

## Profiles for CNC machining

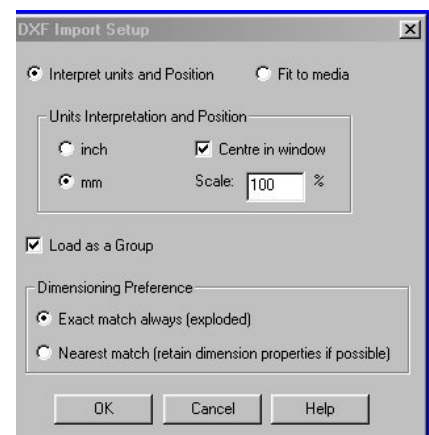
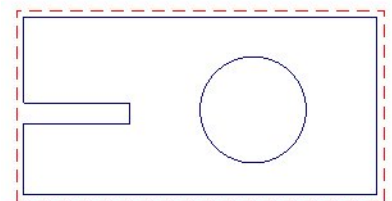
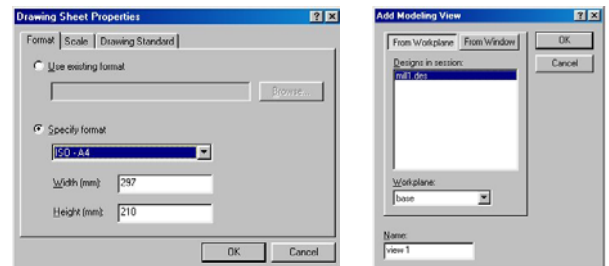
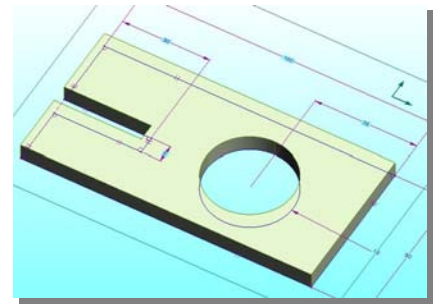
These instructions show you how to take a **profile** drawn in Pro/DESKTOP and convert your design into a file suitable for milling out from Techsoft 2D Design tools.

1. Draw your shape in Pro/DESKTOP **Design**. (Remember that the maximum size of your profile will depend upon your CNC machine. Find out your maximum machining sizes before starting your design.)
2. Save the design. **File, Save**.
3. **File, New Drawing**. A properties box lets you choose the paper size, this has to be bigger than the profile to be machined. **Select A4 or A3. Remember your max machining size**
4. Select the drop down menu **Drawing, Add modelling view**.
5. In the modelling view properties box select **From Workplane**. **OK**.
6. This brings your profile into the drawing area, you now need to convert the drawing file into a language for 2D design to understand. The file format is **DXF** this is a common drawing format.
7. **File Export, DXF** file.
8. A properties box come up in which you select, **DXF, OK**. The file will be saved in the directory where you saved your original Pro/DESKTOP file.

### Close Pro/DESKTOP and open Techsoft 2D

1. **File, Import file**
2. In the properties box change the **Files of type** to **DXF**. Select the location of your file.
3. Check that the "**DXF Import Setup**" properties box has the following checked:
  - Interpret units and Position
  - Centre in window
  - mm
  - Load as a group
  - Exact match always

**NOTE** DXF (Direct exchange file) conversions are not totally reliable, some files that have used spline or ellipse will not always convert .

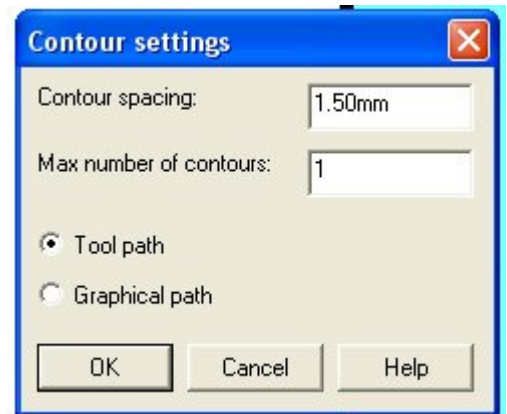


# Profiles for CNC machining

## CNC Machining



1. You can now position your profile on the page.
2. Select the contour icon. This produces a radius around sharp corners; the line drawn represents the centre of the cutting tool.
3. In the properties box set the contour spacing. The size of the spacing will depend upon the size of the cutter you are going to use. If you are using a 3mm diameter cutter you will need to set the contour spacing to 1.5 mm. This draws a toolpath line which is the centre line of the cutting path.
4. Change the contour line colour to red. This indicates the line to be cut when you set up your CNC machine.



The Techsoft drawing can now be machined directly on your Suregrave® machine. Ask your teacher for help.

## Setting tool paths

In Techsoft 2D you can set the tool paths to be on the **inside** or the **outside** of the profile. This is important when making two components fit perfectly, as in a Jig saw on the right where **A** and **B** have to be tight interference fits.

**Fig 1** shows the tool path cutting on the **INSIDE**, this will produce component **A**

**Fig 2** shows the tool path cutting on the **OUTSIDE**, this will produce component **B**

