

INSTALLATION

Insert the QuickTURN software CD into the PC.

The installation menu should automatically start – if it doesn't, double-click the file **Start.exe**. Click **Install QuickTURN 2D Design** from the menu and follow the installation instructions.



When installation is complete, you will be asked to insert your software license media. This will normally be provided on floppy disk or USB memory key. Click the removable media option and browse for the file **seckey.tff**

NB Keep the license disk safe. We charge for replacement license files. However, the software CD can be downloaded free of charge from www.denford.co.uk



RUNNING THE SOFTWARE & CREATING A TURNED PROFILE

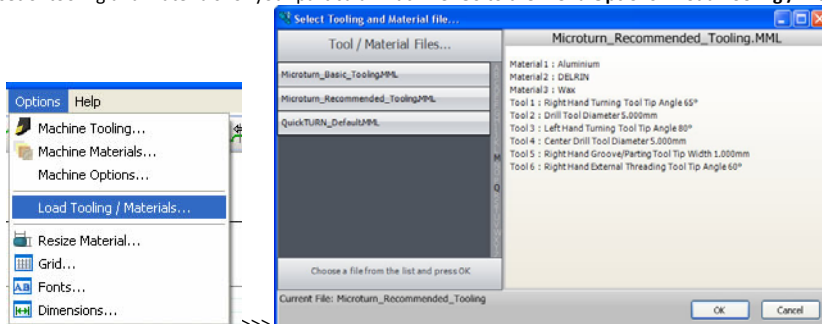
From the All Programs menu, locate the Denford software folder and the QuickTURN 2D Design shortcut :



The first thing the software needs to know is the size of the round material you will be using. Enter something like 25mm diameter bar, 50mm long and hit **OK**




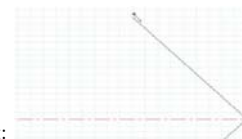
The default tool and material settings are for a Denford back turning lathe with Automatic turret. If your machine is different, you will need to choose the correct set of tooling and materials for your particular machine. Go to the menu **Options > Load Tooling / Materials**




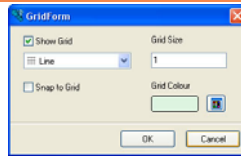
Tip: If you wish to add your own Tooling and Material settings file for different machines, then just copy and rename an existing .mml file in the directory:

C:\Documents and Settings\All Users\Application Data\Denford\Machines the new file will now appear in the selection. Once the settings file is chosen, any further settings changes will be saved to that same file.

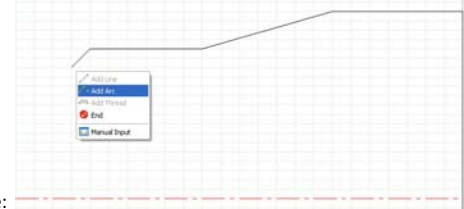
To start creation of turned profile, hit the line tool button (or press **L**)  and a line will appear from the centre point:



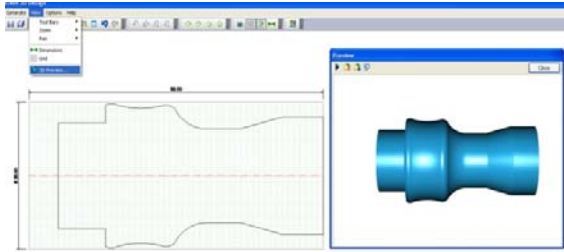
Drawing arcs and lines will snap to the grid by default. Turn this on and off by clicking the snap tool button 



Modify the grid settings from the Menu: **Options > Grid...**



Build up a 2D profile by adding lines, right mouse click to activate the popup menu to switch to arc mode:
(Alternatively press the **A** key)



Here the profile has been finished and a 3D preview of the part shown by selecting
Menu: **View > 3D Preview...**

Now would be a good time to save your profile using either Menu: **File > Save As...** or the tool button:



TOOLING AND MATERIAL SETTINGS

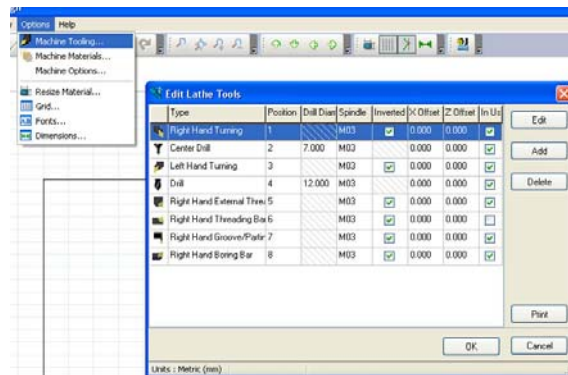
It is very important to set up the tool types, tip geometries and material settings you intend to use.

Tool geometry alters how tool paths are calculated for a number of reasons:

- Tool nose radius compensation is automatically applied when calculations are being made
- The clearance angle of turning/boring tool tips will affect which parts of the profile can be reached
- The width of grooving tools needs to be defined in order to create an accurately sized part
- Drill diameters and lengths must be defined accurately, as the software will not create internal boring passes if the drill is not long or wide enough
- The toolpath 3D simulation also needs accurate information to simulate cuts accurately

Material settings will determine:

- How much material is removed in each roughing pass
- How much material to leave on for finishing passes
- The axes feed rate and spindle speed required for roughing and finishing passes



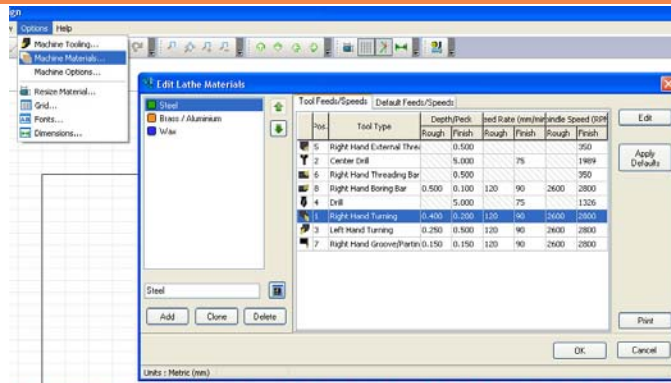
Choose the menu: **Options > Machine Tooling...**

here you can add, remove or disable tools.

Disable tools simply by un-checking them in the **In Use** column. Edit tools by double-clicking them in the list, or selecting and pressing **Edit**.

Tool position will determine the tools number in the automatic toolchanger (turret). Although the software aims to hold tool geometry details for the default tooling supplied by Denford Ltd, you may need to refer to your tool's data sheet to make sure the geometry is correct (in particular the tool tip radius)

Once you are happy the tooling is set up correctly, the material settings need to be checked:



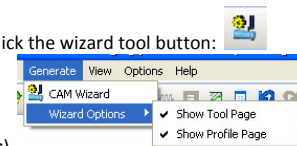
Choose the menu: **Options > Machine Materials...**

Choose one of the default materials which is closest to the material you wish to cut (Steel=Hard; Brass/Aluminium=Soft; Wax=Very soft)

Select each of the tools you wish to use in turn and click **Edit**, confirming the cutting depths, feeds and speeds look correct. As with tool geometry, it may be worth consulting with the tooling manufacturer to get the best cutting parameters for your particular tool and material types.

RUNNING THE CAM-CNC WIZARD

Once the turned profile has been defined, start the CAM wizard in order to produce a CNC G-Code file. Click the wizard tool button:

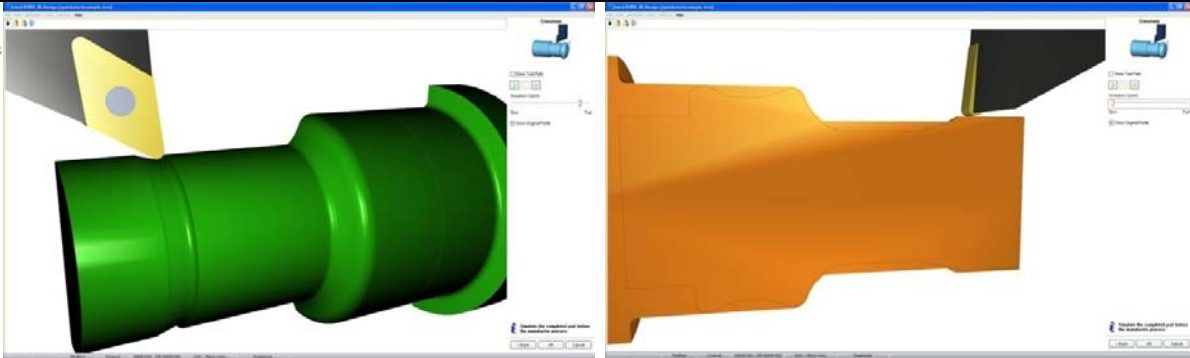


(Some Wizard pages can be turned off to keep things simpler with the menu: **Generate > Wizard Options**)




The following table shows each step of the CAM Wizard, with a brief description:

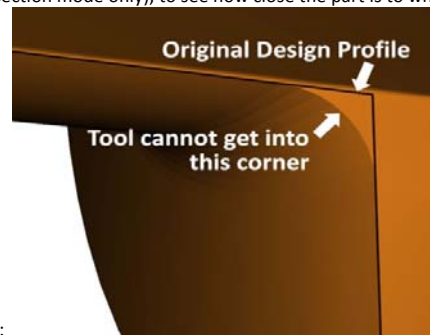
<p>1</p> <p>Pick the material settings you wish to use.</p>	<p>2</p> <p>Now you are able to define your stock material size - especially useful for creating extra clearance passes to remove stock. By default, it will be sized the same as the design.</p>	<p>3</p> <p>This next stage allows you to deselect tools you do not wish to use in tool path generation. Highlight any tool and click Show to confirm the tool's geometry is correct.</p>
<p>4</p> <p>Click Generate to create toolpaths for all available tools. Some tools may not be utilised -depending upon the profile shape, whether internal cuts are required or whether pre-requisite tools are available. For example, internal boring would not be generated if a suitable drill was not available. A forward roughing tool is essential for most jobs. Some finishing passes may be created which are duplicated by different tools, but these may be turned off later.</p>	<p>5</p> <p>At this stage you are able to see how each tool's operation is broken down into separate profiles. All profiles can be disabled. However, if you disable any roughing passes, then consequent operations will also be disabled.</p> <p>Click the section tool button at the top left in order to see the toolpaths more clearly. Clicking on profiles in the list will highlight them in red.</p>	<p>6</p> <p>Now click browse to choose where to post the CNC file to. Then click Post Process to create the G Code file.</p>

7



Left mouse click and hold on the 3D viewer to rotate the view. Hold right mouse button and move to zoom in and out. Hold both mouse buttons and move to pan left/right up/down. **Double-click on the viewer to reset to default view** (just in case you lose the part).

The final stage is to run a 3D simulation of the CNC file that has just been created. Use the Play, Stop and Rewind buttons    to start, stop and reset the simulation. You can choose to see the original design profile (visible in section mode only), to see how close the part is to what you designed – here you can see



that the tool is unable to fit into a sharp corner because of its tip radius: