
DENFORD

# CNC Turning Training


www.denford.co.uk

# Table of contents

# Introduction

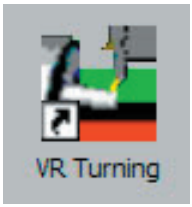
VR CNC Turning is a Windows based software package allowing full editing and control of CNC files, either offline (away from the CNC machine) or online (controlling the operation of a CNC machine).

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

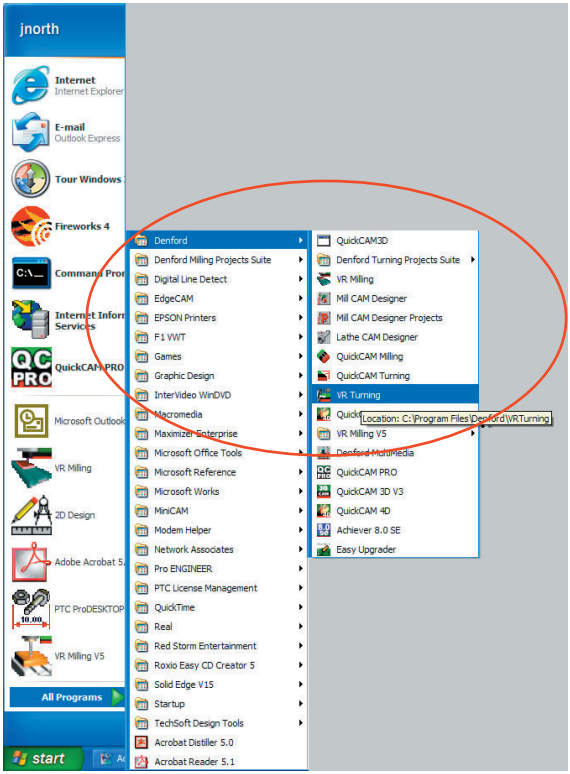
Check that you have the latest version of the VR Turning software by going to the 'Downloads' section of the Denford Website (www.denford.co.uk). You can download the latest version from the website, you will need your licence disk to be able to run the software.

# Step1 - Start the VR Turning software

To start the VR Turning software double-click the VR Turning shortcut icon (if available) on your desktop.




If the shortcut is not available, click “Start” on your “Windows” Start button followed by the “Programs” option, the program group “Denford” and finally the “VR Turning ” icon.



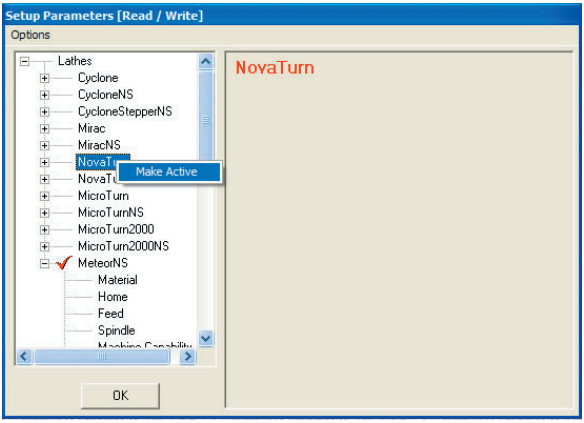
# Step 2 - Configure the software for the machine

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 “NEWTURN270NS” text indicates that a Denford NEW TURN270 can be controlled by the software. The model of the lathe you are using is indicated on the information plate on the side of the machine.


To change the name 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. You may need to look at the CE identification panel on your Denford CNC machine and choose the matching name from the parameter list
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.

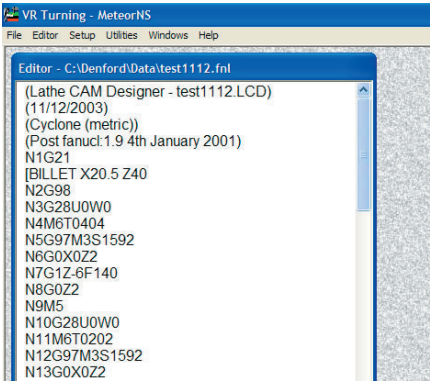


# Step 3 - Load your CNC file

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.




# Step 4 - 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..."

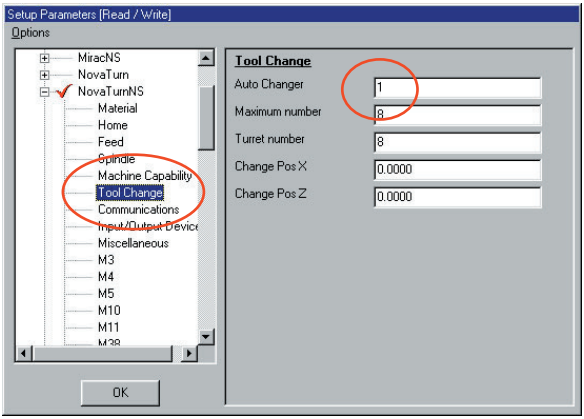
In the 'Password' window enter your password (the default is **denny**).

The “Setup Parameters [Read/Write]” window will appear.

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.

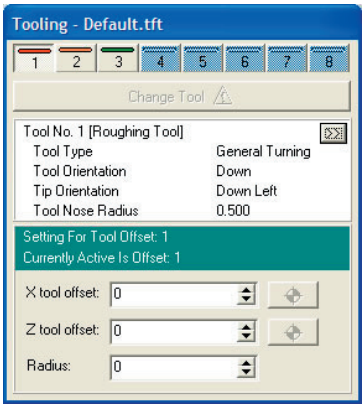


# Step 4 - Configure the tooling

Click the [Tooling] button, to display the “Tooling” window.

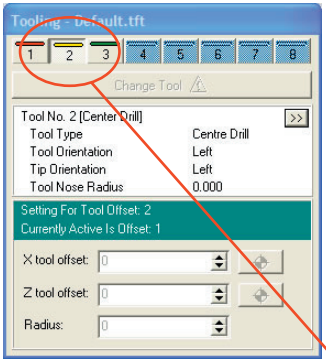

Within the tooling window are 8 tool positions. If you have an ATC CNC Lathe set the tooling to correspond with the tool positions in the turret on the machine.

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.



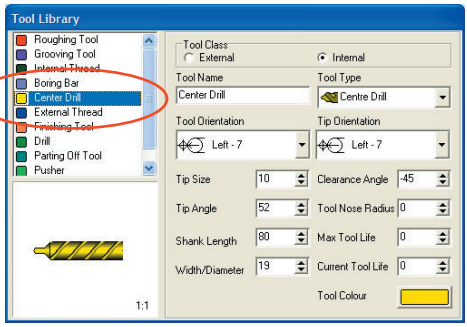

If you have a manual machine set the tooling in the tooling window and assign the corresponding tool numbers to the tools in their holders. It is recommended that you stick numbers onto the tool holders.


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.)

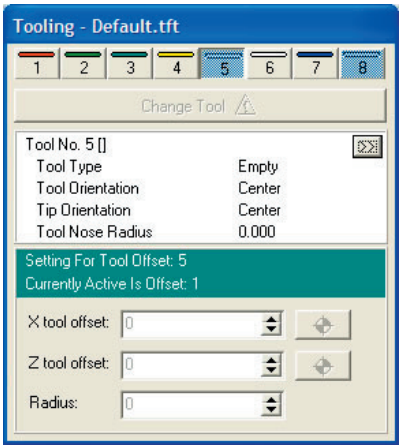
To change the tooling; drag the tool you want from the "Tool Library" and drop it on the appropriate tool number in the "Tooling" window.




Left Click, hold the mouse button and drag the tool from the "Tool Library" to the tool number in the "Tooling" window, release the mouse button to drop the tool in the required tool position.

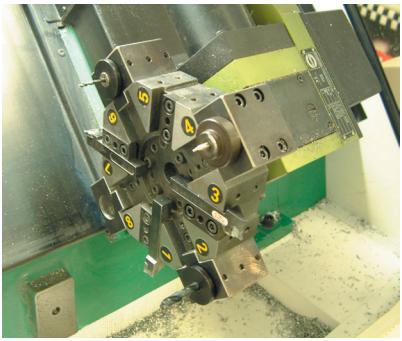
In this example 6 different tools have been configured in the tooling window. Tool positions 5 and 8 are empty.

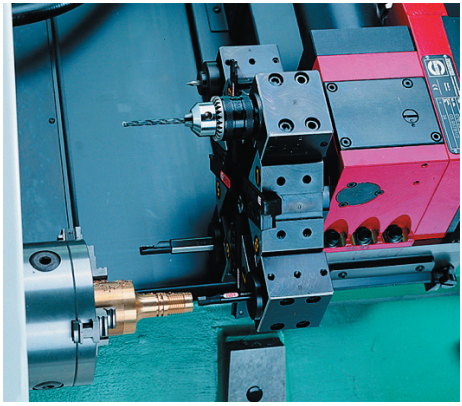
Tools cannot be reused i.e: they can only be put in one position. If you want to use more than one drill for example you will have to create a new tool for each drill.




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

1. Roughing/Finishing Tool
2. 8mm Drill
3. Parting Off Tool
4. Center Drill
5. *Empty*
6. 3.3mm Drill
7. External Thread Tool
8. *Empty*





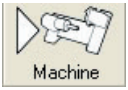
In the example pictured left, all the tool positions are occupied. A number of Boring Bars are fitted along with a Jacobs Chuck containing a drill.

If using the Jacobs Chuck take care to ensure it does not collide with the Machine Chuck when using one of the other tools such as the 'Parting Off' tool. The position of tools may have to be changed to avoid this.

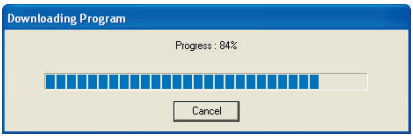
# Step 5 - Connect to, and Home the CNC machine

At this point ensure that the cable is connected from the PC to the Machine and that the machine is switched on.

To connect to the CNC machine, left click the [Machine] button.




A progress bar will appear, allow this to reach 100% and a connection will be established between the machine and the PC.





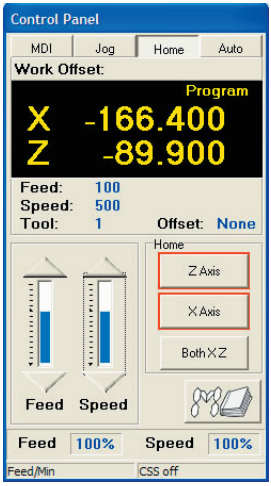

If you see the error message (left) it is either because the Emergency Stop button is pressed or that the drives are not enabled.

If necessary release the Emergency Stop button. On some models it you will need to press the 'Drives On' button to enable the drives. (On most machines the drives power up automatically)

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.

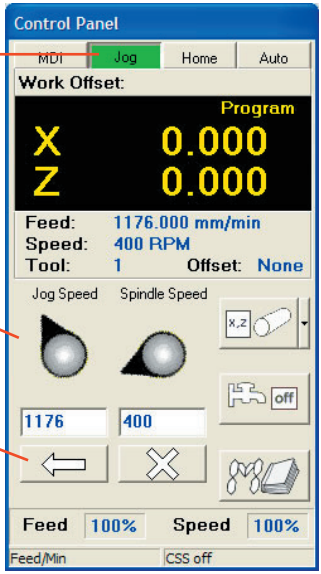


# Step 6 - Move the cutting tool

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.

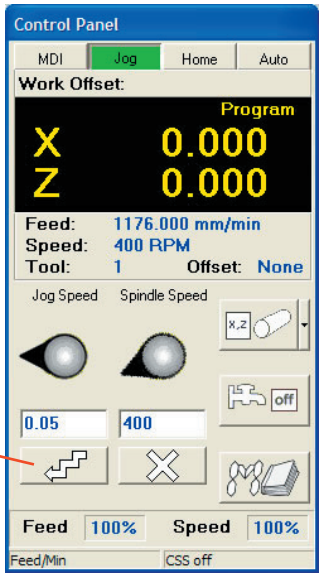



Click and drag the jog speed knob to adjust the Jog speed. The feedrate value is shown in the readout below the jog speed knob.

To change the position of the machine head quickly, click the [Jog] button until a straight arrow is displayed, signifying 'Jog Continuous' mode.

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

To change the position of the machine head with smaller, more accurate movements, click the [Jog] button until a stepped arrow is displayed, signifying Jog Step mode, as shown in the screenshot right.



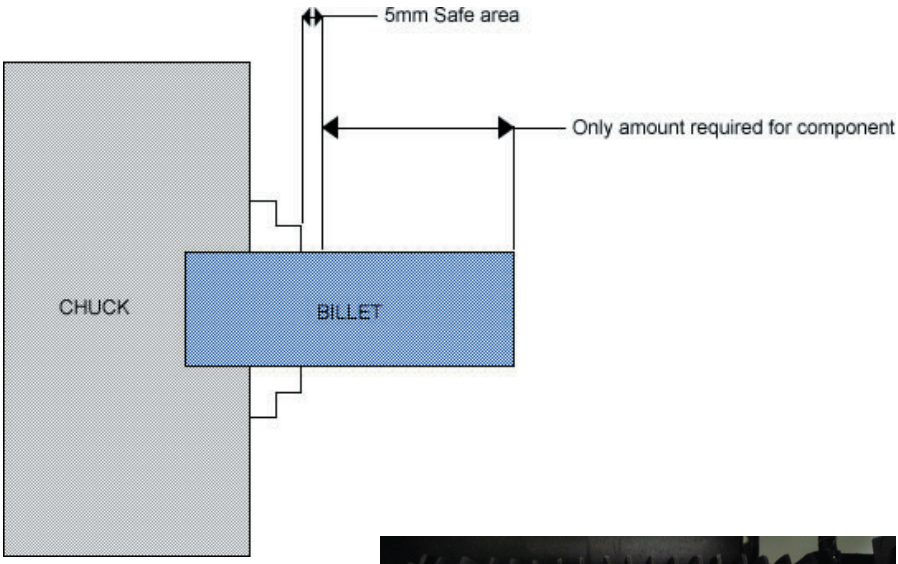

Click and drag the jog speed knob to the required part of the scale. The value in the readout below the jog speed knob indicates how far the axis will move each time a jog computer key is pressed.

# Step 7 - Fit the material in the machine.

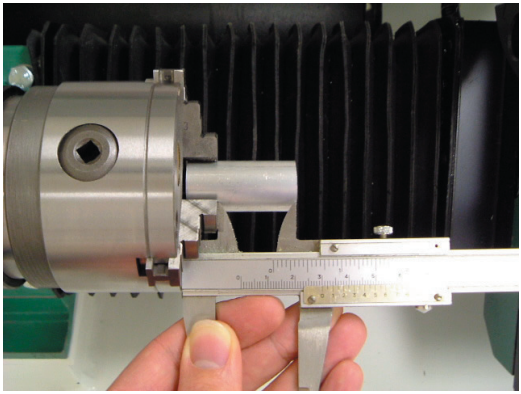
Fit your billet in the chuck of the CNC lathe. Make sure there is enough of the billet protruding to cut your design & 5mm safe area to allow for 'Parting Off'. For example if your finished component will be 40mm long, leave 45mm of billet protruding from the jaws of the chuck. Do not have more material than is necessary to create your component protruding from the jaws of the chuck.

You may have to go back to your CAD package to check the amount of material required.

The maximum diameter that can be fitted will depend on the size of the machine chuck but the standard parting off tool can only part off a billet of 1 1/4" (32mm) or less.




You will have to measure the amount of material protruding from the jaws of the chuck.



# Step 8 - Performing a Tool Change

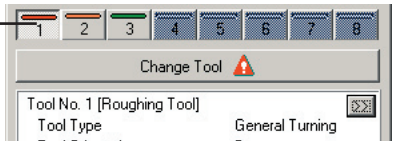
### Change to Tool Number 1

To perform a tool change use the following procedure:

Note: The guard must be closed for this operation.

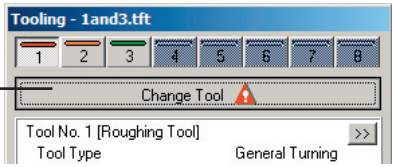
1. Open the tooling window


2. Click on the tool number



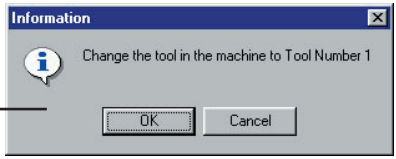

3. Click on the Auto tab in the 'Control Panel'


4. Click the 'Change Tool' button to perform the toolchange.




If your machine has an ATC the tool post will traverse to the tool change position, then rotate to change the tool.

If you have a manual tool post the machine will move to the tool change position, a message will then appear to prompt you to change tools, at this point you will have to open the door and manually change tools.

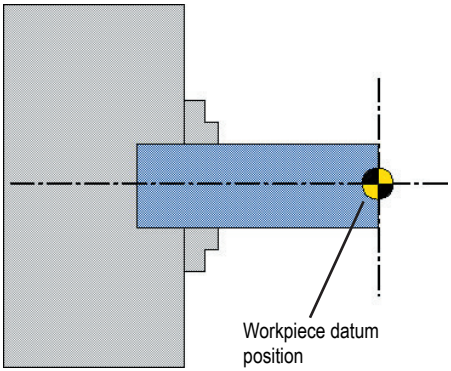


# Step 9 – Set the Work Piece Offsets

### What are offsets?

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

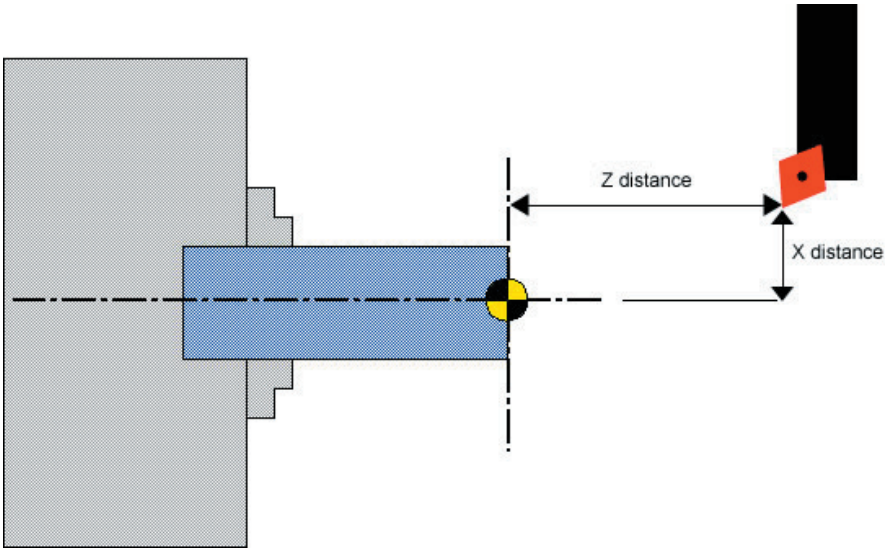
The workpiece datum is usually the centre of the billet on the front face.
(The datum position is defined by the software used to create the program)




### Work Piece Offsets and Tool Offsets

To tell the machine where the material is we will change to Tool number 1 and move the tool to the front face of the billet, this is the Z datum. We will then find the centre of the billet, this is the X datum. The Z and X datum positions will be stored, this is the Work Piece Offset.

If more than one tool is to be used we will change to each tool and set the positions for each tool relative to Tool number 1. These are the Tool Offsets.



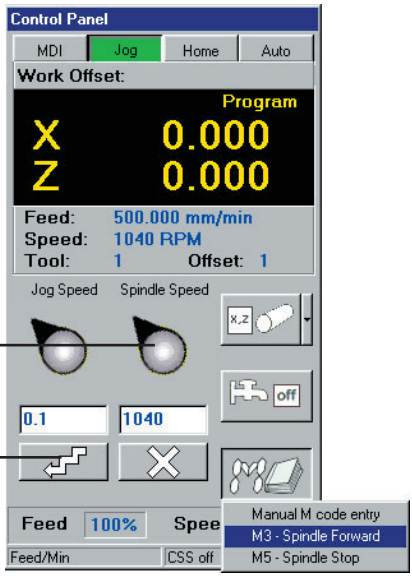
# Set the Z Offset

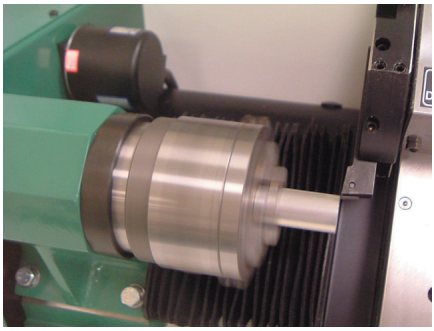
Using the techniques described in Step 6 of this guide, 'jog' Tool number 1 near to the end of the billet without touching it.

Start the spindle and set the speed to approximately 1000 RPM. This can be done by clicking the [M codes] button in the 'Jog' panel and selecting 'M3 - Spindle Forward'. 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.

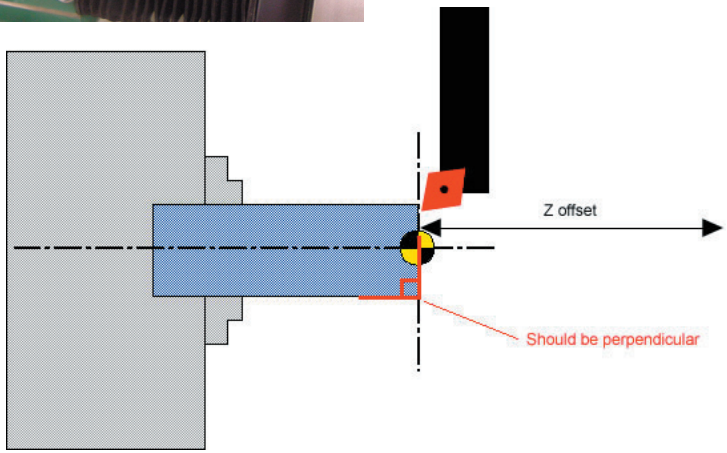


If you like you can go back to continuous jog and at approx 100mm/min jog the cutter in the X axis to 'Face-off' the end of the billet.

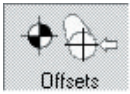
Repeat this process until the end of the billet is flat (Perpendicular to the length of the billet).

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



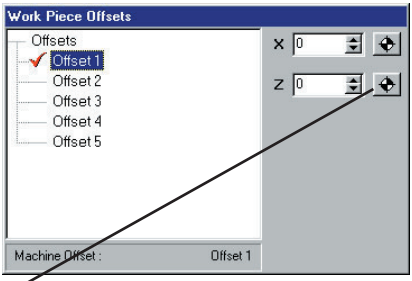
# Set the Z Offset

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

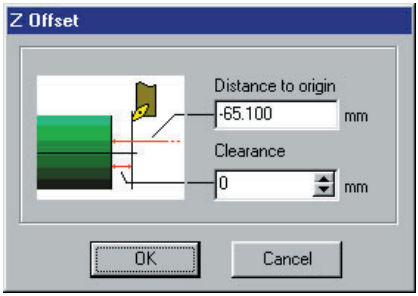



The 'Work Piece Offsets' window will open.

Ensure that 'Offset 1' is currently highlighted and that there is a red tick next to it. This indicates that Offset 1 is the position that the machine will use. It is possible to store a number of positions and swap between them for different jobs.

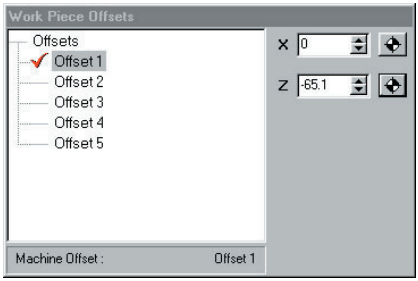



Click on the button next to the Z position.




The 'Z Offset' window will open. In the top dialogue box is the distance the tool has travelled from it's Home position to its current position.

Click [OK] to store the position.

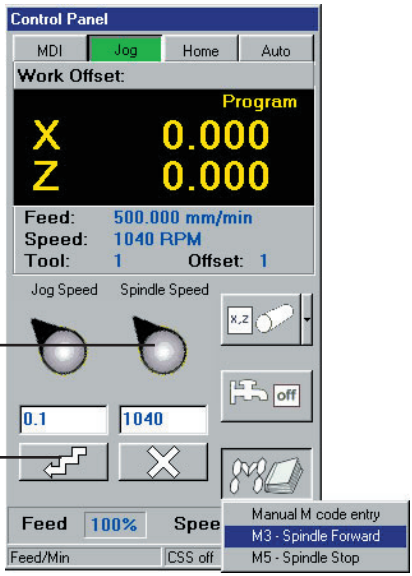



The value will be transfered into the 'Work Piece Offsets' window.

The Z Offset is now set.

# Set the X Offset

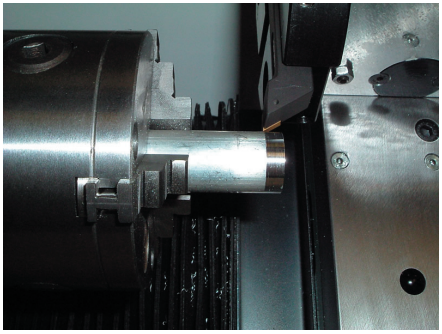
Using the techniques described in step 6 of this guide, 'jog' Tool number 1 near to the diameter of the billet without touching it.




Start the spindle and set the speed to approximately 1000 RPM. This can be done by clicking the [M codes] button in the 'Jog' panel and selecting 'M3 - Spindle Forward'. 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.

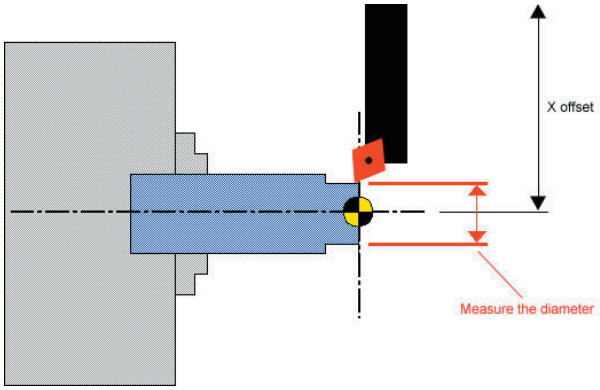
You will be able to hear as soon as the cutter touches the rotating billet.



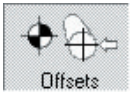
Go back to continuous jog and at approx 100mm/min jog the cutter in the Z axis to 'Turn down' a section of the diameter of the billet.

This process may have to be repeated in 0.1mm steps until the bar has material removed all the way around the circumference. (Until the billet is round)

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



# Set the X Offset



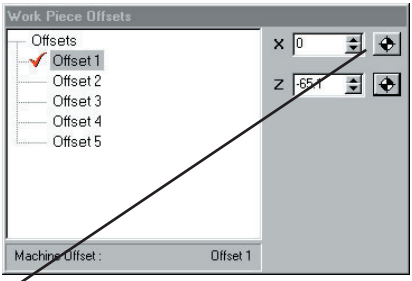

With the cutter in position on the diameter of the billet click the [Offsets] button.

The 'Work Piece Offsets' window will open.

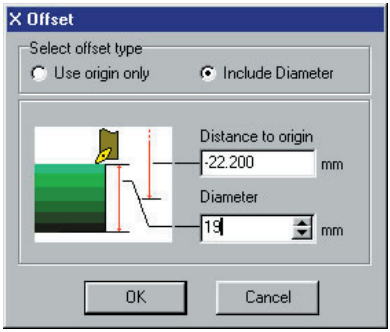
Ensure that 'Offset 1' is currently highlighted and that

there is a red tick next to it. This indicates that Offset 1

is the position that the machine will use. It is possible to store a number of positions and swap between them for different jobs.

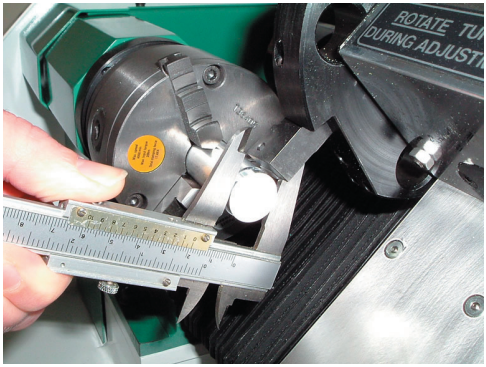


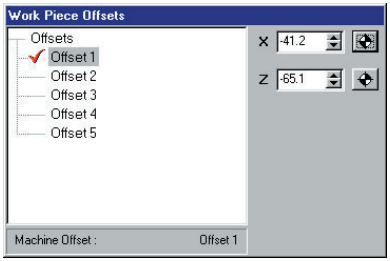

Click on the button next to the X position.




The 'X Offset' window will open. In the top dialogue box is the distance the tool has travelled from it's Home position to its current position.

In the bottom dialogue box enter the diameter of the area of the billet which has been turned down. You will need to measure the diameter.



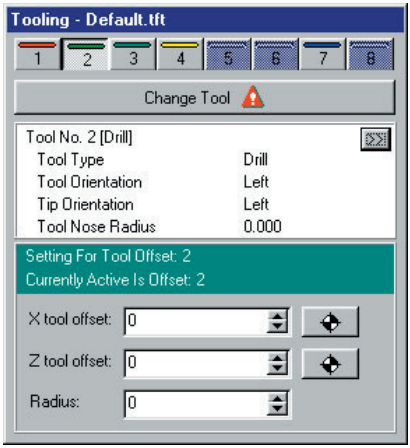

The value will be transfered into the 'Work Piece Offsets' window.

The X Offset is now set.

# Step 10 - Set the Tool Offsets

If your program uses tools other than tool 1, you will need to set 'Tool Offsets'. Tool Offsets allow a variety of tool profiles to be used together on the same CNC program. This is achieved by off setting their differences in position against a fixed reference point, this fixed point will be Tool 1.

Change tools as described in 'Step 8' of this guide.




In this example there are 5 tools configured:

Tool 1 is the Roughing/Finishing Tool

Tool 2 is a Drill

Tool 3 is the Parting Off Tool

Tool 4 is the Centre Drill

Tool 7 is the External Threading Tool

The other tool positions are empty. This corresponds with the Tools in the machines ATC.

In this example we will change to Tool 2 - Drill and set the tool offset.

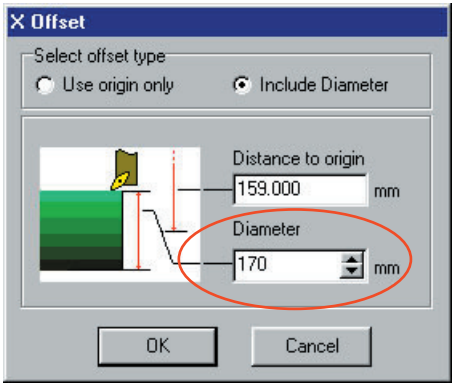
The way in which we set the X offset for a drill varies slightly depending upon the machine you are using. The aim is to find the centre-line of the machine chuck. If you are using a Meteor, Cyclone or Mirac then you can enter a number found on the door of the machine, for other models you need to touch the drill on the billet.

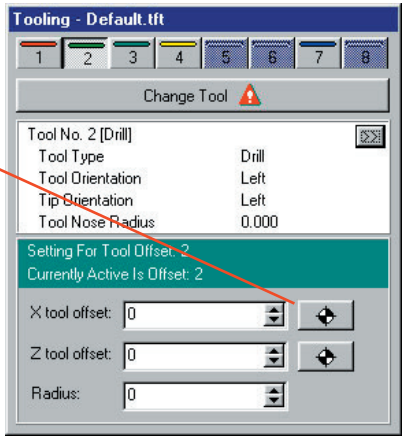
If using a Meteor, Mirac or Cyclone click the datum button next to the X tool offset dialogue box within the 'Tooling' window.

Type the value that is on the sticker on the door of the CNC machine in the 'Diameter' dialogue box in the 'X Offset' window.

In this example the value is *170*, this will vary from machine to machine.

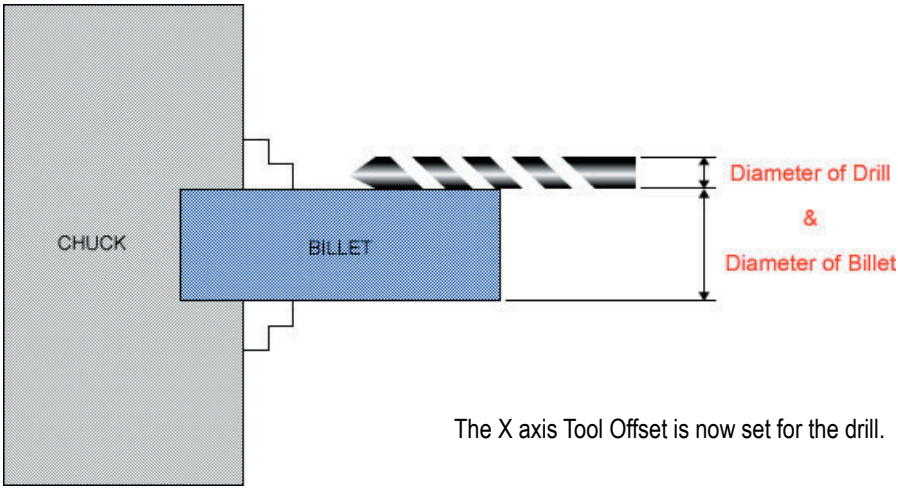
Click [OK].



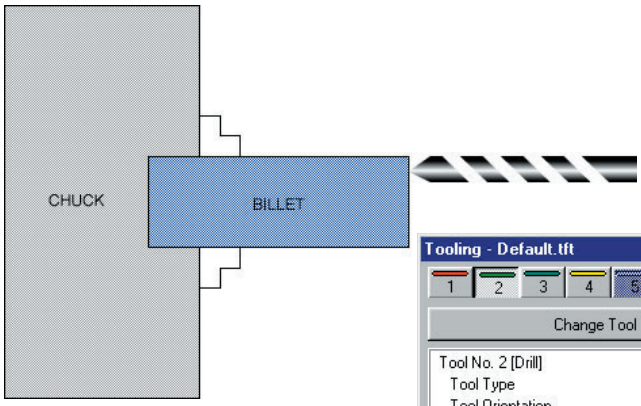

For other models you will have to 'jog' the drill and touch on the diameter of the billet.

With the drill in the position shown in the diagram below, enter the sum of the diameter of the billet + the diameter of the drill into the 'Diameter' dialogue box in the 'X Offset' window.

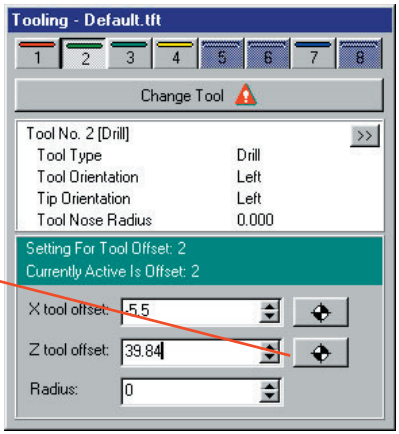



The X axis Tool Offset is now set for the drill.

To set the Z axis Tool Offset for the drill, touch the tip of the drill on the end of the billet

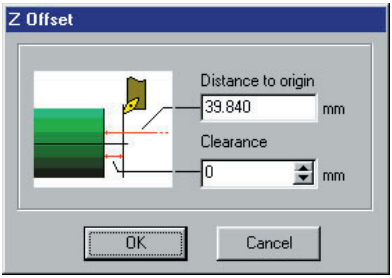


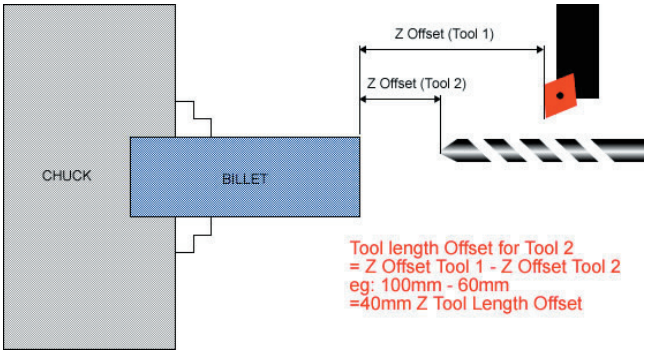

Click the datum button next to the Z tool offset dialogue box within the 'Tooling' window.




When the 'Z Offset' window appears their will be a value in the 'Distance to origin' dialogue. This is the distance the machine has to travel to reach the billet, relative to Tool 1. If the machine has to travel further it will be a negative number, if it has to travel less it will be a positive number.

In this example the value is 39.84, this means the tool doesn't have to travel as far to reach the billet as Tool 1.



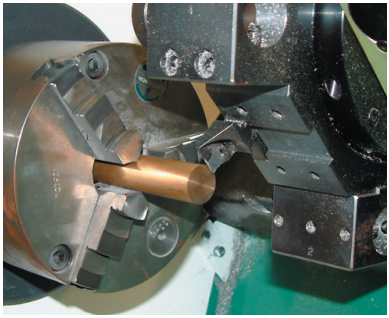

Tool length Offset for Tool 2
= Z Offset Tool 1 - Z Offset Tool 2
eg: 100mm - 60mm
=40mm Z Tool Length Offset

# Setting Tool Length Offsets for Parting Off and Threading Tools

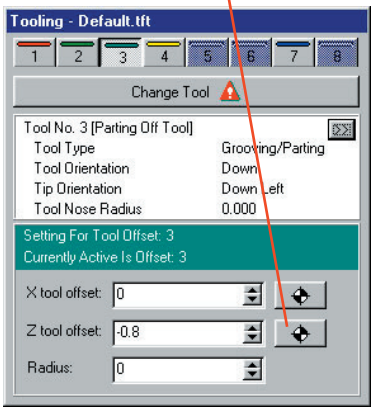
Change tools to the Parting Off Tool as described in 'Step 8' of this guide. In this example we have the Parting Off Tool set as Tool 3.

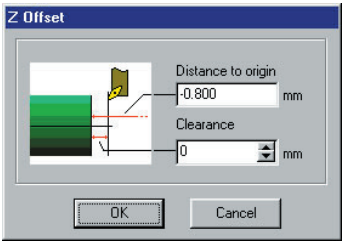
## Set the Z axis Tool Offset for the Parting Off Tool

Jog the Parting Off Tool and touch the end of the billet. Be very careful with the Parting Off Tool as it is more fragile than the Roughing Tool.



Click the Datum Button in the 'Tooling' Window and set the 'Z tool offset'.



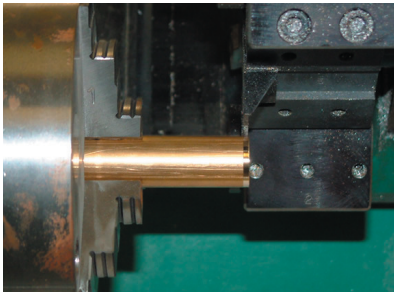

Click [OK] to accept the value

## Set the X axis Tool Offset for the Parting Off Tool

Start the Spindle at approx 1000 RPM and Jog near to the diameter of the billet. Using the incremental Jog facility, touch the workpiece and remove a small amount of material from the diameter. Be careful when using the Parting Off tool not to hit the billet too quickly or at too large an increment.

Measure the diameter where the material has been removed.

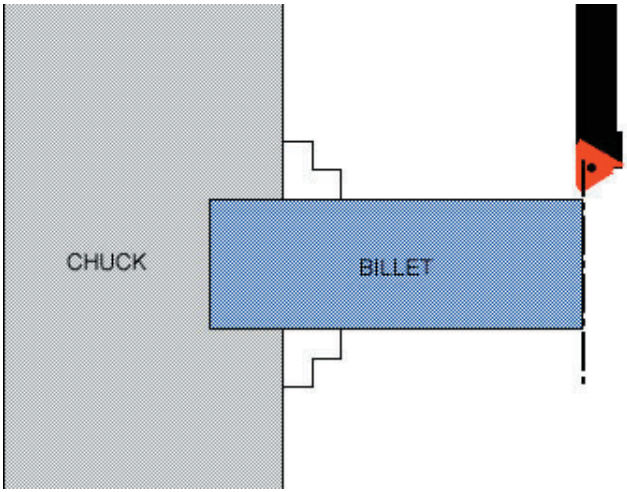
Set the offset as before inside the 'Tooling' window, not the 'Offsets' window.



Change tools to the Threading Tool as described in 'Step 8' of this guide. In this example we have the Threading Tool set as Tool 7.

## Set the Z axis Tool Offset for the Threading Tool

Setting the Z axis Offset for the Threading Tool is slightly different than for the other tools because you cannot touch the end of the billet with the threading tool. Instead we have to line up the tip of the Threading Tool with the end of the billet. (Doing this by eye is usually adequate)




NOTE: It is better to have the tool tip away from the end of the billet, rather than to far towards the chuck.

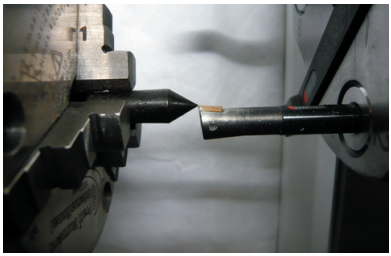
## Set the X axis Tool Offset for the Threading Tool

Start the Spindle at approx 1000 RPM and Jog near to the diameter of the billet. Using the incremental Jog facility, touch the workpiece and touch the diameter.

# Setting Tool Length Offsets for Boring Bar

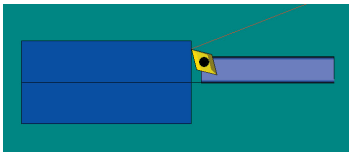
Change tools to the Boring Bar Tool as described in 'Step 8' of this guide.

When setting the boring bar for the first time, you may need to check the centre height. To do this, place a pointed object in the chuck (even a sharpened pencil will do). losen the grub screws holding the boring bar, rotate the boring bar within it's sleve until both tips meet and re-tighten the grub screws agin.

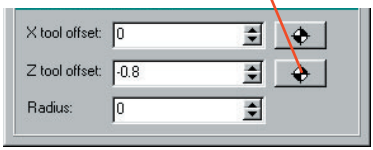


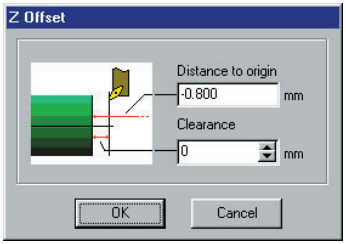
# Set the Z axis Tool Offset for the Boring Bar Tool

Jog the Boring Bar Tool and touch the end of the billet.



Click the Datum Button in the 'Tooling' Window and set the 'Z tool offset'.



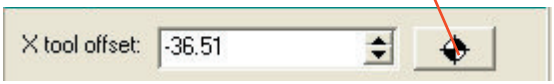



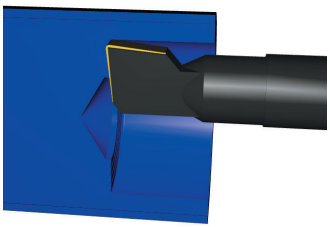

Click [OK] to accept the value

# Set the X axis Tool Offset for the Boring Bar Tool

To do this we need to touch on an internal surface, so you will need a peice of tubing or bar that is already drilled or bored out with minimum hole diameter 11mm, enough for the bar to fit inside. Start the Spindle at approx 1000 RPM and Jog near to the diameter of the billet. Using the incremental Jog facility, touch the workpiece and remove a small amount of material from the inside diameter.

Measure the inside diameter where the material has been removed and click here to set the X datum as before.

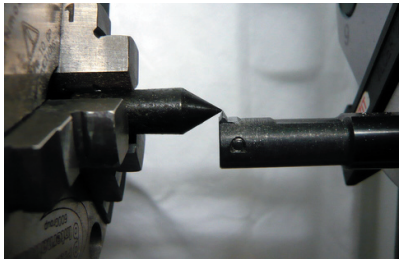


# Setting Tool Length Offsets for Internal Threading

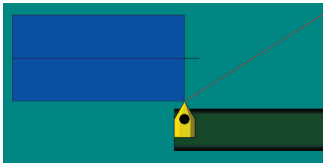
Change tools to the Internal Threading Tool as described in 'Step 8' of this guide.

When setting the internal threading tool for the first time, you may need to check the centre height. To do this, place a pointed object in the chuck (even a sharpened pencil will do). losen the grub screws holding the boring bar, rotate the internal threading bar within it's sleve until both tips meet and re-tighten the grub screws agin.

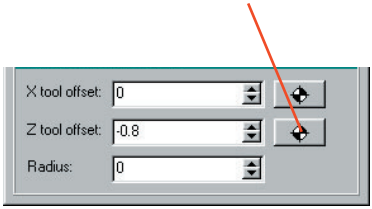


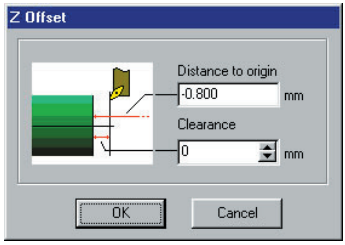
# Set the Z axis Tool Offset for Internal Threading

Jog the internal threading Tool and align the tip to the edge of the billet.



Click the Datum Button in the 'Tooling' Window and set the 'Z tool offset'.



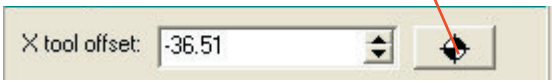



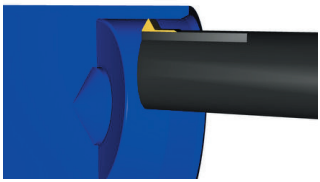

Click [OK] to accept the value

# Set the X axis Tool Offset for Internal Threading

To do this we need to touch on an internal surface, so you will need a peice of tubing or bar that is already drilled or bored out with minimum hole diameter 11mm, enough for the bar to fit inside. Start the Spindle at approx 1000 RPM and Jog near to the diameter of the billet. Using the incremental Jog facility, touch the workpiece and remove a small amount of material from the inside diameter.

Measure the inside diameter where the material has been removed and click here to set the X datum as before.



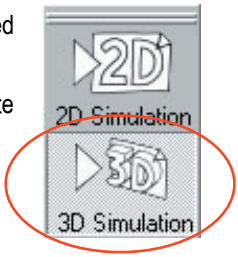



# Step 11 - Run a simulation.

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.

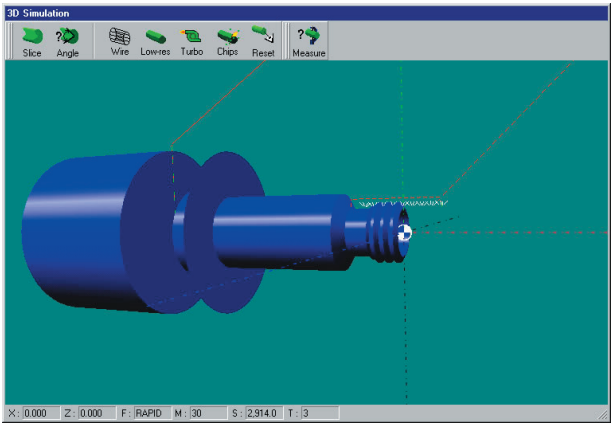
The simulation uses the information in the tooling window to generate the graphics. It is therefore important to have the tools set correctly in the tooling window to generate an accurate simulation.



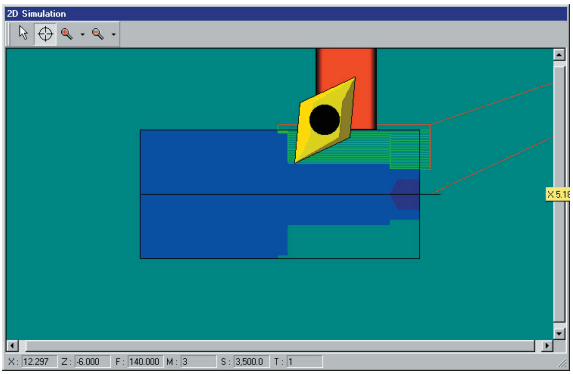

Click on the 3D simulation button.

The 3D simulation window will appear.

Using the 'File Control', press Stop, Rewind and then Play to run the simulation.

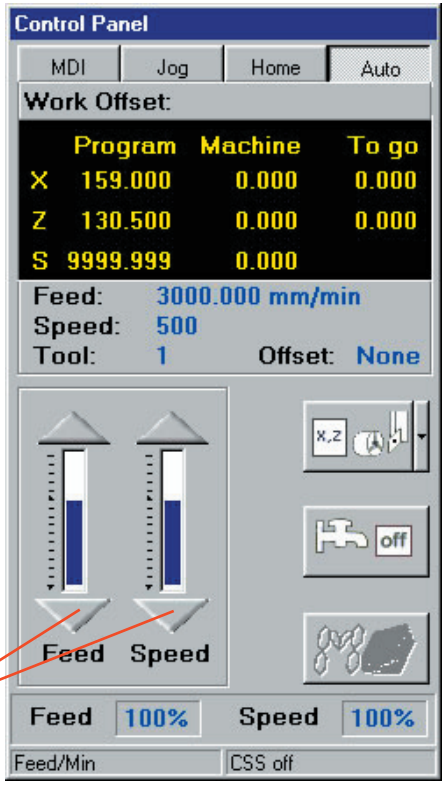



Alternatively a 2D simulation can be run.



# Step 12 - Run the program.

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 here.

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