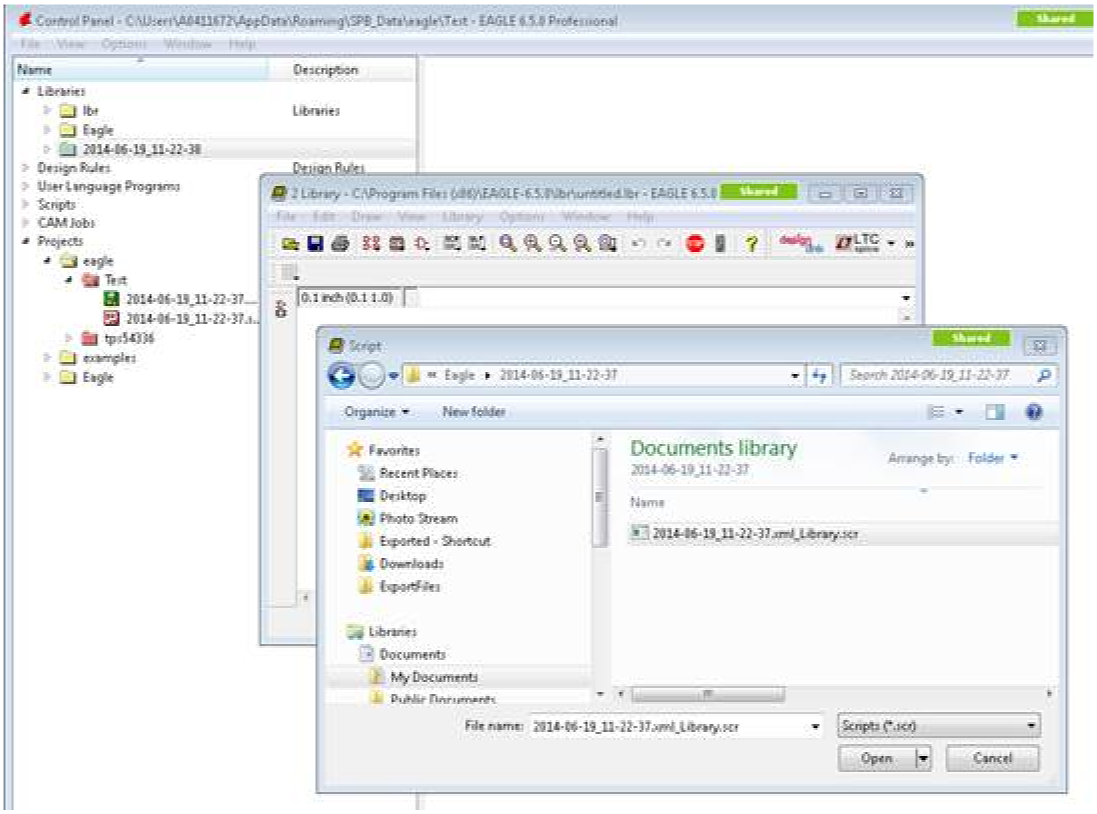
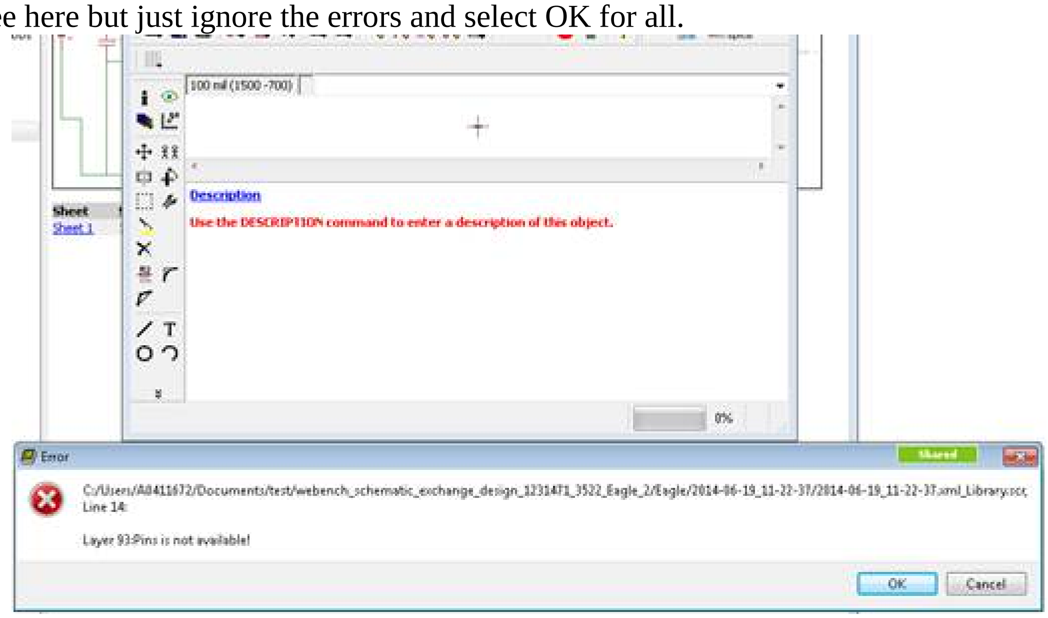
Below are the steps to integrating the Eagle library for Schematic and convert it to board netlist:

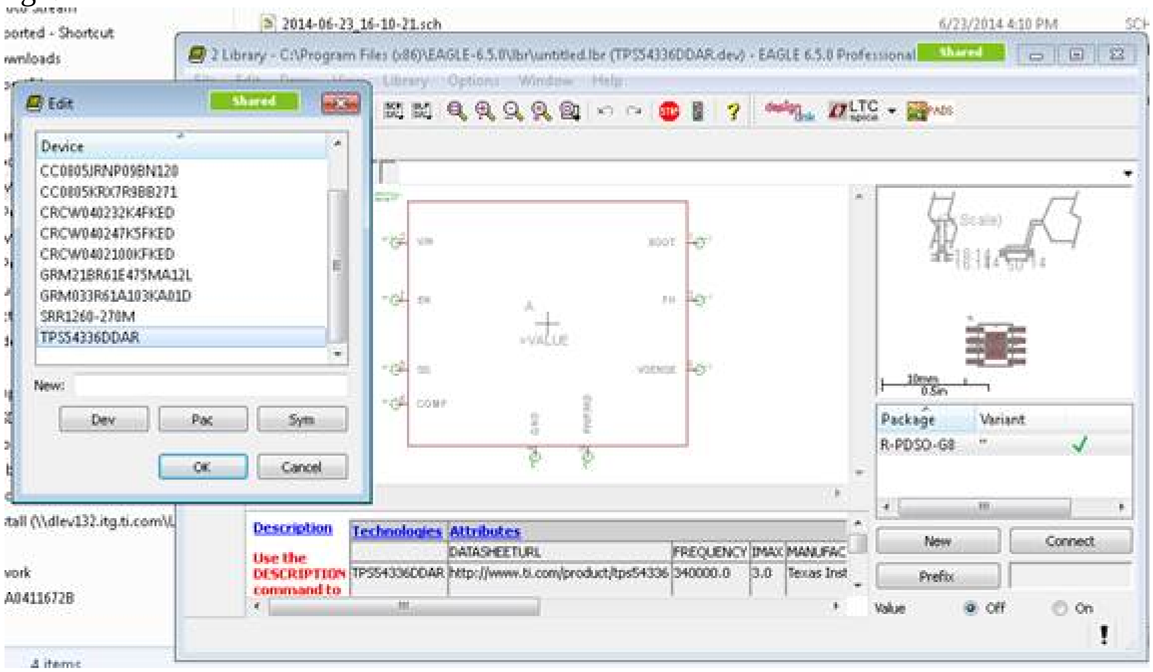
1. Open a new library window and go to File and select Execute Script. Select the downloaded library.scr file.



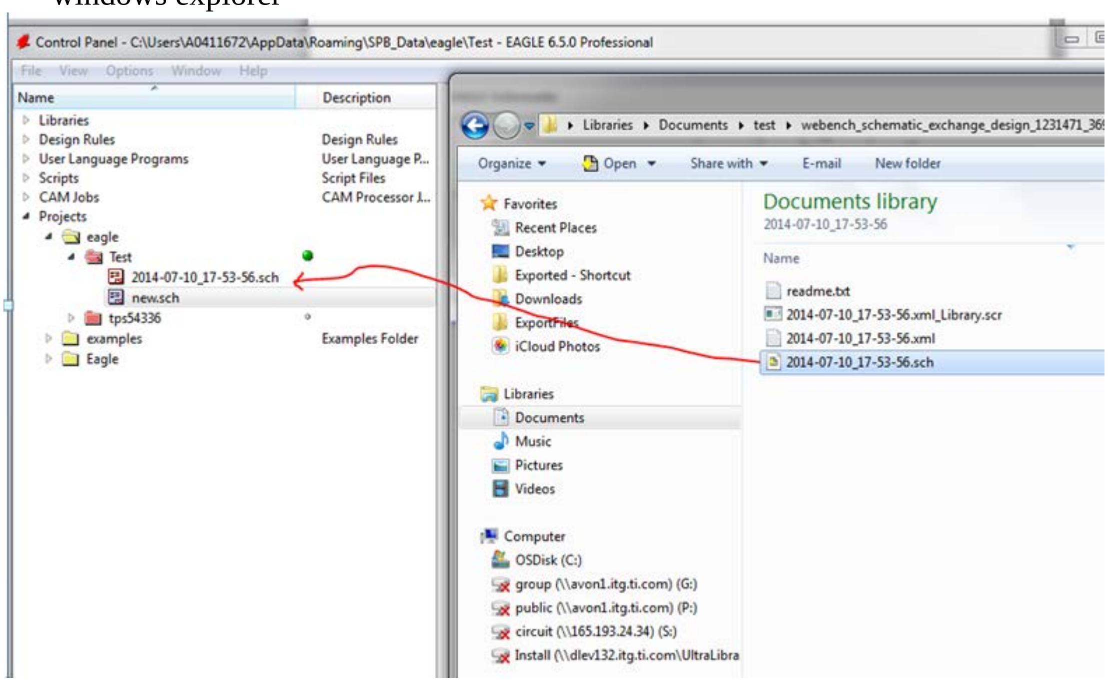
2. Yes, at this point, the library conversion starts and there will be some errors as you can see here but just ignore the errors and select OK for all.



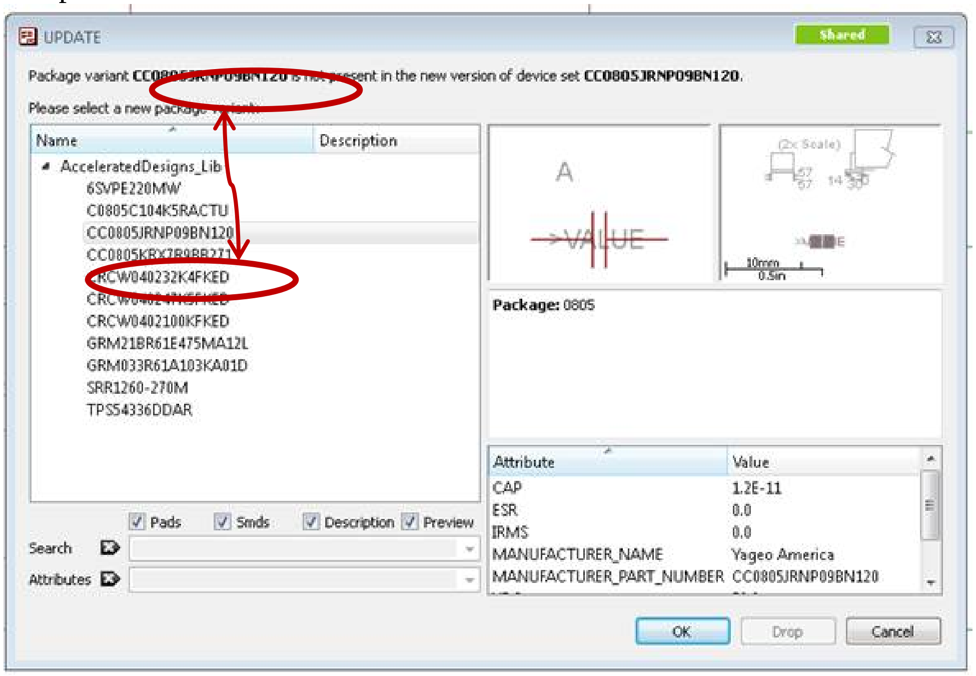
3. Once the library conversion is completed, please save the library as "AcceleratedDesigns_Lib.lbr" so the schematic can link to the default name of the library into all Eagle library directory. Here is a sample of the imported components from the design.



4. Create a project folder "test" and drag the schematic file into the project folder using windows explorer



5. Go to the schematic and select Library and select Update; find the newly imported AcceleratedDesigns_Lib.lbr. It will ask you update each component in the schematic (Note: Match the part # as shown above the library list) and make sure it's done for all components.



6. Save the schematic, select **File → Switch to Board**. It should generate the pcb board with the component netlist (ratsnest) with footprints showing for all the components.