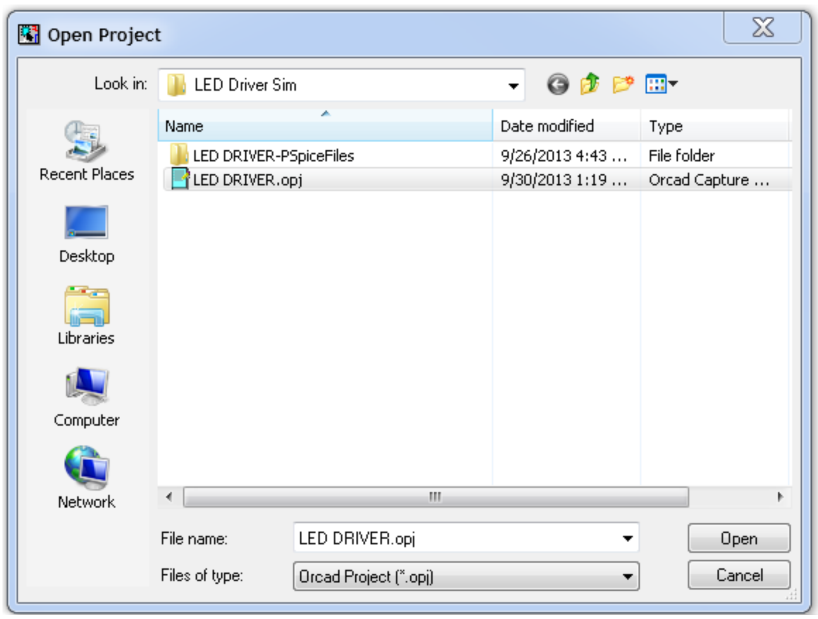
There are three files that are required:

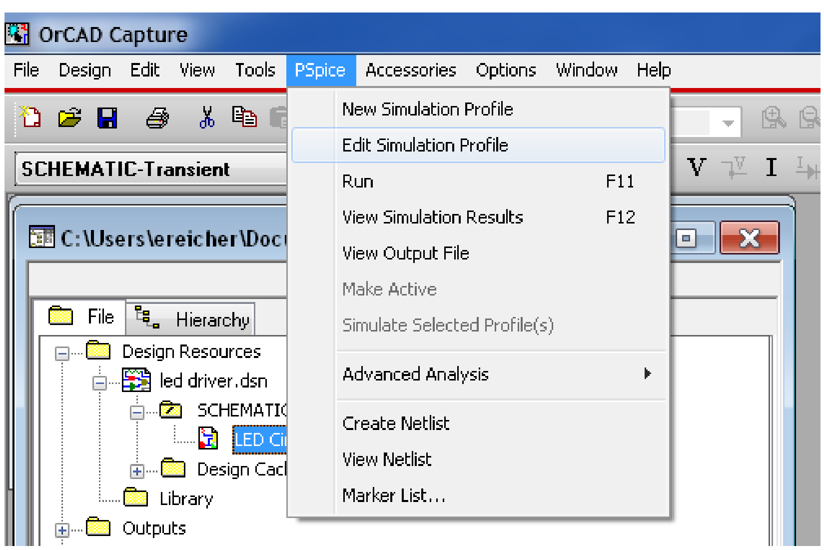
| | |
|---|---|
| A6211.LIB | This file contains the encrypted A6211 Pspice model. |
| A6211.OLB | This file contains the Pspice schematic capture symbols.<br>**Unzip these 2 files to c:\OrCAD\OrCAD_xxx\tools\pspice\UserLib**<br>Where “xxx” is your version of Pspice. |
| LED DRIVER.ZIP | This file contains a sample Pspice project and schematic using the A6211.<br>**Unzip these files to a working directory for your design.** |

**Step-by-step installation instructions:**

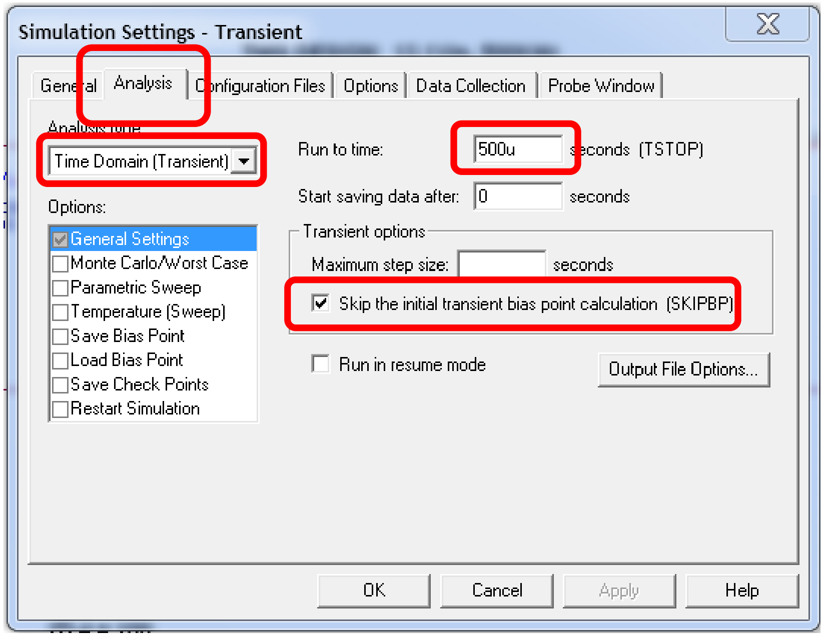
1) After un-zipping the files to their correct locations, start Pspice and use **File > Open > Project** and open the LED DRIVER.opj file.



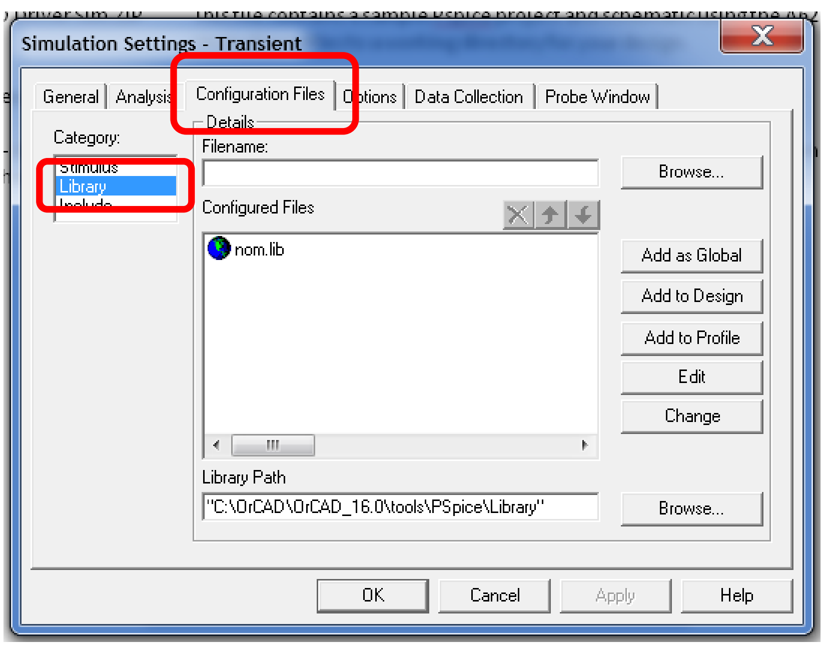

2) From the drop down menus, choose **Pspice** > **Edit Simulation Profile**.



3) Choose the **Analysis** tab, then select **Time Domain (Transient)** and enter the following TSTOP (500u).  Also, check the box for “**Skip the initial transient bias point calculation**”



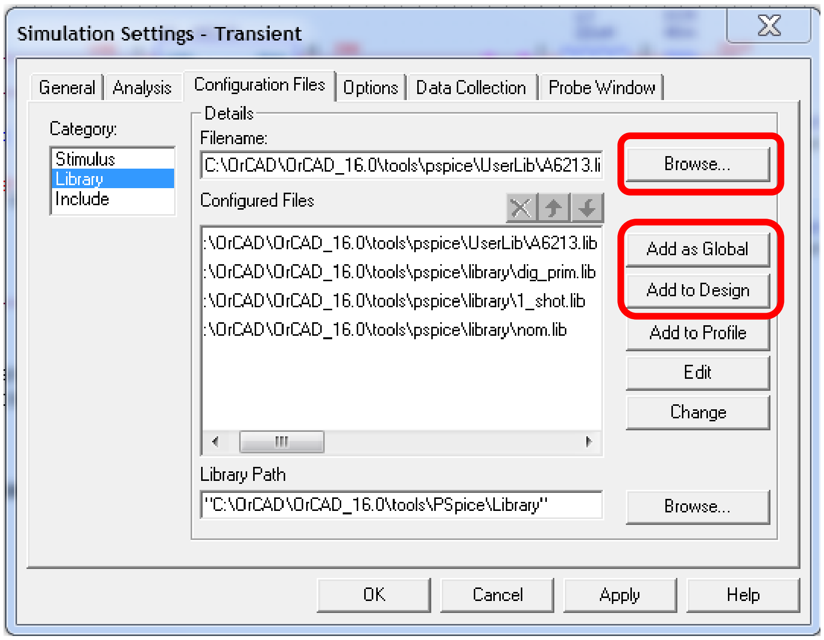

4) Choose the **Configurations Files** tab.  Then select the **Library** category.



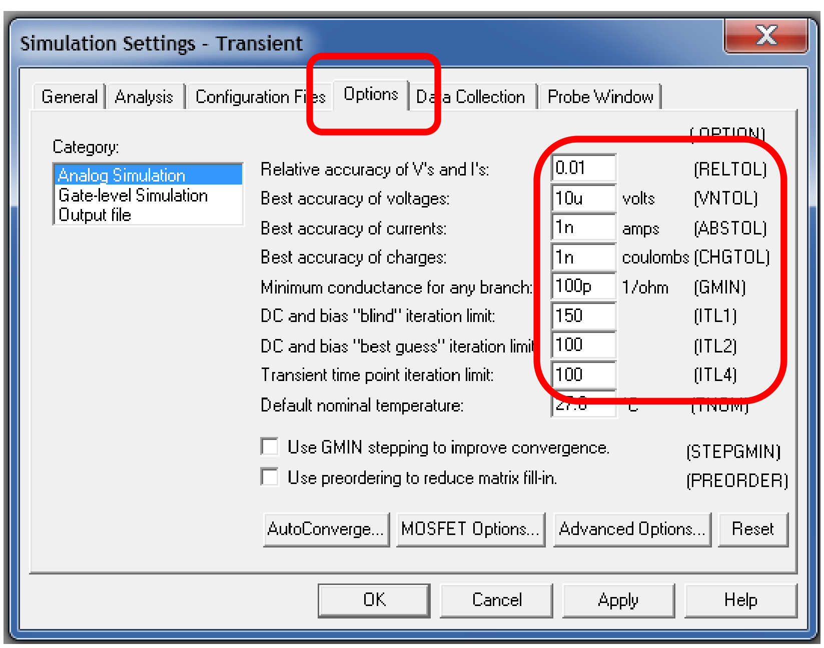
5) Use the **Browse** button to locate and then **Add to Design** (or **Global**) for <u>each</u> of the following libraries:

| | |
|---|---|
| c:\\OrCAD\OrCAD_xxx\tools\pspice\library\1-shot.lib | Add to Design (or Global if you wish) |
| c:\\OrCAD\OrCAD_xxx\tools\pspice\library\dig_prim.lib | Add to Design (or Global if you wish) |
| c:\\OrCAD\OrCAD_xxx\tools\pspice\library\nom.lib | Add as Global (usually already installed) |
| c:\\OrCAD\OrCAD_xxx\tools\pspice**UserLib**\A6211.lib | Add to Design |

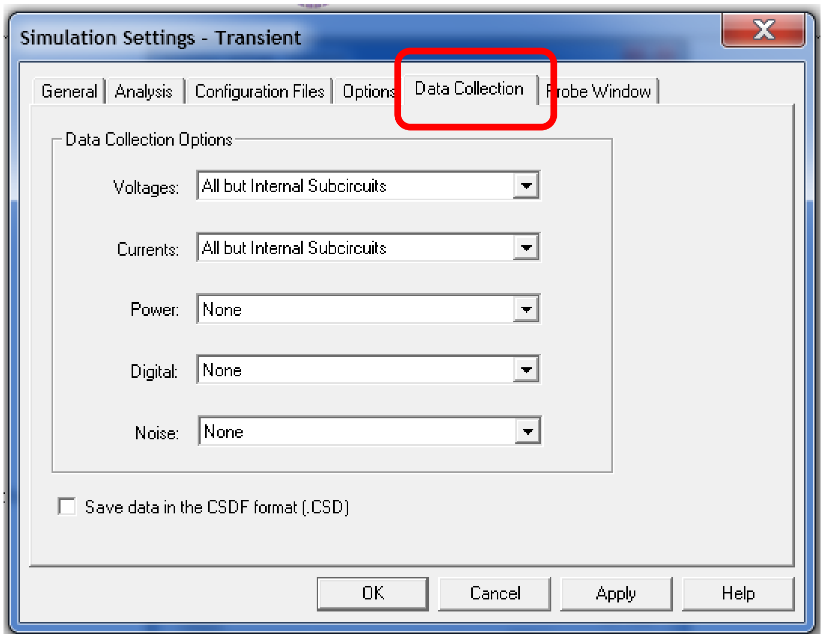
Your Simulation settings should look like this when done correctly:



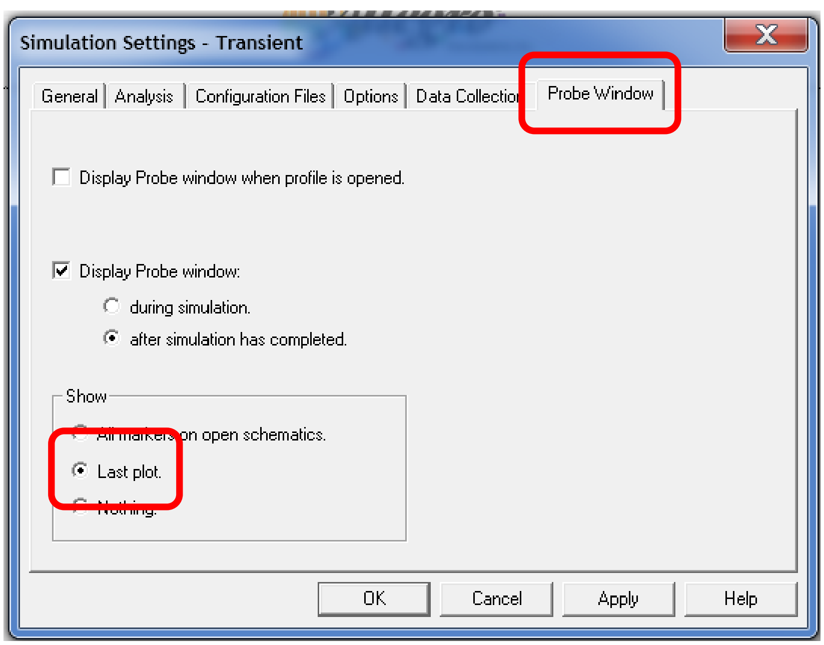

6) Click on the **Options** tab. Make sure your simulation options (RELTOL, VNTOL, …, ITL4) are all set like this or you probably will have convergence problems.



7) Click on the **Data Collection** tab.  We recommend the following settings to save disk space:

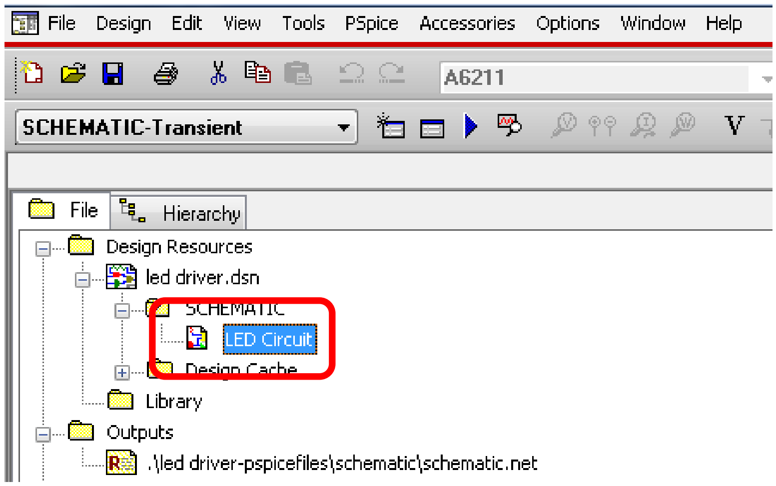



8) Finally, click on the **Probe Window** tab and select "**Last plot**".

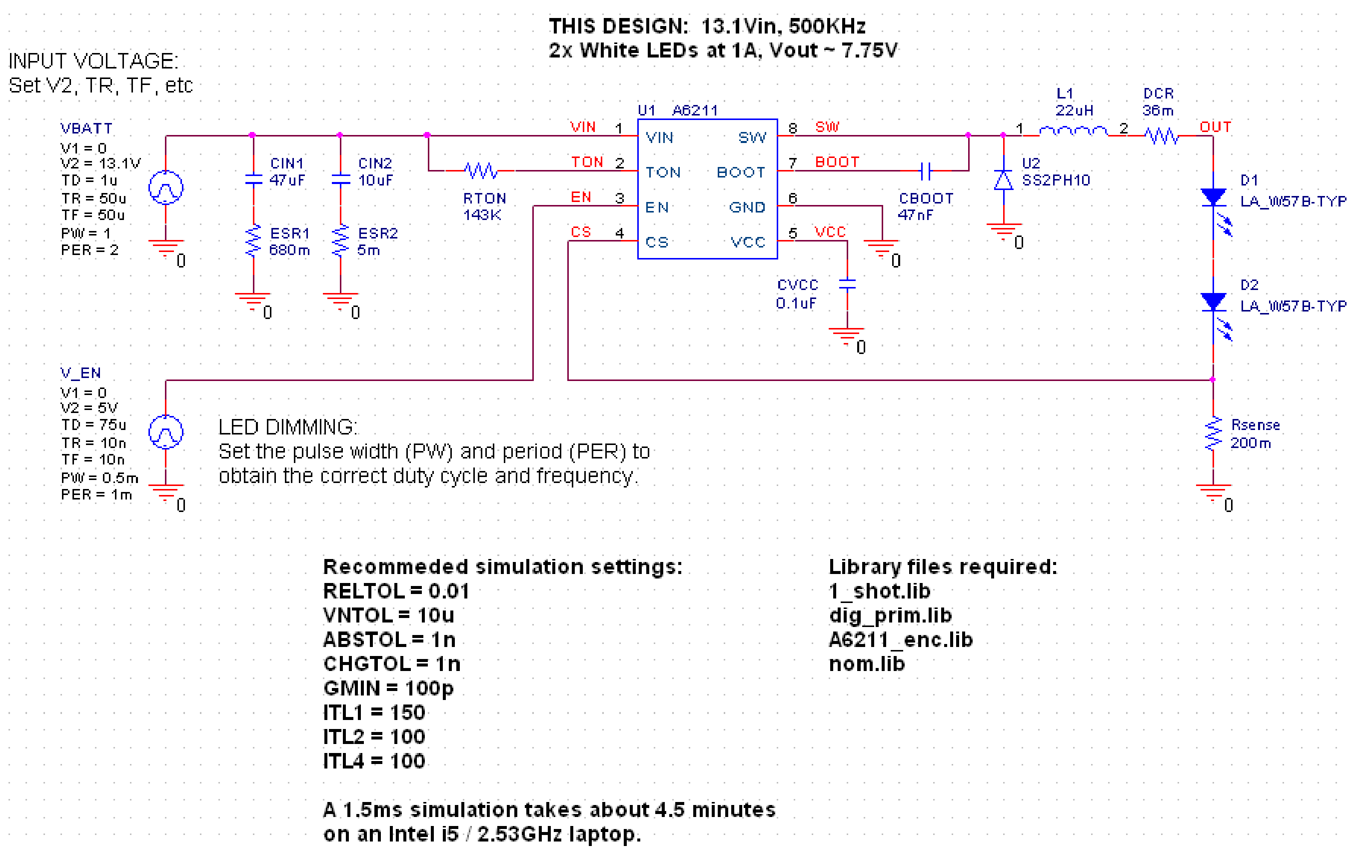



9) Click on **OK.**

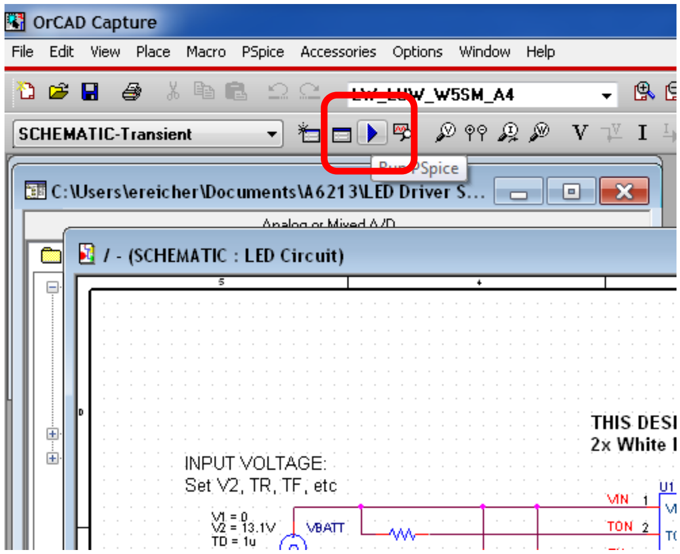
10) Double clicking the LED Circuit icon will bring up the A6211 schematic.



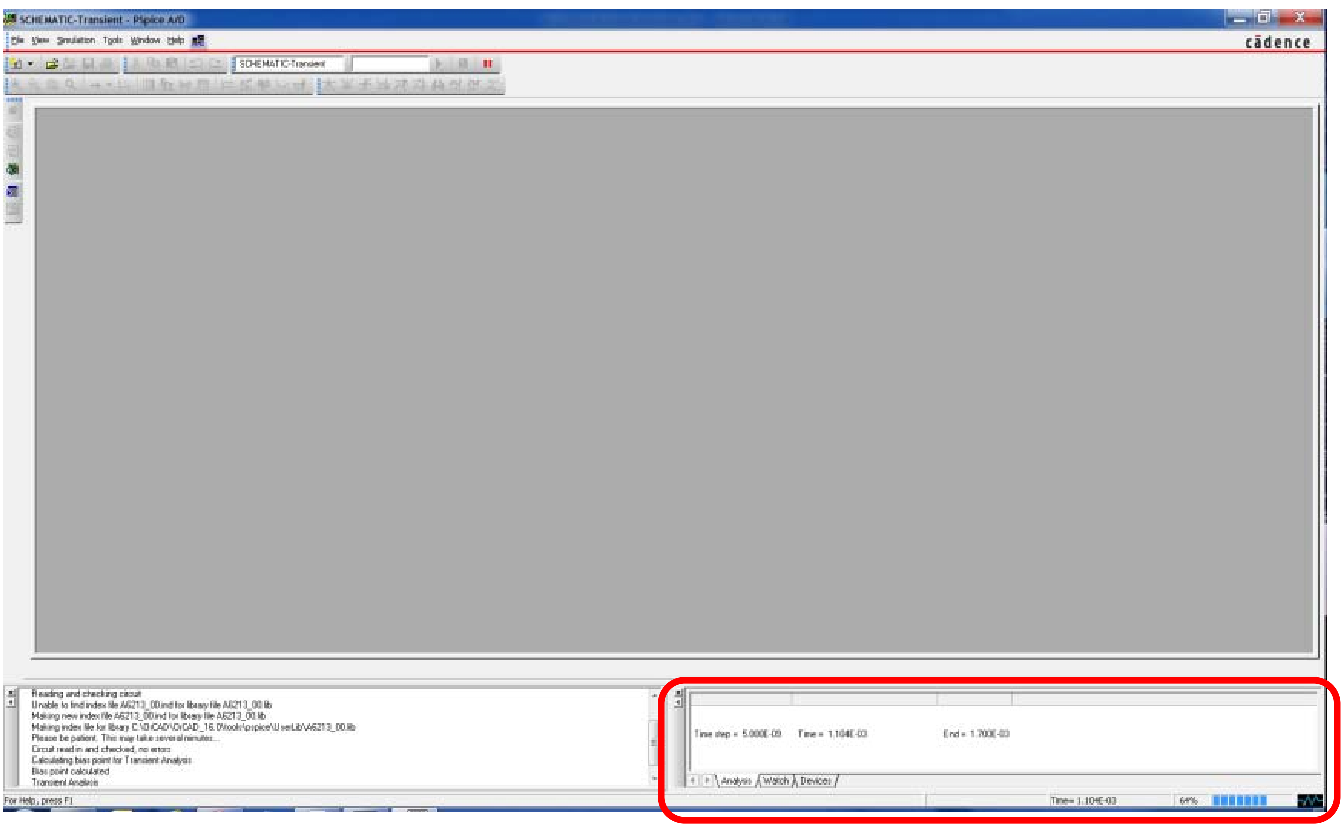

11) You should now see the following A6211 schematic:



12) Lastly, click on **the Run Pspice** icon (the **blue arrow**) to start the simulation:

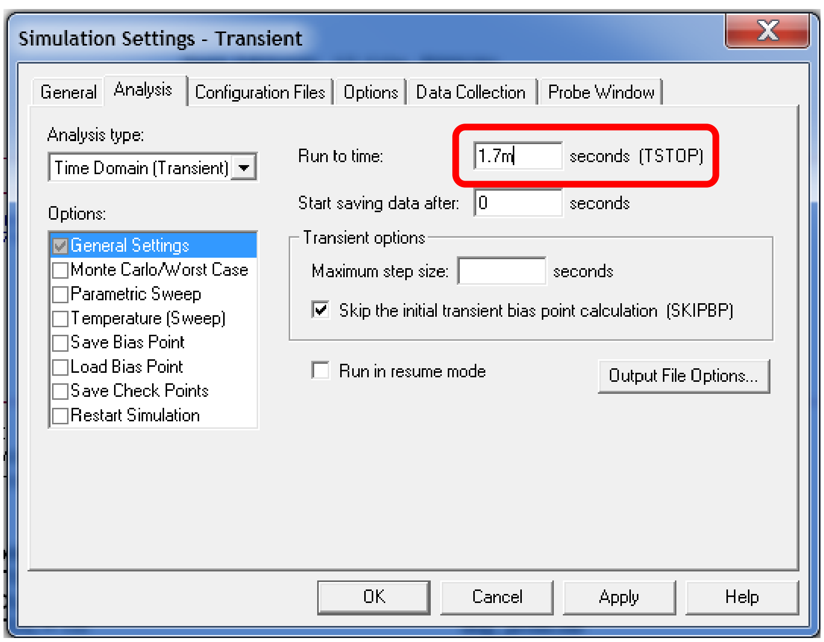


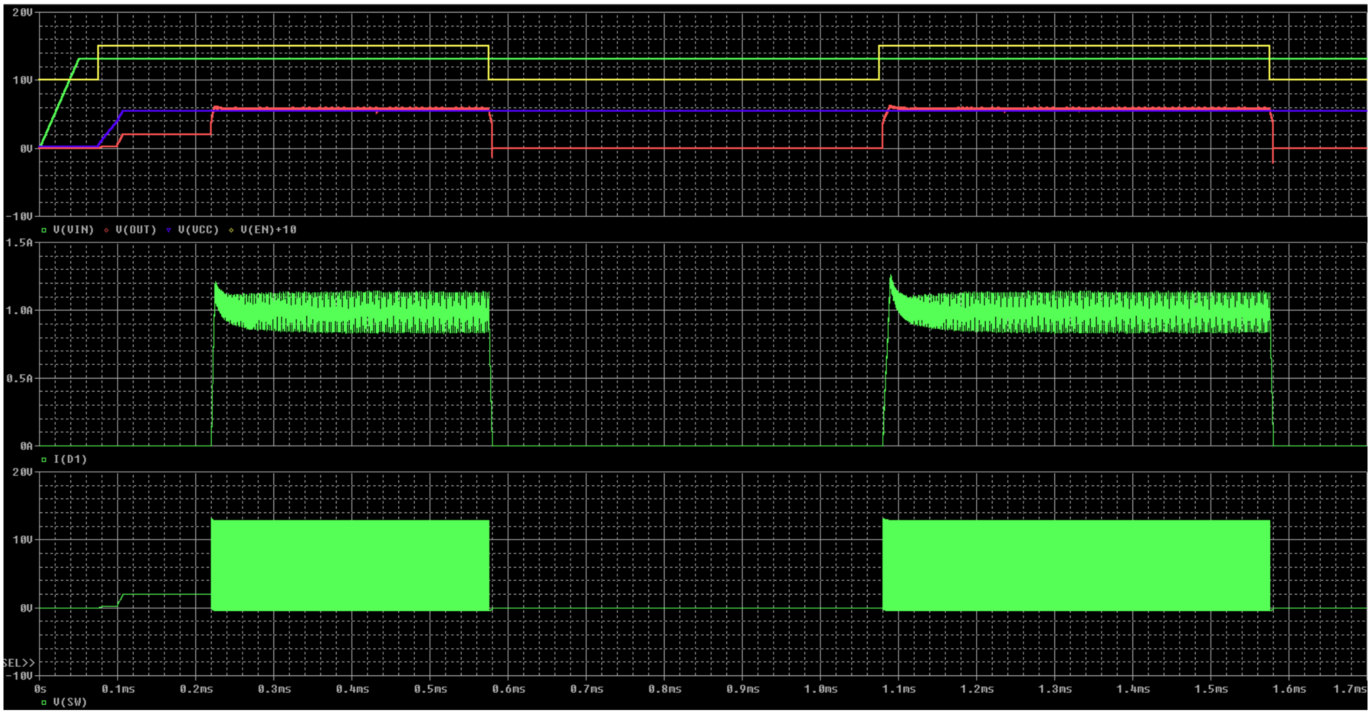

13) The Transient Analysis window should open. The lower right corner shows the simulation progression.



14) When this simulation is complete you should see this plot showing startup and operation to 500us.


15) If you change the TSTOP to 1.7ms and re-run the simulation, you will see the plot on the next page. This plot shows the use of PWM dimming via the EN duty cycle.






16) Congratulations.  You may now use the model to modify the external components to simulate your own design.