## **Understanding G-Code**

Having some understanding of the G-Code that your CAM Software creates, and that your machine understands, can help the operator in day to day operation of the machine.

Below is an example of a G-Code file and a simple explanation of some of the commands.

Studying this example and familiarising yourself with the list of supported G&M Commands on the following page can give the operator a useful knowledge base to call on.

This knowledge can be particularly useful in checking settings such as plunge depths before running a job.



So, from this example we can see that the units are set to metric (G71), the machine will do a tool change (M06) and choose tool 1 (T1), the spindle will switch on (M03) and run at 18000RPM (S18000).

The Machine will then execute a rapid move (G00) to X-15.000 Y1800.000 Z20.000, the machine will then execute a linear move (G01) to -10.5mm below the surface of the material at a feed rate (FZ) of 500mm/min.

The machine will then move to X-15.000 Y-50.000 Z-10.500 at a feed rate (FX) of 1500mm/min.





www.icam.co.nz

Measurements are in