

CREATING G-CODE

- Use VCarve or Carbide Motion to create toolpaths from vectors; imported vectors made in Illustrator or Rhino or coding applications are best for complex designs
- Set depth of stock to 0.01" and Max Depth of "cut" to Stock Bottom
- Toolpaths can specify any tool or tool diameter; edit feed rate for tool to 100 inches/ minute
- Save toolpath

CNC OPERATION

- Open Carbide Motion
- Turn on Shapeoko with rocker switch
- plug into laptop (or nearby desktop by rolling CNC)
- on first use, configure machine to Shapeoko Z-belt drive
- on first use, configure model as XXL
- Home/ Initialize the spindle if it does not home automatically
- tape stock to spoilboard, making sure it is square
- Change/ insert correct drawing/ etching tool, securing with set screws using 2mm allen wrench
- Zero your X, Y, and Z dimensions
- Select Run Job
- Do not plug in or turn on router
- Start job
- Stay in the nodelab to monitor the initial operation of the job

FINISHING

- jog spindle to back of CNC and turn off when done running jobs
- remove stock from spoilboard

- Clean up after yourself, return drawing tool and other equipment to designated storage areas
- throw out non-usable scraps and stock