Steps for producing a PCB in the

CircuitCAM (big buttons work like a wizard)

- Import Gerber files into correct layer:
  - TopLayer (gtl), BottomLayer (gbl), BoardOutline (gm1), DrillPlated (txt)
- Contour Routing
  - Select automatic break-outs to avoid next step
- Define Breakout Tabs
- Define Rubout Areas
  - Small area around all solder pads ONLY!
- Insulate all Layers
  - Calculates machine paths
- Export Files
  - LMD & CAM

Initialize System

- Load LMD file onto host PC
- Turn power on S100
- Start BoardMaster
  - Check that no tool is in holder before beginning homing procedure.
  - Make sure vacuum system is in AUTO mode.
- Do a scratch test
  - Trace should be 8mil wide (use microscope)

Configure Job

- Open a new job file
  - 1 or 2 Sided + ProConduct (2x sided board)
  - 1 or 2 Sided (single sided board)
- Define usable work area on stage
  - Bottom Left & Top Right
  - Avoid collisions with alignment pins
  - Avoid overlapping used up areas on blank
- Import LMD File
  - Move / rotate to desired location
- Import Feducial.lmd (located on desktop)
  - Duplicate into 2x2 grid
  - Move feducials so that they surround board (best accuracy)
- Save job file

Execute Job

- Load material
- Select Phase
  - Marking Drill is usually first one
- Select all
- Start
  - Press a number of times
- When phase ends, select next phase and repeat until done.

Clean Up

- Move machine to Park position
- Vacuum metal scrap from table
- Shut down BoardMaster software
  - Machine will retract table
- Turn power off on machine
- Vacuum metal scrap now that table is out of the way
- Close the lid