Ordering PCBs from JLCPCB using Altium Designer 20.1.14

Instructions

I. Manufacturing
   A. Gerber Files
   B. NC Drill Files

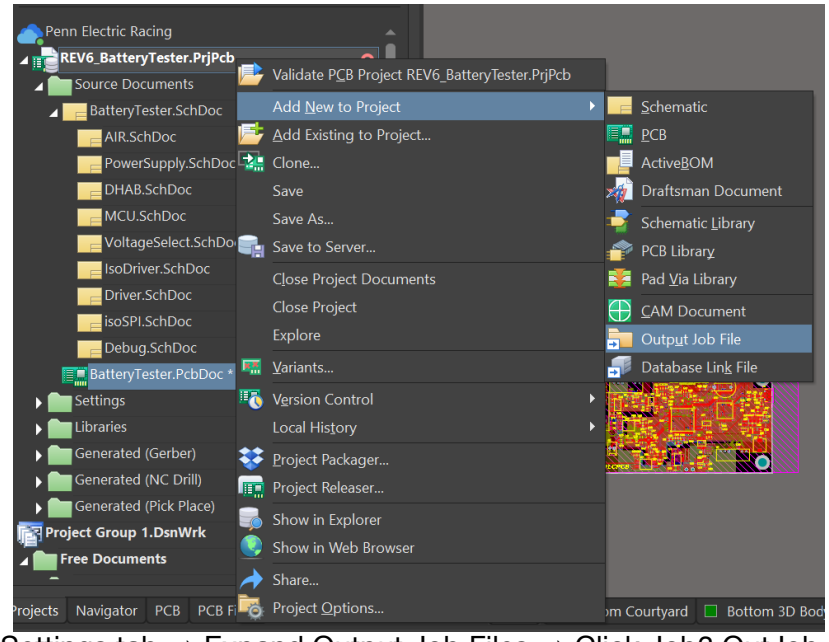
II. Assembly
   A. Pick and Place Files
   B. Bill of Materials Files

Manufacturing Outputs

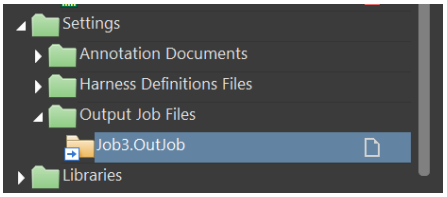
Double check that you have added a board outline to your project

- Go to your PCB -> Panels -> View Configuration -> Right click -> Add Mechanical Layer
  - Layer Name: Mechanical 32; Layer Number 32
- Design -> Board Shape -> Create Primitives from Board Shape
  - Width: 10mil; Layer: Mechanical 32

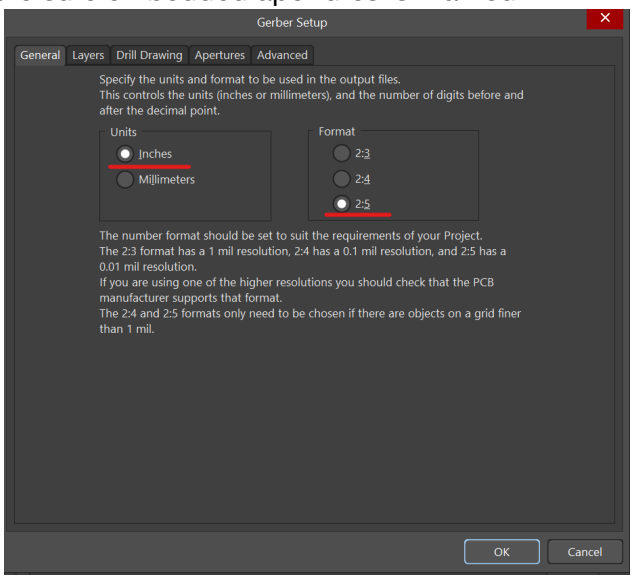
1. Add output job to your project
   1. Right click your project → Add New to Project → Output Job File



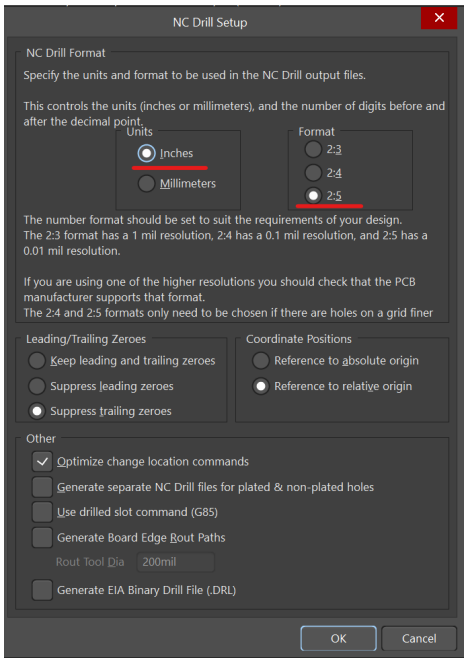
2. Expand the Settings tab → Expand Output Job Files → Click Job3.OutJob



3. Under Fabrication Outputs → Add New → **Gerber Files** → [PCB Document]
    1. Layers Tab → Plot Layers → Used On
    2. Apertures → Make sure embedded apertures is marked



4. Under Fabrication Outputs → Add New → NC Drill Files → [PCB Document]



5. Click New Folder Structure under Output Containers, then enable your Gerber Files and NC Drill Files by hitting the circle under the enabled column (should then display 1, 2)
6. Under Output Containers → Generate content
7. Go to C drive > Users > Public > Public Documents > Altium > Project > Project Outputs
    1. Copy all of the gerber and NC drill files into a folder
    2. Zip folder

Assembly Outputs

1. Under your output job, head to Assembly Outputs
    1. Add New Assembly Output → Generates pick and place files
    2. Units: Metric
    3. ***UNCHECK*** “Show Units”
    4. Separator: .
    5. Formats: CSV
    6. Add to Output Container
2. Under Report Outputs, Bill of Materials → [Project]
    1. Double click
    2. Under Properties → Columns → Sources → Check the PCB Parameters button
    3. Under Columns → unhide the Layer Column
    4. Sort by Layer by hitting the Layer column *label*
    5. Export, source parts for Bottom or Top Layer in new column JLC part numbers

Order Boards!