

Workflow for converting files in mods and milling them on CNC

Authored by Grant and Graham

- Open Your Eagle CAD PCB board
 - Go to the layers in the left corner and hide everything except top layer (pads and traces)
 - Go to export - Export as image
 - Change settings to Monochrome and bump up resolution (500 DPI)
 - Rename your file and select a location
 - Open up <http://fabmodules.org>
 - Click the input tab and select image .PNG
 - Search your image and import the image
 - Invert the image in the menu on the right hand side
 - Select output up top and change it to Other Mills .nc (for Candle)
 - Under process select trace 1/64
 - Afterwards under right menu change tool width to .127mm
 - Finally click calculate to view toolpath
 - Click save in order to save the .nc file
 - Open candle and select your com port for CNC
 - Fasten the PCB material to the CNC
 - Home your machine so that Spindle barely touches PCB using X Y and Z axes
 - Open the desired file
 - Send the file to the CNC and watch as it mills
 - Abort if any problems and vacuum up leftover dust
-
- For profile to cut out in eagle select view and hide all except outline
 - Repeat steps 2-11 above
 - For process select PCB outline 1/32
 - Change the bit size .3mm
 - Select calculate to view cut path
 - Save the file and import into candle
 - Home the spindle having not moved PCB material
 - Send the file to be milled
 - Raise up the Z axes after finished
 - Remove the PCB