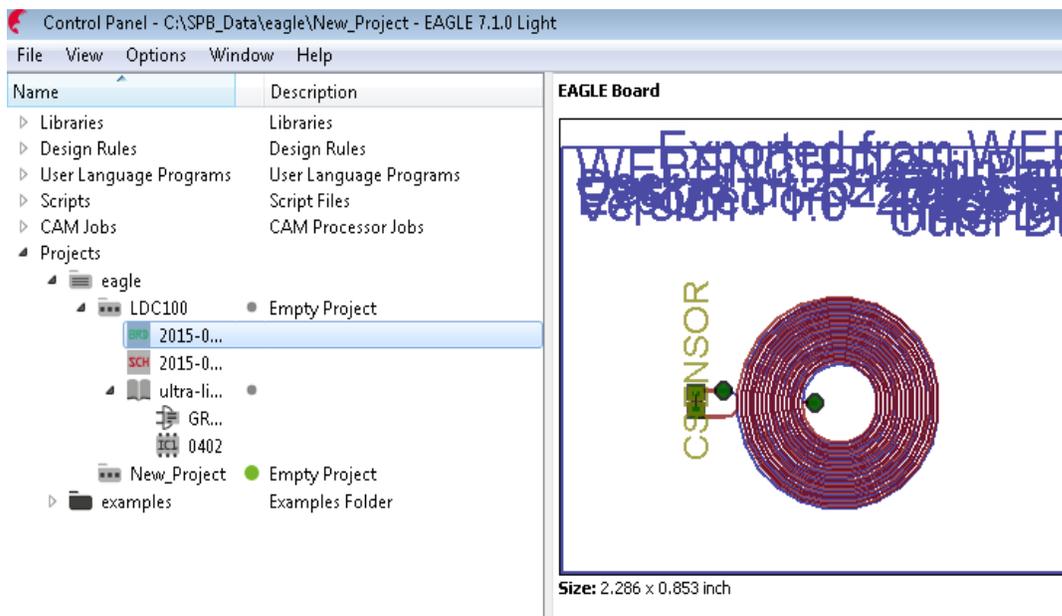
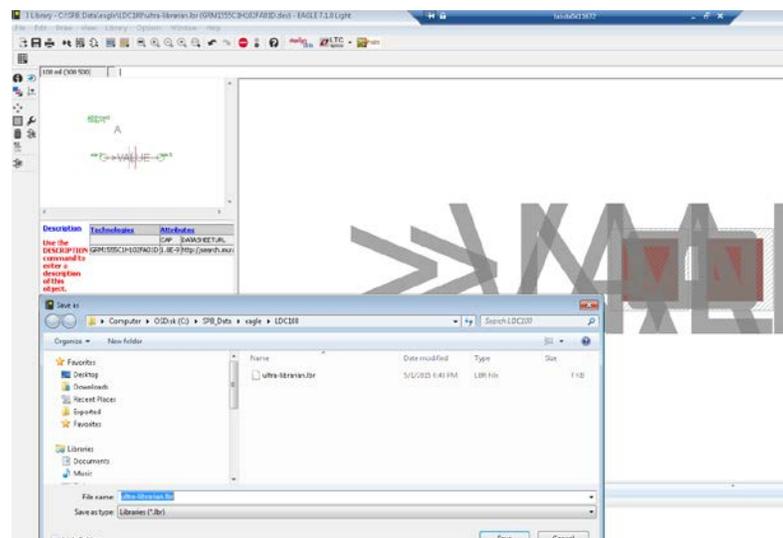


## LDC1000 Eagle - Basic Instructions to link the Schematic to Board

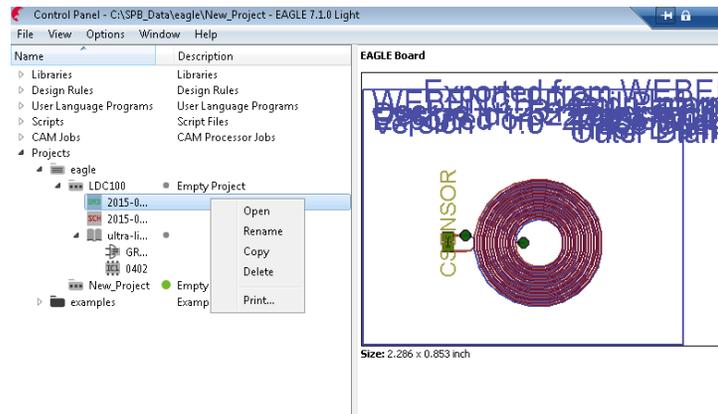
- Unzipped the download file and open the .brd file in Eagle.
- In Eagle Control Panel window, create a new project folder and Save.
  - Use Windows Explore to drag the .brd file into the newly created project folder.



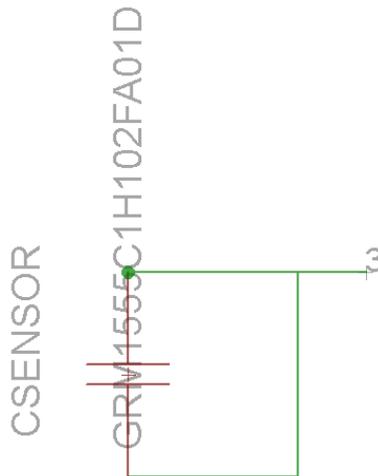
- In the same Control Panel window, go select File → New → Library
  - A new blank library window will open; Go to File and select Execute Script
  - Load the library file which is in the unzipped folder.
  - After the file is loaded; Save the file as “ultra-librarian.lbr” in project folder as shown below:



- Go back to the Control Panel window, select the board file and Open it.



- 
- In the opened Board window; go to File → Switch to schematic
  - A new blank schematic window will open.
- In the blank schematic, place new component which in this case is a cap from the ultra-librarian.lbr and rename it as Csensor. Place a wire connects the two ends of the cap which represents the coil and assign a new net. See below.



Now, the schematic BOM is linked with the PCB and they can be integrated into a larger project.