



MERCHANT DICE LTD.



CNC Machine Guide

- Software

The Process (3 Separate Programs)

Drawing → Converting → Running

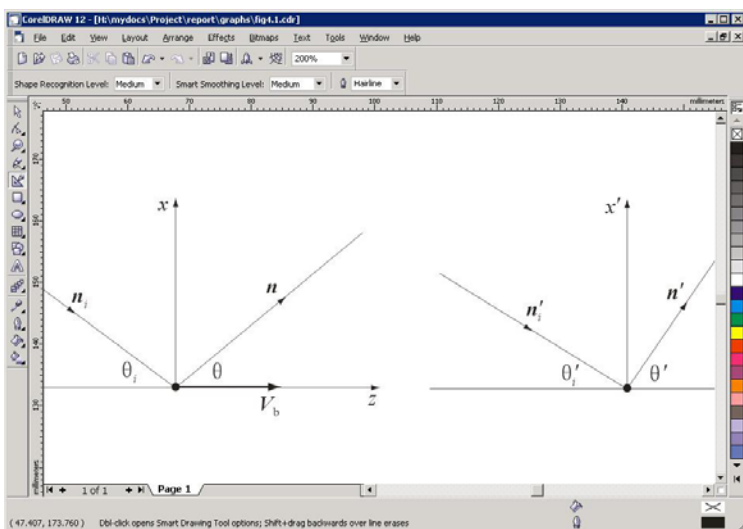
Software we use:

CorelDRAW → SheetCAM → Mach3

Stage 1 - Drawing

The first stage is to produce a drawing of the required work. There are many programs this can be achieved, most common are CorelDRAW and AutoCAD however any drawing package can be used as long as it can produce a file in one of the following file formats (for use in SheetCAM):

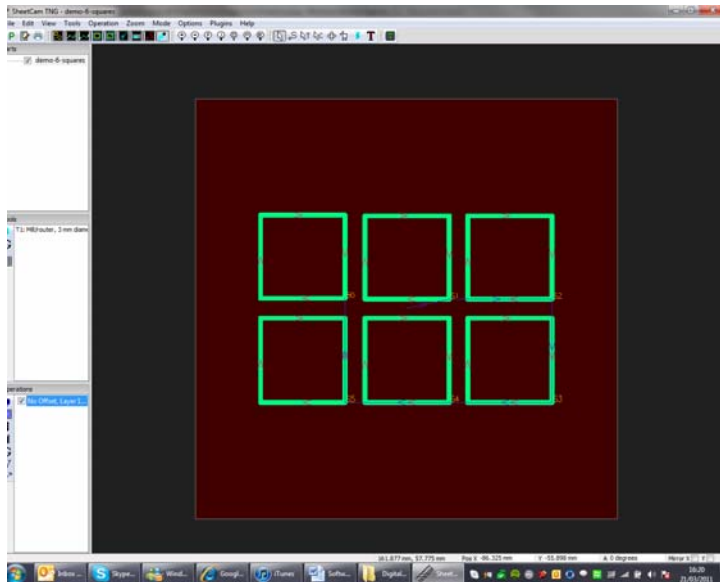
- .DXF (Most common)
- .SVG
- .HPG



(CorelDRAW Screen)

Stage 2 – Converting the Drawing into G-code

The second stage is to convert the drawing into G-code that the machine software can understand and can run. This process can be completed using a number of programs including SheetCAM, LazyCAM, Vectric Aspire and VCarve Pro. Our recommended piece of software is SheetCAM. During this process you will import the drawing and enter the variables about your piece of work which includes details about the cutter you will be using as well as dimensions of your material. You will also create the toolpaths that the machine will follow to complete your project. You will then run the inbuilt Mach3 post processor which will convert all of the above settings, information and toolpaths into G-code specifically for Mach3 (file output .tap or .txt).

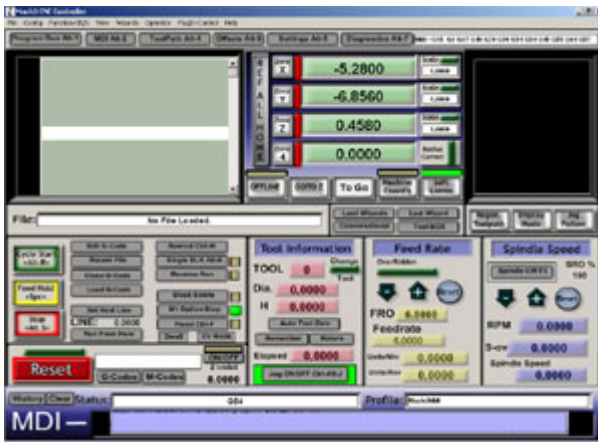


(SheetCAM Screen)

Stage 3 – Mach3 – Running the G-code

Mach3 is the software that directly controls the CNC machine and has the machine settings and parameters built in.

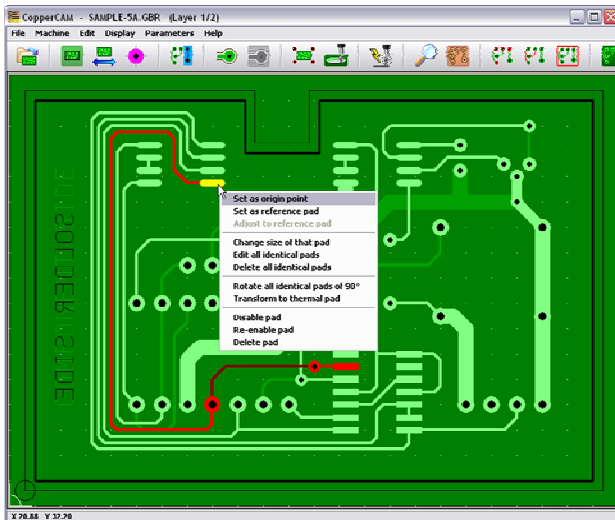
The final stage is to import and run the G-code file (usually .tap or .txt) generated by the SheetCAM Mach3 post processor.



(Mach3 Screen)

Other Software

CopperCAM has been specially created for the design and production of printed circuit boards.



(CopperCAM Screen)