

CNC Turning Training Guide



Table of contents

Introduction	3
1) - Start the VR Turning Software	3
2) - Software Start-up.....	4
3) - Configure the Software for the Machine	5
4) - Load a CNC file	6
5) - Configure the Tooling.....	7
6) - Connecting to and Home the CNC Machine	9
7) - Move the Cutting Tool.....	11
8) - Fit the Material in the Machine	12
9) - Performing a Tool Change.....	12
10) - Set the Workpiece Offsets	13
11) - Set the Tool Offsets.....	18
12) - Run a Simulation.....	23
13) - Run the Machine	24
14) - Fine Tuning Offsets.....	25
15) - Summary.....	26

Introduction

VR CNC Turning 6 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).

The VR Turning software contains detailed help files including tutorials. Access these by going to Help on the menu.

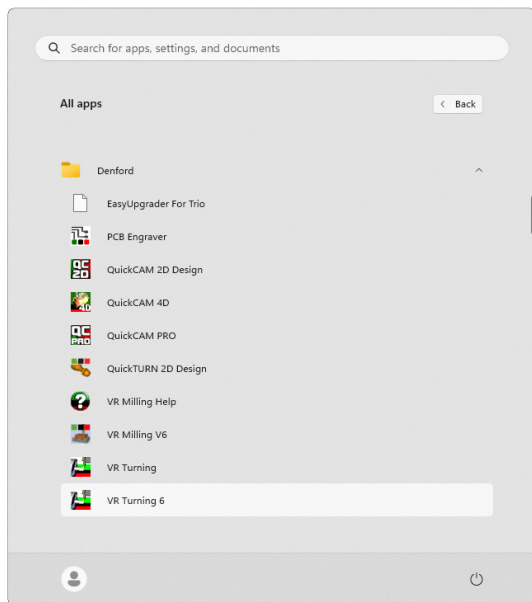
Check that you have the latest version of the VR Turning 6 software by going to the 'Downloads' section of the Denford Website (www.denford.co.uk). You can download the latest version from the website, you will need your licence disk to be able to run the software.

1) - Start the VR Turning software

To start the VR Turning software double-click the VR Turning shortcut icon (if available) which is located within the Denford Applications folder on your desktop.



If the shortcut is not available, click your Windows icon on the Task Bar followed by "All Apps" option, the program group "Denford" and finally the "VR Turning 6" icon.



2) - Software Start-up

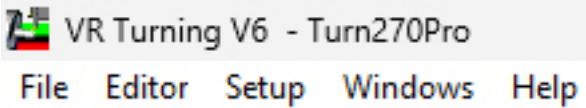
When the software starts up the below splash screen will be displayed. Double click on the top banner to close the splash screen quicker. DO NOT click on the update licence button on the bottom right hand corner unless you wish to change the licence details.



3) - Configure the Software for the Machine

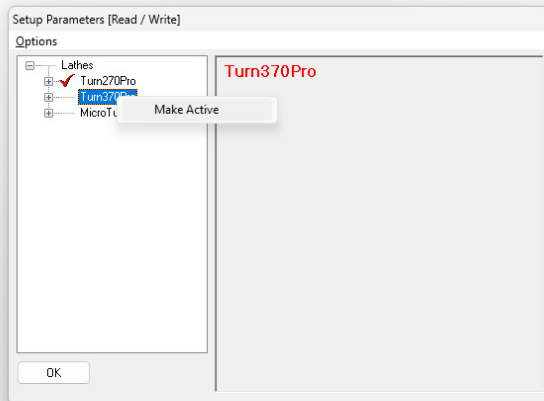
Ensure that the software is configured for the machine you are going to use.

The text at the end of the main title-bar indicates the type of Denford CNC lathe that you are currently able to control with the software. In the example screenshot below, the “Turn270Pro” text indicates that a Denford Turn 270 Pro can be controlled by the software. The model of the lathe you are using is indicated on the serial number plate on the side of the machine. If an incorrect machine is selected then a warning will be displayed indicating that the selected machine doesn't match the connected machine.



To change the model of the Denford CNC machine that can be controlled by the software:

1. Click the “Setup” menu and choose “Machine Parameters ...”
2. In the 'Password' window enter your password (the default is **denny**).
3. In the “Setup Parameters [Read/Write]” window, highlight the name of the required CNC machine.
4. Right-click over the name, then select the "Make Active" option from the pop-up menu that appears.
5. Click the [OK] button to apply the changes.

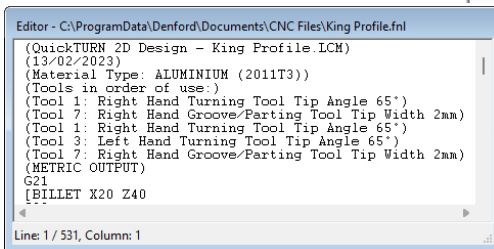
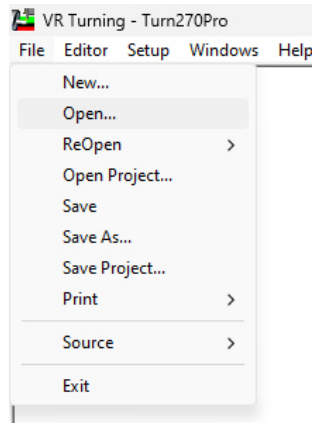


4) - Load a CNC file

Click the "File" menu and select the "Open" option.

Browse to the drive and folder containing your CNC file - look for files with the extension letters ".fnl" - then [Open] the file.

The contents of your CNC file will be displayed in the "Editor" window. As the name suggests, any loaded CNC files can be further edited here or you could even write one from scratch.



5) - Configure the Tooling

Before configuring the Tooling in the software, verify whether you are using a machine with a manual toolpost. The default setting within VR Turning is for a machine with an ATC (Automatic Tool Changer).

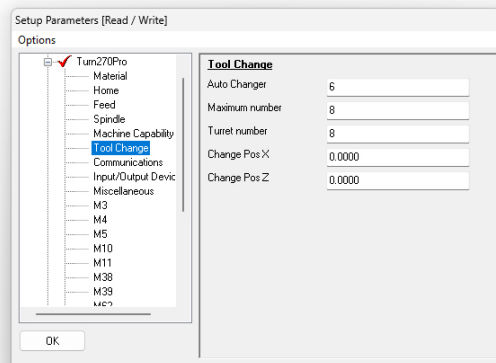
If you have a manual toolpost go to the menu at the top of the screen and select "Setup" and the option "Machine Parameters..."

In the 'Password' window enter your password (the default is **denny**).

Click the + next to your machine name to expand the file tree.

Highlight "Tool Change" in the file tree and change the 1 in the "Auto Changer dialogue" to 0.

The 0 means 'false' - there is no Auto Changer.



5) - Configure the Tooling

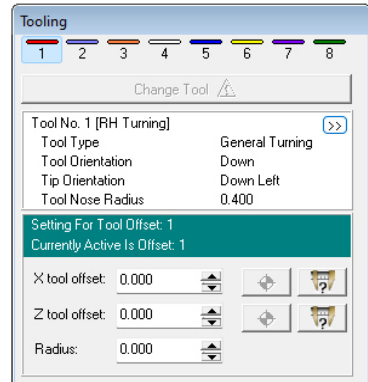
Click the [Tooling] button, to display the "Tooling" window.



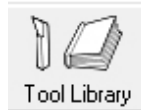
Within the tooling window are 8 tool positions. The tooling will default to that of a machine fitted with an 8 station turret.

Remember that on an ATC machine odd numbers are assigned to external tools such as Roughing, Finishing and Threading Tools and even numbers are assigned to internal tools such as Boring Bars and Drills.

If you have a manual machine set the tooling in the tooling window and assign the corresponding tool numbers to the tools in their holders. It is recommended that you stick numbers onto the tool holders.

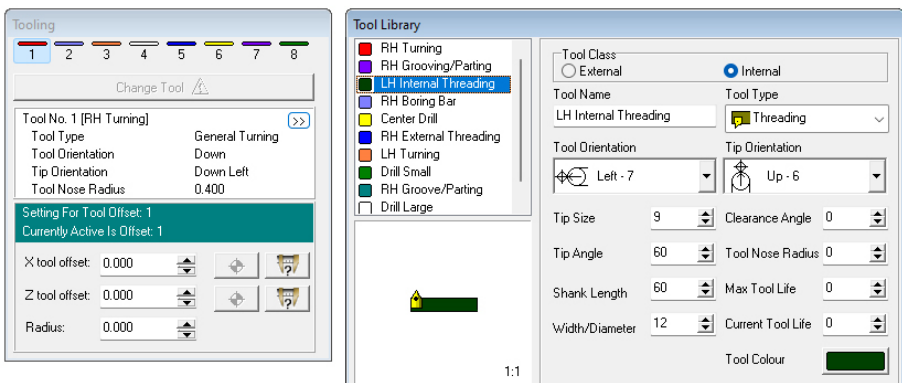


To change tools click on the [Tool Library] button to display the "Tool Library" window.



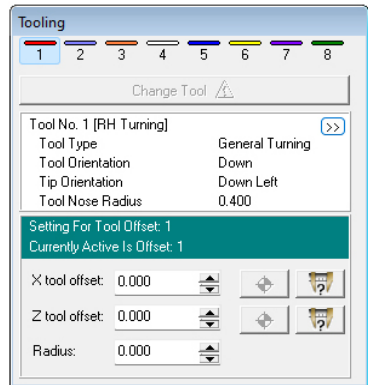
Within the tool library window is a list of predefined tools. All the tools you are likely to use are defined here. New tools can be added if required (see software help files for details.)

To change the tooling; left click, hold and drag the tool you want from the "Tool Library" and drop it on the appropriate tool number in the "Tooling" window.



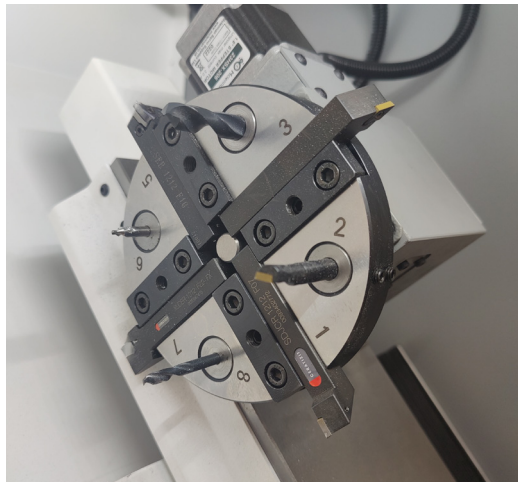
This example shows the Comprehensive Tooling Package that can be optionally added to a Turn 270 Pro.

If you wish to make any change from this tooling package, please note that each tool can only be placed one within the tooling window, if for any reason two tools of the same profile are to be fitted to the turret then a new tool must be created



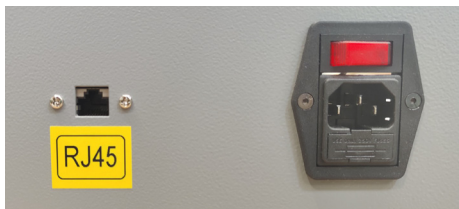
The tools as configured in the 'Tooling Window' and in the ATC on the machine are:

1. Right Hand Turning
2. Right Hand Boring Bar
3. Left Hand Turning
4. Drill Large (10mmØ)
5. Right Hand External Threading
6. Centre Drill (5mmØ)
7. Right Hand Grooving/Parting
8. Drill Small (5mmØ)



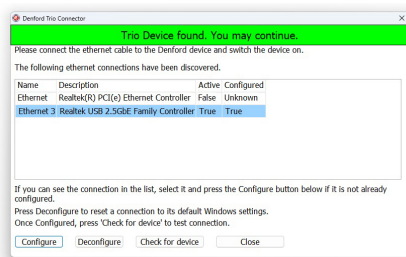
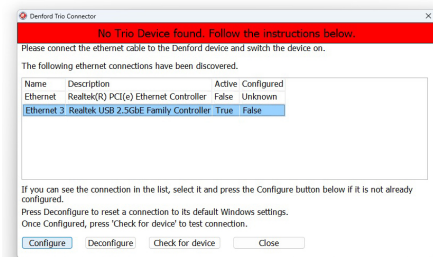
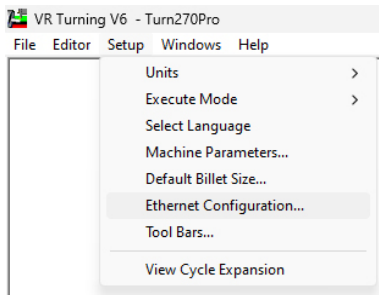
6) - Connect to, and Home the CNC Machine

At this point ensure that the power cable is connected to the machine and is switched on and that the Ethernet (RJ45) cable is connected to the machine and PC either directly into the Ethernet port or via the supplied Ethernet to USB converter.



On first connecting your PC to the machine you will need to configure the Ethernet port. This is done by selecting 'Setup' and 'Ethernet Configuration' as shown right.

This opens the Denford TRIO Connector, if the ethernet is not configured then the red banner will be displayed, click on the configure icon (VR Turing 6 may need to be run in Administrator mode) with the ethernet port to be used highlighted.

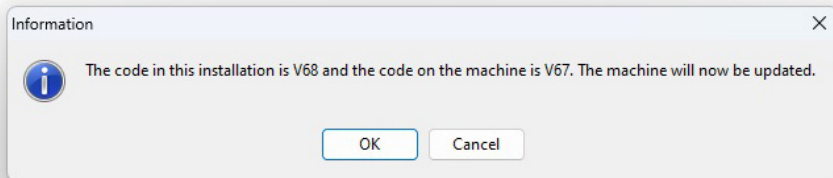


Once configured the software will automatically look for a TRIO controller, if detected then the banner will turn green. if the machine is not detected, ensure that the machine is powered on, the ethernet cable is connected and the correct ethernet port is selected.

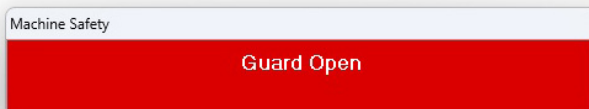
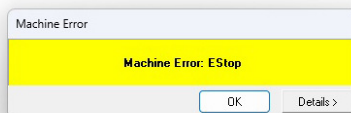
With this completed, to connect to the CNC machine, left click the [VR Machine] button.



Depending on the software version installed on the machine a prompt to update the machine code may appear. click ok and once completed the PC will be connected to the machine.



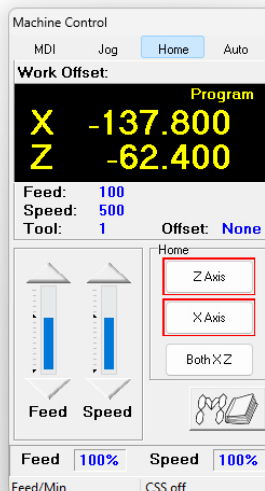
You may see one of the following error messages, either the E-stop button is depressed or the guard is not closed.



Once a connection has been successfully established, the machine “Control Panel” window will appear.

At the moment, only the “Home” mode tab is available.

Click the [Both XZ] button to home both machine axes.



7) - Move the Cutting Tool

After homing, the “Jog”, “Auto” and “MDI” mode tabs become available.

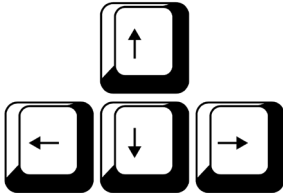
The position of the cutting tool can be manually controlled using Jog mode. In the “Control Panel” window, click the “Jog” tab to select Jog mode.

When Jog mode is active, the Jog panel text is displayed on a green background, as shown in the screenshot right.

Click and drag the jog speed knob to adjust the Jog speed. The feedrate value is shown in the readout below the jog speed knob.

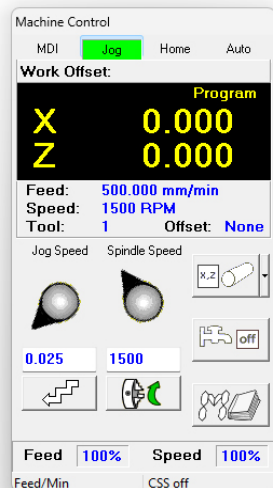
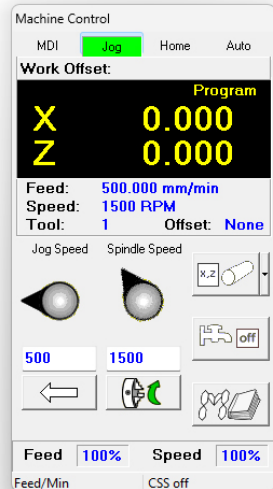
The machine will default to ‘Jog Continuous’ this is indicated by the straight arrow underneath the job speed knob. This allows fast movement of the machine axes.

The four [cursor arrow] computer keys are used, by default, to control the X and Z axes. Press and hold the appropriate computer key to move the required axis.



To change the position of the axes with smaller, more accurate movements, click the straight arrow once, this will now change to a stepped arrow, signifying Jog Step mode, as shown in the screenshot right.

Click and drag the jog speed knob to the required part of the scale. The value in the readout below the jog speed knob indicates how far the axes will move each time a jog computer key is pressed.



8) - Fit the Material in the Machine.

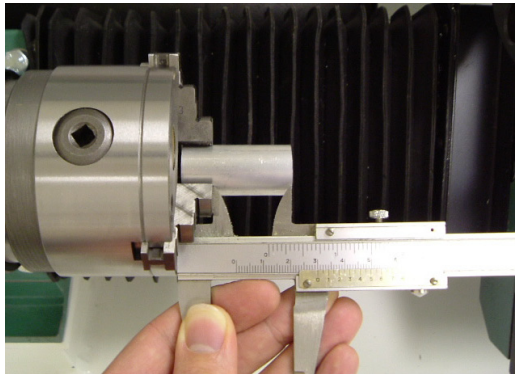
Fit your billet in the chuck of the CNC lathe. For the ease of managing the work offsets this guide will start with 50mm of material protruding from the chuck jaws. From this offset, other offsets can be created easily with some simple maths.

The maximum diameter that can be fitted will depend on the machine in use, for the standard range of machines a:

- A) - Microturn pro can part off a maximum billet of
- B) - Turn 270 Pro can part off a maximum billet of 25mm
- C) - Turn 370 Pro can part off a maximum billet of 35mm

Denford Lathe work offsets are set using tool 1 as the reference tool, this tool will have a 0,0 tool offset.

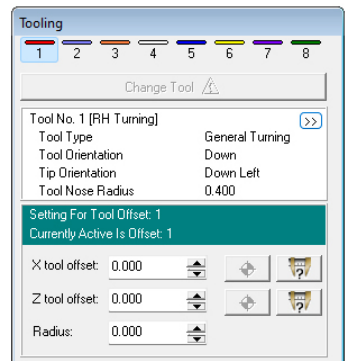
Using vernier callipers set the billet out by $50.20\text{mm} \pm 0.20\text{mm}$ from the chuck jaws.



9) - Performing a Tool Change

Ensure the selected tool is tool 1,
to carry out a tool change:

- A) -Open the tooling window
- B) - Click tool number 1
- C) - Click 'Change Tool'



A manual tool change can only be carried out in the Jog window, if the 8 Station turret is fitted to the machine, the machine will move to the set safe position and change to tool 1. If the manual tool change system is fitted then the machine will move to the tool change position and prompt on the screen to fit tool number 1, once the correct tool has been fitted press the OK button.

10] - Set the Workpiece Offsets

What are offsets?

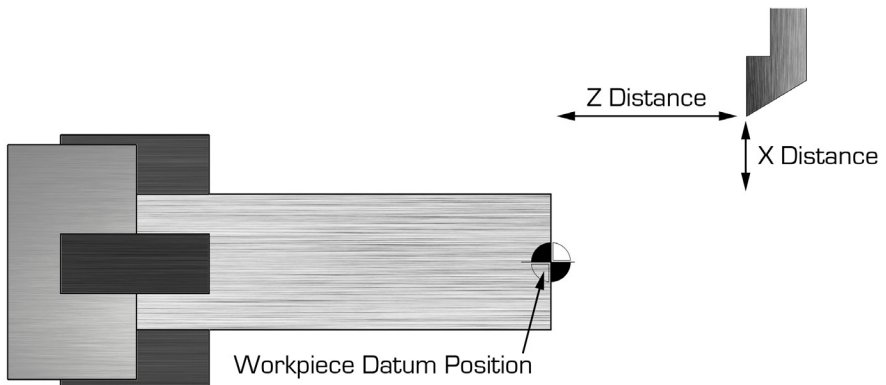
Offsets are the distances the cutting tool needs to travel, from its 'Home' position to the Workpiece datum in X & Z.

The workpiece datum is usually the centre of the billet on the front face. (The datum position is defined by the software used to create the program)



Work Piece Offsets and Tool Offsets

To tell the machine where the material is Tool number 1 will be used to face off the billet, an offset will be saved from this, this is the Z datum. To find the centre of the billet, material will be turned down, measured and saved, this is the X datum. Both the Z and X datum positions will be stored, this is the Work Piece Offset. each tool will then be touched onto these machined surfaces and the difference from Tool 1 saved. These are the Tool Offsets.



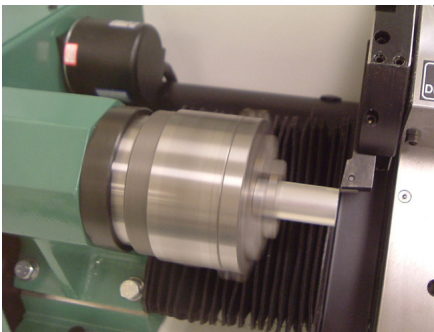
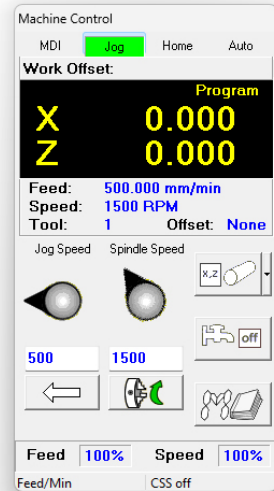
Set the Z Offset

Using the techniques described in Step 6 of this guide, 'jog' Tool number 1 near to the end of the billet without touching it.

Start the spindle and set the speed to approximately 1500 RPM. This can be done by clicking the start spindle icon below the Spindle Speed dial. The speed can be adjusted with the Spindle Speed dial.

Using incremental jog set at 0.1mm touch the cutter onto the end of the bar.

You will be able to hear as soon as the cutter touches the rotating billet.

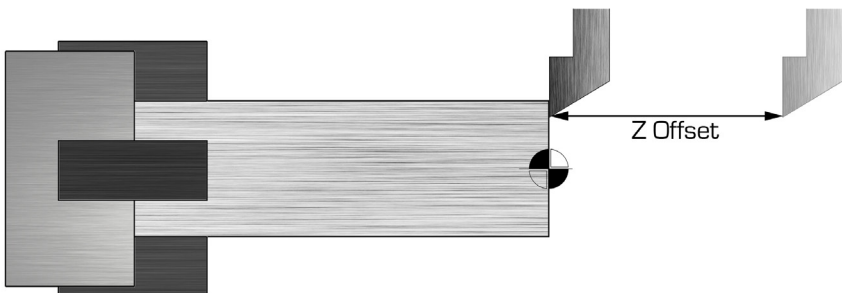


Go back to continuous jog and at approx. 100mm/min jog the cutter in the X axis to 'Face-off' the end of the billet.

Repeat this process until the end of the billet is flat (Perpendicular to the length of the billet).

Do not move the cutter in the Z axis because we need to store this position.

Take the vernier callipers again and measure how much of the billet is protruding from the chuck jaws, the aim here is to have a billet at $50.00\text{mm} \pm 0.02\text{mm}$. If the billet is greater than this remove some more material. if it is less than this go back to step 8 and repeat until the billet is protruding the desired amount.



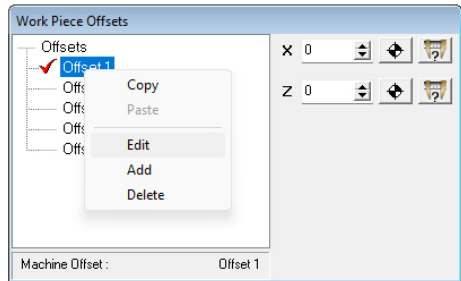
Set the Z Offset

With the cutter in position at the end of the billet click the [Offsets] button.

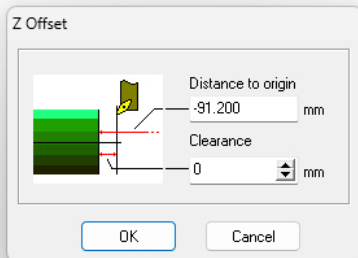


This will open the 'Work Piece Offsets' window.

Right click on the currently active offset, this is identified by the red tick. navigate to edit and rename the offset to 50mm Bar, this will help in identifying the correct offset in the future.



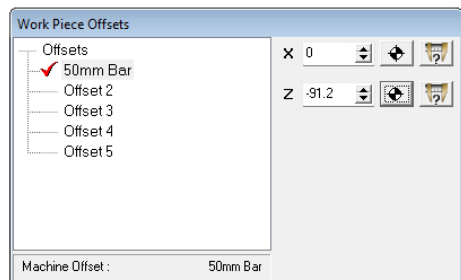
Click on the datum icon  button next to the Z position.



The value will be transferred into the 'Work Piece Offsets' window.

The Z Offset is now set.

The 'Z Offset' window will open. The Distance to origin box is the distance the tool has travelled from it's Home position to its current position. the Clearance box is how far from the face of the billet the tool is, as the tool is touching this is 0. Click [OK] to store the position.



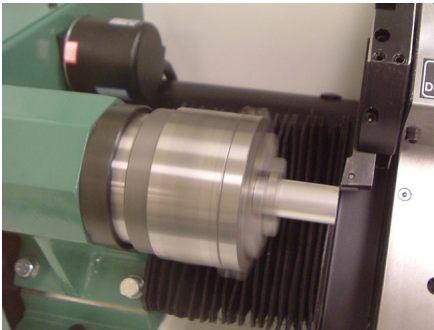
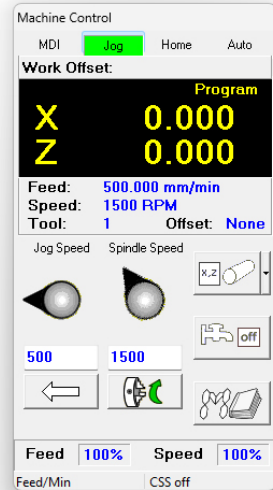
Set the X Offset

Using the techniques described in Step 6 of this guide, 'jog' Tool number 1 near to the diameter of the billet without touching it.

Start the spindle and set the speed to approximately 1500 RPM. This can be done by clicking the start spindle icon below the Spindle Speed dial. The speed can be adjusted with the Spindle Speed dial.

Using incremental jog set at 0.1mm touch the cutter onto the diameter of the bar.

You will be able to hear as soon as the cutter touches the rotating billet.

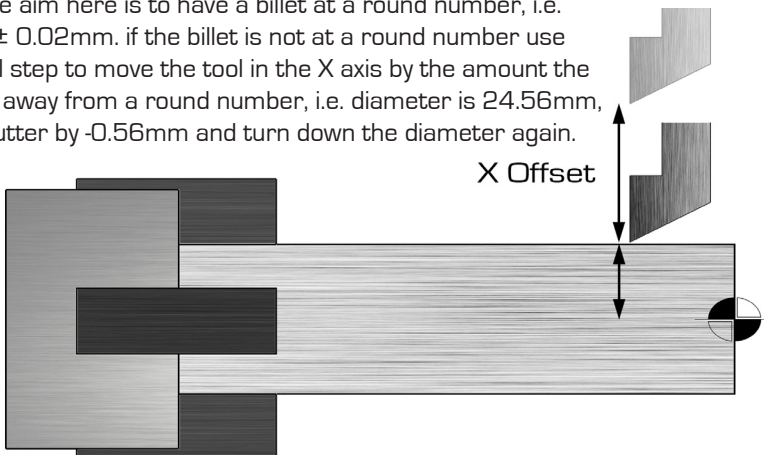


Go back to continuous jog and at approx. 100mm/min jog the cutter in the X axis to 'Turn down' a section of the diameter of the billet.

Repeat this process until the end of the billet is round and all of the diameter has had material removed.

Do not move the cutter in the X axis because we need to store this position.

Take the vernier callipers again and measure the diameter of the billet, the aim here is to have a billet at a round number, i.e. 24.00mm \pm 0.02mm. If the billet is not at a round number use incremental step to move the tool in the X axis by the amount the diameter is away from a round number, i.e. diameter is 24.56mm, move the cutter by -0.56mm and turn down the diameter again.



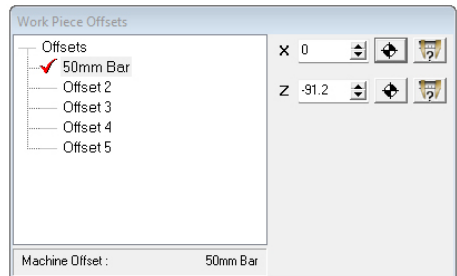
Set the X Offset

Once a round number is measured on the diameter of the billet, keep the cutter in position on the diameter of the billet and click the [Offsets] button.

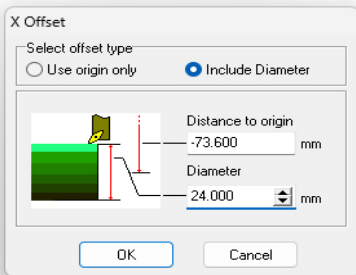


This will open the 'Work Piece Offsets' window.

Left click on the currently active offset, this is identified by the red tick.



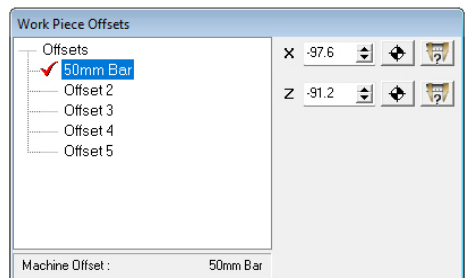
Click on the datum icon  button next to the X position.



The 'X Offset' window will open. The Distance to origin box is the distance the tool has travelled from its Home position to its current position. the Clearance box is the diameter of the billet which has just been turned down (also known as the distance to centre), type here the measured diameter.

The value will be transferred into the 'Work Piece Offsets' window.

The X Offset is now set.



11) - Set the Tool Offsets

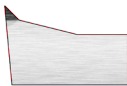
As highlighted earlier in this guide the Tool Offsets are referenced from tool 1, In order use other tools the 'Tool Offsets' need to be set. Tool Offsets allow a variety of tool profiles to be used together on the same CNC program. This is achieved by offsetting their differences in position against a fixed reference point, Tool 1.

The tools listed below are included in both the Turn 270 Pro and the Turn 370 Pro Comprehensive Tooling Package. This guide will set 1 external and 1 internal tool. To learn how to set the rest of the tools, please refer to the Lathe Tooling Guide.

1. Right Hand Turning



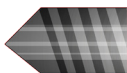
2. Right Hand Boring Bar



3. Left Hand Turning



4. Drill Large (10mmØ)



5. Right Hand External Threading



6. Centre Drill (5mmØ)



7. Right Hand Grooving Parting



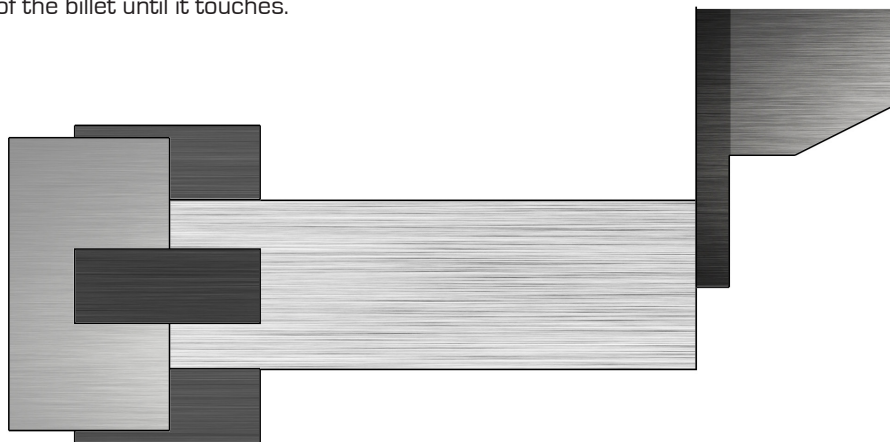
8. Drill Small (5mmØ)



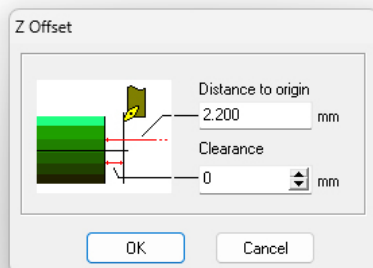
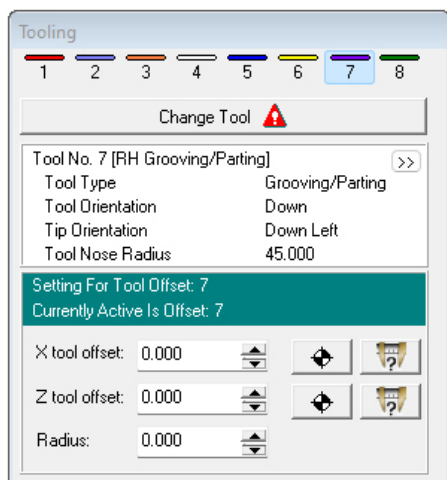
For the majority of the external tools, the setup is the same, in this example tool number 7 (RH Grooving Parting) will be set as this tool is also supplied as standard with both the Turn 270 Pro and the Turn 370 Pro. With reference to section 9 change to tool number 7.

Set the Z axis Tool Offset Tool 7

With reference to section 10, start the spindle and jog the Parting tool to the face of the billet until it touches.

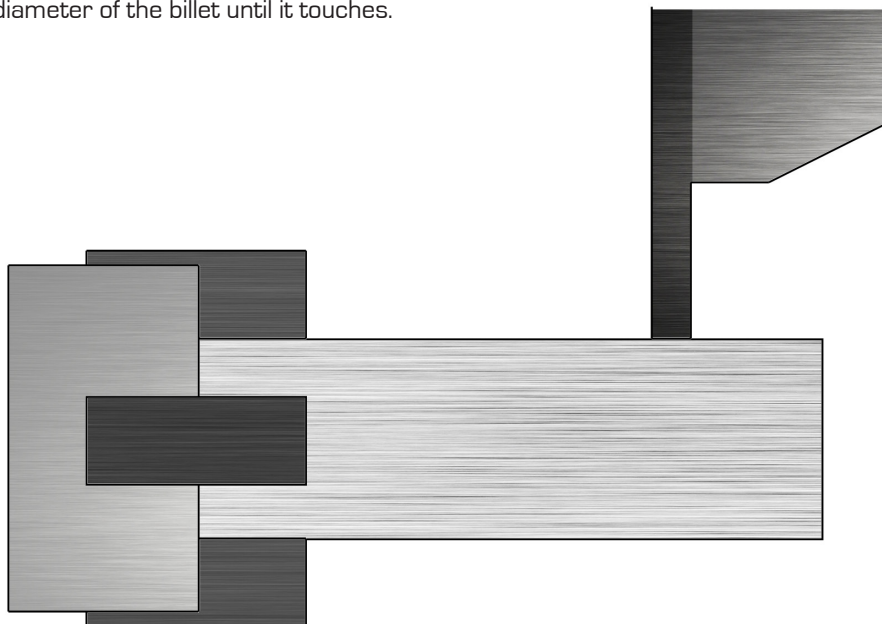


Open the 'Tooling' window and click on the datum icon next to the Z tool offset to set the offset, as it is touching the Clearance value is 0.

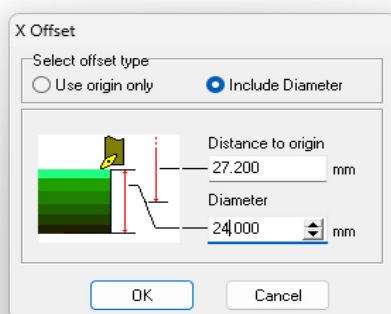
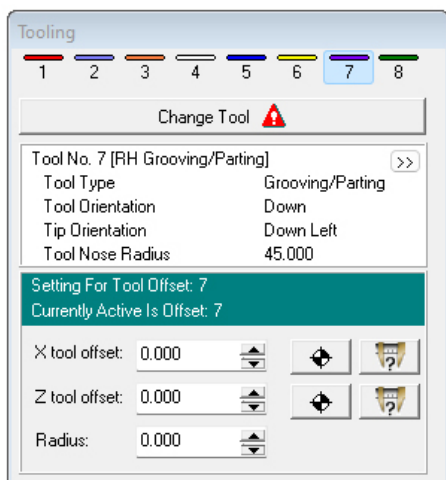


Set the X axis Tool Offset Tool 7

With reference to section 10, start the spindle and jog the Parting tool to the diameter of the billet until it touches.



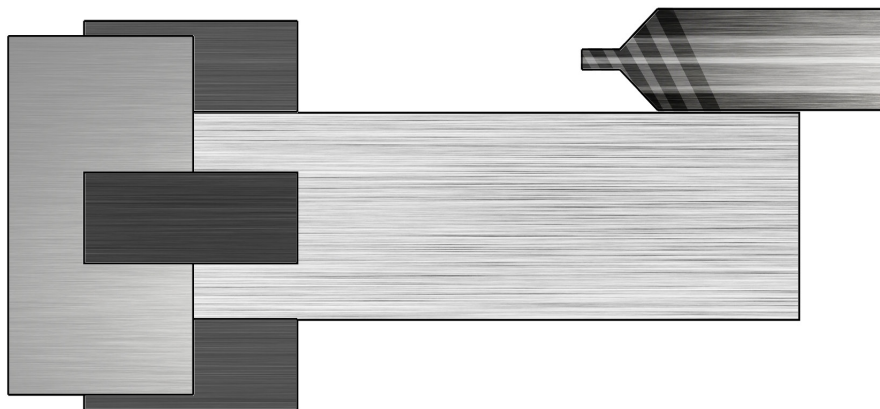
Open the 'Tooling' window and click on the datum icon next to the X tool offset to set the offset, type in the measured diameter from section 10 in the Diameter box.



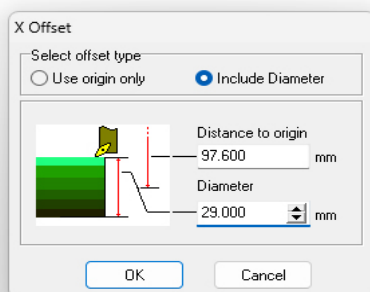
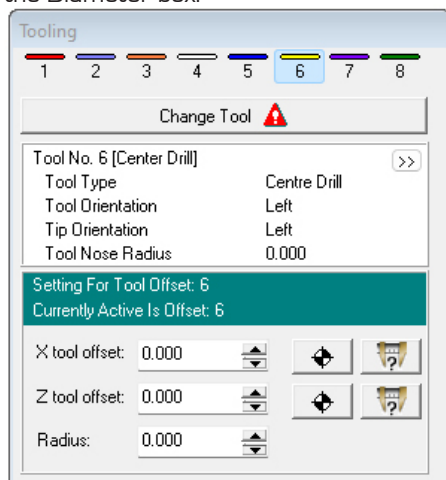
For the majority of the internal tools, the setup is the same, in this example tool number 6 (Centre Drill) will be set. With reference to section 9 change to tool number 6. The X offset will be set first to ensure when the Z is set the tool is in the centre.

Set the X axis Tool Offset Tool 6

With reference to section 10, start the spindle and jog the Centre Drill to the Diameter of the billet until it touches.

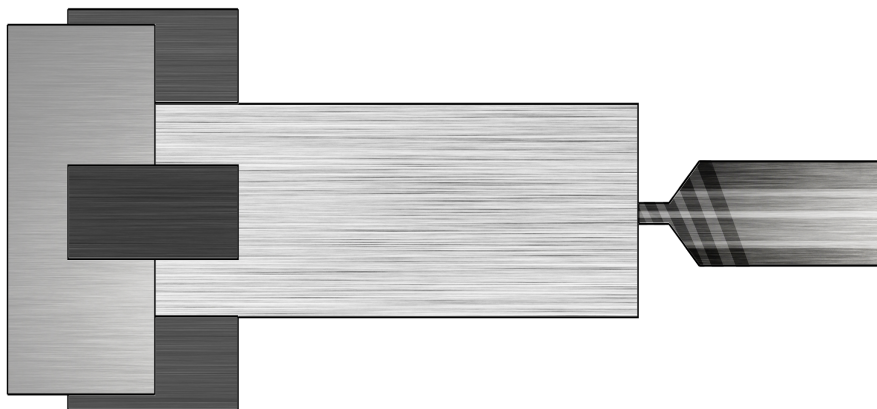


Open the 'Tooling' window and click on the datum icon next to the X tool offset to set the offset, as the centre of the tool is not touching the diameter, the diameter of the tool also needs to be taken into account, add the tool Diameter (5mm) and the billet diameter from section 10 (24mm) together. Type in the sum (29mm) in the Diameter box.

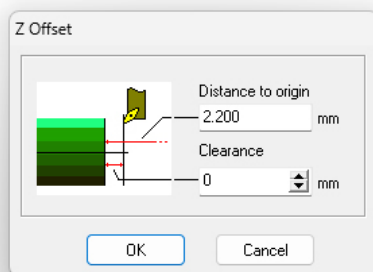
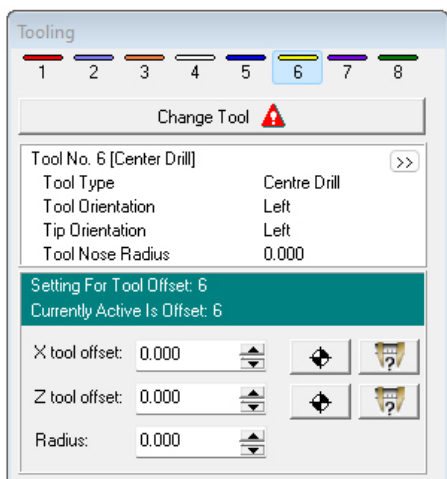


Set the X axis Tool Offset Tool 6

With reference to section 10, start the spindle and jog the Centre Drill to X 0 to ensure that the tool is on the centre, visually confirm that the tool is at the centre before jogging the Z axis to the face of the billet until it touches.



Open the 'Tooling' window and click on the datum icon next to the Z tool offset to set the offset, as it is touching the Clearance value is 0.



12) - Run a Simulation

To verify the program a 2D or 3D simulation can be run prior to manufacturing.

You can simulate the contents of the CNC file, if there is an unwanted move in the program, this will show in the simulation.

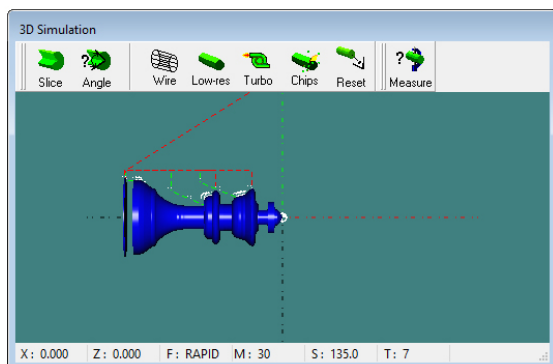
The simulation uses the information in the tooling window to generate the graphics. It is therefore important to have the tools set correctly in the tooling window to generate an accurate simulation.

Click on the 3D simulation button.

The 3D simulation window will appear.

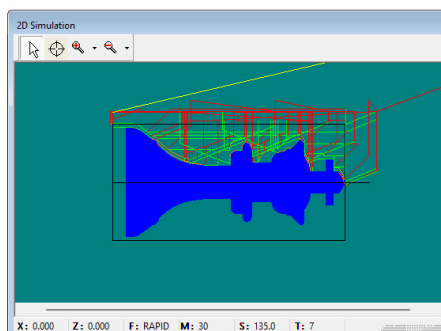
Using the 'File Control', press Stop,

Rewind and then Play to run the simulation.



The menu at the top has multiple options for different views, try the different ones when running the simulation.

Alternatively a 2D simulation can be run.



13) - Run the Machine

The program is now ready to be run.
To run the machine, you must click the 'Auto' tab in the Control Panel.

The program must be run from the beginning, to ensure this is the case click the Stop button, followed by Rewind and finally the Start button.

The program will begin to run.

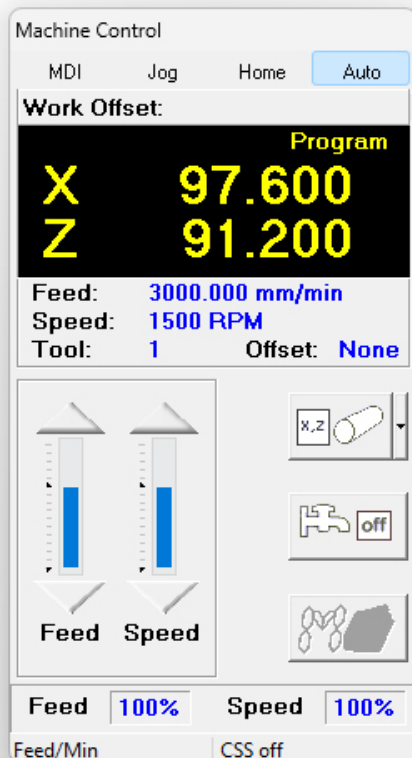
If you have a manual toolpost a message may appear asking you to change tools. Check you have the correct tool and click [OK]. The spindle will start and the program will begin.

In 'Auto' mode there are the Feed and Speed override indicators.

If the machine you are using is fitted with potentiometers it is these that are used to override the Feed rate and Spindle speed.

If your machine does not have potentiometers, you can click the arrows above the Feed and Speed text.

note: If there is no response, check that the 'Feed pot' and 'Speed pot' parameters are unticked in the Machine setup parameters 'machine capability' settings.



14] - Fine Tuning Offsets

When the program is first run, take note of which tool carries out which operation. If when the part has been finished the measured size is not the programmed size, the offsets can be corrected easily. please take note that if the offset for tool 1 is changed then it is advisable to re-run the program and check the measurements again before making adjustment to other tools.

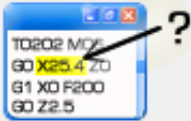
Tool 1 should only ever need correction in the X axis, if the part is parted off at a length different to the programmed length, then the Parting Off Tool needs correction.

To correct the tool 1 X axis, open the Offsets window and click on the calliper icon next to the X.

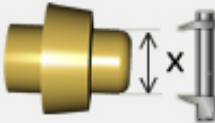


This will open the Measured difference window shown below. Enter the Programmed Size in the top box and the Measured Size in the bottom box. this will adjust the offset to compensate for the error.

Measured Difference



Programmed Size:

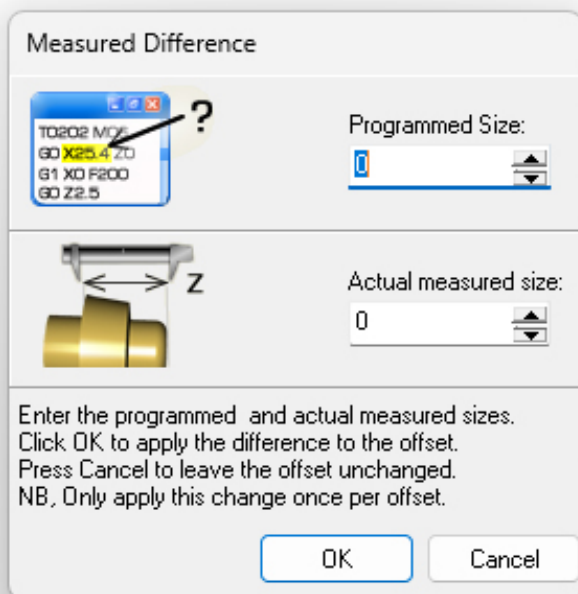


Actual measured size:

Enter the programmed and actual measured sizes.
Click OK to apply the difference to the offset.
Press Cancel to leave the offset unchanged.
NB, Only apply this change once per offset.

To carry out adjustments on the different tools. Again, measure the profile cut with the tool in question. select the tool within the tooling window and click on the Z or X calliper icon depending on which measurement is incorrect.

Within the Measured Difference window, the menu will prompt on which measurement to take.



15] - Summary

This guide is intended as a refresher after attending one of the Denford Training courses. if you require any further assistance, please:

Visit our Forum - <https://www.denfordata.com/bb/>

Email Technical - technical@denford.co.uk

Call our office - 01484 728000