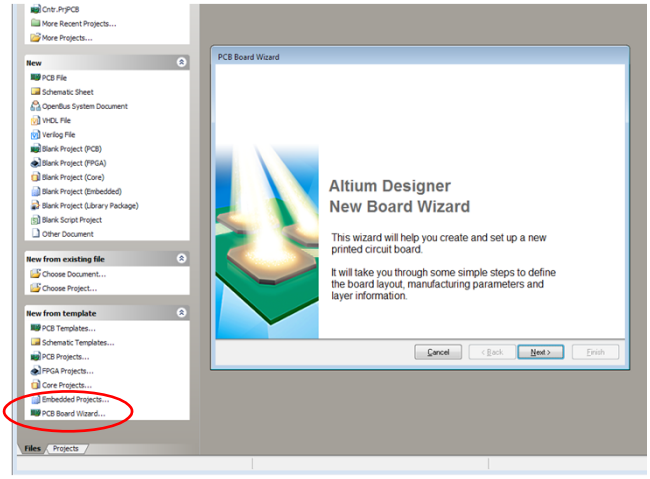
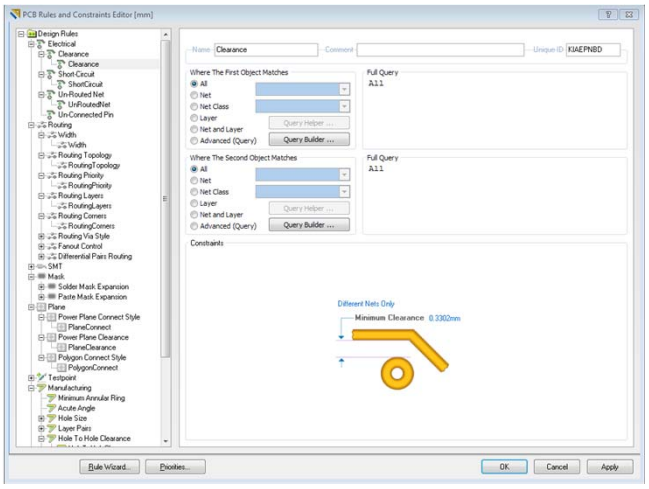
Add a PCB using Wizard



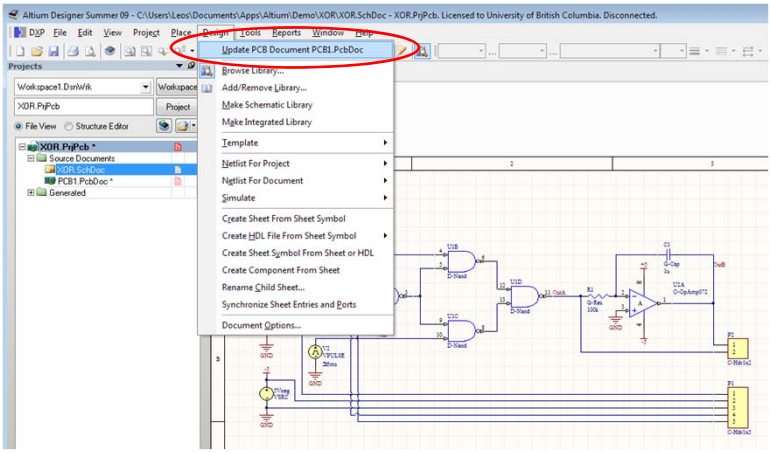
PCB Specifications

- Metric
- Custom Size
  - Be generous, it is easy to make it smaller later
- Dimension Layer = Mechanical Layer 1
  - Uncheck all boxes (title / legend / dim lines)
- 2 Signal Layers / 0 Power Planes
- Through-hole vias & components
- Auto-Routing Specifications
  - One Track between adjacent pads
  - Min Track Size / Clearance = 0.2mm (or greater)
- Make sure drill available for via hole
  - 0.8 mm is ok
  - Refer to LPKF web page
- Move new PCB document into Project Source Documents & SAVE Project

To Modify: Design / Rules …



Add Components to PCB



Route PCB

- Change Room definition (burgundy area)
- Drag components
  - Space bar to rotate
- Auto Route / All …
- Select layer and modify nets
  - Highlight layer you are working on or …
  - Hide layer you are not working on
- Interactive Routing

Make Changes to Schematic

- Schematic
  - New components
  - Change footprints
  - Design / Update PCB Document …
- PCB
  - Tools / Unroute / All
  - Auto Route / All …

Modify Board Shape

- Select Mechanical Layer 1
- Erase & Redraw Desired Board Shape
  - Place / Line
- Redefine board
  - Edit / Select / All on Layer
  - Design / Board Shape / Define from selected objects
  - Adjust “Keep Out” layer

Save Gerber Files

- File / Fabrication Outputs / Gerber Files
  - General : Inches, 2:4
  - Layers : GTL, GBL, GM1
  - Apertures: Embed apertures (RS274X)
  - Gerber Files in CAMtastic! Documents directory
- File / Fabrication Outputs / NC Drill Files
  - Inches, 2:3
  - Generate separate NC Drill files for plated & non-plated holes
  - De-select “Generate Board Edge Rout Paths”
- CAMtastic###.cam
  - File / Export / Gerber …

PCB Production Flowchart

- Layout circuit
- Simulate circuit
- Correct Output? **No**
- Build circuit on breadboard
- Correct Output? **No**
- Layout PCB
- Submit drawing for review
- Submit Gerber files for production