

Configuring a Tool Offset File:

- 1) Click the [Tooling] button, from the "Options" toolbar, to display the "Tooling" window. The table of numbered tool offsets is displayed in the lower half of the window. To create a new "Tool Offset" file, right click on any numbered tool offset button and select "Add".
- 2) Highlight the required tool offset number button, so the red arrow marker is displayed on the button itself. Any subsequent data entries will now be placed in this file. Note - when a tool change is requested, the software automatically activates the tool offset file with the same number as the tool profile called (see note*).
- 3) Jog the tool to a chosen X tool offset reference position. This same X position must be used by ALL tool profiles, when configuring their own tool offset files. Note - Axes can only be manually moved when the "Jog" tab is coloured green.
- 4) There are two [datum graphic] buttons, positioned above the X and Z tool offset value columns, in the "Tooling" window. Click the X [datum graphic] button to display the "Set Offset" window - a value is automatically suggested. If you jogged to a known position, don't forget to account for your known value. Click the [OK] button.
- 5) Jog the tool to a chosen Z tool offset reference position. This same Z position must be used by ALL tool profiles, when configuring their own tool offset files.
- 6) Click the Z [datum graphic] button to display the "Set Offset" window - a value is automatically suggested. If

- you jogged to a known position, don't forget to account for your known value. Click the [OK] button.
- 7) Repeat steps 2) to 6) for all remaining tool profiles.
- 8) To save your tool offset files, ensure "Tooling" is the active window in the software, then click "Tooling | Save As". Tool offset files (together with their numbered tool profiles) are stored using the file extension ".lft".

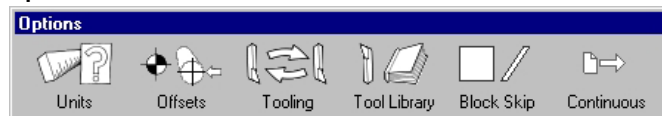
*Note - Specific tool offset file numbers can be manually activated in the software, for occasions when the tool offset file number is different to the tool profile number. Highlight the required tool offset number button, then click on any other tool offset number button. Click the [Yes] button in the "Confirm" window. The tool offset number in the statusbar area of the "Control Panel" window will change to show the required tool offset file is active (ie, it's effect on the "Program" co-ordinate display can be viewed).

Running a CNC program - Auto Mode:

- 1) Check that all the tools used by your CNC file are configured correctly in the "Tooling" window.
- 2) Highlight the "Control Panel" window and select the "Auto" tab.
- 3) In the "Editor" window, position the cursor at the beginning of the first line of your CNC file.
- 4) From the "File Control" toolbar, click the triangular [Play] button to start running your CNC file.

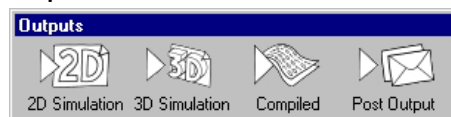
Toolbar Definitions.

Options.



Used to switch the units of measurement between Metric (mm) and Imperial (Inch).	Displays the "Work Piece Offsets" window, used for configuring global workpiece offset values.	Displays the "Tooling" window, showing available tools that can be added to the ATC and any tool offset data.	Displays the "Tool Library" window, listing all tools that can be added to the ATC in the "Tooling" window.	When active, any CNC file lines beginning with a / mark are ignored.	Runs the CNC file from start to finish (Continuous, shown above) or line by line (Single Step).
--	--	---	---	--	---

Outputs.



Displays the "2D Simulation" window. Outputs the CNC file on a 2D side view of the billet.	Displays the "3D Simulation" window. Outputs the CNC file on a 3D billet.	Outputs the CNC file as a Denford Post Processor file (.gnc).	Outputs the CNC file as a Denford Post Processor file (.gnc).
--	---	---	---

Machine Control.

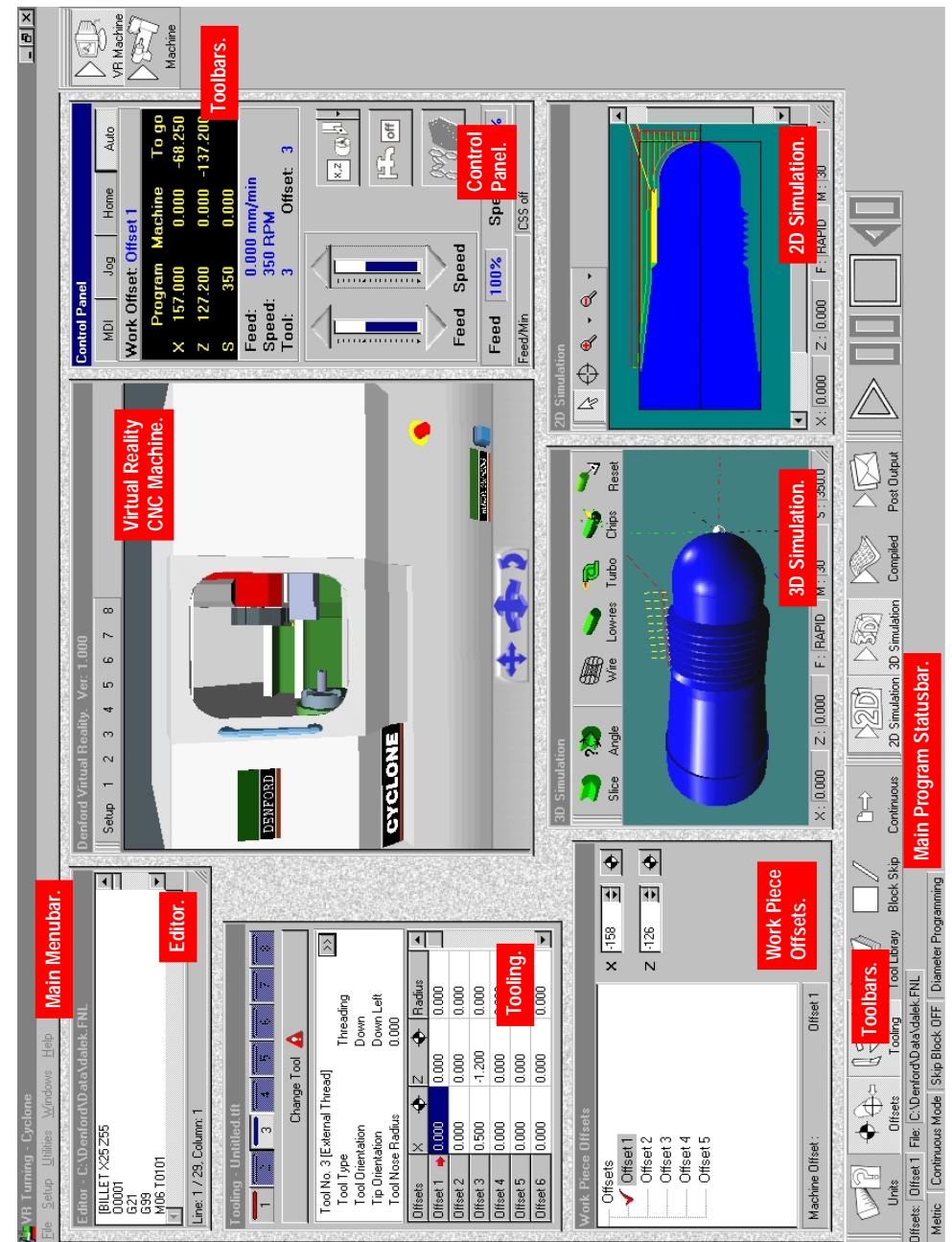


Switches on a virtual reality CNC machine, displaying the "Control Panel" and "Virtual Reality" windows.	Opens communications between any attached real CNC machine, displaying the "Control Panel" window.
--	--

File Control.



Run the CNC file from the current cursor position in the "Editor" window.	Pause a running CNC file at the end of the current CNC file line.	Stop the running of a CNC file.	Rewind the cursor in the "Editor" window back to the beginning of the CNC file.
---	---	---------------------------------	---

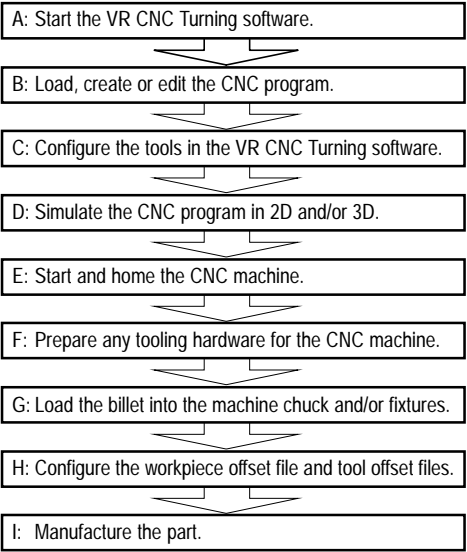


Virtual Reality CNC Turning Software Quickstart Reference Card.

Note: Not all windows and main menus are shown in the example screenshot. Note: The positions of the various windows and toolbars may be different on your VR CNC Turning software (since their positions are user defined).

Recommended Order of Manufacturing.

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.



Remember, Safety First, when using a real CNC machine:

- 1) Ensure you know the position of the Emergency Stop button, before beginning any work.
- 2) Never place any objects in the working area of the CNC machine that could interfere with its safe operation.
- 3) Never attempt to enter the working area of the CNC machine when the spindle (chuck) and/or machine axes are in motion.

Starting and closing the VR CNC Turning Software:

- 1) If you want to control a real CNC machine with the software you MUST start the real CNC machine FIRST.
- 2) From your windows startbar, click "Start | Programs | Denford | VR Turning". Note: this sequence only applies when installing the original program files to their defaults.
- 3) To close VR CNC Turning, click "File | Exit". Note: never attempt to exit the software when real machining operations are being performed.

Opening a CNC program:

- 1) From the main menubar, click "File | Open". CNC programs are stored using the file extension ".fml".
- 2) Select the drive and directory where your CNC program is stored, highlight its title and click the [Open] button.

Editing a CNC program:

The text from any CNC program is displayed in the "Editor" window. The editor behaves in a similar way to a simple word processor, like Windows Notepad. Additionally, global modifications can be made to highlighted text, using options such as "Line Numbering" (eg. N0010), "Line End Tokens" (end of line symbols) and "Padding Tokens" (spaces between characters and words). These options are available in the "Editor | Modify" sub-menu.

Saving a CNC program:

- 1) From the main menubar, click "File | Save As". CNC programs are stored using the file extension ".fml".
- 2) Select the drive and directory where you want to save your CNC program, enter a title and click the [Save] button.

Configuring Tools ready for use:

- Tools must be transferred from the "Tooling Library" to "Tooling", where they are assigned a tool number and become ready for use. Assign tool numbers to profiles according to the tool number definitions used in your CNC program.
- 1) Click the "Tool Library" button, from the "Options" toolbar, to display the "Tooling Library" window.
 - 2) A default set of tools is available. To create a new tool not specified in the list, right click on any tool title and select "Add Tool", then configure the data panel for your new tool.
 - 3) Click the "Tooling" button, from the "Options" toolbar, to display the "Tooling" window.
 - 4) Click and hold on a tool title in the "Tooling Library". Drag the tool title out of the "Tooling Library" window and over the required tool number in the "Tooling" window. Release the left mouse button to drop the tool onto the number. The tool profile is now assigned that tool number and is ready for use.
 - 5) Repeat step 4) for all remaining tools used by your CNC program.

Manual Tool Changes:

- 1) Click the "Tooling" button, from the "Options" toolbar, to display the "Tooling" window.
- 2) Click the "Auto" tab, in the "Control Panel" window.
- 3) Using the "Tooling" window, click the number of the tool you want to change to, then click the [Change Tool] button. Manual toolposts: Follow any on-screen prompts to change the tool.
ATC tooling: All tooling operations are performed automatically.

Simulating a CNC file in 2D:

- 1) Check that all the tools used by your CNC program are configured correctly in the "Tooling" window.
- 2) Click the "2D Simulation" button, from the "Outputs" toolbar, to display the "2D Simulation" window.
- 3) From the main menubar, click "2D Simulate | Use Offsets", so a tickmark is NOT displayed next to the title.
- 4) In the "Editor" window, position the cursor at the beginning of the first line of your CNC program.
- 5) From the "File Control" toolbar, click the triangular [Play] button to run a 2D simulation of your CNC program.

Simulating a CNC file in 3D:

- 1) Check that all the tools used by your CNC program are configured correctly in the "Tooling" window.
- 2) Click the "3D Simulation" button, from the "Outputs" toolbar, to display the "3D Simulation" window.
- 3) From the main menubar, click "3D Simulate | Use Offsets", so a tickmark is NOT displayed next to the title.
- 4) In the "Editor" window, position the cursor at the beginning of the first line of your CNC program.
- 5) From the "File Control" toolbar, click the triangular [Play] button to run a 3D simulation of your CNC program.

Starting a Virtual Reality CNC machine:

- 1) Click the "VR Machine" button, from the "Machine Control" toolbar, to display the "Control Panel" and "Virtual Reality" windows.
- 2) To move around the virtual reality world, click, hold, then drag any of the three movement icons at the base of the "Virtual Reality" window.
- 3) Predefined viewpoints can also be selected by clicking the numbered buttons, displayed at the top of the "Virtual Reality" window.
- 4) Home the CNC machine before starting any work.

Communicating with a real CNC machine:

- 1) Click the "Machine" button, from the "Machine Control" toolbar, to display the "Control Panel" window.
- 2) Home the CNC machine before starting any work.

Homing the CNC machine - Home Mode:

- The CNC machine must always be homed after first being switched on. Homing is also recommended after configuring any offsets, or just before the running of a CNC program.
- 1) Highlight the "Control Panel" window "Home" tab.
 - 2) Click the [X Axis] button to home the X slide.
 - 3) Click the [Z Axis] button to home the Z slide. Alternatively, click the [Both] button to home both slides.

Jogging (manually moving) the machine axes - Jog Mode:

- 1) Highlight the "Control Panel" window and select the "Jog" tab, so the tab title is highlighted in green.
- 2) Jog each axis using the following keys:
X Axis: [Cursor Up] or [Cursor Down]
Z Axis: [Cursor Right] or [Cursor Left]

Types of Jog Movement:

- Select the "Jog" tab in the "Control Panel" window. In the "Jog Speed" area, click the [Straight/Stepped Arrow] button to switch between the two modes. Use the "Jog Speed" rotary control to set the required movement speed.
- i) Jog Continuous - [Straight Arrow] button. Continuous movement is obtained when a jog key is pressed and held down. Use this mode for fast, rough positioning.
 - ii) Jog Step - [Stepped Arrow] button. One single increment movement is obtained on each press of a jog key. Use this mode for fine adjustments to positions.

Manually entering an M code:

- M codes can be entered to manually, to operate powered parts of the CNC machine, such as the safety guard door, vice and ATC (Automatic Tool Changer).
- 1) Highlight the "Control Panel" window and select the "Jog" tab.
 - 2) Click the [M Codes] button, then click on the required M code function from the dropdown list.

What are Offsets?

Offsets are used to describe the position of the workpiece datum. This is the place where you want any machining co-ordinates to begin. Two types of offset file are used in combination to describe the workpiece datum position:

- i) The Workpiece Offset file. This file contains two values relating to the global shift that can be applied along the X and Z machine axes.
- ii) The Tool Length Offset files. Each tool profile has its own tool offset, used to compensate for the differences in length between all the various tools used together on one job.

When you specify the position of your workpiece datum on your billet, it's important that this matches the part datum used when creating your CNC file. Most CNC programs use the face end and chuck/billet centreline of the imaginary billet as their part datum.

Configuring a Workpiece Offset File:

- 1) Click the [Offsets] button, from the "Options" toolbar, to display the "Work Piece Offsets" window. To create a new "Offset" file, right click on any numbered offset title and select "Add Offset".
- 2) Activate the required numbered workpiece offset file by right clicking on the title of your new "Offset" and selecting "Make Current". A red tickmark shows the currently active offset file. Any values in the workpiece offset file are applied to the "Program" co-ordinate display in the "Control Panel" window.
- 3) Highlight the required numbered workpiece offset file by clicking on the title of your new "Offset", so the title name is displayed in white letters on a blue background. Any subsequent data entries will now be placed in this file.
- 4) Highlight the "Control Panel" window and select the "Jog" tab. Set the "Control Panel" window to display "Program" co-ordinates, by clicking the [Co-ordinates Display Select] button.
- 5) Jog the tool to the exact position of your X workpiece datum (eg, the chuck centreline). If the exact position cannot be reached, jog to a known position (eg, a known dimension on the billet). Note - Axes can only be manually moved when the "Jog" tab is coloured green.
- 6) There are two [datum graphic] buttons, positioned to the right of the X and Z offset display panels, in the "Work Piece offsets" window. Click the X [datum graphic] button to display the "Set Offset" window - a value is automatically suggested. If you jogged to a known position, don't forget to account for your known value. Click the [OK] button.
- 7) Jog the tool to the exact position of your Z workpiece datum (eg, the face end of the billet). If the exact position cannot be reached, jog to a known position (eg, a known dimension on the billet).
- 8) Click the Z [datum graphic] button to display the "Set Offset" window - a value is automatically suggested. If you jogged to a known position, don't forget to account for your known value. Click the [OK] button.
- 9) To save your workpiece offset file, ensure "Work Piece Offsets" is the active window in the software, then click "Offsets | Export". Workpiece offset files are stored using the file extension ".tol".