Running HOMAG & Weeke CNC Routers

The Homag and Weeke CNC Routers are controlled using the WoodWop software program that is developed by the Homag Group in Germany.

Important!

If you are unsure about running toolpaths on your router please contact your Router supplier for detailed advice on how to run VCarve Pro toolpaths on your CNC machine.

1. Saving Toolpaths from VCarve Pro

Save each toolpath using the **2 Weeke postp's** – each with the same file name.

> **Weeke MPR**
> This saves the Job information – Material size and Thickness etc.
> Save this file into the folder named,
>
> C:\ww4\a1**MP4**
>
> **Weeke PLY**
> This saves the 3D Toolpath moves X, Y & Z coordinates
> Save this file into the folder named,
>
> C:\ww4\a1**ML4**

2. Opening the toolpath files in WoodWop
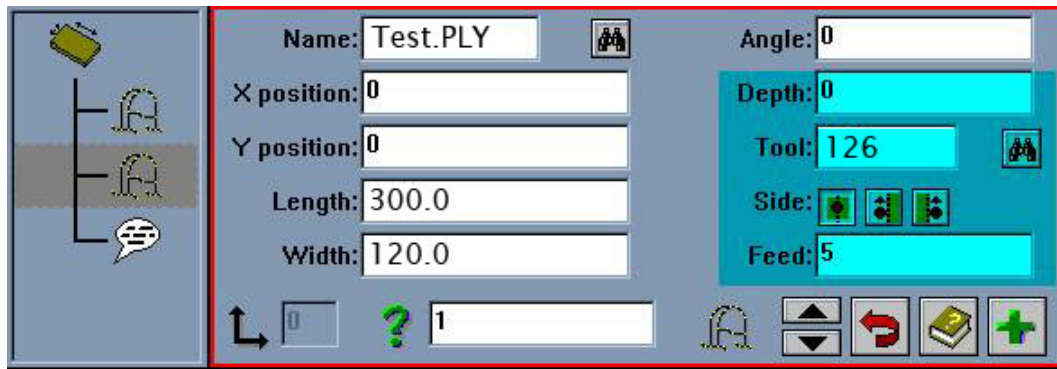
This is a 2-stage process.

> Start the WoodWOP control program and select,
>
> > **File > Open** (Select the .MPR file)
>
> A Macro for each toolpath will appear in the macro list.

Important Check that the correct tool number and associated cutting parameters are correct in each macro before generating the CN-code.

> The associated .PLY file should be loaded and displayed as shown below.

For more information about the WoodWop software please contact your supplier or visit the HOMAG web site