

DENFORD

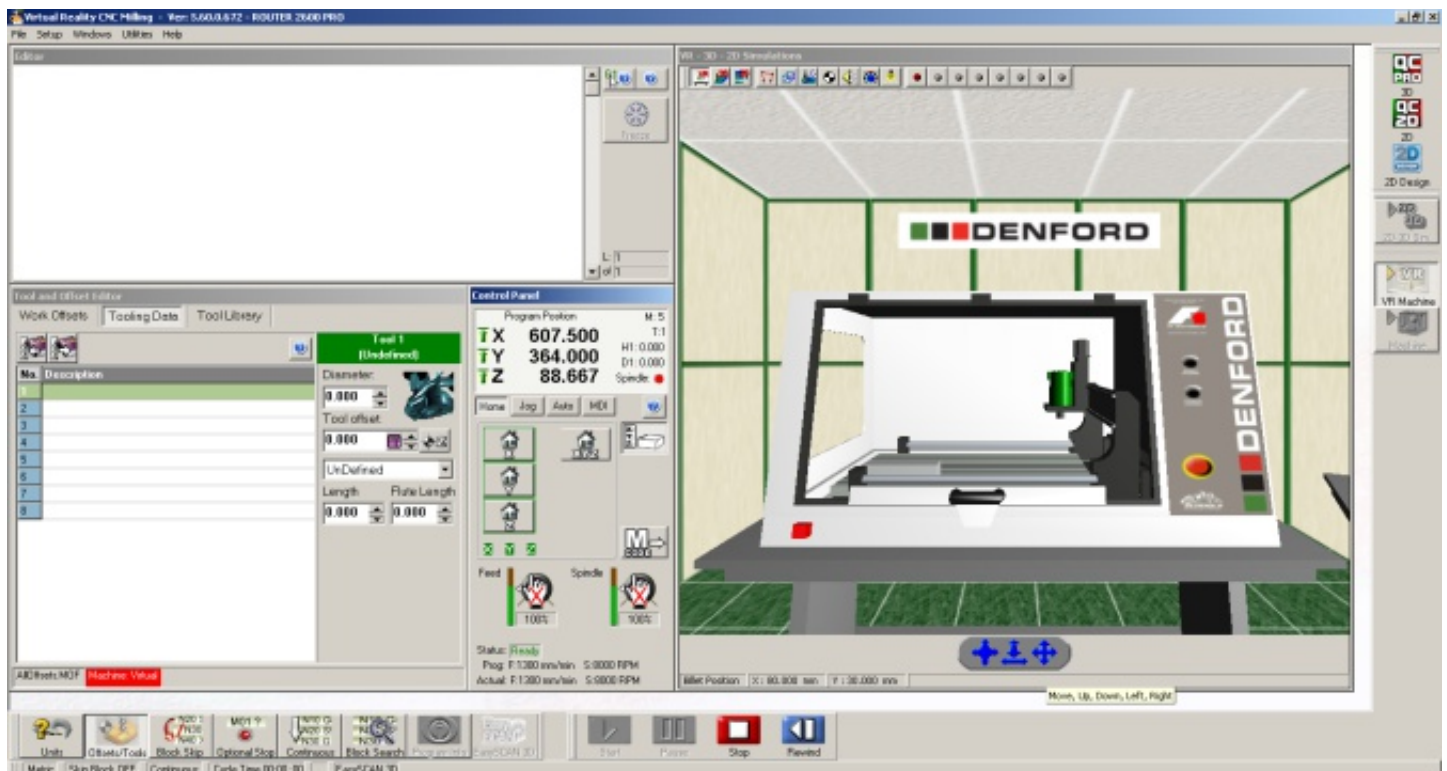
CAD/CAM Solutions & Projects for Education

VR Milling V5

Router and Mill Tool Change

Training Guide

VR Milling V5 (V5.61)



DENFORD

Router 2600 ATC & Router 6600 ATC

3 AXIS CNC ROUTERS WITH 5 STATION ATC

NEW



5 Station Automatic Tool Changer

**QUALITY, PRECISION,
MAINTENANCE FREE
ROUTING**

Router 2600 ATC shown with optional universal bench, optional computer support extension and optional integrated Dust Pro 100 (PC not included)

These 3 axis CNC Routers with totally enclosed interlocking guard and complete with 5 Station Automatic Tool Changer are suitable for all levels of education and are ideal for cutting a range of resistant materials such as hard and soft wood, plastic, modelling foam, acrylic, prototyping material and non-ferrous metals.

Additionally, the Router 6600 ATC is a high speed machine, complete with built-in machine bench, offering large machining capacity (table size 1080 x 640mm) at an exceptional price.



Ideal for use in conjunction with



Contents

About	4
Introduction	4
Configuring Tools	5
Adding Tools	6
Editing Tool Names	7
Setting Tool Offsets	8
Setting the Tool Offset with an ATC	8
Tool Changing with an ATC	9
Setting the Tool Offset with an ATC (continued)	10
Setting the Tool Offset with a Manual Tool Change	11
Router	11
Mill	11
Both	12
Tool Changing with a Manual Tool Change	12
Saving Tool Offsets	13
Loading Tool Offsets	14

About Tool Changing

Denford Routers and Mills can be equipped with Automatic Tool Changers (ATC) or without. If you have an ATC or manual tool change you are able to write programs which call for tool changes.

The aim of this training guide is to illustrate how to set-up your tool library to enable tool changing and how to perform a tool change.

Introduction

The aim of this guide is to take you step by step through the process of Tool Changing on Denford routers or mills.

This guide makes use of screen shots where possible and will use the following conventions:

Instructions will be in this format

Text to be typed will be in this format

Any software buttons to be pressed, a picture of the button will follow the instruction

This guide assumes that your software has already been installed and your machine has been commissioned.

If any of the features described in this guide are not operating as described please check that the version number you are using is the same as that shown on the front cover.

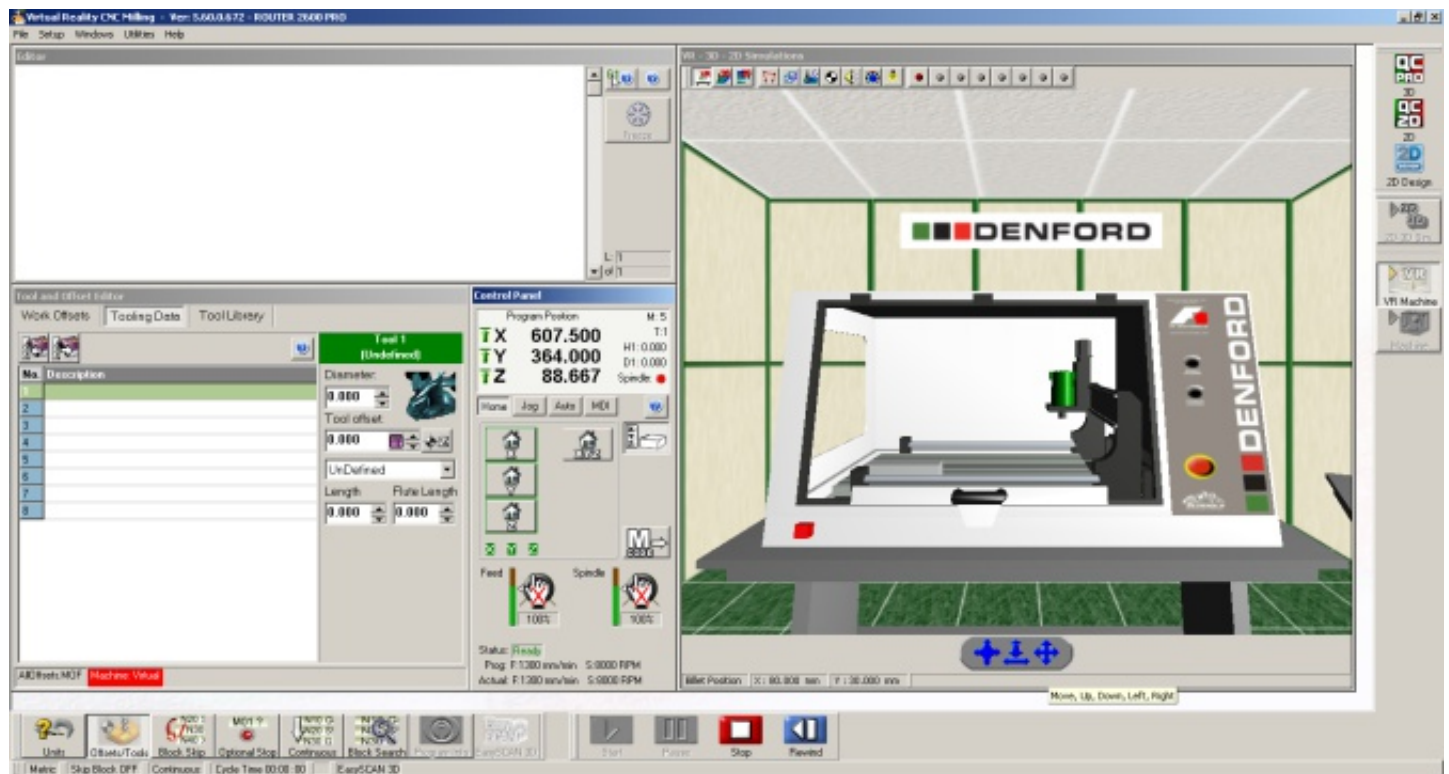
Version number is written on the title bar of the main software window.

Denford provide machine training and it is recommended that you undertake the training and use this guide as a revision guide after completion of the machine training.

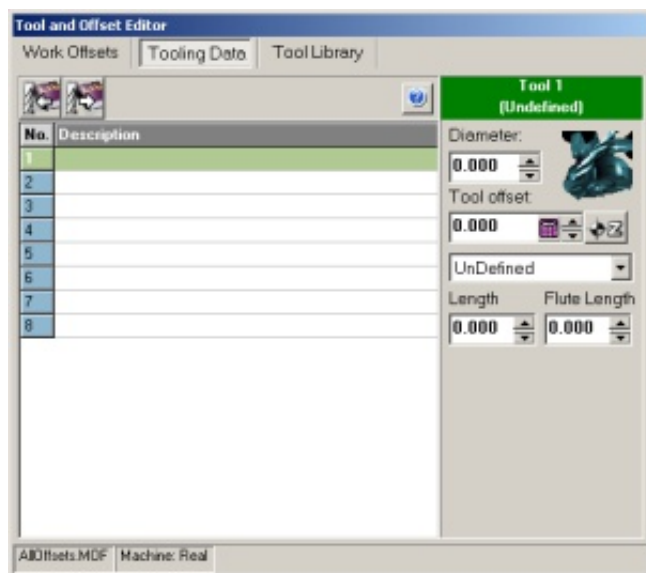
Configuring Tools

To set-up tools in your router / mill you need to open VR Milling V5

Open VR Milling V5



Click on the "Offset / Tools" button



The Tool and Offset Editor window will open, as shown in the image on the left.

There are 3 tabs and you want the middle tab, which is the Tooling Data tab.

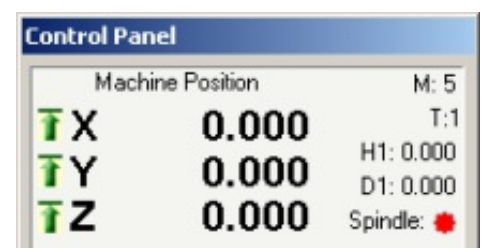
Click on the "Tooling Data" tab

Tooling Data

The Tooling Data tab allows you to configure 8 tools for use in a single program.

If using an ATC then the number of tools is dependant on which ATC you have.

The current tool is highlighted in green, the image above shows tool 1 as the currently selected tool and the image on the right also shows that tool 1 is selected in the Control Panel

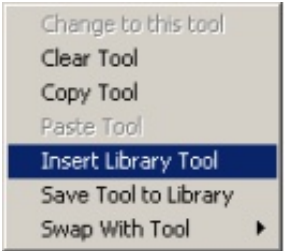


Adding Tools

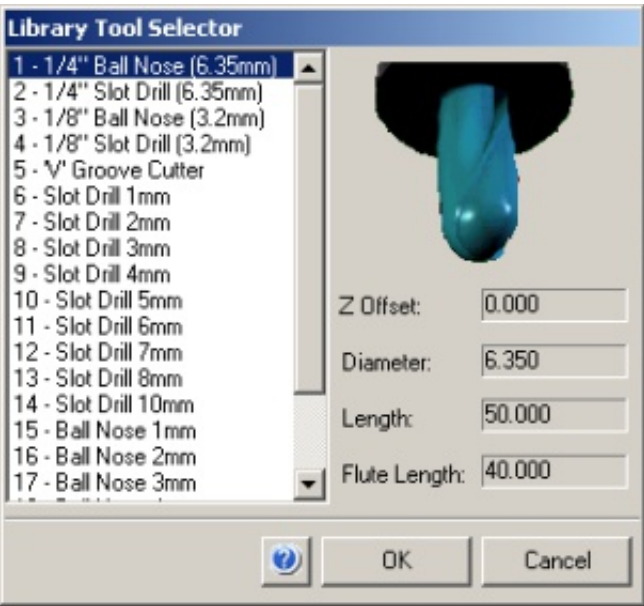
To add a tool into 1 of the 8 slots available in the Tooling Data tab, right click on the description section of the empty slot

Right click on slot number 1

The menu shown on the right will appear, you should insert a tool from the tooling library.



Click on "Insert Library Tool"

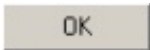


The Library Tool Selector will appear.

For this guide we are going to set up the tools for a router based on the Denford Recommended Tooling Package.

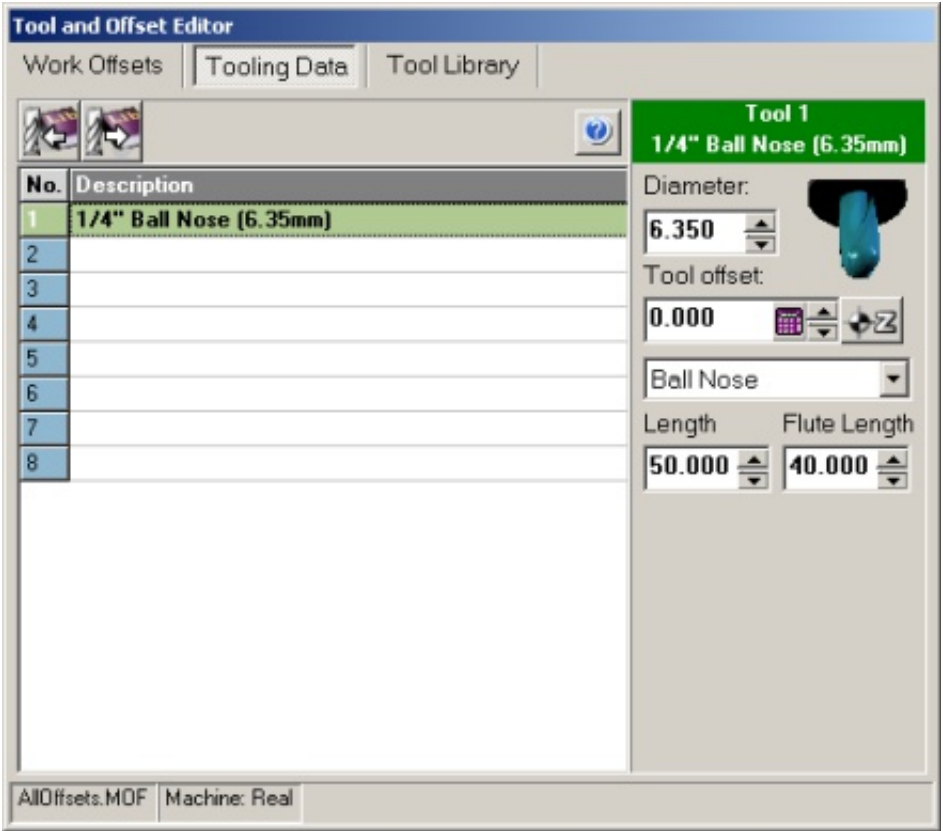
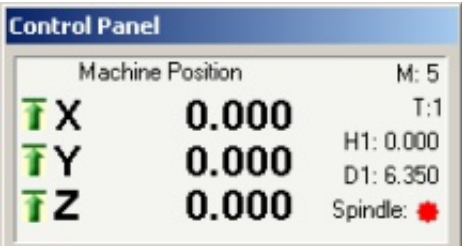
Click on tool number 1 so that the 1/4" Ball Nose is highlighted in blue, like the image on the left.

Click the "OK" button



The Tool and Offset Editor window should now have the 1/4" Ball Nose set in slot 1 of the Tooling Data tab, like the image on the right.

The Control Panel should now show the tool diameter as D1, like the image below.



Editing Tool Names

There may be times when you want to edit the name of the tool in the Tooling Data tab without creating a new tool in the Tool Library.

To do this you double left click in the Description section of the tool slot so that the tool is highlighted blue with a flashing cursor after the last letter, type the new name and click on another tool slot to save the edited text.

The Denford Recommended Tooling Package contains 2 1/4" Ballnose cutter, one of them is the Solid Carbide Extra Long Series cutter, we will set this as tool number 1 and we should change the name so that we can differentiate between the 2 when writing a CNC program.

Double left click on the Description of tool 1

No.	Description
1	1/4" Ball Nose (6.35mm)

Press the "right cursor", to remove the blue highlight.

Press the "space bar" and type the text below:

Long Series

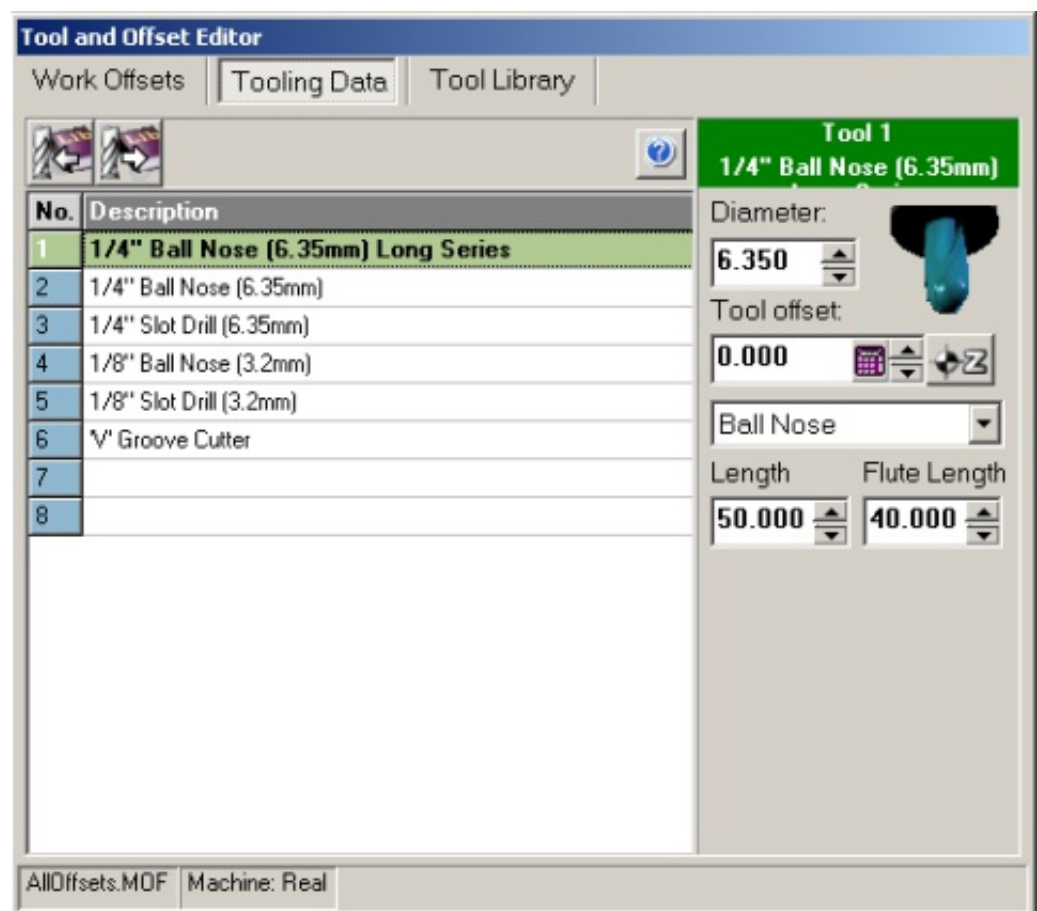
Click on slot 2 to save the name for tool 1

No.	Description
1	1/4" Ball Nose (6.35mm) Long Series

Now that you have edited tool 1 you should set the rest of the tools by right clicking on each slot and following the same method as used for tool 1

Set tools 2 to 6 in the Tooling Data Tab

Your tool data tab should now look like the image to the right.

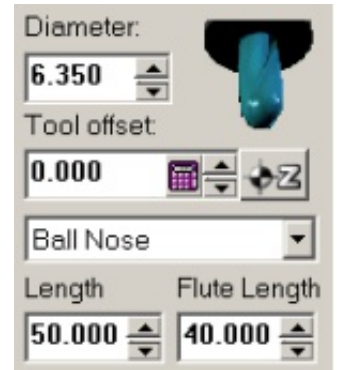


Setting Tool Offsets

Tools 1 to 6 are of different lengths, we need to inform the CNC machine of this difference in length.

We do this by setting Tool Offsets

In the Tool Data tab there is a setting for the Tool Offset for each tool, this is just below the diameter as illustrated by the image on the right.



There are a couple of ways to set the tool offset.

With a manual tool change on a router it is better to use a depth gauge to set the length of tool sticking out from the spindle nut in order to set the Offset.

With an ATC router or mill it is easier to touch on the same point for each tool to set the Offset.

This guide shows both methods.

Setting the Tool Offset with an ATC

To set the tool offset with an ATC the easiest way to set the Offset is to touch on the same point. To do this you first need to set the Z0 position for tool 1.

With tool 1 selected, drive the Z axis down until the tool just touches the part

In the Tool and Offsets Editor, click on the Work Offsets tab



Set the current position as Z0 by clicking the "Z Datum 0" button



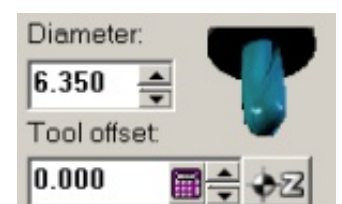
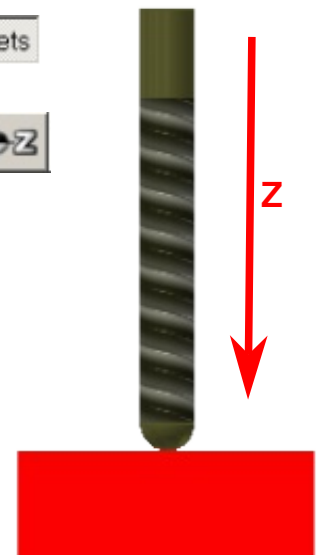
You have now set the position of tool 1 as Z0, you will not use the Work Offsets tab again, from now on all settings will be done in the Tooling Data tab.

Click the Tooling Data tab



From now on we will be using the Tool Offset section of the Tooling Data tab to set the height offset between the current tool and the Z0 datum position.

We do this by jogging the tool down until it touches the same point as we set the Z0 datum to and clicking the "Z0" button under Tool Offset. As tool 1 is still touching on the point that we set as Z0, there is no need to create an offset for Tool 1 as the offset would be 0.



Tool Changing with ATC

To change a tool with the ATC the Machine Guard must be closed and the Control Panel must be in either the Jog or Auto tab.

To change tool, select the tool you wish to change to and click the "Change Tool" button

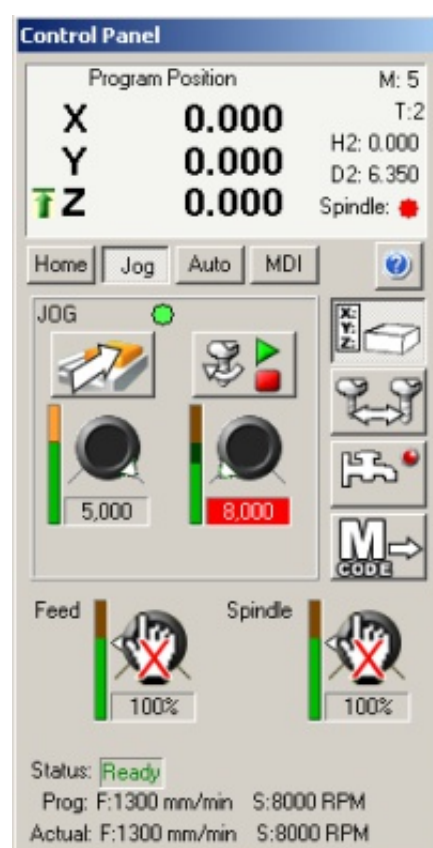
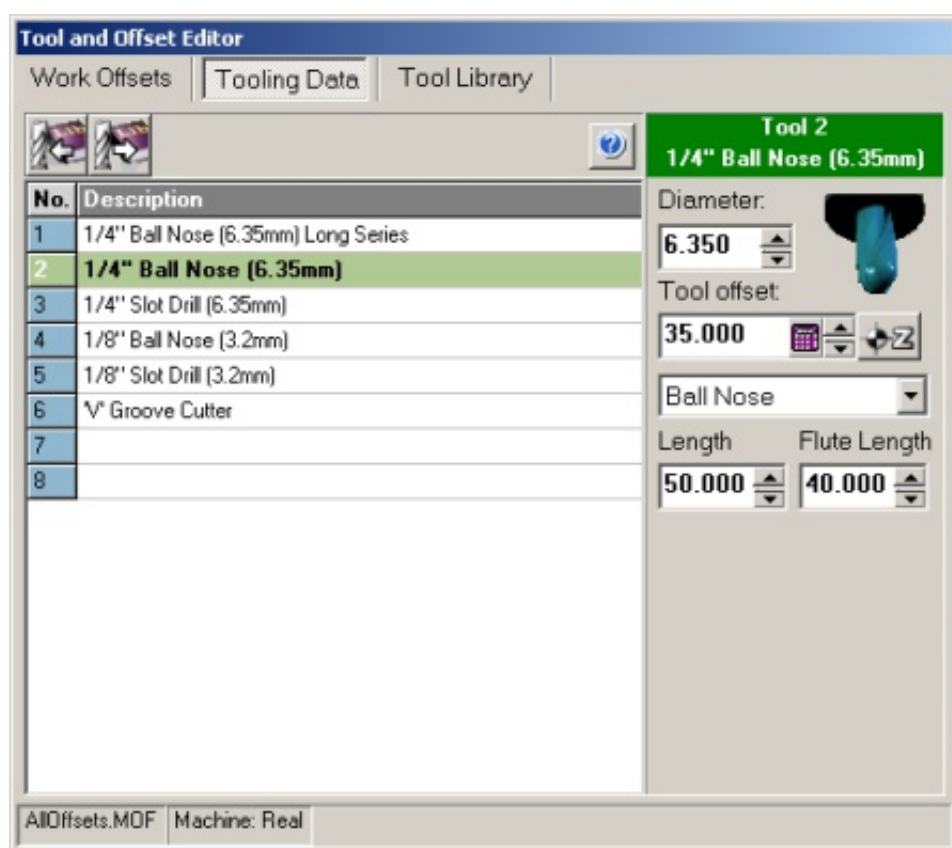
Click on tool 2

No.	Description
1	1/4" Ball Nose (6.35mm) Long Series
2	1/4" Ball Nose (6.35mm)

Click the "Tool Change" button

Change Tool

The machine will now perform a tool change, when complete Tool 2 will be highlighted green in the Tooling Data tab and the Control Panel will indicate it has Tool 2 fitted



Setting the Tool Offset with an ATC (continued)

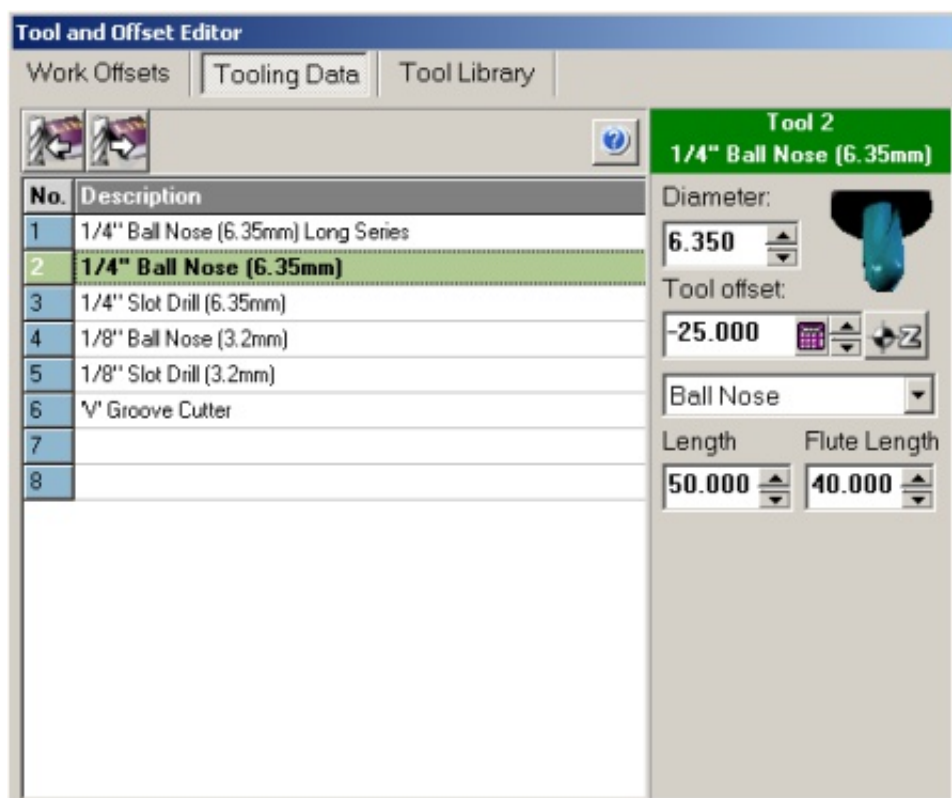
Now that you have changed to tool 2, you should jog Tool 2 down until it touches the same point that you touched on with Tool 1 and then set the Tool Offset for Tool 2.

Jog the Z axis down until Tool 2 touches the point used for Tool 1

Click the "Z0" button under Tool Offset



Notice that there is now an offset value shown under the Tool Offset section, the offset is also showing in the Control Panel as H2



Repeat the process above until you have set all Tools

Click the "Tool / Offset" button to close the Tool and Offset editor.

All of your tools are now set and you are able to create a CNC program using multiple tool changes.

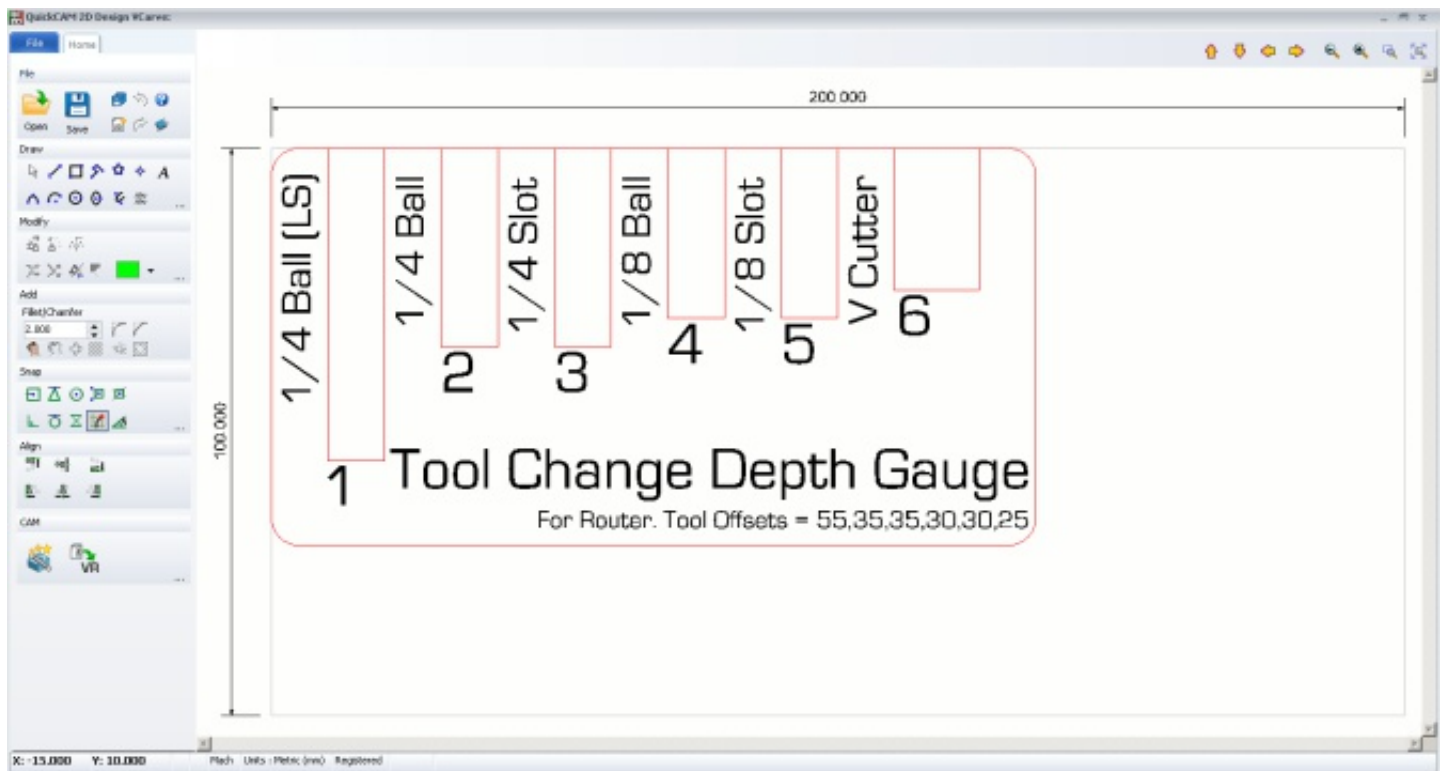
Setting the Tool Offset with a Manual Tool Change

Router

With a Denford Router you do not have tool holders, the tool is held in the spindle by a collet in the spindle nut.

As each tool is of a different length there will be a different amount of tool stuck out from the spindle nut for each tool.

The easiest way to set this is to make a depth gauge like the one below on the laser cutter. It has a pocket for each tool and the top of the pocket is butted up against the spindle nut whilst the tool is held against the bottom of the pocket until the spindle nut has been tightened. The depth of the pocket is the tool offset.



Mill

The VMC1300 CNC Mill uses the same tool holders for the ATC and the Manual tool change, the tool holders are released by a hydraulic draw bar on the ATC and a lever operates the drawbar on the Manual.

For the Manual tool change you should measure the length of the complete tool holder and tool and record the values of each one, we will need the values for the next step.



Both

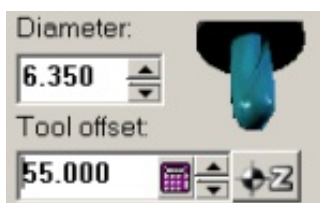
Now that you have the values of the Tool Offset for all your tools the values need to be entered into the Tooling Data tab.

We will set Tool 1 first, ensure the slot for Tool 1 is highlighted green when setting the offset.

Double click on the Tool Offset value so it is highlighted blue as illustrated in the image below.



Type in the Tool Offset value



For Tool 1 on the router this value will be 55mm and is the depth of the 1st pocket on the depth gauge.

Click on the slot for Tool 2, it should highlight yellow

No.	Description
1	1/4" Ball Nose (6.35mm) Long Series
2	1/4" Ball Nose (6.35mm)

Type in the Tool Offset value



For Tool 2 on the router this value will be 35mm and is the depth of the 2nd pocket on the depth gauge.

Repeat the process until you have entered the values for all tools.

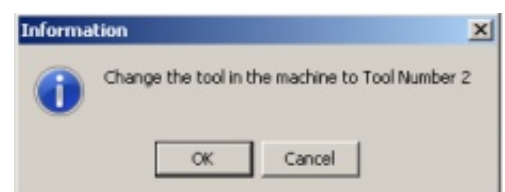
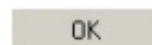
All of your tools are now set and you are able to create a CNC program using multiple tool changes.

Tool Changing with a Manual Tool Change

When a tool change is required the spindle will stop, the spindle will move to the tool change position and the guard will unlock.

A dialogue on screen will instruct which tool is being called. You should change to the required tool, close the guard.

Click the "OK" button when the guard is closed.



The machine will resume your CNC program.

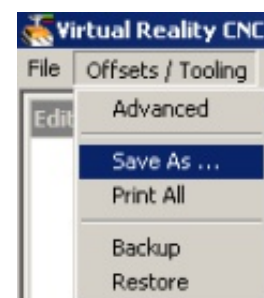
Saving Tool Offsets

Having set up all of your tools it is a good idea to save them.

With the Offset and Tool Editor window open and selected, the 2nd heading on the taskbar will read "Offsets / Tooling"

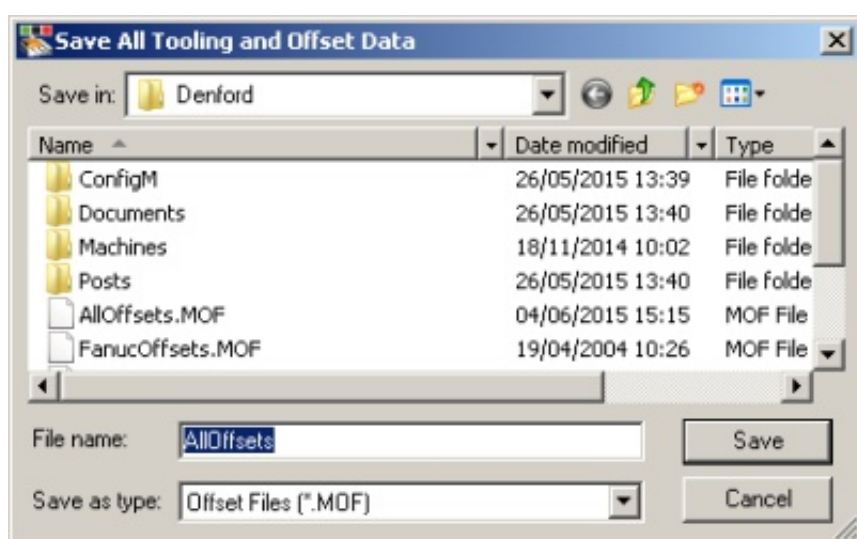
Click the "Offsets / Tooling" heading 

Scroll down to "Save As ..." and click it 



The Save all Offset and Tooling Data window, shown below, will appear.

Type a name for your offset in the File Name dialogue box



Click the "Save" button 

Loading Tool Offsets

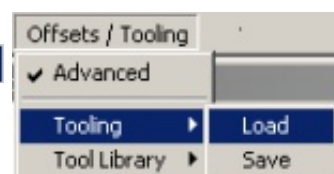
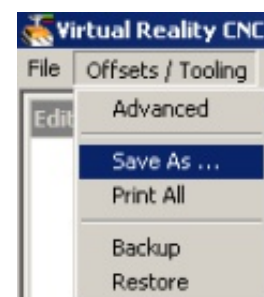
With the Offset and Tool Editor window open and selected, the 2nd heading on the taskbar will read "Offsets / Tooling"

Click the "Offsets / Tooling" heading 

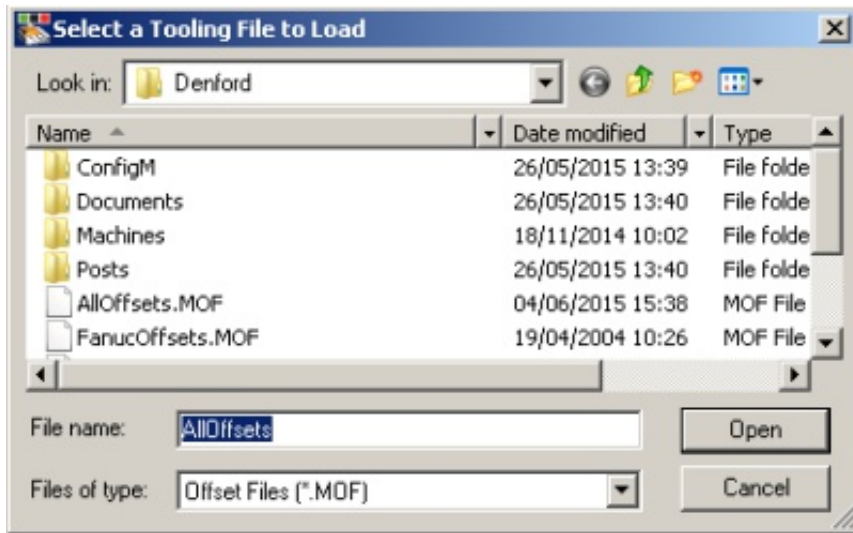
Click the "Advanced" heading 

Click the "Offset / Tooling" heading again 

Scroll down to "Tooling" and then select the "Load" heading 

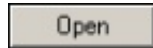


The Select a Tooling File to Load window, shown below, will appear.



Select the Tooling File you wish to load

Click the "Open" button



You have now opened a saved tooling file.

If you are creating CNC programs on another PC away from the router it is a good idea to make sure that the PC being used for creating the CNC program has the same tooling file as the CNC machine otherwise tools may be selected in the wrong order and this may cause breakages.

DENFORD

VMC 1300/1300 PRO

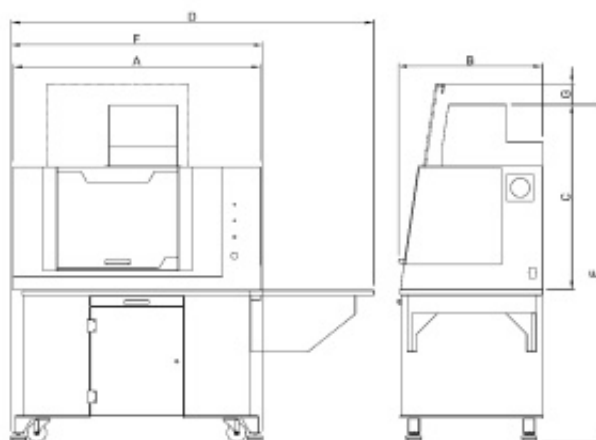
CNC MILLING MACHINE



AVAILABLE WITH:

- Flood Coolant & Industrial Cabinet Base
- 6 or 8 Station Automatic Tool Changer
- Automatic Lubrication System

Shown with optional universal bench and optional computer support extension. (PC not included)



Machine Dimensions.

A 3 axis CNC milling machine available either floor standing or for bench mounting, with totally enclosed high visibility interlocking guard, suitable for all levels of education and training. Programmable spindle speeds and feedrates make the VMC 1300 ideal for cutting a range of resistant materials such as wax, plastic, acrylic, free cutting alloys, aluminium and steel. The VMC 1300 Pro has a more powerful, higher speed spindle (6000 rpm) for heavy duty cutting. Now available with 6 or 8 Station Automatic Tool Changer and the option of Flood Coolant with Industrial Cabinet Base.

CE
APPROVED

denforddata.com/bb/

On-Line Technical Forum

TECHNICAL SUPPORT AVAILABLE 24 HOURS A DAY, 7 DAYS A WEEK

Denford's Technical Forum is a free of charge on-line technical support service that is available to Denford customers 24 hours a day, 7 days a week.

"The technical forum has provided a wealth of information and support for our 20-year-old Denford CNC machine, in fact just as good as the support we receive for our brand new CNC Router!"



Denford's On-Line Technical Forum is a free of charge service that can be accessed 24 hours a day, 7 days a week.

The On-Line Technical Forum is available to Denford customers, old and new, and it couldn't be easier to use. Just visit <http://www.denforddata.com/bb/> and register on line.....it's that simple.

Denford's On-Line Technical Forum opens up the traditional communication channels that can restrict customer and technical support, due to availability of staff, teaching commitments or different time zones.

A multitude of topics relating to Denford machines and software (both new and old) are covered within the forum, which is simple to search, and easy to use.

Denford's Technical Team and Denford customers from around the world regularly log on to the forum to offer support and advice and, most importantly, post a solution for all to see.

As well as offering comprehensive technical support, Denford's On-Line Technical Forum enables customers to share ideas and projects with other users. Media such as teaching material, project work, PDFs, images, drawings and text documents are easily attached to messages for all users to view and comment on.

You can also read the latest Denford news before anyone else, and keep track of machine and software upgrades, some of which can be downloaded direct from the Technical Forum web site.

The On-Line Technical Forum has proved to be hugely popular with customers. One recent user posted a note to inform us that the Technical Forum has "provided a wealth of information and support for our 20-year-old Denford CNC machine, in fact just as good as the support we receive for our brand new CNC Router!"

Of course the traditional methods of phone and email are still available, but try out this new service by simply logging on to www.denforddata.com/bb/ and register.



Scan this to view our website



denford.co.uk

ISO9001 Compliant

DENFORD
T: +44(0)1484 728000
F: +44(0)1484 728100 Email: info@denford.co.uk
Denford Limited, Arnytage Road, Brighouse, West Yorkshire H6 1GF, England