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 Turning software.

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



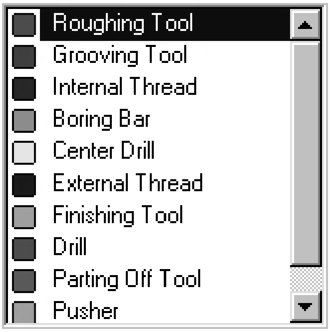
To close the "Tool Library" window (shown above), click the [Tool Library] button, from the "Options" toolbar.

6: General Layout of the Tool Library Window

Note

If data is edited in the "Tool Library" window on a profile already being used in the "Tooling" window, then the data of the profile in the "Tooling" window will also change.

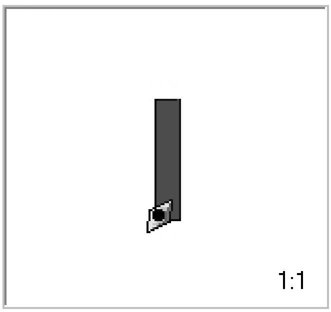
Tool Profile List Panel.



The upper left panel of the "Tool Library" window displays a list of available tool profiles. To view the information relating to a tool profile, click on the required title from the list. Highlighted tool profiles are shown using white title text on a blue background.

To add a new tool profile to the list, right click in the tool profile list to display the pop-up menu (shown right), then select "Add". The new tool profile will be created at the bottom of the current list.

Tool Profile Graphic Panel.



The lower left panel of the "Tool Library" window displays a graphic of the currently selected tool profile. The position and shape of the graphic is updated according to the settings within the Tool Profile Data panel.

6: General Layout of the Tool Library Window

Note

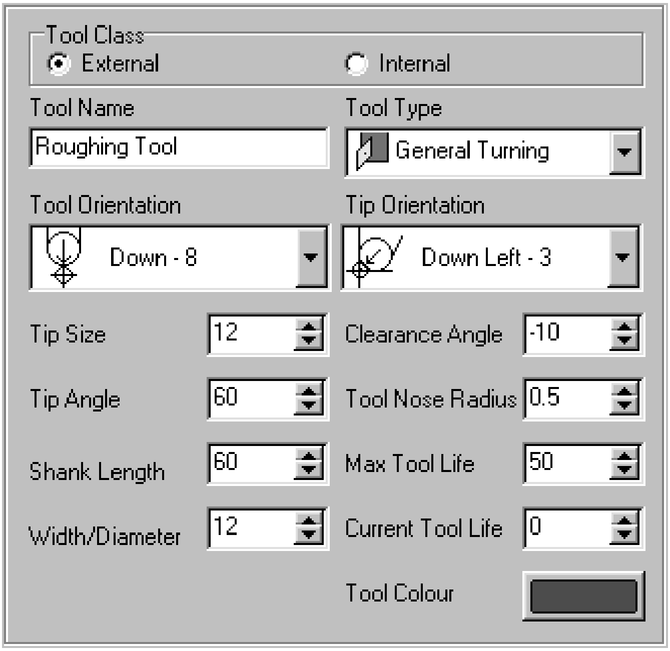
If data is edited in the "Tool Library" window on a profile already being used in the "Tooling" window, then the data of the profile in the "Tooling" window will also change.

Tool Profile Data Panel.

The right panel of the "Tool Library" window displays data relating to the currently selected tool profile.

Before editing any tool profile data:

- Check that the units of measurement set for the VR CNC Turning software matches the units used by any tooling profiles. The units of measurement setting for the VR CNC Turning software is configured using the [Units] button on the "Options" toolbar. For example, if you are editing a metric tool, the VR CNC Turning software units of measurement must be set to metric.



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

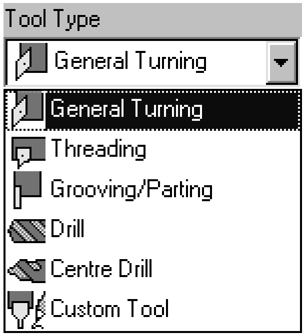
Tool Class: Select "External" for tools designed for machining around the outside surfaces of the billet, or "Internal" for tools designed for machining inside surfaces (ie. drilling, boring). The black bullet indicates the current setting of this option.

Tool Name: Characters typed into this dialogue box define the name of the tool that appears in any subsequent tool lists and descriptions.

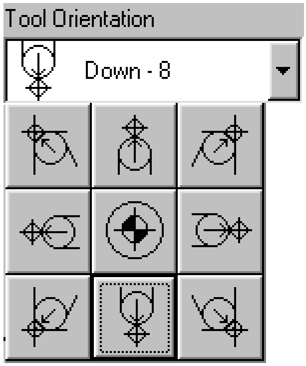
continued...

continued...

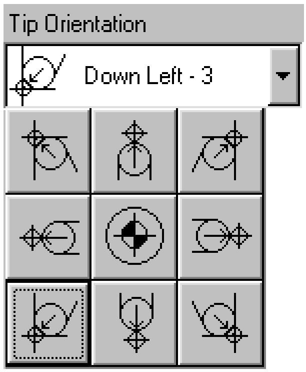
Tool Type: Indicates the machining function assigned to the tool profile. To change the description, click the [down arrow] button and select the new function from the six options (shown right).



Tool Orientation: Indicates the direction of the tool nose in relation to the billet position. To change the direction, click the [down arrow] button and select the new orientation from the nine options (shown right).



Tip Orientation: Indicates the direction of the tool tip mounted in the tool shank, in relation to the billet position. To change the direction, click the [down arrow] button and select the new orientation from the nine options (shown right).

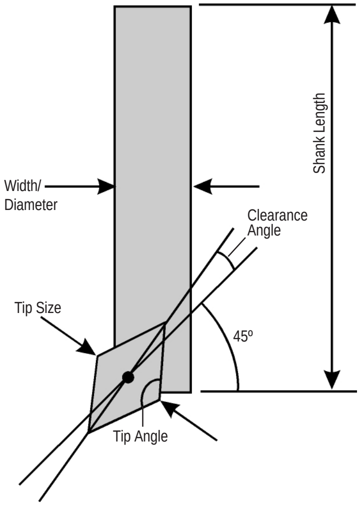


continued...

6: General Layout of the Tool Library Window

continued...

Diagram:
Tool Profile Data Definitions applied to the example from the screenshot on page 36.



Tip Size: Indicates the width of the tool tip (see example in diagram above). Click the [arrow] buttons to edit the values.

Tip Angle: Indicates the tool tip angle (see example in diagram above). Click the [arrow] buttons to edit the values.

Shank Length: Indicates the length of the tool shank (see example in diagram above). Click the [arrow] buttons to edit the values.

Width/Diameter: Indicates the width of the tool shank or diameter of the drill shaft (see example in diagram above). Click the [arrow] buttons to edit the values.

Clearance Angle: Indicates the tool tip clearance angle (see example in diagram above). This angle prevents the cutting edge of the tool from rubbing against the side of the workpiece. Click the [arrow] buttons to edit the values.

Tool Nose Radius: Indicates the value of the tool nose radius. Each tool is assigned a radius value to help strengthen the tip and increase tool lifespan. To find the correct amount of tool nose radius for a real tool tip, refer to the labelling on the original tool delivery box. Click the [arrow] buttons to edit the values.

Max Tool Life: The life of the tool, displayed in hours. Click the [arrow] buttons to edit the values.

Current Tool Life: The current life of the tool, displayed in hours. When the current tool life = the max. tool life, the tool must be replaced. Click the [arrow] buttons to edit the values.

Tool Colour: Displays the colour assigned to the tool profile graphic and squares next to the tool profile titles, in the "Tool Library" window. Click the coloured square to display the colour palette window, select your new colour, then click the [OK] button to change. Note that this palette colour is also applied to the [tool number] button bars, in the "Tooling" window, as individual tool profiles are added or updated.

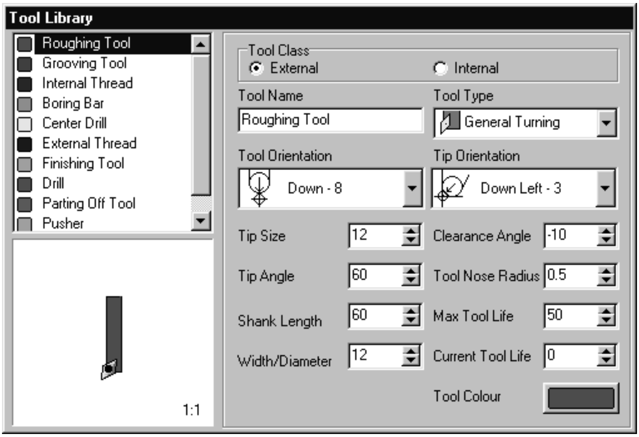
6: Tool Profiles for the Sample CNC File

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

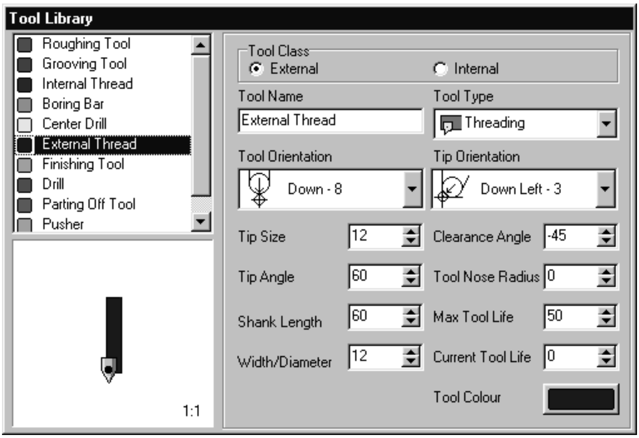
Check that the units of measurement are set to "Metric". The units of measurement setting for the VR CNC Turning software is configured using the [Units] button on the "Options" toolbar.

The sample CNC file uses two tool profiles:

- General Roughing/Finishing Turning tool.
  The default data values for this tool profile are used and shown below:



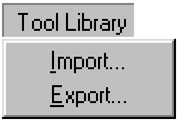
- Threading tool.
  The default data values for this tool profile are used and shown below:



6: Loading and Saving Tool Library Window Data

Note

In order to view the "Tool Library" menu, the "Tool Library" window must be the active window in the VR CNC Turning software.



To Open saved data in the "Tool Library" window:

From the VR CNC Turning software menubar, click "Tool Library | Import...", to load a previously saved library of tool profiles.

To Save data from the "Tool Library" window:

From the VR CNC Turning software menubar, click "Tool Library | Export..." to save the current library of tool profiles, with a user defined filename.

Tool profile library files are saved using the file extension ".TTD".

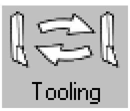
6: Displaying the Tooling Window

Displaying the "Tooling" window

Individual tool profiles from the "Tool Library" window are added to the "Tooling" window, where they are assigned a tool number, ready for use with any CNC files.

When a programmable turret is fitted to a VR CNC machine, the tool profiles are also added to the appropriate station numbers on the VR turret.

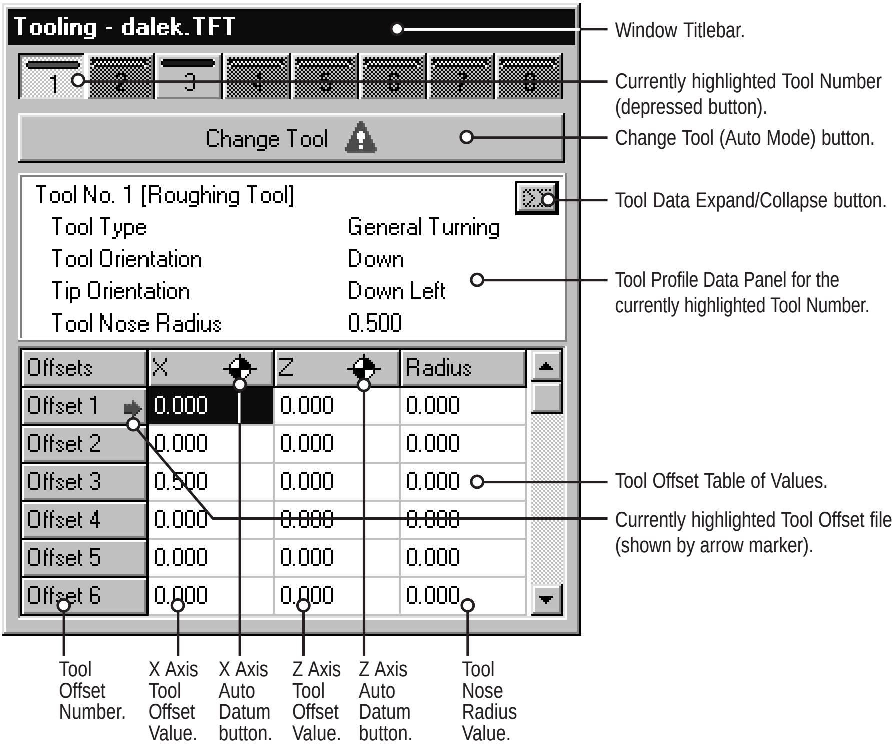
When a programmable turret is fitted to a real CNC machine, the correct tool profiles must be installed in the correct (tool) numbered stations on the turret.



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

6: General Layout of the Tooling Window

General Layout



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

6: General Layout of the Tooling Window

continued...

The options available in the "Tooling" window are as follows:

> **Note**
>
> The depressed [tool number] button indicates the currently highlighted tool profile number - data relating to this tool profile is shown in the "Tooling" window. It does NOT show the tool profile number currently active with the CNC machine controller (this is shown in the "Control Panel" window statusbar).

Tool Number buttons: Each button represents a tool number. If a tool number slot is a filled, the thin bar above the number on the button displays a colour. This is the colour associated with the tool profile from the "Tool Library" window. If a tool number slot is empty, the button number is greyed out.

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 T01 as a roughing/finishing profile, then a roughing/finishing profile must be transferred to tool number 1 in the "Tooling" window. When a programmable turret is used, this number will also be the station position.

Tool No. 1 [Roughing Tool]	
Tool Type	General Turning
Tool Orientation	Down
Tip Orientation	Down Left
Tool Nose Radius	0.500
Tip Angle	60
Width/Diameter	12.000
Length	60.000
Clearance Angle	-10
Tool Colour	
Max Tool Life	50
Current Tool Life	0

> **Note**
>
> For more information on manually requesting a tool change operation, refer to pages 70-71.

Change Tool button: The Change Tool button causes the CNC machine to the start the tool change sequence (eg. Turret rotation) and is active under the following conditions:

1) A VR or Real CNC machine is connected and running.

2) The "Control Panel" window is operating in Auto Mode.

3) The tool number required is not the current active tool number.

Tool Profile Data Panel: This panel displays tool profile data associated with the tool number button currently depressed - it does not necessarily show the data relating to the tool number currently active on the CNC machine. To expand the data panel (as shown below), click the [>>] button.

continued...

6: General Layout of the Tooling Window

Note

Configuring tool offsets will take into account the currently active workpiece offset - be sure to set your workpiece offsets first.

Note*

The red arrow marker indicates the currently highlighted tool offset file. It does NOT show the number of the tool offset file currently active with the CNC machine controller (this is shown in the "Control Panel" window statusbar).

Note

Advanced Users - Changing the Active Tool Offset File.
When a tool change is requested manually, the software always activates the tool offset file with the same number as the tool profile called. Different tool offset file numbers can be manually activated in the software, for occasions when the tool offset file number is different to the tool profile number. Refer to page 98 for this procedure.

Tool Offset Table of Values: The table displays a number of columns relating to tool offsets (listed from right to left):

Offsets	X	Z	Radius
Offset 1	0.000	0.000	0.000
Offset 2	0.000	0.000	0.000
Offset 3	0.500	0.000	0.000
Offset 4	0.000	0.000	0.000
Offset 5	0.000	0.000	0.000
Offset 6	0.000	0.000	0.000

Column 1: Offset Number. This is the number used to identify specific tool offset files, each containing an X, Z and tool radius value.

The currently highlighted tool offset number is shown using a red arrow marker (see note* left). Any data entries made via the [Auto-Datum] buttons are placed in the highlighted file.

Column 2: X Axis Tool Offset value and [X Auto-Datum] button. Values can be entered directly, by clicking and overtyping in the appropriate value box, or by using the [X Auto-Datum] button.

Pressing the [X Auto-Datum] button will display the "X Offset" window. When "Use Origin Only" is selected, the offset is calculated from the X axis origin (usually the centreline of the billet). When "Include Diameter" is selected, the diameter of the billet is taken into account when calculating the X offset value.

Column 3: Z Axis Tool Offset value and [Z Auto-Datum] button. Values can be entered directly, by clicking and overtyping in the appropriate value box, or by using the [Z Auto-Datum] button.

Pressing the [Z Auto-Datum] button will display the "Z Offset" window. When "Use Origin Only" is selected, the offset is calculated from the Z axis origin (usually the face end of the billet). When "Clearance" is selected, a user defined clearance value is taken into account when calculating the Z offset value.

Column 4: Tool Nose Radius value. Indicates the value of the tool nose radius. Each tool is assigned a radius value to help strengthen the tip and increase tool lifespan. To find the correct amount of tool nose radius for a real tool tip, refer to the labelling on the original tool delivery box.

6: Loading, Saving and creating New Tooling Window Data

Note

In order to view the "Tooling" menu, the "Tooling" window must be the active window in the VR CNC Turning software.

To create a New "Tooling" window:

From the VR CNC Turning software menubar, click "Tooling | New". A new, blank tool numbers/profiles list and blank tool offset table will be created.

To Open saved data in the "Tooling" window:

From the VR CNC Turning software menubar, click "Tooling | Open...", to load a previously saved collection of tool offset files and tool numbers/profiles.

To Save data from the "Tooling" window:

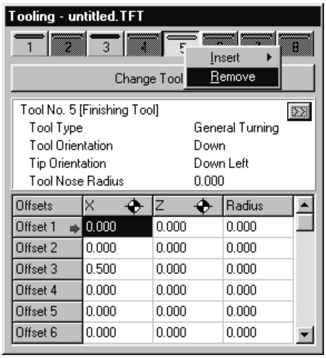
From the VR CNC Turning software menubar, click "Tooling | Save" to save the current collection of tool offset files and tool numbers/profiles, using the filename displayed in the "Tooling" window titlebar. Any data previously saved using this filename is automatically overwritten.

From the VR CNC Turning software menubar, click "Tooling | Save As..." to save the current collection of tool offset files and tool numbers/profiles, with a user defined filename.

Collections of tool offset files are saved, together with appropriate tool numbers and profile data, using the file extension ".TFT".

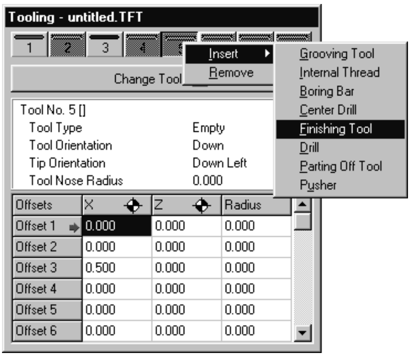
6: Transferring Tools into the Tooling Window

Deleting Tools from the Tooling window



Position your mouse cursor over the numbered button holding the tool profile you want to remove. Click the right mouse button to display the pop-up menu, then highlight and click the "Remove" option. The numbered button will become greyed out to indicate the position is now empty.

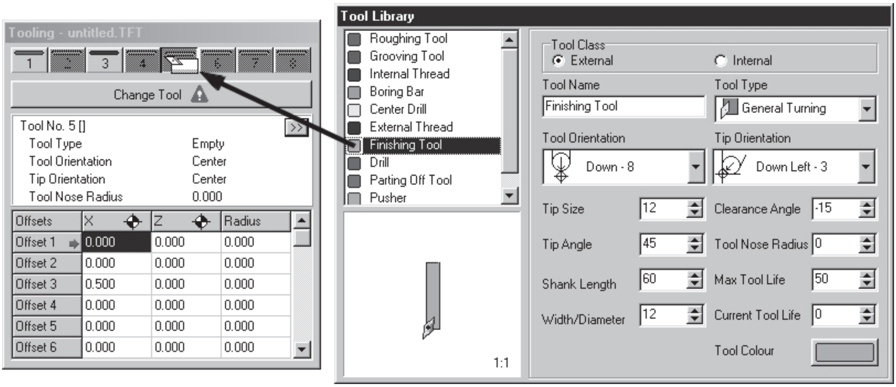
Adding Tools into the Tooling window



Position your mouse cursor over the numbered button where you want to add a tool profile. Click the right mouse button to display the pop-up menu, then highlight the "Insert" option and move your mouse cursor across to the right to display the expanded list of available tool profiles. Highlight and click on the required tool profile name from the list. The selected tool will then be assigned the specified tool number and turret station position. If a new tool profile is added to a position that is already filled, your new tool profile data will overwrite any old tool profile data.

6: Transferring Tools into the Tooling Window

Drag and Dropping Tool Profiles into the Tooling window



In the "Tool Library" window, click and hold down the left mouse button on the title of the tool profile you want to add to the "Tooling" window. The list of tool profiles is displayed in the top left corner of the "Tool Library" window. Note that if you hold your mouse cursor stationary over a tool profile title, a hint caption will display whether the tool is currently being used.

Whilst continuing to hold down the left mouse button, drag the tool out from the "Tool Library" window and into the "Tooling" window. Position the cursor over the required tool number and release the left mouse button. The selected tool profile will then be assigned with the specified tool number and turret station position.

6: Configuring the Tooling window with the sample CNC file

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

The sample CNC file uses two tool profiles:

- General Roughing/Finishing Turning tool.
  This profile should be assigned as tool number 1.
- Threading tool.
  This profile should be assigned as tool number 3.

When a programmable turret is fitted to a real CNC machine, the correct tool profiles must be installed in the correct (tool) numbered stations on the turret. Tool number 1 must be fitted to turret station number 1 and tool number 3 must be fitted to turret station number 3. This is completed automatically on VR CNC machines.

7: Displaying the 2D Simulation Window

The "2D Simulation" window provides a side view of the billet, displaying any machined parts when the CNC file is executed.

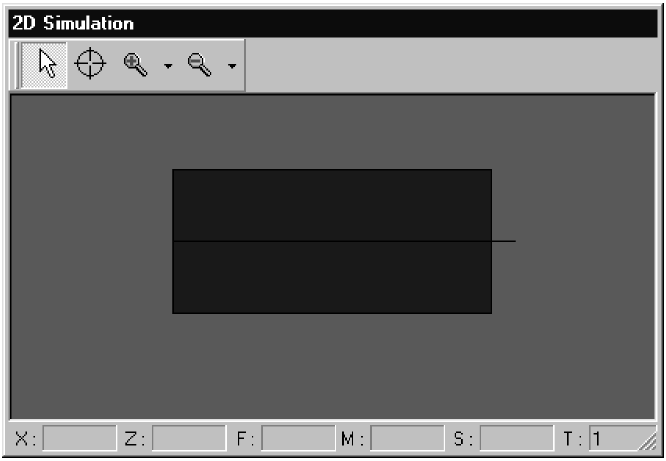
To display the "2D Simulation" window, click the [2D Simulation] button, shown above, from the "Outputs" toolbar.

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

Note

Click "2D Simulate | Options" on the menubar to display the "2D Simulation Setup" window, allowing the tool movement, tool graphic and billet colours used in the "2D Simulation" window to be configured individually.

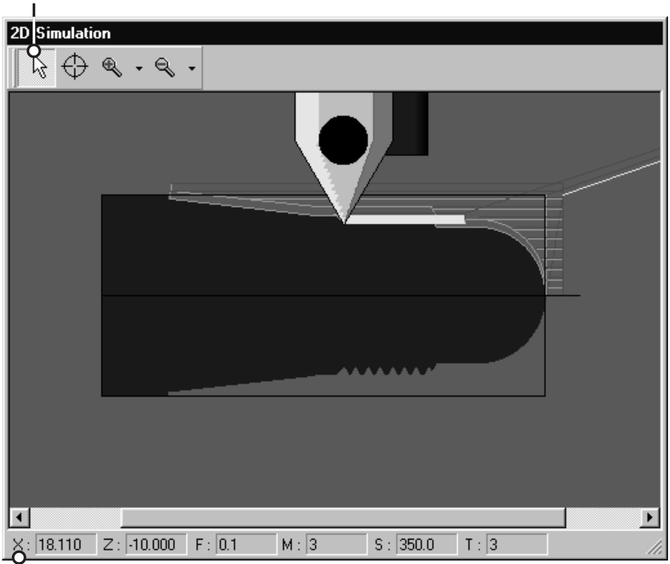
In order to view the "2D Simulate" menu, the "2D Simulation" window must be the active window in the VR CNC Turning software.



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

7: General Layout of the 2D Simulation Window

Zoom/Position Toolbar.



Window Statusbar.

The Zoom/Position Toolbar.

The options available in the Zoom/Position Toolbar allow the 2D display to be changed. Note that this toolbar can be undocked from the main "2D Simulation" window.

The options available in the Zoom/Position Toolbar are as follows (listed from left to right):

[Cursor] button: Select to show the normal arrow cursor within the window.

[Co-ordinates] button: Select to show the X and Z Co-ordinates as the cursor is moved around the window.

[Zoom In] button: Clicking the button once enlarges the size and detail shown on the billet graphic, according to the magnification amount set in the dropdown menu (to the right of the [Zoom In] button).

[Zoom Out] button: Clicking the button once reduces the size and detail shown on the billet graphic, according to the magnification amount set in the dropdown menu (to the right of the [Zoom Out] button).

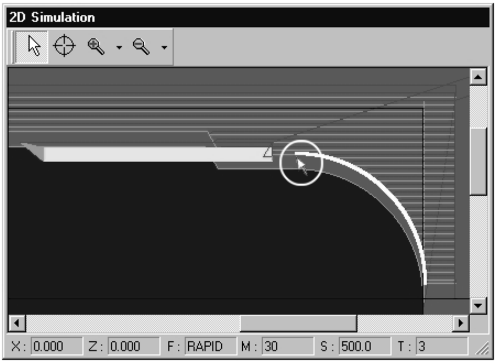
Statusbar.

The "2D Simulation" window also contains a statusbar at the bottom of the window. It displays updated information for the current X co-ordinate, Z co-ordinate, feedrate, M code, spindle speed and tool number.

7: Activating SourceTrack Technology™

SourceTrack™ Technology is a unique feature that allows you to back track into your CNC program, from any move completed in the "2D Simulation" window. Using SourceTrack™ Technology is a great way to debug CNC programs - especially the more complex ones.

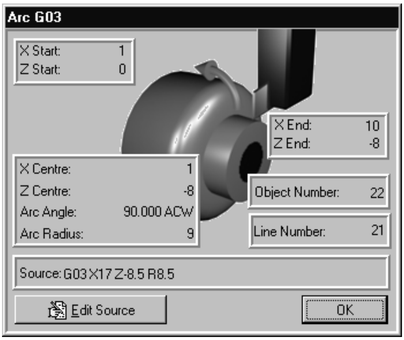
To activate SourceTrack™, move your mouse cursor around the linear and arc movement lines displayed in the "2D Simulation" window. When your cursor nears the end or beginning points of a movement line, the line itself will be highlighted by a different colour and thicker line width. In the screenshot below, the circled mouse cursor has highlighted the outer arc movement line.

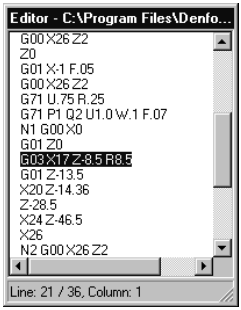


Click the left mouse button over the highlighted movement line to display the SourceTrack™ Information window, shown below left. Clicking the [Edit Source] button will highlight the line of the CNC file in which the element appears, shown below right. Note that the "Line Number" refers to the text line number in the "Editor" window and not the program line number (the "N" address letter).

Note

The layout and complexity of each SourceTrack™ Information window will vary, according to the type of move highlighted.





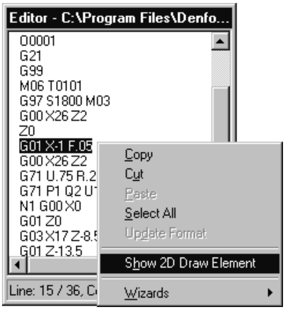
7: Activating SourceTrack Technology™

SourceTrack™ Technology can also be used in reverse, to find out what part of the graphic display in the "2D Simulation" window was created by a selected line in your CNC program.

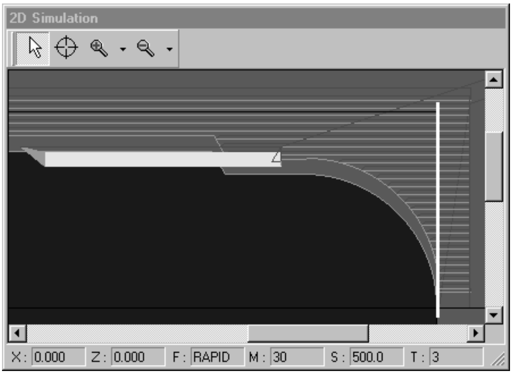
Note

In order to highlight a movement graphic in the "2D Simulation" window from a line in your CNC program, ensure you have completely run the CNC program so all possible movement lines are displayed in the "2D Simulation" window.

Highlight any CNC program line in the "Editor" window. Click the right mouse button to display the pop-up menu, then highlight and click the "Show 2D Draw Element" option, as shown below. The movement graphic matching the chosen CNC program line will be highlighted in the "2D Simulation" window.



In the screenshot below, the highlighted vertical movement line (at the right end of the billet) matches the CNC program line "G01 X-1 F.05" selected from the screenshot above.



7: Running in 2D without Simulated Offsets

Note

In order to view the "2D Simulate" menu, the "2D Simulation" window must be the active window in the VR CNC Turning software.

Note

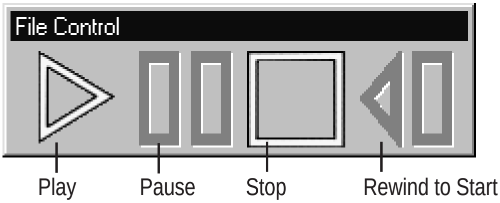
Before running the 2D simulation:

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

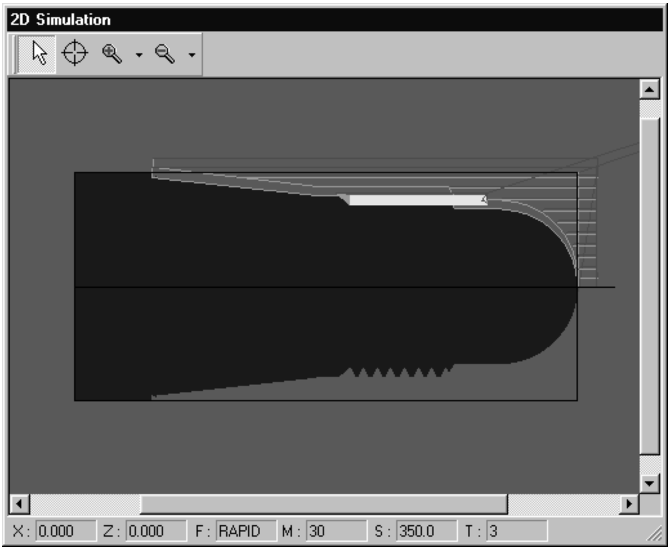
Click the "Use Offsets" option on the "2D Simulate" menu, so a tick mark is not shown next to the title, as shown left. This will display the 2D simulation without using an offset datum.

The default (zero) datum position is the billet centreline for the X axis and the right-hand end of the billet face for the Z axis.

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: Running in 2D using Simulated Offsets

Note

In order to view the "2D Simulate" menu, the "2D Simulation" window must be the active window in the VR CNC Turning software.

Click the "Use Offsets" option on the "2D Simulate" menu, so a tick mark is shown next to the title, as shown left. This will allow the 2D simulation to be displayed with a user defined offset datum.

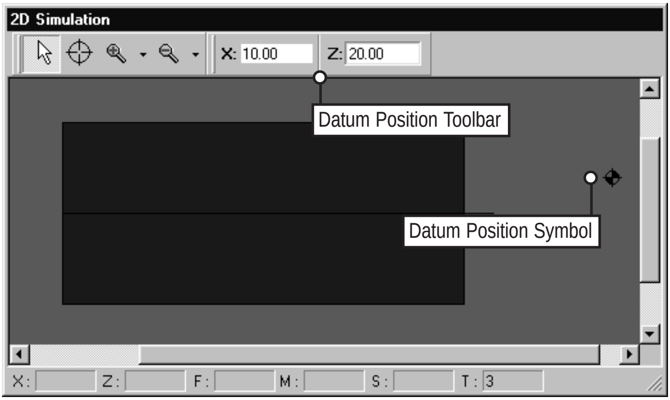
Note

Before running the 2D simulation:

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

When simulated offsets are used, an additional "Datum Position" toolbar is added to the right of the normal buttons in the "2D Simulation" window.

The default (zero) datum position is the billet centreline for the X axis and the right-hand end of the billet face for the Z axis. To move (offset) this datum position, enter the appropriate values in the "X" and "Z" dialogue boxes. As each value is registered, the position of the datum will be updated.

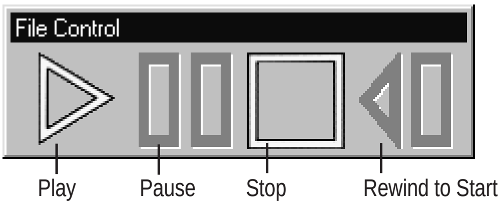


Jargon Buster

A datum is the point (co-ordinate) from which a series of measurements are taken.

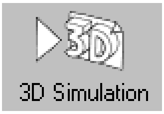
An offset involves shifting (offsetting) the entire two dimensional co-ordinate grid system of the CNC machine.

Ensure the "Editor" window cursor is positioned at the start of the first line of the CNC file.



To run the CNC file, click the triangular [Play] button from the "File Control" toolbar, shown above.

7: Displaying the 3D Simulation Window

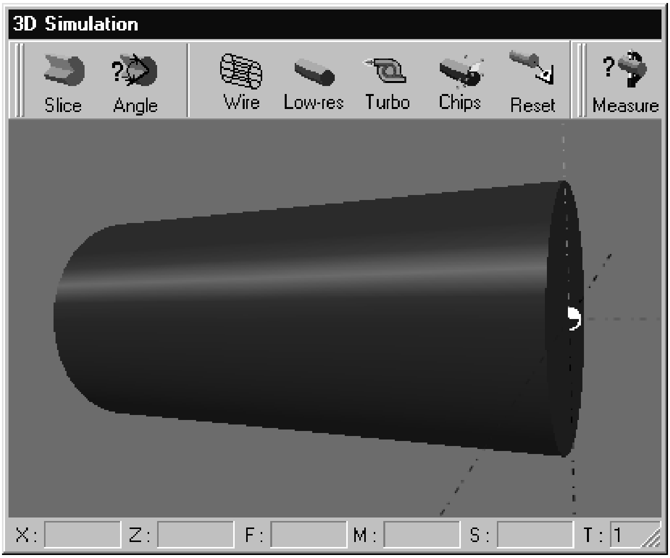


The "3D Simulation" window provides a three dimensional view of the billet, displaying any machined parts when the CNC file is executed.

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

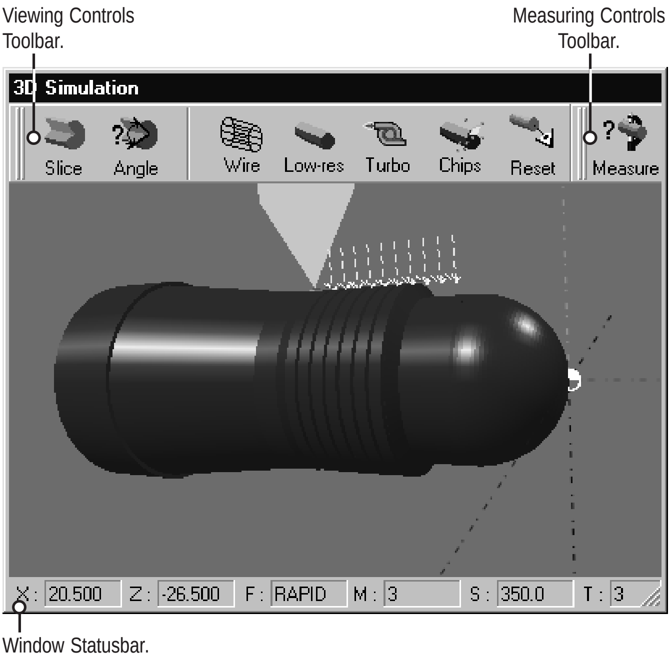
The "3D Simulation" window will show a fullsize view of the billet, as shown below.

> **Note**
>
> Click "3D Simulate | Options" on the menubar to display the "3D Simulation Setup" window, allowing the tool movement, tool graphic and billet colours used in the "3D Simulation" window to be configured individually.
>
> In order to view the "3D Simulate" menu, the "3D Simulation" window must be the active window in the VR CNC Turning software.



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

7: General Layout of the 3D Simulation Window



The Viewing Controls Toolbar.

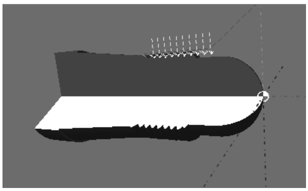
The options available in the Viewing Controls Toolbar allow the 3D model display to be changed. Note that this toolbar can be undocked from the main "3D Simulation" window.

The options available in the Viewing Controls Toolbar are as follows:

Note

If the [Slice] button is depressed, you will still be able to Zoom and Pan the billet but not rotate. Simply press the [Slice] button again to turn the Slice feature off.

[Slice] button: When this button is depressed, a section of the billet is removed (shown right). This feature is useful for displaying any internal details.

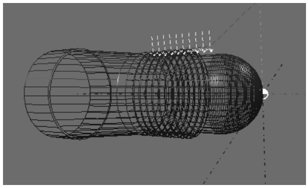


[Angle] button: Clicking this button allows the slice angle to be changed. This changes the amount of material that is removed when using the [Slice] button.

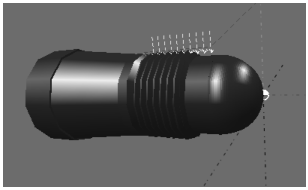
continued...

7: General Layout of the 3D Simulation Window

[Wire] button: When this button is depressed, the billet is displayed as a wireframe model (shown right).

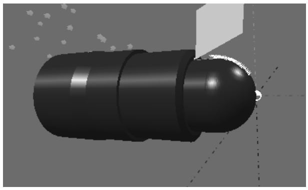


[Low-res] button: When this button is depressed, the billet is displayed in low resolution, ie. with less detail (shown right).



[Turbo] button: When this button is depressed, Turbo Mode will be active. When only the "3D Simulation" window is being used as an output window, ie. the VR or real CNC machines are inactive, any machining results will be generated faster.

[Chips] button: When this button is depressed, material chips will be shown in the "3D Simulation" window, as the tool machines parts of the billet (shown right).



[Reset] button: Clicking this button resets the view size and angle back to their default positions.

continued...

7: General Layout of the 3D Simulation Window

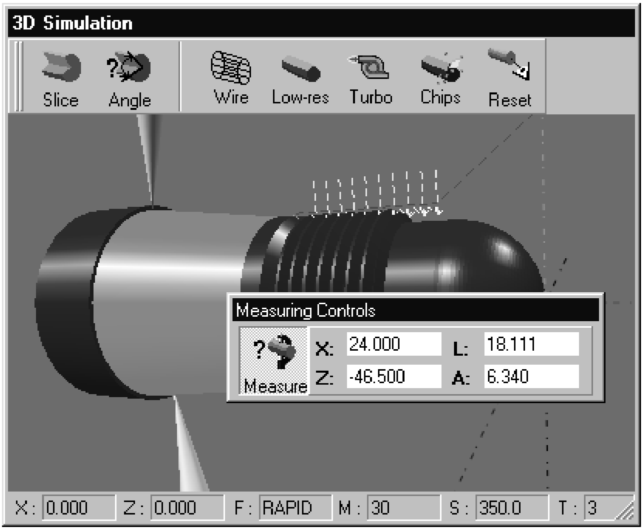
> **Note**
>
> If the [Measure] button is depressed, you will still be able to Zoom and Pan the billet but not rotate. Simply press the [Measure] button again to turn the Measure feature off.

The Measuring Controls Toolbar.

Once a CNC program has been successfully executed and the 3D billet model has been created, it is possible to measure diameters, lengths and angles of your part. Note that this toolbar can be undocked from the main "3D Simulation" window.

The options available in the Measuring Controls Toolbar are as follows:

[Measure] button: Clicking this button displays an information box to the right of the button (shown undocked in the screenshot below). Measuring probes also appear on the 3D model, indicating which point on the surface you are measuring. The section being measured is shown using a paler colour than the 3D model. To move the probes to a different part of the 3D model, press and hold the left mouse button, then move the mouse left or right to the required section.



The Information Box:

X: The current diameter as determined by the point of the measuring probes.

Z: The current Z dimension in relation to the front datum point.

L: The length of the item that is highlighted in a paler colour.

A: The angle of the item that is highlighted in a paler colour.

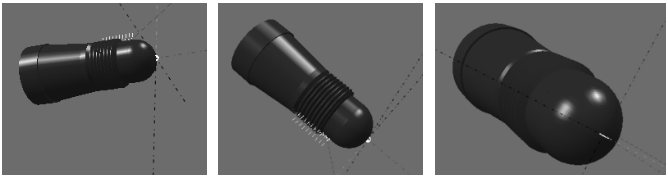
continued...

7: General Layout of the 3D Simulation Window

Statusbar.

The "3D Simulation" window also contains a statusbar at the bottom of the window. It displays updated information for the current X co-ordinate, Z co-ordinate, feedrate, M code, spindle speed and tool number.

7: Moving the 3D Model in the 3D Simulation Window



The view in the "3D Simulation" window can be changed by zooming, panning and rotating the 3D model using your mouse:

Zoom In: Hold down the right mouse button and move the straight arrowed cursor upwards.

Zoom Out: Hold down the right mouse button and move the straight arrowed cursor downwards.

Pan Left: Hold down the right mouse button and move the straight arrowed cursor to the left.

Pan Right: Hold down the right mouse button and move the straight arrowed cursor to the right.

Rotate around the X axis: Hold down the left mouse button and move the curved arrowed cursor to the left or right.

Rotate around the Z axis: Hold down the left mouse button and move the curved arrowed cursor up or down.

Raise Viewpoint: Hold down the middle mouse button and move the straight arrowed cursor upwards.

Lower Viewpoint: Hold down the middle mouse button and move the straight arrowed cursor downwards.

(if you do not have a middle mouse button, hold down the **[Shift]** key whilst using your right mouse button).

7: Running in 3D without Simulated Offsets

Note

In order to view the "3D Simulate" menu, the "3D Simulation" window must be the active window in the VR CNC Turning software.

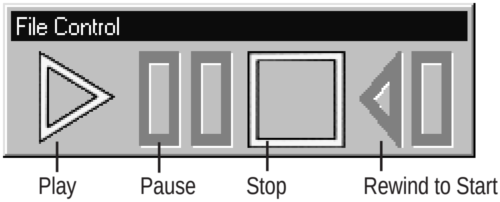
Click the "Use Offsets" option on the "3D Simulate" menu, so a tick mark is not shown next to the title, as shown left. This will display the 3D simulation without using an offset datum. The X and Z datum is always the default position of X on the centreline and Z on the right-hand end face of the billet.

Note

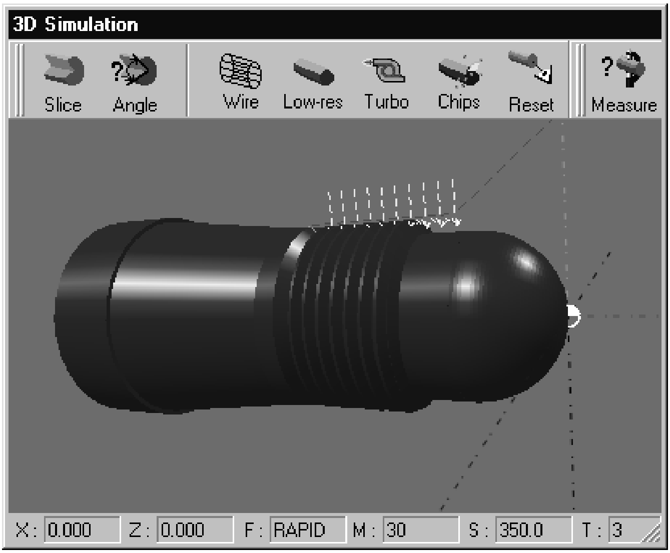
Before running the 3D simulation:

- Check that the units of measurement set for the VR CNC Turning software matches the units used in both the CNC file and any tooling profiles. The units of measurement setting for the VR CNC Turning software is configured using the [Units] button on the "Options" toolbar.
- Check that the tool numbers and tool profiles used in the "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: Running in 3D using Simulated Offsets

Note

In order to view the "3D Simulate" menu, the "3D Simulation" window must be the active window in the VR CNC Turning software.

Note

Before running the 3D simulation:

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

Jargon Buster

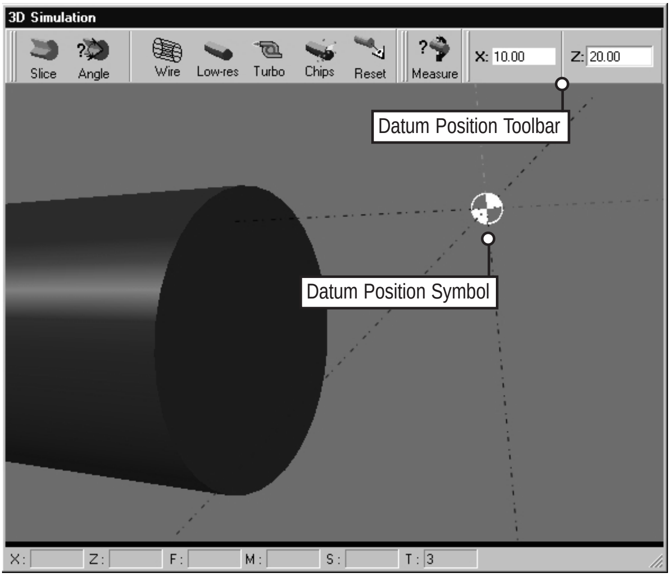
A datum is the point (co-ordinate) from which a series of measurements are taken.

An offset involves shifting (offsetting) the entire two dimensional co-ordinate grid system of the CNC machine.

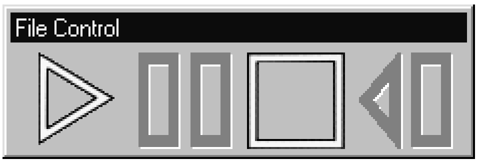
Click the "Use Offsets" option on the "3D Simulate" menu, so a tick mark is shown next to the title, as shown left. This will allow the 3D simulation to be displayed with a user defined offset datum.

When simulated offsets are used, an additional "Datum Position" toolbar is added to the right of the normal buttons in the "3D Simulation" window.

The default (zero) datum position is the billet centreline for the X axis and the right-hand end of the billet face for the Z axis. To move (offset) this datum position, enter the appropriate values in the "X" and "Z" dialogue boxes. As each value is registered, the position of the datum will be updated.



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



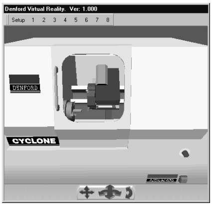
Click the triangular [Play] button from the "File Control" toolbar, shown left.

8: Starting a CNC Machine

Before starting any CNC Machine:

Check that the units of measurement set for the VR CNC Turning software matches the units used in both the CNC file and any tooling profiles. The units of measurement setting for the VR CNC Turning software is configured using the [Units] button on the "Options" toolbar.

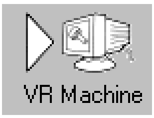
Check that the tool numbers and tool profiles used in the "Tooling" window match those used by your CNC file.

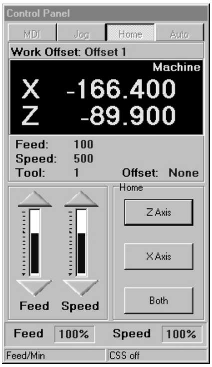


Above: The "Denford Virtual Reality" Window.

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. If a programmable turret is fitted, any tools present in the "Tooling" window will be automatically loaded into the appropriate station numbers. The "Control Panel" window will also open. This window is used for controlling the movements of the VR CNC machine.



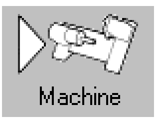


Above: The "Control Panel" Window.

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 Turning software.
4) Establish communications to the real CNC Machine by clicking the [Machine] button, shown below, from the "Machine Control" toolbar.

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



8: General Layout of the Control Panel Window

> **Note**
>
> The "MDI", "Jog" and "Auto" tabs cannot be accessed until the machine has been configured by homing both machine axes.

The "Control panel" window contains four tabs, just below the window titlebar:

- MDI: MDI (Manual Data Input) mode, used for typing in small CNC programs that can be run in automatic single step mode.
- Jog: Jog Mode, used for manually moving the machine within its co-ordinate working envelope.
- Home: Home Mode, used for configuring the machine, before it can be fully used. This finds the Machine Datum and limits of co-ordinate movement.
- Auto: Auto Mode, used for controlling the machine with a CNC file.

Numerous information statusbar panels are positioned around "Control panel" window (listed from top to bottom).

Directly above Co-ordinate Display window:

- Workpiece Offset: The number of the offset file currently active in the "Workpiece Offsets" window.

Directly below Co-ordinate Display window:

- The last programmed Feedrate setting: Denoted by "Feed:" and a numerical value. 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. If feedrate mode is per revolution, then the description changes accordingly, eg."Feed: 0.05mm/rev".
- The last programmed Spindle Speed setting: Denoted by "Speed:" and a numerical value. The spindle speed is measured using revolutions per minute or Metres per minute for Constant Surface Speed (feet per minute if units are Imperial).
- The last programmed Tool Change: Denoted by "Tool:" and the number of the tool.
- The last programmed Tool Offset: Denoted by "Offset:" and the number of the currently active tool offset file.

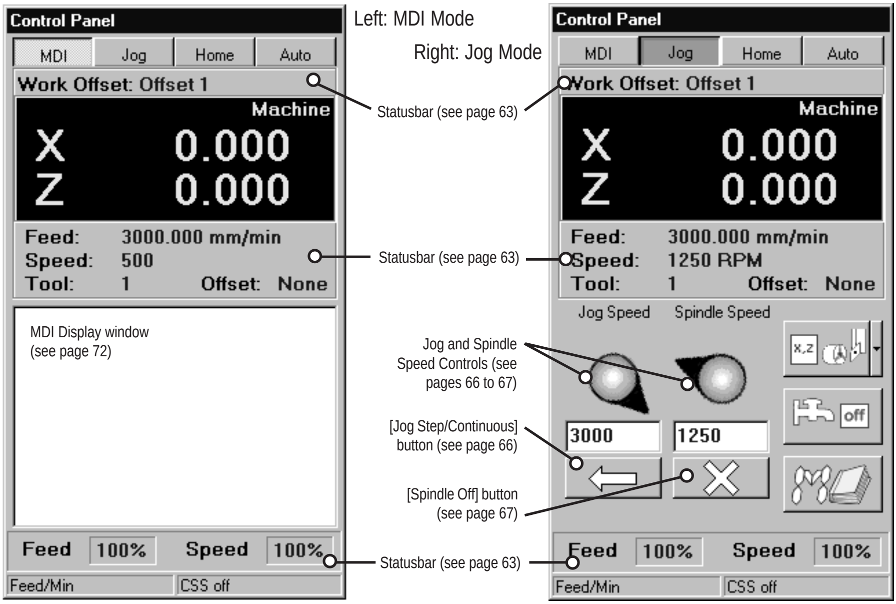
> **Note**
>
> When starting a CNC machine, the initial tool offset value is set to "None".
>
> To manually change the active tool offset file, refer to page 98.

> **Note**
>
> Overrides above 100% are shown in red text.

At the bottom of the "Control panel" window:

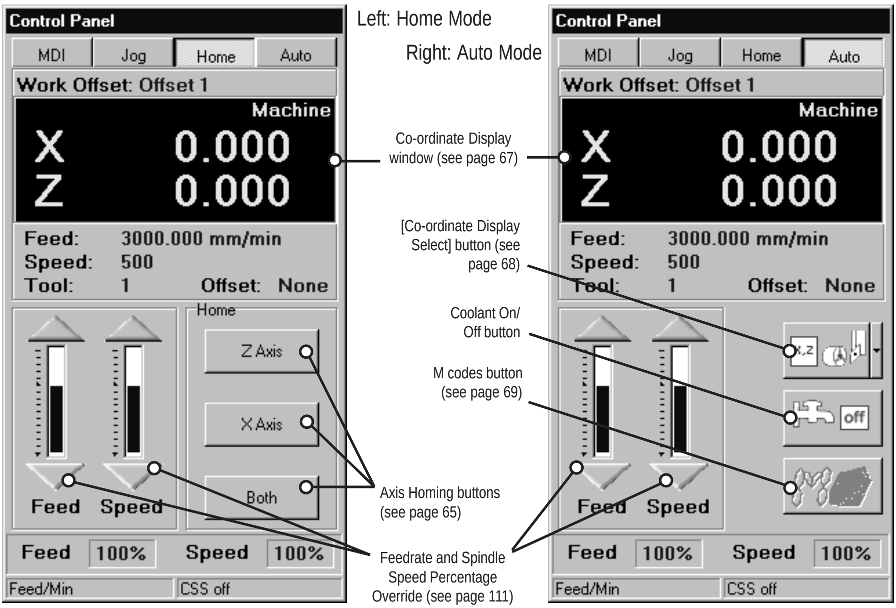
- Feedrate Override: Denoted by "Feed:" and a numerical value, indicating the current amount of percentage override that would be applied to a programmed feedrate.
- Speed Override: Denoted by "Speed:" and a numerical value, indicating the current amount of percentage override that would be applied to a programmed spindle speed.
- Current Feedrate setting: Denoted by "Feed/min" (feedrate per minute) or "Feed/rev" (feedrate per revolution).
- Current CSS (Constant Surface Speed) setting: Denoted by "CSS on" or "CSS off". When active, constant surface speed automatically adjusts the spindle speed for the given diameter, so the billet material passes the tool tip at the same speed.

8: General Layout of the Control Panel Window



MDI Mode is used for manually entering short independent CNC programs.

Jog Mode is used for manually moving the machine axes within their working envelope.



Home Mode is used for configuring the machine before it can be fully used.

Auto Mode is used for controlling the machine when running a CNC program.

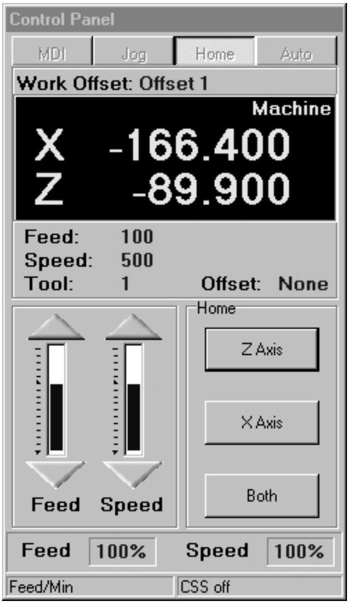
8: Homing a CNC Machine (Home Mode)

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 Turning software and any offsets being used on your computer system.

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.



When a CNC machine is first started, the "Control Panel" window will be displayed with only the "Home" tab active, as shown left.

Home Mode 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. Both machine axes are sent to their fixed zero positions. This defines the two 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. The position of the machine datum can be viewed when the X and Z values in the co-ordinate display both read zero (note that the co-ordinate display panel must be set to read "Machine").

Homing the CNC Machine Axes

Note

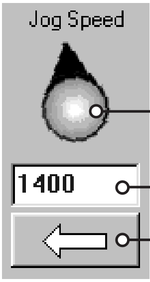
The "MDI", "Jog" and "Auto" tabs cannot be accessed until the machine has been configured by homing both machine axes.

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

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

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

8: Moving the Axes (Jog Mode)



Jog Speed Control.

Jog Control Value
Rotary Dial.

Jog Control Value Dialogue Box.

[Jog Step/Continuous] button.

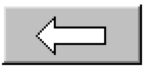
> **Note**
>
> In order to move any of the machine axes, the background of the "Jog" tab must be highlighted in <u>green</u> - this shows that Jog Mode is active. If the background is grey, simply click on the "Jog" tab to make it active.

Jog Mode is used for manually moving the CNC machine axes within their co-ordinate working envelope. To activate the Jog mode panel, click the "Jog" tab in the "Control Panel" window.

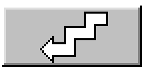
The machine turret can be moved, or jogged, using two different methods; jog continuous or jog step. To change between these two methods, click the [Jog Step/Continuous] button.

To change the jog control value, click and hold down the left mouse button on the triangular pointer of the rotary dial and drag the pointer to the new position. Alternatively, enter a new value in the jog control value dialogue box. 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.

Jog Continuous: In jog continuous mode, the selected machine axis will move at the speed displayed in the jog control value dialogue box, when one of the machine axis movement keys is pressed and held down. The selected machine axis will continue to move until the key is released. The maximum jog control speed is dependant on the capability of the CNC machine. When Jog Continuous is active, the [Jog Step/Continuous] 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 dialogue box), each time the selected axis movement key is pressed. Jog increments of 0.050, 0.100, 0.500, 1.000 and 2.000 units can be set. When Jog Step is active, the [Jog Step/Continuous] button graphic will be displayed as shown below.



8: Moving the Axes (Jog Mode)

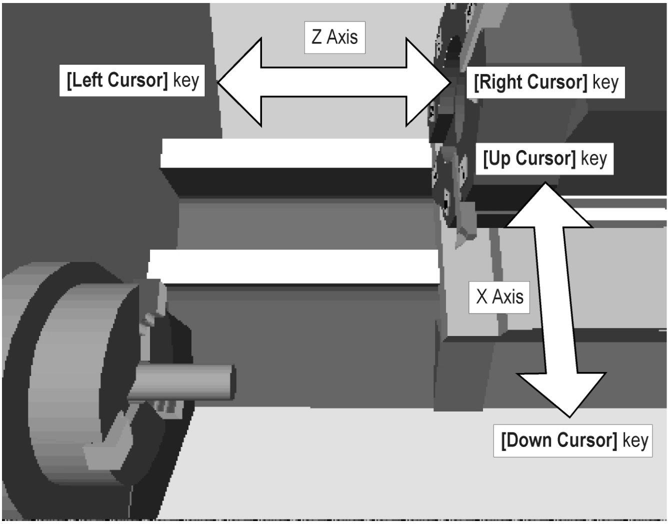
Jog Control Movement Keys.

There are four Jog Control movement keys:

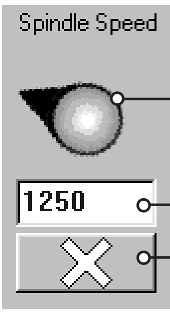
To move the X machine axis use the **[Up Cursor]** and **[Down Cursor]** arrow keys, with the "Jog" tab active (highlighted in green).

To move the Z machine axis use the **[Left Cursor]** and **[Right Cursor]** arrow keys, with the "Jog" tab active (highlighted in green).

> **Note**
>
> In order to move any of the machine axes, the background of the "Jog" tab must be highlighted in green - this shows that Jog Mode is active. If the background is grey, simply click on the "Jog" tab to make it active.



Spindle Speed Control.



To change the spindle speed value, click and hold down the left mouse button on the triangular pointer of the rotary dial and drag the pointer to the new position. Alternatively, enter a new value in the spindle speed control value dialogue box. The spindle speed is measured using revolutions per minute. To stop a running spindle, click the [Spindle Stop] button.

8: Co-ordinate System Display (Jog 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 Turning software and any offsets being used on your computer system.

> **Note**
>
> When starting a CNC machine, the initial tool offset value is set to "None".
>
> To manually change the active tool offset file, refer to page 98.

The [Co-ordinates Display Select] button is used to switch between the different systems used for displaying the co-ordinate positions.

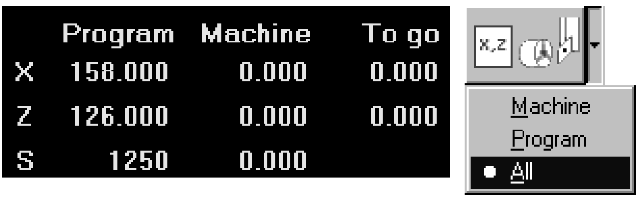
When the [Units] of Measurement are set to "Inch" the co-ordinates are displayed using inches. When the [Units] of Measurement are set to "Metric" the co-ordinates are displayed using millimetres.

The co-ordinate display options are:

Machine (shown below): This system displays the co-ordinate position relative to the fixed machine datum. When the Co-ordinate System Display Mode is set to Machine, the button graphic shows the X, Z symbol with a picture of the chuck and turning tool. The display panel shows the text "Machine".

Program (shown below): This system displays the co-ordinate position relative to the moveable workpiece datum. When the Co-ordinate System Display Mode is set to Program, the button graphic shows the X, Z symbol with a picture of the work piece (billet). The display panel shows the text "Program".

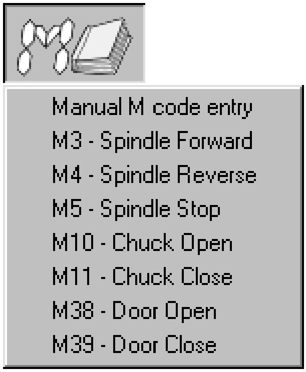
All (shown below): This system displays both Machine values and Program values. There is also an additional "To go" column, displaying the distance to go for each axis. When running a CNC program in auto mode, this column shows how far the current operation has to move until it is complete. The programmed and actual spindle speed is also shown (actual being the accurate chuck speed read from the machines encoder).



8: Selecting M Codes (Jog and Auto Modes)

Selecting M Codes.

M codes are used for miscellaneous functions, such as switching the spindle on and off, or opening and 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. Note that the options available on the M codes list will depend on the options fitted to the CNC machine being controlled.

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

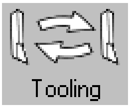
Note

If the M code you require is not displayed in the dropdown list, enter the M code using MDI mode.

For more information on running an independant program in MDI mode, refer to page 72.

8: Changing Tools (Auto Mode)

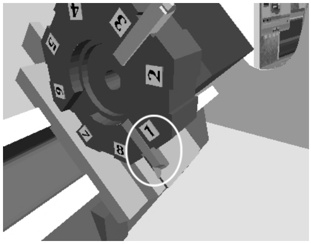
The "Tooling" window is used to manually request a tool change operation.



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

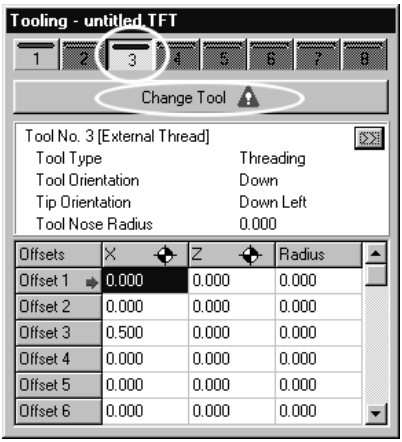
Note

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.



The current tool profile and tool offset file numbers are shown in the statusbar area of the "Control Panel" window, as shown in the screenshot left.

Tool changes can only be performed when the "Control Panel" window is set to Auto mode, achieved by clicking the "Auto" tab.

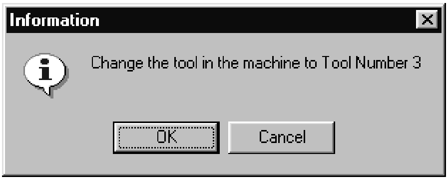


In the "Tooling" window, click the numbered button containing the tool profile you want to change to, shown as tool number 3 - a threading tool, in the screenshot left.

To start the tool change sequence, click the [Change Tool] button.

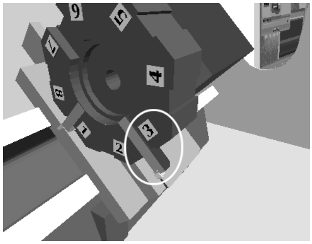
continued...

8: Changing Tools (Auto Mode)



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 current tool 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.

On CNC machines fitted with a programmable turret, the tool change operation will be performed automatically.



Following completion of the tool change operation, the new tool number is shown in the statusbar area of the "Control Panel" window, as shown in the screenshot left.

The VR CNC Turning software automatically activates the tool offset file with the same number as the tool profile called.

Note

To manually change the active tool offset file, refer to page 98.

8: Entering Data Manually (MDI Mode)

MDI Mode.

MDI mode is used for entering small CNC programs which can only be run one block (program line) at a time, independantly of the main CNC file currently loaded.

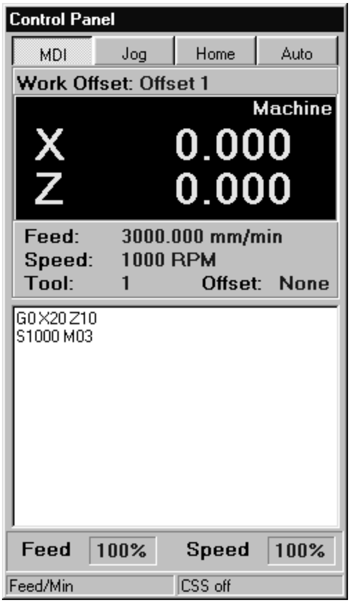
To activate the MDI mode panel, click the "MDI" tab in the "Control Panel" window. Type any commands into the white box.

Use the buttons on the "File Control" toolbar to run your MDI program:

1) Postion the cursor at the beginning of your MDI program and press the traingular [play] button.
2) On completion of first block, the MDI program will pause at the beginning of the next block.
3) The triangular [play] button must be pressed to run each block, until the MDI program has completed.
4) Pressing the [reset/rewind] button will clear the MDI window of all your commands.

Note

Example MDI program:
G0 X20 Z10
S1000 M03



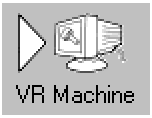
In the example left, the first block will rapid to a position near the workpiece and the second block will start the spindle at 1000RPM. This would be an ideal position from which to activate Jog mode, for touching onto your billet (workpiece and tool offsets configuration).

9: Starting a VR CNC Machine

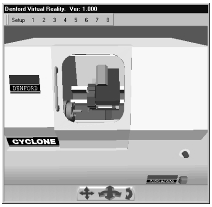
Before starting a Virtual Reality CNC Machine:

Check that the units of measurement set for the VR CNC Turning software matches the units used in both the CNC file and any tooling profiles. The units of measurement setting for the VR CNC Turning software is configured using the [Units] button on the "Options" toolbar.

Check that the tool numbers and tool profiles used in the "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.

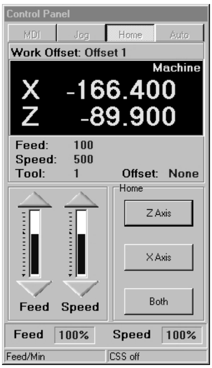


Above: The "Denford Virtual Reality" Window.

The "Denford Virtual Reality" window will open. This window is used for viewing the 3D model of the VR CNC machine. If a programmable turret is fitted, any tools present in the "Tooling" window will be automatically loaded into the appropriate station numbers.

Note

For more information regarding the "Control Panel" window, please refer to Section 8 - CNC Machine Control.



Above: The "Control Panel" Window.

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

9: General Layout of the Virtual Reality Window

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 Cyclone CNC Lathe is being controlled through the virtual reality interface.

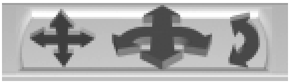
The "Denford Virtual Reality" window, shown above, is split into three basic areas:

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.

9: Moving around the Virtual Reality World

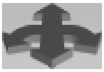


The movebar tools allow you to freely 'fly' around the virtual world, using your mouse. Try to think of the display in the Main Viewing Area as being the view from a floating camera head, which you can control using the three movement icons. Each icon is used to control a different type of movement in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

Using the movebar to adjust your viewpoint.

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

 : Moves the viewpoint in the vertical plane only (ie. up, down, left or right, whilst pointing straight ahead).

 : Moves the viewpoint in the horizontal plane only (ie. forwards, backwards, turning left and turning right).

 : Tilts the viewpoint plane (ie. moves the viewing angle up or 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.

9: Interactive Objects

Some objects on the VR CNC machines are interactive:

- **Chuck Control.**

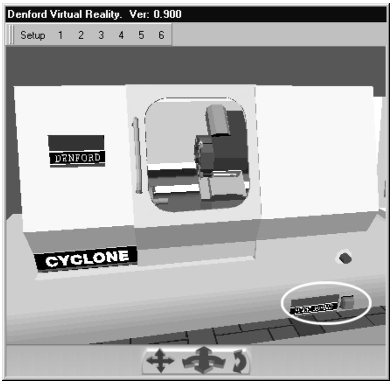
  Right click on the chuck to close the jaws. Left click on the chuck to open the jaws.

- **Safety Door Control.**

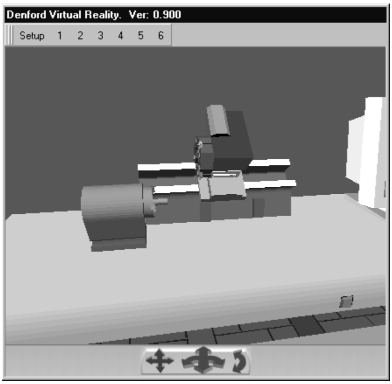
  Left click on the door handle to open or close the safety guard door.

- **Remove CNC Machine Cabinet.**

  The CNC machine cabinet can be removed to access a clearer view of the internal components - this can be useful when configuring offsets. This feature is available when a blue square button is fitted to the front edge of the table on which the CNC machine is mounted. To remove or refit the cabinet left click the blue square button.

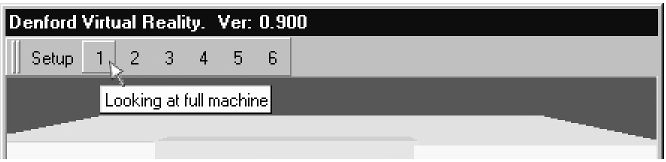


Left click the blue square button next to the logo containing the text "remove guarding", on the front edge of the table (shown circled in the screenshot left)...



...to remove the CNC machine cabinet from view.

9: Using the Predefined Viewpoints



Note

Many of the assigned viewpoints are designed to help during configuration of the 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.

The following six viewpoints are available for the Novaturn, Mirac and Cyclone CNC Lathes:

Button [1]: Looking at full machine.

Button [2]: Look at guard remove switch.

Button [3]: Looking at chuck and billet.

Button [4]: Close up of turret.

Button [5]: Close up of job.

Button [6]: Overall view of machine.

The following five viewpoints are available for the Microturn CNC Lathe:

Button [1]: Looking at full machine.

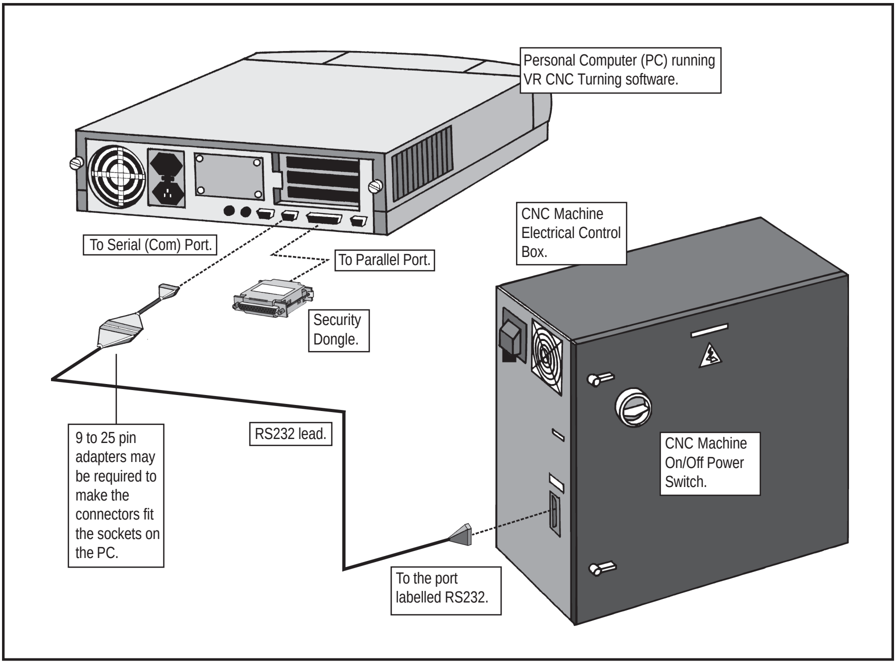
Button [2]: Looking at chuck and billet.

Button [3]: Close up of turret.

Button [4]: Close up of job.

Button [5]: Overall view of machine.

10: 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 Turning 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.

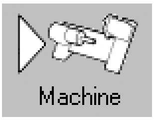
The opposite end of the RS232 lead fits into the serial (COM) port on your CNC machine casing or electrical control box.

Note

For more detailed information regarding CNC hardware connections, please refer to your separate CNC Machine User's Manual.

10: 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 Turning software.
4) Before attempting to establish a connection to a real CNC Machine:
   - Check that the units of measurement set for the VR CNC Turning software matches the units used in both the CNC file and any tooling profiles. The units of measurement setting for the VR CNC Turning software is configured using the [Units] button on the "Options" toolbar.
   - Check that the tool numbers and tool profiles used in the "Tooling" window match those used by your CNC file.
   - When a programmable turret is fitted, check that the correct tool profiles are fitted to the station number positions, according to the definitions defined in the "Tooling" window.
5) To start the real CNC Machine, click the [Machine] button, shown below, from the "Machine Control" toolbar.



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

Note

For more information regarding the "Control Panel" window, please refer to Section 8 - CNC Machine Control.

11: 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 - Workpiece offsets are used to apply a global shift to the co-ordinate grid system of the CNC machine. In other words, the X and Z values contained in the currently active workpiece offset file are used with EVERY tool profile.

ii) The tool offset files - Tool offsets are used to compensate for the differences in size between all the various tool profiles used to machine a part. In other words, the X and Z values contained in each tool offset file are associated for use ONLY with SPECIFIC tool profiles.

How is an offset calculated?

The X position of the workpiece datum is defined by the X value entered in the currently active workpiece offset file and the X value entered in the individual tool offset file associated with the tool profile currently in use.

The Z position of the workpiece datum is defined by the X value entered in the currently active workpiece offset file and the Z value entered in the individual tool offset file associated with the tool profile currently in use.

Jargon Buster

The moveable workpiece datum defines the zero point on our billet (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 two dimensional co-ordinate grid system used by the machine.

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 CNC program. In other words, the position of the workpiece datum will determine the place on the CNC machine where our part is manufactured.

What actually happens when I program my workpiece datum position?

Configuring the workpiece datum position allows us to temporarily move the zero point of the machine (the machine datum) to the zero position used by our CNC program with the billet (the workpiece datum). This is achieved by shifting, or offsetting, the entire two dimensional co-ordinate grid system used by the CNC machine, along the X and Z axes.

11: 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 LatheCAM Designer, the part datum would automatically be set along the cylindrical billet centreline, at the face end of the billet. In this case, the workpiece datum would be positioned on the real billet in the same location - along the chuck (same as the cylindrical billet) centreline and at the face end of the billet.

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

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

Are standard offset files supplied?

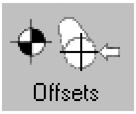
Note

For more information regarding Offsets and CNC terminology, please refer to Section 14 - CNC Theory.

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 tool profiles are used with the same tool numbers.
- The billet size does not change.
- The billet is always placed in the chuck/fixture in exactly the same position.

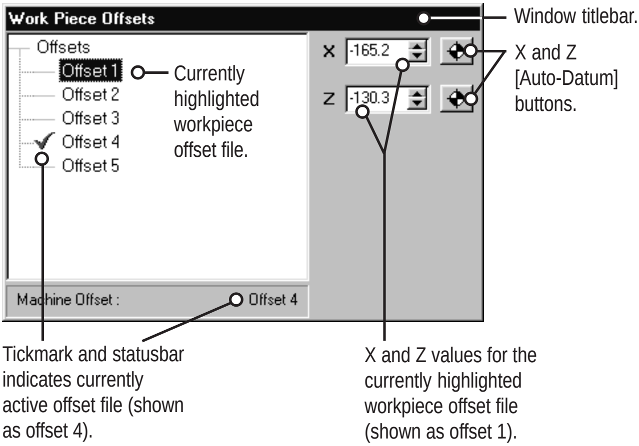
11: The Work Piece Offsets Window



The "Work Piece Offsets" window displays the various numbered collections of workpiece offsets available.

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

General Layout of the "Work Piece Offsets" Window.



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

11: Highlighting a Workpiece Offset File

Note

It is important to remember that the values shown in the "X" and "Z" readout boxes relate to the currently highlighted workpiece offset. This may not necessarily be the currently activated workpiece offset, which is shown using the red tickmark.

In order to view any values contained in a workpiece offset file, highlight the required file by clicking on its title. Highlighted workpiece offset files are shown using white title text on a blue background.

The co-ordinate readout boxes, to the right of the workpiece offsets list, display any "X" and "Z" values entered into the highlighted workpiece offset file. Any values entered into the "X" and "Z" readout boxes, either manually, or via the [Auto-Datum] buttons are registered to the currently highlighted workpiece offset file.

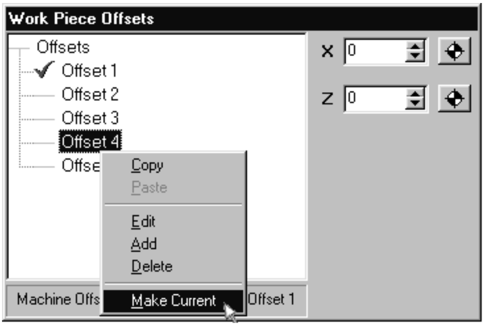
11: Activating a Workpiece Offset File

In order to view the effects of a workpiece offset file in the "Program" co-ordinate display of the "Control Panel" window, the required workpiece offset file title must be activated. A red tick mark is used to indicate the currently activated workpiece offset, also shown in the statusbar, positioned at the bottom of the "Work Piece Offsets" window.

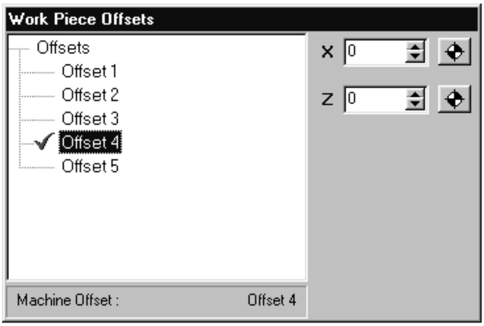
To activate a new workpiece offset file:

The example screenshots show changing the currently active offset file from number 1 to number 4.

1) Highlight the title of the required workpiece offset file.
2) Click the right mouse button on the title to display the pop-up menu.
3) Highlight and click the "Make Current" option, from the pop-up menu that is displayed, as shown below.

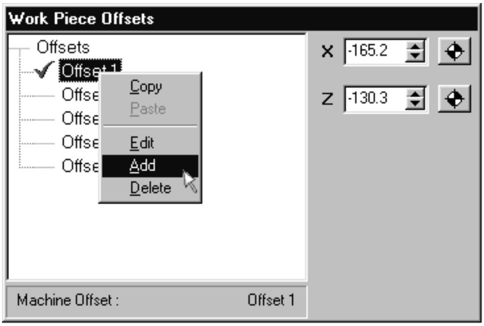


4) The red tickmark will now be displayed next to the title of the selected workpiece offset file, indicating it is currently active, as shown below.

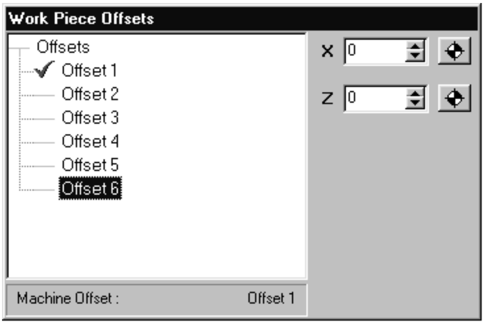


11: Creating a New Workpiece Offset File

1) Highlight any title from the workpiece offset list, then click the right mouse button to display the pop-up menu. Highlight and click the "Add" option, as shown below.



2) The new workpiece offset file will be added to the end of the workpiece offsets list. In the example below, there were 5 original workpiece offset files, so the new workpiece offset is numbered 6.



11: Loading and Saving Workpiece Offset Files

Note

In order to view the "Offsets" menu, the "Work Piece Offsets" window must be the active window in the VR CNC Turning software.

To Open a collection of Workpiece Offset Files:

From the VR CNC Turning software menubar, click "Offsets | Import...", to load a previously saved collection of workpiece offset files.

To Save a collection of Workpiece Offset Files:

From the VR CNC Turning software menubar, click "Offsets | Export..." to save the current collection of workpiece offset files, with a user defined filename.

Collections of workpiece offset files are saved using the file extension ".TOL".

11: Preparing for Workpiece Offsets

Note

For more information regarding preparation of tool profiles, please refer to Section 8 - Configuring the Tooling - pages 36 to 48.

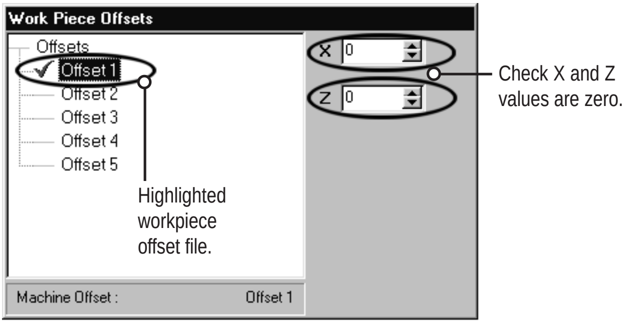
Prepare any Tool Profiles.

Click the [Tooling] button to display the "Tooling" window. Load or configure the tool profiles and tool numbers used with your CNC program.

Prepare the Workpiece Offset File.

Click the [Offsets] button to display the "Work Piece Offsets" window.

Highlight the title of the required workpiece offset, so any values can be registered in the correct numbered file.



If you want to view the true effects of any offset programming in the "Program" co-ordinate display of the "Control Panel" window, the following offset files must also be activated:

- The required workpiece offset file (refer to page 83).
- The tool offset file associated with the chosen tool number (refer to page 98).

We also recommend that any previous values entered in your workpiece and tool offset files are changed to zero before starting any offset configuration procedure.

Change the setting of the Co-ordinate Display Area.

Click the [Co-ordinates Display Select] button, in the "Control Panel" window, so "Program" is shown in the co-ordinates display area. This system shows any co-ordinate positions relative to the moveable workpiece datum (see page 68).

11: Preparing for Workpiece Offsets

Note

For more information regarding the theory of different offset programming methods, refer to pages 121-122.

Note

Notice that when the CNC machine is first started, the statusbar in the "Control Panel" window will always display "Tool: 1" with Tool "Offset: None".

In order to view the effects of the tool offset file on the "Program" co-ordinate display in the "Control Panel" window, the tool offset file associated with tool number 1 must be manually activated (refer to page 98).

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.

Home the CNC Machine.

We recommend the CNC machine is homed before starting any offset configuration procedure. In the "Control Panel" window, click the "Home" tab to enter Home mode, then click the [Both] button to home both the X and Z axes.

If you have just started a VR CNC machine, or connected to a real CNC machine, The "Control Panel" window will automatically open in Home mode. In this case, both axes MUST be homed before the CNC machine can be controlled manually.

Preparing a Reference Tool.

A reference tool can be used to help program any offsets. When a reference tool is used, all other tool profiles are measured in relation to the size of this reference tool.

If you are following our tutorials, we will use this method to program our offsets. Our reference tool is a roughing profile with tool number 1.

If you intend to use a reference tool, ensure that any "X" and "Z" values in the tool offset file associated with the reference tool number are set to zero.

Manually requesting a Tool Change.

In the "Control Panel" window, click the "Auto" tab to enter Auto mode.

In the "Tooling" window, click the numbered button containing the profile you want as your new tool, then click [Change Tool]. The tool change operation will be performed or prompted. Notice that the VR CNC Turning software automatically activates the tool offset file with the same number as the tool profile called, both items being displayed in the statusbar of the "Control Panel" window. To manually change the active tool offset file, refer to page 98.

If you are following our tutorial, change to tool number 1.

11: Configuring a Workpiece Offset (X Value)

> **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 Turning software and any offsets being used on your computer system.

> **Note**
>
> In order to move any of the machine axes, the background of the "Jog" tab must be highlighted in green - this shows that Jog Mode is active. If the background is grey, simply click on the "Jog" tab to make it active.
> To move the X machine axis use the **[Up Cursor]** and **[Down Cursor]** arrow keys.
> To move the Z machine axis use the **[Left Cursor]** and **[Right Cursor]** arrow keys.

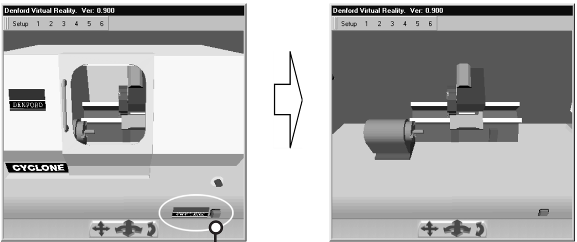
Pages 88 to 91 explain how to configure the X workpiece datum position, for use with the sample Metric CNC file, listed on pages 14 to 17. In our example, the X workpiece datum position is the centreline of the chuck.

We must jog (move) the reference tool to align with the X workpiece datum position before we can enter any values.

Sometimes the X workpiece datum position can be difficult to exactly locate. In our example, the X workpiece datum position is the centreline, which is invisible. In cases such as this, it is easier to jog and touch the tool onto a known physical position in the X axis. We would then have to measure and account for this known X position, when calculating the X workpiece offset value.

In our example, we will touch the reference tool (tool 1) on the outside surface of the billet diameter.

If you are using a VR CNC machine, click the blue square button next to the logo containing the text "remove guarding", on the front edge of the table to remove the CNC machine cabinet from view. This will make it much easier to see the billet and tool.



This button removes the machine cabinet from view.

In the "Control Panel" window, click the "Jog" tab to enter Jog mode - the tool can be moved manually in this mode.

Select Jog Continuous mode, by clicking the [Jog Step/Continuous] button, so a straight arrow graphic is displayed. When an axis key is pressed and held down, the tool will move in the appropriate direction, at the speed indicated in the "Jog Speed" readout box.

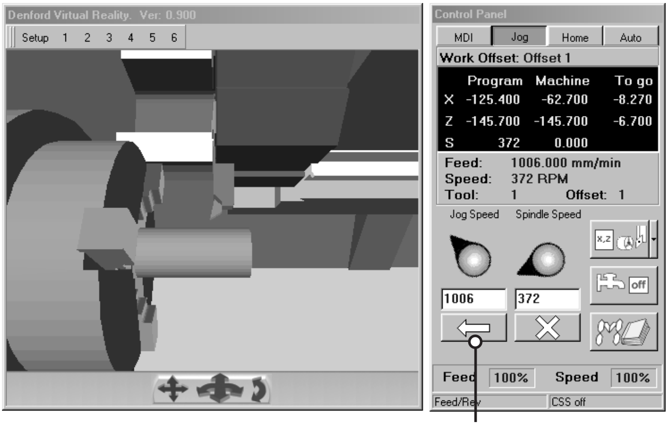
Move the tool until it is fairly close to the chosen X position (as shown in the screenshot at the top of the next page).

continued...

11: Configuring a Workpiece Offset (X Value)

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 Turning software and any offsets being used on your computer system.

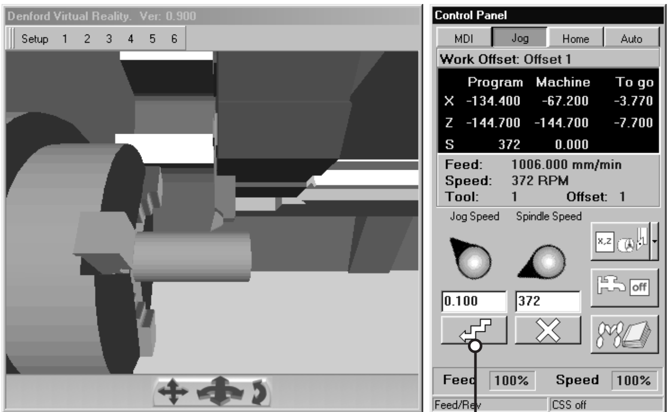


Jog Continuous Mode.

Change to Jog Step mode, by clicking the [Jog Step/Continuous] button, so a stepped arrow graphic is displayed. The tool will move one step in the appropriate direction, each time an axis key is pressed. The step amount is indicated in the "Jog Speed" readout box.

Move the tool until it is exactly aligned with (ie. just touching) the chosen X position. In our example screenshot below, the tool tip is just touching the outside diameter of the billet.

If you are using a real CNC machine, ensure (when possible) the spindle is running at an appropriate speed. To switch the spindle on, click the [M Codes] button and select the required M code. This will make it much easier to determine when the tool touches the billet and will also prevent damage to the tool tip.



Jog Step Mode.

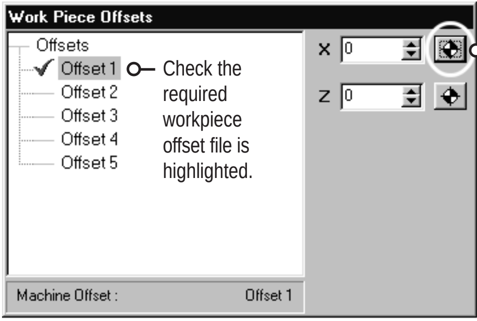
11: Entering the Workpiece Offset X Value

Note

Values can also be manually entered into the "X" and "Z" readout boxes of the highlighted workpiece offsets file. To register these new values with the co-ordinate display area of the "Control Panel" window, ensure the correct workpiece offset file is activated (see page 83), then close and reopen the "Work Piece Offsets" window, using the [Offsets] button.

Once the reference tool has been aligned either directly with the X workpiece datum position, or to a known position on the X axis, values can be entered into the "Work Piece Offsets" window.

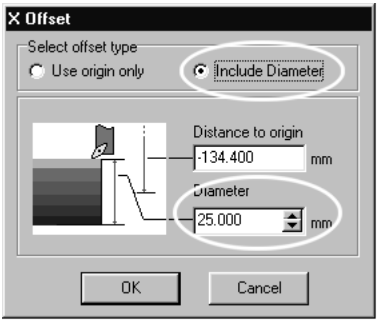
Click the [X Auto-Datum] button, to the right of the "X" readout box, in the "Work Piece Offsets" window.



[X Auto-Datum] button.

Note

Remember - Any X axis entries must always be input as diameter, NOT radius values.



The "X Offset" window will be displayed. Notice that the VR CNC Turning software automatically transfers a suitable value into the "Distance to origin" dialogue box.

When "Include Diameter" is selected, the "Diameter" dialogue box becomes active (as shown left).

Note

Important.
If a known point on the tool (rather than the tool tip itself) has been used to locate the X workpiece datum position, you must also manually add this known value to any other values in the "Diameter" dialogue box. For example, if we touched the side of a 6mm diameter drill onto the outer surface of a 25mm diameter billet, the "Diameter" dialogue value entered would be 31mm (6mm + 25mm).

Use this option when the tool has been touched onto a known position (such as the billet diameter), instead of directly onto the X workpiece datum position. If you are using a real CNC machine, measure the diameter of the billet to find the value entered into this dialogue box. If you are using a VR CNC machine, use the billet value contained at the beginning of your CNC program.

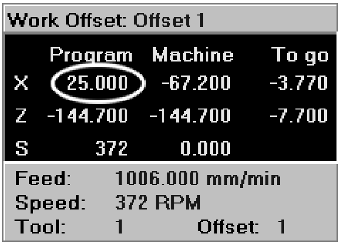
Click the [OK] button to confirm and apply the X workpiece offset value.

continued...

11: Entering the Workpiece Offset X Value

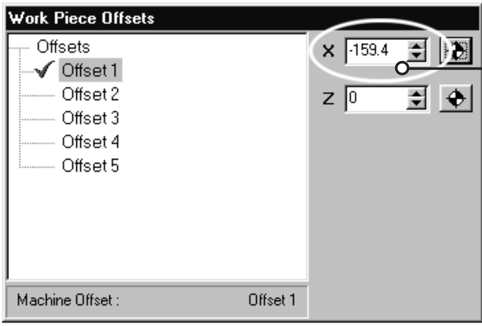
Note

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



Notice that the X co-ordinate in the "Program" display area of the "Control Panel" window display changes, indicating the X workpiece offset value has been registered.

In the example above, the display reads "25" (the "diameter" value entered into the "X Offset" window on the previous page), indicating that the tool is just touching the billet.



Workpiece Offset X Value.

The X workpiece offset value is automatically transferred into the "X" readout box, in the "Work Piece Offsets" window.

The value is sign sensitive (+ or -), indicating the direction of the position along the X axis, in relation to the machine datum.

11: Configuring a Workpiece Offset (Z Value)

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 Turning software and any offsets being used on your computer system.

Note

In order to move any of the machine axes, the background of the "Jog" tab must be highlighted in green - this shows that Jog Mode is active. If the background is grey, simply click on the "Jog" tab to make it active.
To move the X machine axis use the **[Up Cursor]** and **[Down Cursor]** arrow keys.
To move the Z machine axis use the **[Left Cursor]** and **[Right Cursor]** arrow keys.

Pages 92 to 95 explain how to configure the Z workpiece datum position, for use with the sample Metric CNC file, listed on pages 14 to 17. In our example, the Z workpiece datum position is the face end of the billet.

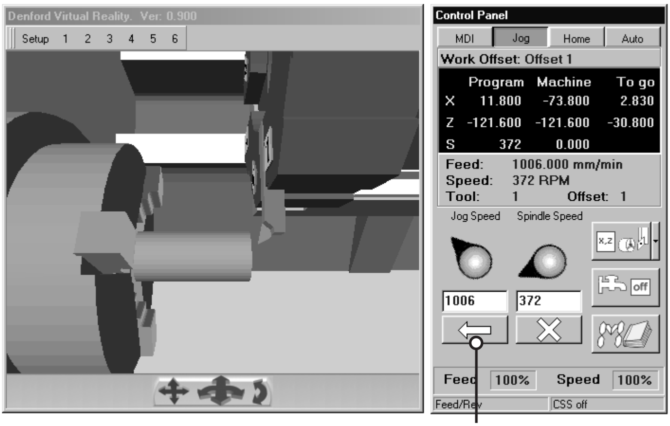
We must jog (move) the reference tool to align with the Z workpiece datum position before we can enter any values.

In our example, we can physically touch onto the Z workpiece datum position. If this were not possible, we would have to touch onto a known position in the Z axis, then measure and account for this known Z position, when calculating the Z workpiece offset value.

In the "Control Panel" window, click the "Jog" tab to enter Jog mode - the tool can be moved manually in this mode.

Select Jog Continuous mode, by clicking the [Jog Step/Continuous] button, so a straight arrow graphic is displayed. When an axis key is pressed and held down, the tool will move in the appropriate direction, at the speed indicated in the "Jog Speed" readout box.

Move the tool until it is fairly close to the chosen Z position (as shown in the screenshot below).



Jog Continuous Mode.

continued...

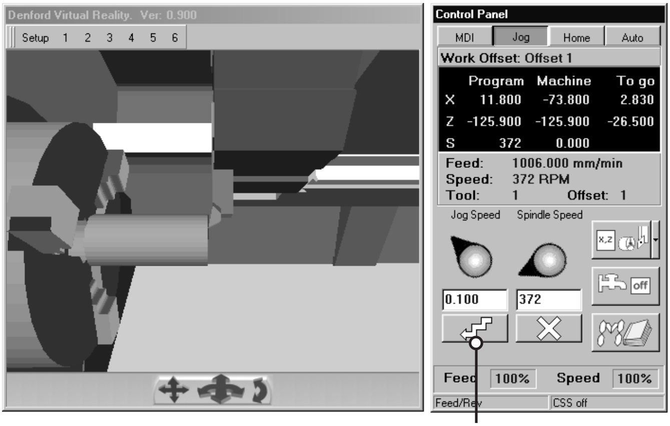
11: Configuring a Workpiece Offset (Z Value)

> **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 Turning software and any offsets being used on your computer system.

Change to Jog Step mode, by clicking the [Jog Step/Continuous] button, so a stepped arrow graphic is displayed. The tool will move one step in the appropriate direction, each time an axis key is pressed. The step amount is indicated in the "Jog Speed" readout box.

Move the tool until it is exactly aligned with (ie. just touching) the chosen Z position. In our example screenshot below, the tool tip is just touching the face end of the billet.

If you are using a real CNC machine, ensure (when possible) the spindle is running at an appropriate speed - this will make it much easier to determine when the tool touches the billet and will also prevent damage to the tool tip.



Jog Step Mode.

11: Entering the Workpiece Offset Z Value

Note

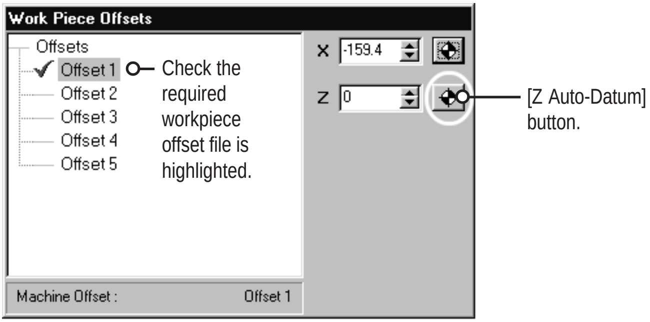
The numerical figures depicted on any screenshots will differ according to the CNC machine type, the units of measurement setting for the VR CNC Turning software and any offsets being used on your computer system.

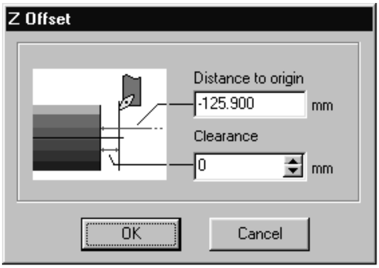
Note

Values can also be manually entered into the "X" and "Z" readout boxes of the highlighted workpiece offsets file. To register these new values with the co-ordinate display area of the "Control Panel" window, ensure the correct workpiece offset file is activated (see page 83), then close and reopen the "Work Piece Offsets" window, using the [Offsets] button.

Once the reference tool has been aligned either directly with the Z workpiece datum position, or to a known position on the Z axis, values can be entered into the "Work Piece Offsets" window.

Click the [Z Auto-Datum] button, to the right of the "Z" readout box, in the "Work Piece Offsets" window.





The "Z Offset" window will be displayed. Notice that the VR CNC Turning software automatically transfers a suitable value into the "Distance to origin" dialogue box.

The "Clearance" dialogue box provides the option to enter a known clearance value, which will be included when calculating the Z workpiece datum. Use this option when the tool has been touched onto a known position, instead of directly onto the Z workpiece datum position. The clearance value can also be used when a known point on the tool (rather than the tool tip itself) has been used to locate the Z workpiece datum position.

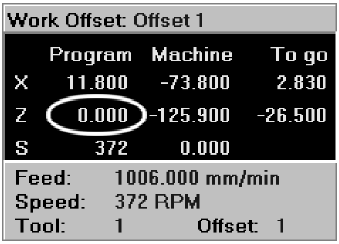
Click the [OK] button to confirm and apply the Z workpiece offset value.

continued...

11: Entering the Workpiece Offset Z Value

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 Turning software and any offsets being used on your computer system.



Notice that the Z co-ordinate in the "Program" display area of the "Control Panel" window changes, indicating the Z workpiece offset value has been registered.

In the example above, the display reads "0", indicating that the tool is just touching the face end of the billet.

The Z workpiece offset value is automatically transferred into the "Z" readout box, in the "Work Piece Offsets" window.

The value is sign sensitive (+ or -), indicating the direction of the position along the Z axis, in relation to the machine datum.

12: What are Tool Offsets?

> **Note**
>
> For more information regarding Offsets and CNC terminology, please refer to Section 14 - CNC Theory.

What are Tool Offsets?

Every tool profile is associated with its own individual tool offset file, containing X and Z offset values. They are used to compensate for the differences in size between all the various tool profiles used to machine a part.

How are they different to workpiece offsets?

Workpiece offsets are used to apply a global shift to the co-ordinate grid system of the CNC machine. In other words, the X and Z values contained in the currently active workpiece offset file are used with EVERY tool profile.

Tool offset files are different, since they are associated for use ONLY with SPECIFIC tool profiles.

How are tool offset values used?

The X position of the workpiece datum is defined by the X value entered in the workpiece offset file and the X value entered in the individual tool offset file associated with the tool profile currently in use.

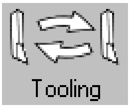
The Z position of the workpiece datum is defined by the Z value entered in the workpiece offset file and the Z value entered in the individual tool offset file associated with the tool profile currently in use.

Example.

It is common practice to use tool number 1 as a reference tool for locating and registering the position of the workpiece datum, using the workpiece offsets file. In this case, the X and Z values in the tool offset file associated with the reference tool would be zero.

Although the same workpiece offset values would be used by all other tool numbers, they would only describe the workpiece datum position for the reference tool profile, rather than their own particular shape. Values would have to be entered into their associated tool offset files to account for the small discrepancies in size between themselves and the reference tool.

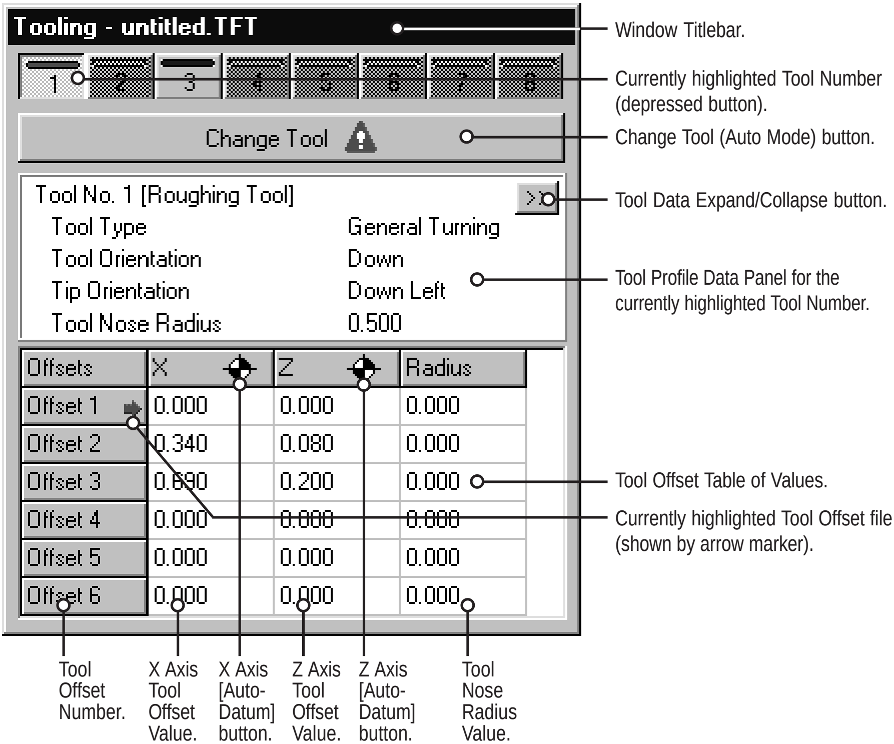
12: The Tool Offsets Window



The table of tool offset files is displayed in the "Tooling" window.

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

General Layout of the "Tooling" Window.



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

12: Highlighting a Tool Offset File

> **Note**
>
> The currently highlighted tool offset file may not necessarily be the currently activated tool offset, which is shown in the statusbar of the "Control Panel" window.

The tool offset table is displayed in the lower half of the "Tooling" window. To highlight a tool offset file, click its numbered button, from the first column in the table. A red arrow marker is used to indicate the currently highlighted tool offset file.

Any values entered into the "X" and "Z" readout boxes, either manually, or via the [Auto-Datum] buttons are registered to the currently highlighted tool offset file.

12: Activating a Tool Offset File

> **Note***
>
> **Important.**
> Configuring tool offsets will take into account the currently active workpiece offset - be sure to set your workpiece offsets first.

> **Note**
>
> The red arrow marker indicates the currently highlighted tool offset file.
>
> It does NOT show the tool offset number currently active with the CNC machine controller (this is shown in the "Control Panel" window statusbar).

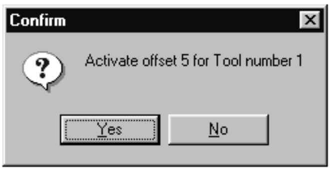
Every tool number used by a CNC program is associated with a numbered tool offset file. Most CNC programs are configured so these numbers are identical. For example, tool number 1 would be associated with tool offset file 1, tool number 2 with tool offset 2, tool number 3 with tool offset 3, etc...

Tool offset files are automatically loaded and activated when a CNC program is run. For example, running the program line M06 T0305 would change to tool profile number 3 and automatically load and activate any data from tool offset file number 5.

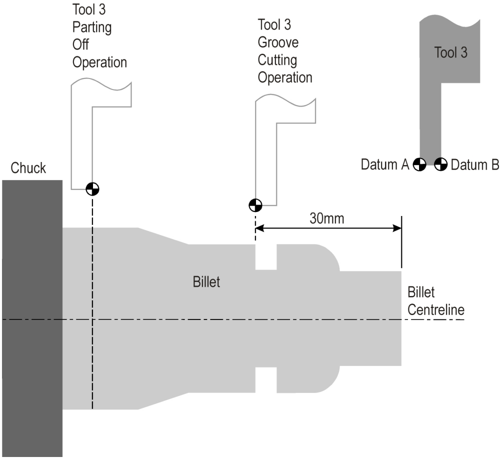
When a tool change is manually requested, the VR CNC Turning software automatically activates the tool offset file with the same number as the tool profile called. However, any of the tool offset files can be manually activated, in order to view their effect in the "Program" co-ordinate display area of the "Control Panel" window (see note* top left).

To manually activate a tool offset file:

1) Click the required [tool offset number] button, so the red arrow marker is displayed on the button itself
2) Make any adjustments, if necessary, to the X, Z and tool radius values.
3) Click on any other [tool offset number] button.
4) A "Confirm" window will be displayed asking whether you want to activate the new tool offset number (an example is shown left). Click the [Yes] button. Notice that the "Program" co-ordinates in the "Control Panel" window display area will change and the tool offset number in the statusbar will update, indicating the new tool offset file has been activated.



12: Activating a Tool Offset File



Example showing when different numbered tool offset files could be used with the same tool profile number:

"Tool 3" is used to cut a groove in the billet, 30mm from the face end. The part of the tool used to configure the tool offset for this grooving operation is datum "a". However, the same tool is also used for parting off. The part of the tool used to configure the tool offset for this parting off operation is datum "b". Although the same tool is used for both operations, two separate tool offset files are required, since two separate datums were used on "Tool 3".

12: Loading, Saving and Creating New Tool Offset Files

Note

In order to view the "Tooling" menu, the "Tooling" window must be the active window in the VR CNC Turning software.

To Open a collection of Tool Offsets:

From the VR CNC Turning software menubar, click "Tooling | Open...", to load a previously saved collection of tool offset files.

To Save a collection of Tool Offsets:

From the VR CNC Turning software menubar, click "Tooling | Save As..." to save the current collection of tool offset files and tool number/profiles, with a user defined filename.

Collections of tool offset files are saved, together with appropriate tool number and profile data, using the file extension ".TFT".

To create a New (blank) Tool Offsets Table:

From the VR CNC Turning software menubar, click "Tooling | New". A new, blank tool numbers/profiles list and blank tool offset table will be created.

12: Preparing for Tool Offsets

Note

Important.
Configuring tool offsets will take into account the currently active workpiece offset - be sure to set your workpiece offsets first.

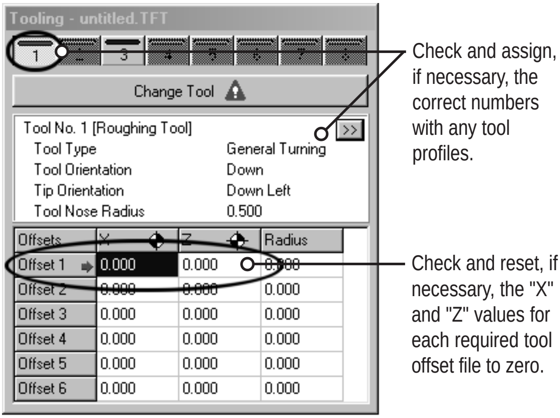
Prepare Tooling and zero any Tool Offset File values.

Click the [Tooling] button to display the "Tooling" window. Assign numbers to tool profiles, according to your CNC program (see pages 47-48).

The tutorial CNC program uses the following tool profiles:

- Tool 1: roughing profile.
- Tool 3: threading profile.

Set the "X" and "Z" values for each numbered tool offset file you want to configure at zero (any previous values can be edited by overtyping).



Change the setting of the Co-ordinate Display Area.

Click the [Co-ordinates Display Select] button, in the "Control Panel" window, so "Program" is shown in the co-ordinates display area. This system shows any co-ordinate positions relative to the moveable workpiece datum (see page 68).

Home the CNC Machine.

We recommend the CNC machine is homed before starting any offset configuration procedure. In the "Control Panel" window, click the "Home" tab to enter Home mode, then click the [Both] button to home both the "X" and "Z" axes.

If you have just started a VR CNC machine, or connected to a real CNC machine, The "Control Panel" window will automatically open in Home mode. In this case, both axes MUST be homed before the CNC machine can be controlled manually.

12: Preparing for Tool Offsets

Note

Important.
Configuring tool offsets will take into account the currently active workpiece offset - be sure to set your workpiece offsets first.

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.

Configuring Tool Offsets for our Tutorial.

Each tool profile number is associated with a numbered tool offset file, according to the definitions in your CNC program.

The tutorial CNC program, listed on pages 14 to 17, uses the following tool offsets:

- Tool 1 is associated with tool offset file number 1.
- Tool 3 is associated with tool offset file number 3.

The X and Z values for tool offset file number 1 must be entered as zero. This is because tool 1 was used as a reference tool for locating and registering the position of the workpiece datum, via the workpiece offsets table.

Remember, any values entered into the active workpiece offset file are used by every tool profile, so tool 3 will use the same workpiece offset file as tool 1. However, these values alone will not describe the exact location of the workpiece datum for tool 3. This is because tool 1 and tool 3 are different sizes. To account for this size difference, we configure tool offset file number 3, since it will only be used with tool 3.

Manually requesting a Tool Change.

In the "Control Panel" window, click the "Auto" tab to enter Auto mode.

In the "Tooling" window, click the numbered button containing the profile you want as your new tool, then click [Change Tool]. The tool change operation will be performed or prompted. Notice that the VR CNC Turning software automatically activates the tool offset file with the same number as the tool profile called, both items being displayed in the statusbar of the "Control Panel" window. If you need to manually change the active tool offset file, refer to page 98.

If you are following our tutorial, change to tool number 3.

Highlight the required Tool Offset Number.

The "Tooling" window contains a table of numbered tool offset files. When [Auto-Datum] buttons are used to input any tool offsets, the values are always entered into the currently highlighted tool offset file, indicated by a red arrow marker. Simply click on the numbered [offset] buttons to change the currently highlighted tool offset file.

If you are following our tutorial, highlight tool offset file number 3.

12: Configuring a Tool Offset (X Value)

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 Turning software and any offsets being used on your computer system.

Note

In order to move any of the machine axes, the background of the "Jog" tab must be highlighted in green - this shows that Jog Mode is active. If the background is grey, simply click on the "Jog" tab to make it active.
To move the X machine axis use the **[Up Cursor]** and **[Down Cursor]** arrow keys.
To move the Z machine axis use the **[Left Cursor]** and **[Right Cursor]** arrow keys.

Note

Important.
Configuring tool offsets will take into account the currently active workpiece offset - be sure to set your workpiece offsets first.

Pages 102 to 109 explain how to configure the tool offsets, for use with the sample Metric CNC file, listed on pages 14 to 17. In our example, the X workpiece datum position is the centreline of the chuck.

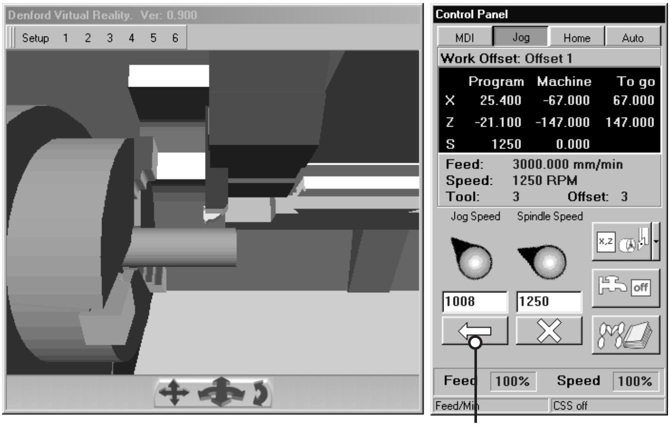
We must jog (move) the tool profile to align with the X workpiece datum position before we can enter any tool offset values. This can be either the X workpiece datum position, or a known position on the X axis.

In our example, we will touch the threading profile (tool 3) on the outside surface of the billet diameter.

In the "Control Panel" window, click the "Jog" tab to enter Jog mode - the tool can be moved manually in this mode.

Select Jog Continuous mode, by clicking the [Jog Step/Continuous] button, so a straight arrow graphic is displayed. When an axis key is pressed and held down, the tool will move in the appropriate direction, at the speed indicated in the "Jog Speed" readout box.

Move the tool until it is fairly close to the chosen X position (as shown in the screenshot below).



Jog Continuous Mode.

continued...

12: Configuring a Tool Offset (X Value)

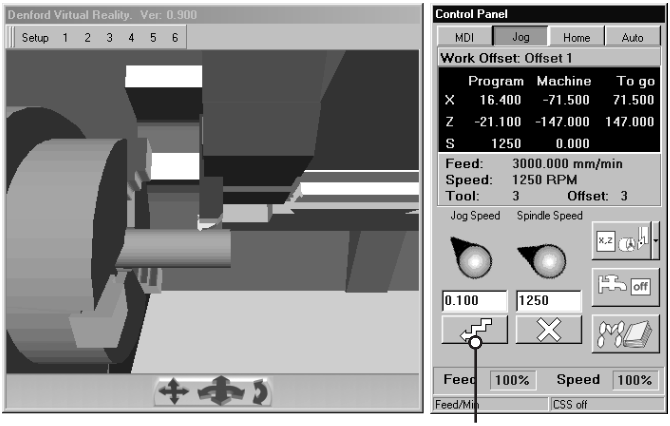
Note

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

Change to Jog Step mode, by clicking the [Jog Step/Continuous] button, so a stepped arrow graphic is displayed. The tool will move one step in the appropriate direction, each time an axis key is pressed. The step amount is indicated in the "Jog Speed" readout box.

Move the tool until it is exactly aligned with (ie. just touching) the chosen X position. In our example screenshot below, the tool tip is just touching the outside diameter of the billet.

If you are using a real CNC machine, ensure (when possible) the spindle is running at an appropriate speed. To switch the spindle on, click the [M Codes] button and select the required M code. This will make it much easier to determine when the tool touches the billet and will also prevent damage to the tool tip.



Jog Step Mode.

12: Entering the Tool Offset X Value

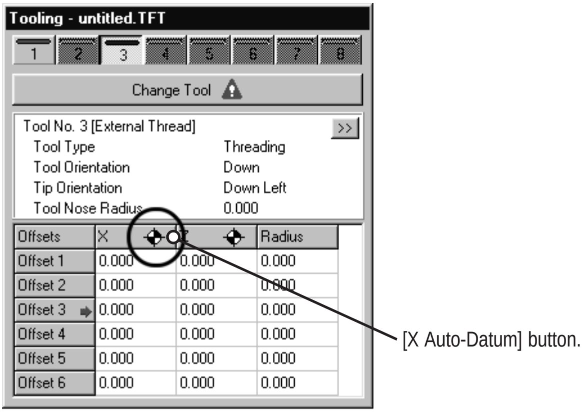
> **Note**
>
> Values can also be manually entered into the "X" and "Z" columns of the highlighted tool offset file.
> To register these new values with the co-ordinate display area of the "Control Panel" window, ensure the correct tool offset file number is activated (see page 98), then close and reopen the "Tooling" window, using the [Tooling] button.

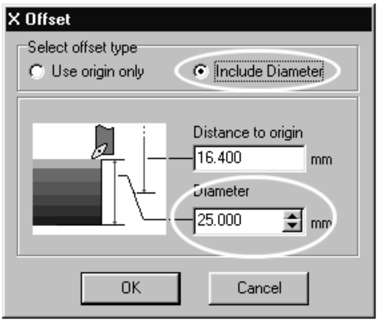
> **Note**
>
> Remember - Any X axis entries must always be input as diameter, NOT radius values.

> **Note**
>
> **Important.**
> If a known point on the tool (rather than the tool tip itself) has been used to locate the X workpiece datum position, you must also manually add this known value to any other values in the "Diameter" dialogue box. For example, if we touched the side of a 6mm diameter drill onto the outer surface of a 25mm diameter billet, the "Diameter" dialogue value entered would be 31mm (6mm + 25mm).

Once the tool has been aligned either directly with the X workpiece datum position, or to a known position on the X axis, values can be entered into the tool offsets table in the "Tooling" window.

Click the [X Auto-Datum] button, at the top of the X value column.





The "X Offset" window will be displayed. Notice that the VR CNC Turning software automatically transfers a suitable value into the "Distance to origin" dialogue box. It is important to remember that this value also takes into account the X component of any active workpiece offsets file.

When "Include Diameter" is selected, the "Diameter" dialogue box becomes active. Use this option when the tool has been touched onto a known position (such as the billet diameter), instead of directly onto the X workpiece datum position. If you are using a real CNC machine, measure the diameter of the billet to find the value entered into this dialogue box. If you are using a VR CNC machine, use the billet value contained at the beginning of your CNC program.

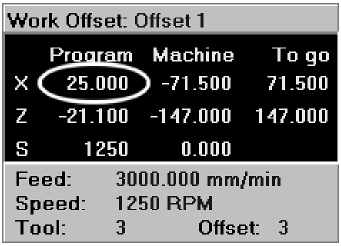
Click the [OK] button to confirm and apply the X tool offset value.

continued...

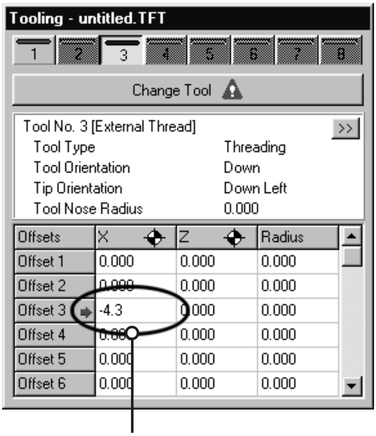
12: Entering the Tool Offset X Value

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 Turning software and any offsets being used on your computer system.



Notice that the X program co-ordinate in the "Program" display area of the "Control Panel" window display changes, indicating the X tool offset value has been registered. In the above example, the display reads "25" (the "diameter" value entered into the "X Offset" window on the previous page), indicating the tool is just touching the billet.



X Value for Tool Offset Number 3.

The X tool offset value is automatically transferred into the "X" column of the tool offsets table. The value is sign sensitive (+ or -), indicating the direction of the position along the X axis, in relation to the machine datum.

12: Configuring a Tool Offset (Z Value)

Pages 102 to 109 explain how to configure the tool offsets, for use with the sample Metric CNC file, listed on pages 14 to 17. In our example, the Z workpiece datum position is the face end of the billet.

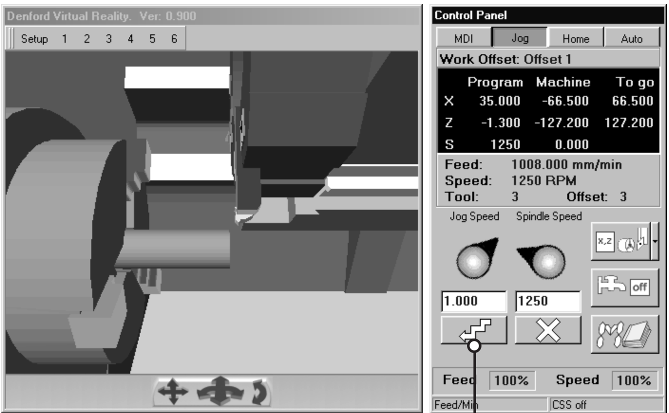
We must jog (move) the tool profile to align with the Z workpiece datum position before we can enter any tool offset values. This can be either the Z workpiece datum position, or a known position on the Z axis.

In our example, we must touch the threading profile (tool 3) on the face end perimeter edge of the billet. Due to the shape of the tool, this is the only part of the face end of the billet that can be reached by the tool tip.

In the "Control Panel" window, click the "Jog" tab to enter Jog mode - the tool can be moved manually in this mode.

Select Jog Continuous mode, by clicking the [Jog Step/Continuous] button, so a straight arrow graphic is displayed. When an axis key is pressed and held down, the tool will move in the appropriate direction, at the speed indicated in the "Jog Speed" readout box.

Move the tool until it is fairly close to the chosen Z position (as shown in the screenshot below).



Jog Continuous Mode.

continued...

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 Turning software and any offsets being used on your computer system.

Note

In order to move any of the machine axes, the background of the "Jog" tab must be highlighted in green - this shows that Jog Mode is active. If the background is grey, simply click on the "Jog" tab to make it active.
To move the X machine axis use the **[Up Cursor]** and **[Down Cursor]** arrow keys.
To move the Z machine axis use the **[Left Cursor]** and **[Right Cursor]** arrow keys.

Note

Important.
Configuring tool offsets will take into account the currently active workpiece offset - be sure to set your workpiece offsets first.

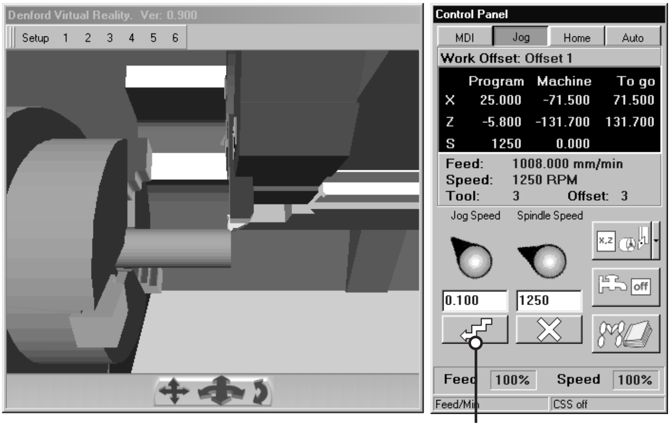
12: Configuring a Tool Offset (Z Value)

> **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 Turning software and any offsets being used on your computer system.

Change to Jog Step mode, by clicking the [Jog Step/Continuous] button, so a stepped arrow graphic is displayed. The tool will move one step in the appropriate direction, each time an axis key is pressed. The step amount is indicated in the "Jog Speed" readout box.

Move the tool until it is exactly aligned with (ie. just touching) the chosen Z position. In our example screenshot below, the tool tip is just touching the face end perimeter of the billet.

If you are using a real CNC machine, ensure (when possible) the spindle is running at an appropriate speed. To switch the spindle on, click the [M Codes] button and select the required M code. This will make it much easier to determine when the tool touches the billet and will also prevent damage to the tool tip.



Jog Step Mode.

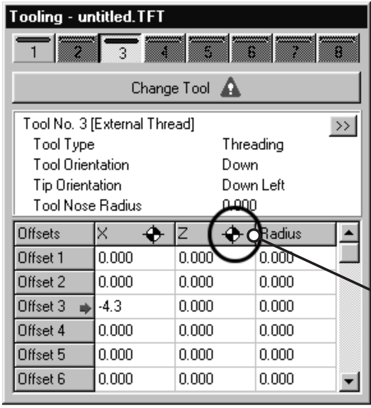
12: Entering the Tool Offset Z Value

> **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 Turning software and any offsets being used on your computer system.

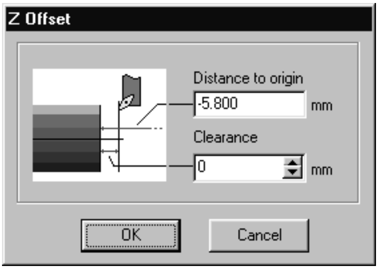
> **Note**
>
> Values can also be manually entered into the "X" and "Z" columns of the highlighted tool offset file.
> To register these new values with the co-ordinate display area of the "Control Panel" window, ensure the correct tool offset file number is activated (see page 98), then close and reopen the "Tooling" window, using the [Tooling] button.

Once the tool has been aligned either directly with the Z workpiece datum position, or to a known position on the Z axis, values can be entered into the tool offsets table in the "Tooling" window.

Click the [Z Auto-Datum] button, at the top of the Z value column.



[Z Auto-Datum] button.



The "Z Offset" window will be displayed. Notice that the VR CNC Turning software automatically transfers a suitable value into the "Distance to origin" dialogue box. It is important to remember that this value also takes into account the X component of any active workpiece offsets file.

The "Clearance" dialogue box provides the option to enter a known clearance value, which will be included when calculating the Z workpiece datum. Use this option when the tool has been touched onto a known position, instead of directly onto the Z workpiece datum position. The clearance value can also be used when a known point on the tool (rather than the tool tip itself) has been used to locate the Z workpiece datum position.

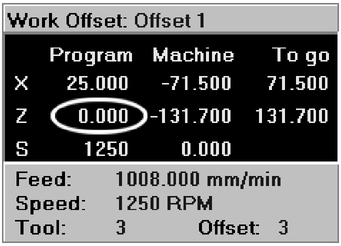
Click the [OK] button to confirm and apply the Z tool offset value.

continued...

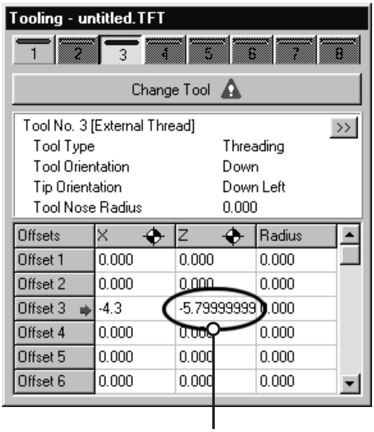
12: Entering the Tool Offset Z Value

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 Turning software and any offsets being used on your computer system.



Notice that the Z program co-ordinate in the "Program" display area of the "Control Panel" window display changes, indicating the Z tool offset value has been registered. In the above example, the display reads "0", indicating the tool is just touching the face end of the billet.



Z Value for Tool Offset Number 3.

The Z tool offset value is automatically transferred into the "Z" column of the tool offsets table. The value is sign sensitive (+ or -), indicating the direction of the position along the Z axis, in relation to the machine datum.

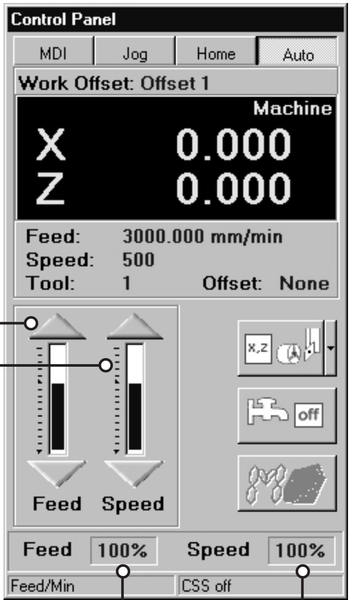
13: Running a CNC file on a CNC Machine (Auto Mode).

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

Auto Mode.

Feedrate override adjustment buttons and slider bar.

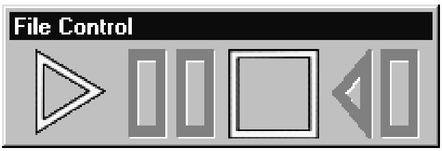
Spindle Speed override adjustment buttons and slider bar.



Feedrate override status.

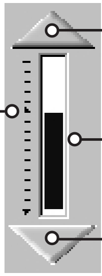
Spindle Speed override status.

Auto Mode is used for controlling the CNC machine when running a CNC program. To activate the Auto mode panel, click the "Auto" tab in the "Control Panel" window.



To start running the CNC program, ensure the "Editor" window cursor is positioned at the start of the first line of the CNC file (click the [Rewind to Start] button, if required). Click the triangular [Play] button from the "File Control" toolbar, shown above, to start the CNC machine in Auto mode.

13: Feedrate & Spindle Speed Overrides.



Left:
There are two sliderbars - the sliderbar above the "Feed" text relates to the feedrate override, the sliderbar above the "Speed" text relates to the spindle speed override.

Note

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

Software Feedrate Override - "Feed" panel.

Feedrate override values between 1-100% are set in the area of the slider bar below the triangular marker, with the associated percentage value indicated in the statusbar using blue text. Feedrate override values between 100-150% are set in the area of the slider bar above the triangular marker, with the associated percentage value indicated in the statusbar using red text.

To increase the feedrate override, click the triangular [Up] button. To decrease the feedrate override, click the triangular [Down] button. The feedrate override can be increased or decreased in 10% steps, one step applied per button click.

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.

Note

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

Software Spindle Speed Override - "Speed" panel.

Spindle speed override values between 1-100% are set in the area of the slider bar below the triangular marker, with the associated percentage value indicated in the statusbar using blue text. Spindle speed override values between 100-150% are set in the area of the slider bar above the triangular marker, with the associated percentage value indicated in the statusbar using red text.

To increase the spindle speed override, click the triangular [Up] button. To decrease the spindle speed override, click the triangular [Down] button. The spindle speed override can be increased or decreased in 10% steps, one step applied per button click.

The spindle speed is measured using revolutions per minute.

14: Homing the Machine

Immediately after being switched on, the X and Z axes of the CNC machine must be homed.

When you home the CNC machine, both axes will move to the furthest positions available on their slides.

On a back turret / slant bed lathe, the turret will move back (away from you) and to the right, when viewed from the front of the machine.

On a standard front turret machine (eg. Microturn), the turret will move forwards (towards you) and to the right, when viewed from the front of the machine.

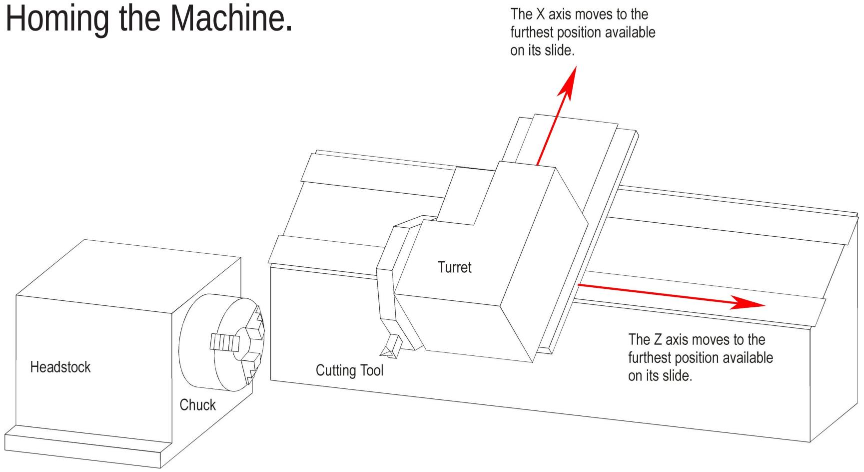
These movements are always positive because all movements into or towards the billet are negative.

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 any offsets.

Homing the Machine.

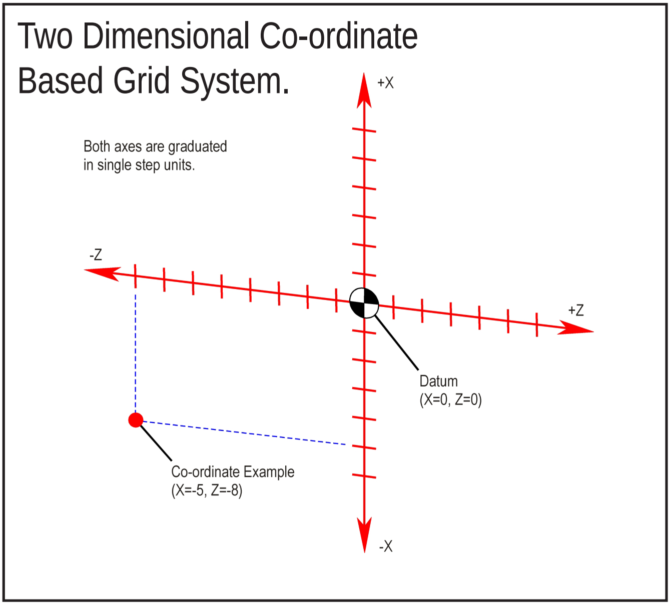


14: The Co-ordinate based Grid System

Precise points are plotted on the CNC machine using the positions of the X and Z axes.

These X and Z values relate to a two 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 two CNC machine axes.



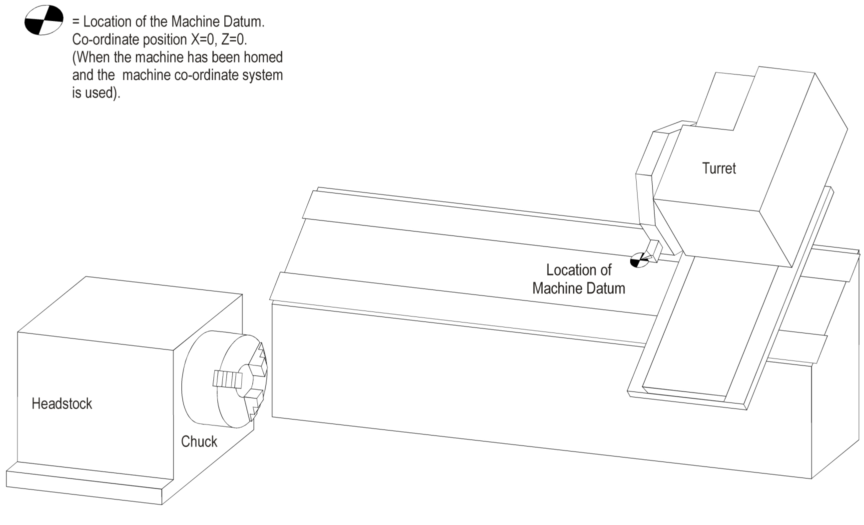
14: The Machine Datum

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 profile in the turret, then home both axes, we can describe the machine datum as being the extreme edge of the cutting tool tip. The position of the machine datum, when using machine co-ordinates, will be X=0, Z=0, as shown in the diagram below.

The Machine Datum.



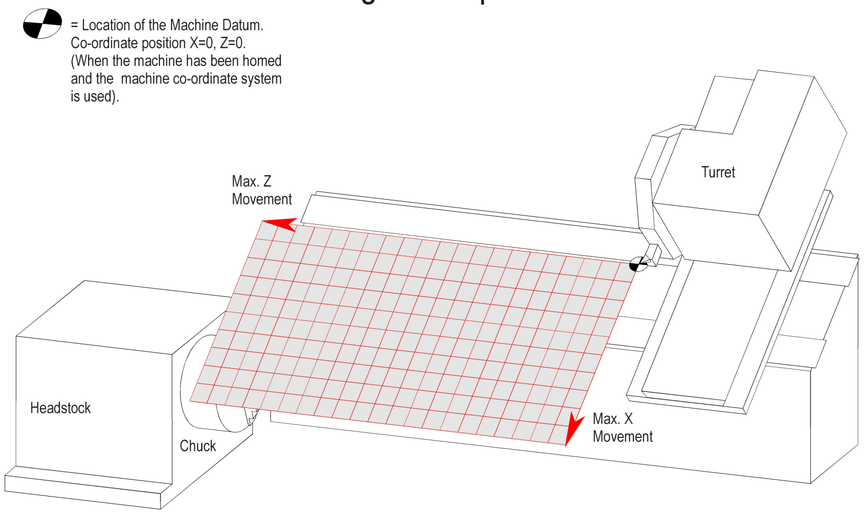
14: The CNC Machine Working Envelope

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

The shaded block represents the maximum working envelope of the CNC machine, effectively the full length of movement in each of the two axes. In other words, most tool tips can effectively be positioned within this shaded block.

Note that the working envelope extends into the headstock area of the CNC machine, so the possibility of crashing the tool into the chuck will always exist. Before machining on a real CNC lathe, always try to use the VR CNC machines and 2D or 3D simulations first, to confirm that your CNC programs will not do anything unexpected.

The CNC Machine Working Envelope.



14: Machine Co-ordinates Display Mode

Co-ordinate System Display Modes

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

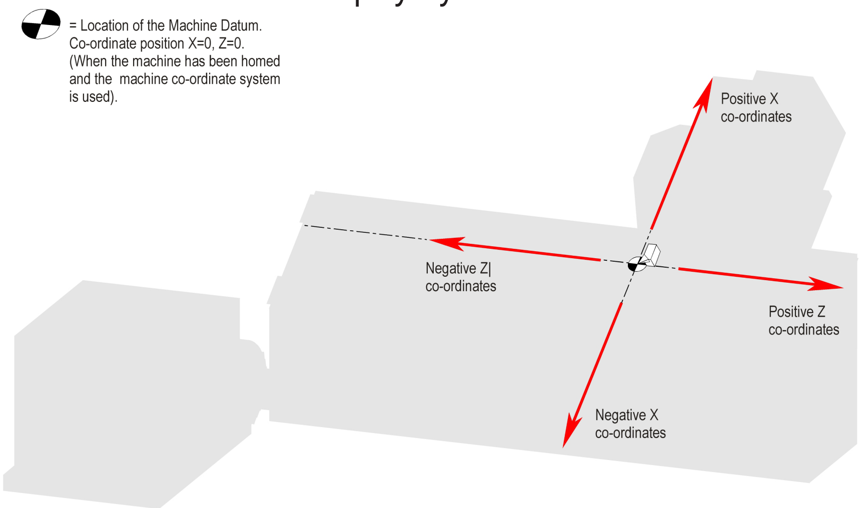
1) **Machine Co-ordinates**: This mode is operating when the word "Machine" is shown in the co-ordinate display panel.

Any co-ordinate values plotted 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 by the position X=0, Z=0.

Machine Co-ordinates Display System.



14: Workpiece Co-ordinates Display Mode

Co-ordinate System Display Modes

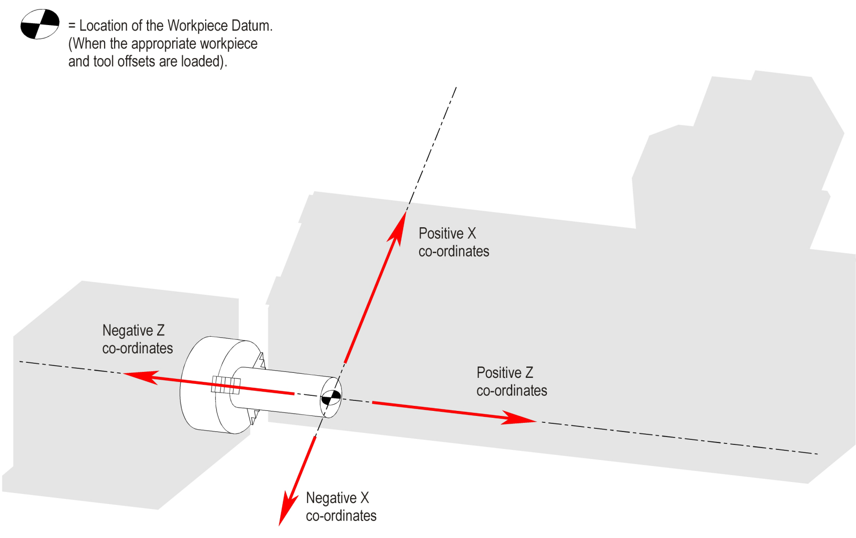
2) **Workpiece Co-ordinates**: This mode is operating when the word "Program" is shown in the co-ordinate display panel.

Any co-ordinate values plotted relate to the programmed workpiece datum. The co-ordinate display shows the position of the machine when any offsets are being used.

The workpiece datum position, described through use of the offset facility, 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.

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

Workpiece (Program) Co-ordinates Display System.



14: Configuring Offsets

The CNC program zero reference - the Part Datum.

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.

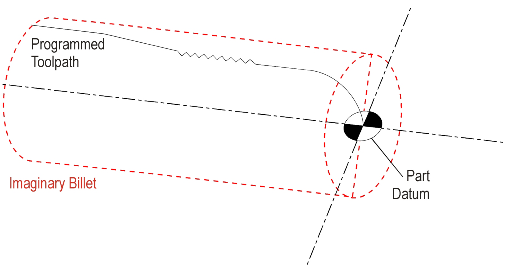
We recommend that the Z co-ordinate of the part datum is set to the front face end of the part. This is the end of the billet that will not be held in the chuck. In doing so, any negative Z values programmed will indicate that the tool is cutting into the billet, any positive Z values programmed will indicate that the tool is clear from the billet.

We recommend that the X co-ordinate of the part datum is set to the centreline of the billet. When the billet is a cylindrical bar, this will also be the centreline of the chuck. In doing so, any X values programmed would signify the diameter or radius of any cutting paths.

However, the part datum can be positioned anywhere. It could be positioned in one of the corners of a part design drawing, or with the chuck face being Z zero. In most CAD/CAM software packages it may be set automatically when the CNC program is generated.

In the example below, we have positioned the part datum at the face end of the cylindrical billet, along the centreline.

Positioning the Part Datum



The CNC machine zero reference - the Machine Datum.

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.

14: Configuring Offsets

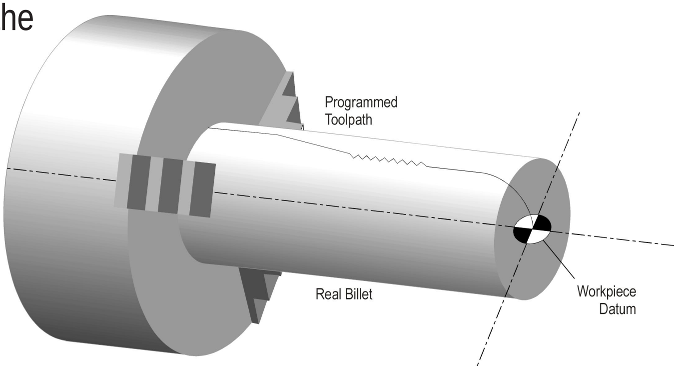
The zero reference for the CNC program on the CNC machine - the Workpiece Datum.

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

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 our example below, with the position of the part datum in the diagram on the previous page - they are identical. If these datums were not identical, the toolpath would be machined in the wrong place on the real billet.

Positioning the Workpiece Datum



14: Configuring Offsets

Two different types of offset files are used, in combination, to describe the exact location of the workpiece datum for each tool profile:

- **Workpiece Offsets**

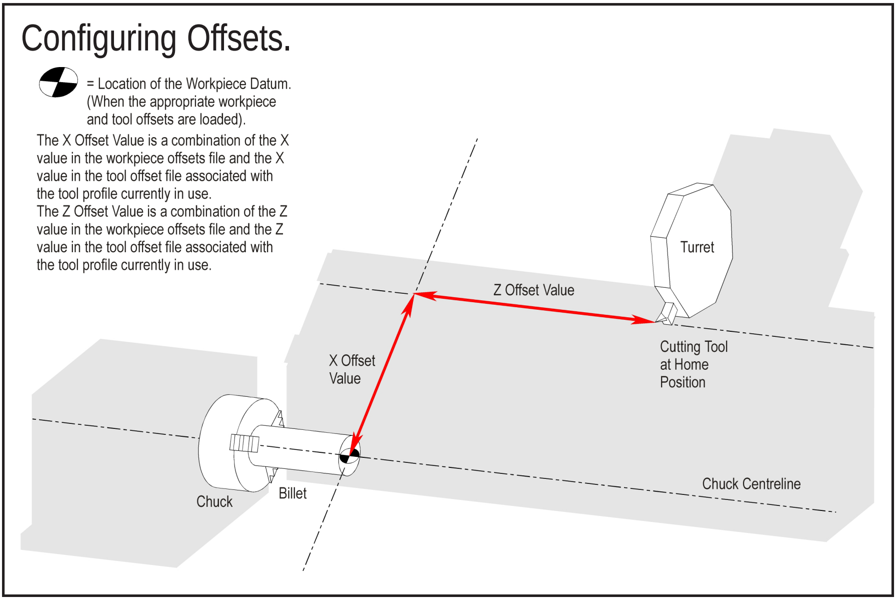
  Configuring a workpiece offset shifts the co-ordinate based grid system of the machine, by the X and Z amounts programmed into the file. The values in the active workpiece offsets file are used by EVERY tool number.

  Usually, only one workpiece offset file needs to be configured for a CNC program. In this case, the values contained in this single workpiece offset file are used by all the tool numbers.

- **Tool Offsets**

  Configuring a tool offset shifts the co-ordinate based grid system of the machine, by the X and Z amounts programmed into the file. The values in each tool offset file are associated for use with SPECIFIC tool numbers. There may be occasions when the same tool number is associated with more than one tool offset (see the example on page 99).

  They allow a variety of tool profiles to be used together on the same CNC program, by offsetting any differences in size against a common fixed reference point. Whenever a tool number is called by the CNC machine, any values in the associated numbered tool offset file are automatically loaded and applied to the CNC machine control systems.



14: Configuring Offsets

Recap.

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 your workpiece and tool offsets, the same files may be used over and over again, as long as the following holds true:

- The same cutting tool profiles are used with the same tool numbers.
- The billet size does not change.
- The billet is always placed in the chuck/fixture in exactly the same position.

Bearing this in mind, collections of offset files can be created and saved for specific projects, to reduce future setup time.

Example - Configuring Offsets using a Reference Tool.

This method is commonly used to configure offsets against a workpiece datum positioned along the billet/chuck centreline, at the exposed face end of the billet.

1) Select a tool profile number as a reference tool. Drive the cutting tip of the reference tool to align exactly against the workpiece datum position.
2) Enter X and Z values in the workpiece offset file. Leave the X and Z values in the reference tool offset file at zero, since the tool profile is already aligned against the workpiece datum.

   If the workpiece datum position cannot be located directly with the reference tool tip, touch onto known points in the X and Z axes, then account for these points when inputting the X and Z values into the workpiece offset file. Similarly, account for any values when a known point on the tool has been used as a touching on point, rather than the tool tip itself.

   For example, use the side of a drill to touch onto the side of the billet, when locating the X workpiece datum position, then account for the diameter of both the drill and the billet when calculating the X workpiece offset file value.
3) Change to the next tool profile number. Drive the cutting tip of this tool to align exactly against the workpiece datum position. Note that the previously configured workpiece offset file values entered in section 2) will now be active for this tool.
4) Enter X and Z values in the tool offset file associated with the tool profile number being used. Remember to account for any known points used to help determine the workpiece datum position.
5) Repeat sections 3) and 4) for all remaining tool profile numbers.

14: Configuring Offsets

Example - Configuring Offsets with a Global Co-ordinate Shift.

This method is commonly used to configure offsets that allow the workpiece datum to be moved at a later date. In order for this to be achieved, the workpiece offset file is used to apply a global shift of identical X and Z values across all tool profiles.

1) Ensure the currently used workpiece offsets file contains zero values before continuing. Drive the cutting tip of the first tool profile number to align against a chosen datum position - our reference datum.
2) Enter X and Z values in the tool offsets file associated with the first tool profile number. If the reference datum position cannot be located directly with the tool tip, touch onto known points in the X and Z axes, then account for these points when inputting the X and Z values into the tool offset file. Similarly, account for any values when a known point on the tool has been used as a touching on point, rather than the tool tip itself.
3) Change to the next tool profile number. Drive the cutting tip of this tool to align against the reference datum used in section 1). Note that the same reference datum must be used by all the tool profile numbers.
4) Enter X and Z values in the tool offsets file associated with that tool profile number. Remember to account for any known points used to help determine the reference datum position.
5) Repeat sections 3) and 4) for all remaining tool profile numbers.
6) At this moment, the workpiece datum for all tool profiles is set at the position of the reference datum from section 1). Remember, the reference datum is used by all tool profiles. In order to shift this to our true workpiece datum, manually enter appropriate X and Z co-ordinate shift values into the workpiece offsets file, then register these values with the CNC machine controller, so they become active.

14: Radius and Diameter Programming

Two different modes are available for reading CNC program information:

- **Diameter Programming.**

  All X values programmed and read by the CNC controller are relative to the overall diameter of the billet.

  This is the most common form of CNC lathe programming because it is much easier to understand where the tool is positioned and what sizes are being cut.

  For example, if your billet was 30mm diameter and you programmed X25 Z0, followed by X25 Z-100 you would turn a 100mm long section of your billet down to a 25mm diameter (see the diagram below).

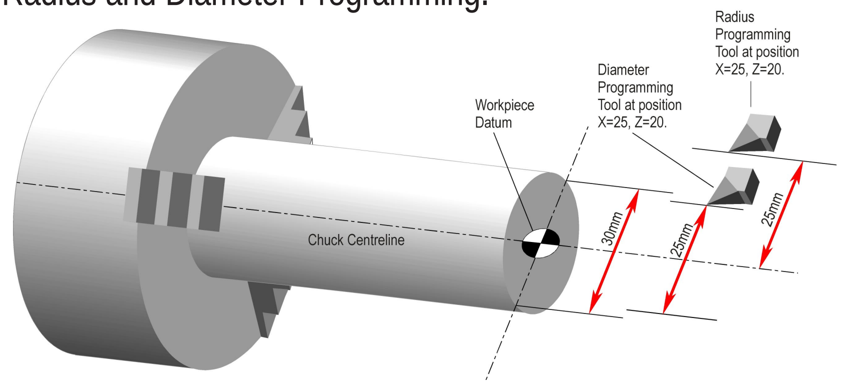
- **Radius Programming.**

  All X values programmed and read by the CNC controller are relative to the radius of the part.

  Radius programming is less common but can be used to make CNC program creation easier if your dimensioned drawings show only the profile of the turned part (ie. all dimensions are shown as radius values).

  For example, if you tried to turn a 30mm diameter billet with a value of X25, no material would be cut. X25 in radius programming would mean cutting a diameter of 50mm (see the diagram below).

Radius and Diameter Programming.



15: Glossary

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

ARC .... A portion of a circle.

ATC .... See programmable turret.

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 lathe (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 Z (left and right, parallel to the chuck centreline) and X (forwards and backwards, at right angles to the chuck centreline). Combinations of both allow precise co-ordinates to be described.

BED .... The base of the CNC lathe.

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.

CAROUSEL .... The rotating section of a programmable turret, containing a number of stations. Also referred to as the turret disc.

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

CHUCK .... The clamping system that holds the billet, driven by the spindle.

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 shortcut keys are used to call a Context Sensitive helpfile, the pages displayed relate directly to the current active elements within the software.

CO-ORDINATES .... Positions or relationships of points or planes. Co-ordinates are usually described using two numbers referring to the X and Z axes, eg. the co-ordinate X-14, Z-10 means -14 units along the X axis and -10 units along the Z axis.

CNC .... Computer Numerical Control.

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

CUTTING SPEED .... The velocity of the rotating workpiece relative to the stationary cutting edge of the tool.

CROSS-SLIDE .... The physical X axis on the CNC lathe.

CSS .... Constant Surface Speed. The spindle is automatically adjusted according to the diameter being machined, so that the material always passes the cutting edge of the tool at a constant rate.

15: Glossary

CYCLE .... A sequence of events or commands.

DATUM .... The zero 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. Replaces the qwerty keyboard and mouse.

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

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

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

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

DWELL .... A programmed time delay.

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

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

FEEDRATE .... The rate, in mm/min or in/min at which the cutting tool is advanced into the workpiece.

FILE .... An arrangement of instructions or information, usually referring to work or control settings.

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

FNL .... FANUC Lathe file, extension ".fnl". 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 (eg. a cutting operation).

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

HOME .... Operation to send the axes of the CNC lathe 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 that determines an auxiliary function (eg. coolant on, tool change etc.).

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

15: Glossary

MACHINE DATUM .... A fixed zero reference point, used to define the co-ordinate based grid system of the CNC lathe. 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.

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 lathe to ensure machining occurs in the correct place on the billet. Offsets shift the entire co-ordinate based grid system of the CNC lathe so the machine recognises our (moveable) workpiece datum as zero, rather than the (fixed) machine datum.

PART DATUM .... Used as a zero reference point in a CNC file. All machining co-ordinates originate from this point within the program. Commonly set at the centreline on the X axis and the face end of the billet on the Z axis.

PC .... Personal computer.

PROFILE .... The outside shape of an object. The term tool profile is used to describe the shapes of the different cutting tool tips.

PROGRAM .... A systematic arrangements of instructions or information to suit a piece of equipment.

PROGRAMMABLE TURRET . Area where cutting tools are held, also referred to as an automatic turret or ATC (automatic tool changer). An 8 station programmable turret could hold a maximum of 8 tool profiles. The tool being used to machine the billet can be automatically changed for any other tool fitted to the programmable turret, using software or CNC commands.

SADDLE .... The moving fixture on the cross-slide (X axis), onto which the toolpost is mounted.

SPINDLE SPEED .... The rate of rotation (velocity) of the chuck, measured in RPM.

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

STATION .... The position of a tool profile in a programmable turret, eg. station 3 would refer to position number 3 on the programmable turret.

TOOL OFFSET .... When machining, allowances must be made for the different sizes of tools used. The tool offset is the amount the X and Z value must be moved (or offset), so that all the different cutting tool tips used with the same CNC program line up with each other against a common reference point.

TOOLPOST .... The area on the CNC lathe where tool profiles are held. Manual toolposts can only hold one tool at a time. A programmable turret can hold a number of different tools and change between them automatically.

TRAVERSE .... Movement of the cutting tool through the 2 machine axes, between cutting settings.

TURRET .... See programmable turret.

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

15: Glossary

WORK (WORKPIECE) .... The actual material being machined. 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. G01, M03 etc.).

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

16: Index

16: Index

16: 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.