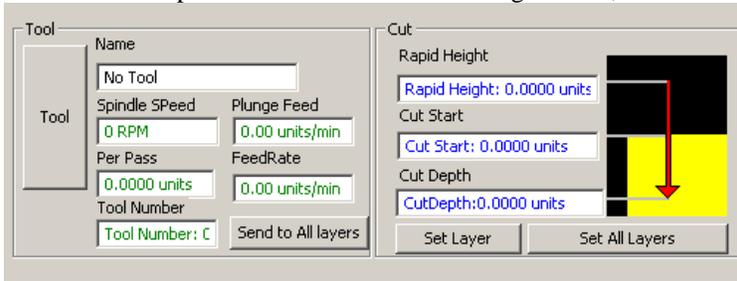
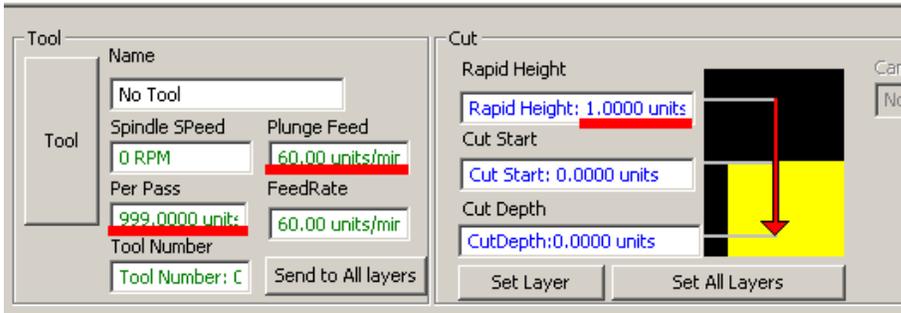


When LC first opens there are no tools or cutting defined,



When a DXF is imported LC provides a default tool and Z rapid plane to each of the created layers / chains in order to show rapid moves from one to another chain.



Notice that the default plunge feed is 60 and the per pass is basically infinity and a rapid of 1 unit is applied.

If you never did anything to the file and posted it to MACH those settings become part of the generated code.

```

N20 (Default Mill Post)
N30 G91.1
N40 G0 Z1.0000 ←
N50 M3
N60 X0.0000 Y0.0000
N70 G1 Z0.0000 F60.00 ←

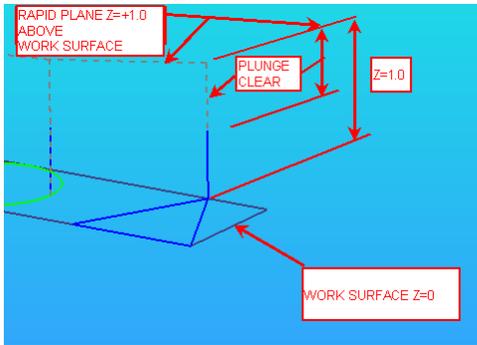
```

The code generated is just a rapid ( G00 ), moving at the max velocity setting you defined for the motors in motor tuning, from the origin to the clearance plane of Z=1.

In this case, since the origin is also at the same location of the first chain to be cut , the tool then does a ( G01 ) interpolated move at a designated feed rate ( F60 ) into the material.

One additional point is that in the setup>posting options, the value for the plunge clearance was set to a zero value. So the Z moves as if it were cutting material all the way down to the piece.

Now if you were to change the Plunge Clearance setting to say a value of ½ the default Z clearance plane value as depicted below, and import the DXF, different code would be generated.



Here is the code after posting to Mach.

```

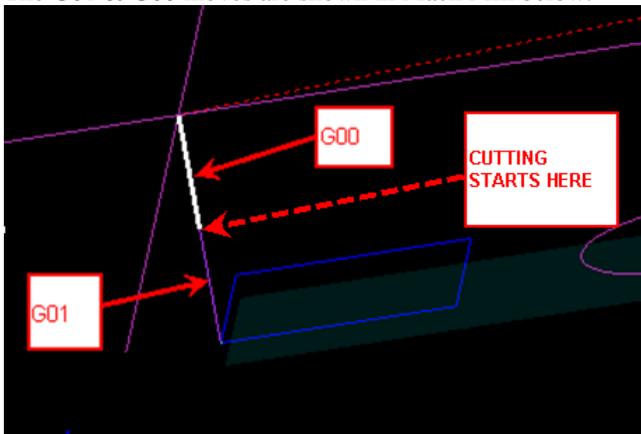
N10 (File Name = SHAPES LCMRD HCIR on Monday, Aug
N20 (Default Mill Post)
N30 G91.1
N40 G0 Z1.0000
N50 M3
N60 X0.0000 Y0.0000
N70 Z0.5000
N80 G1 Z0.0000 F60.00

```

PLUNGE / RAPID DOWN AND THEN AT A FEED RATE

The code generated is just a rapid ( G00 ) , moving at the max velocity setting you defined for the motors in motor tuning, from the origin to the clearance plane of Z=1. In this case, since the origin is also at the same location of the first chain to be cut , the tool then does a rapid ( G00 ) down to Z=+5 and then a ( G01 ) interpolated move / cutting at a designated feed rate ( F60 ) into the material.

The G01 & G00 moves are shown in Mach Mill below.



Your rapid velocity is usually much faster than when cutting. As an example, if the rapid is 100 inches / min and the cutting feed velocity is only 1 inch / min, it would take a lot of time for the Z axis moves and especially if they were long moves. The cutting starts / is done with a G01 move.

**Plunge Clearance** - The distance ABOVE the Cut starting point where Mach can safely rapid down to without hitting the workpiece. IF set to ZERO it turns OFF the function and mach feeds from the SafeZ height ( Rapid Plane ) to the cut level instead of rapids.

