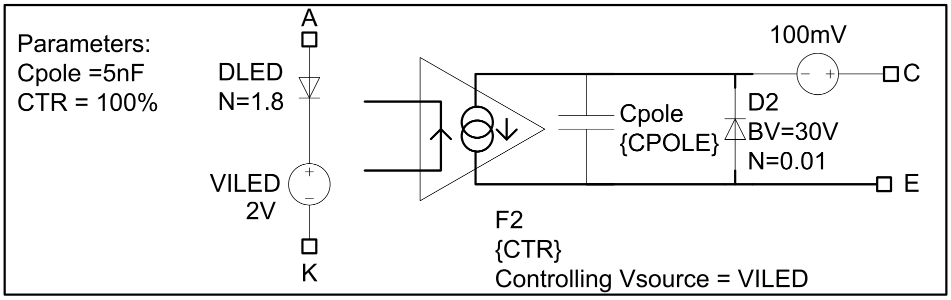
Creating SPICE Model Parameters for Orcad PSpice

There be SPICE model text files all over the internet. Thanks to the ltspice group for this one, http://tech.groups.yahoo.com/group/LTspice/ , a simplified spice model for a optocoupler Basso made for switching power supply modeling.

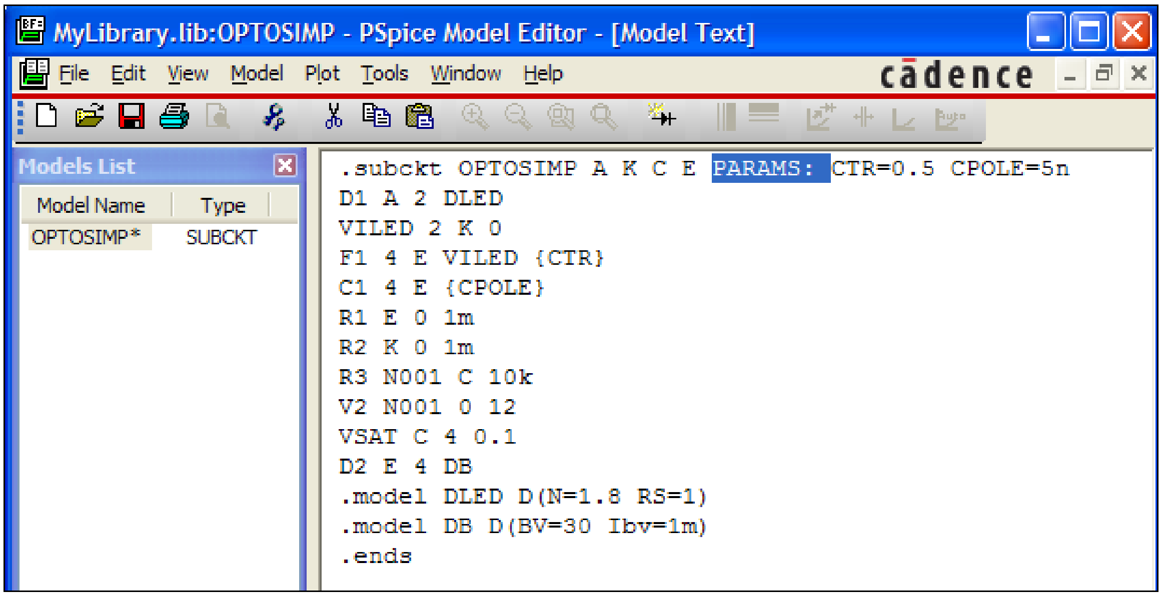



```
.subckt OPTOSIMP A K C E CTR=1.0 CPOLE=5n
D1 A 2 DLED
VILED 2 K 0
F1 4 E VILED {CTR}
C1 4 E {CPOLE}
R1 E 0 1m
R2 K 0 1m
R3 N001 C 10k
V2 N001 0 12
VSAT C 4 0.1
D2 E 4 DB
.model DLED D(N=1.8 RS=1)
.model DB D(BV=30 Ibv=1m)
.ends
```

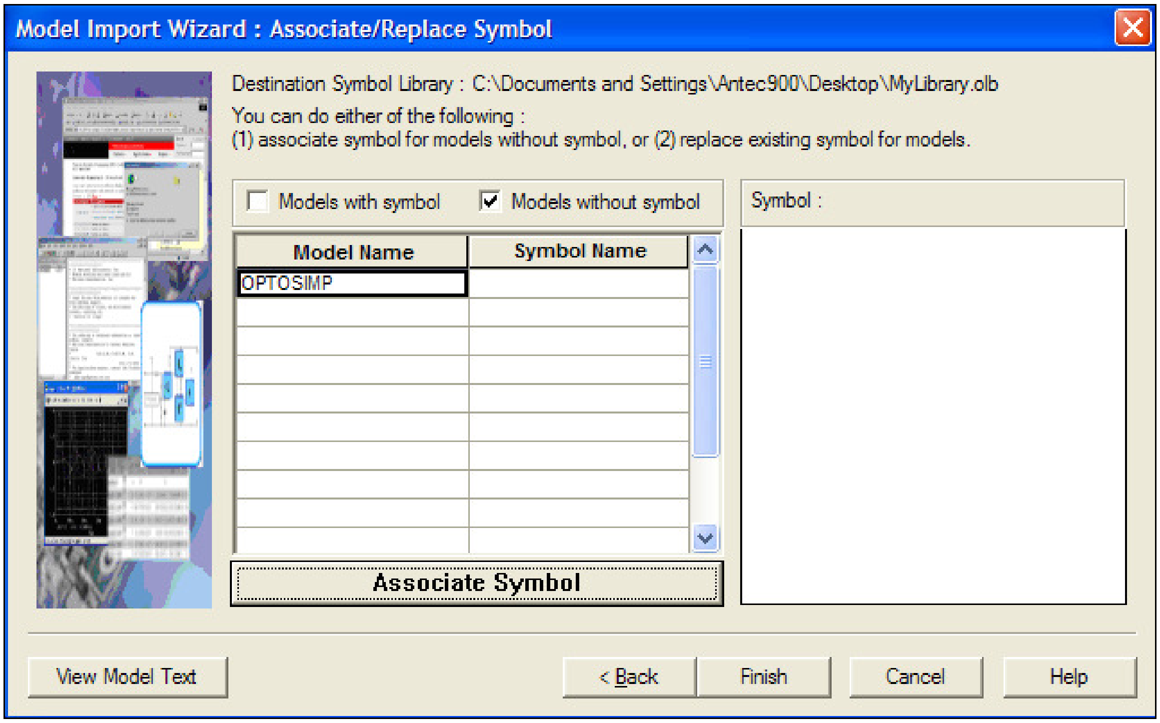
For this model, there will be two variables, the optocoupler CTR and the output capacitor pole, these will be the model's parameters, shown in brackets,{ }. The default values are shown to be a CTR of 100% (1.0) and Cpole of 5nF (5n).
Place the SPICE code into a text file and save it someplace on your computadora.

For more detailed PSpice help
Google search for '*PSpicePrimer.pdf*'
Google search for '*sloa070.pdf*'

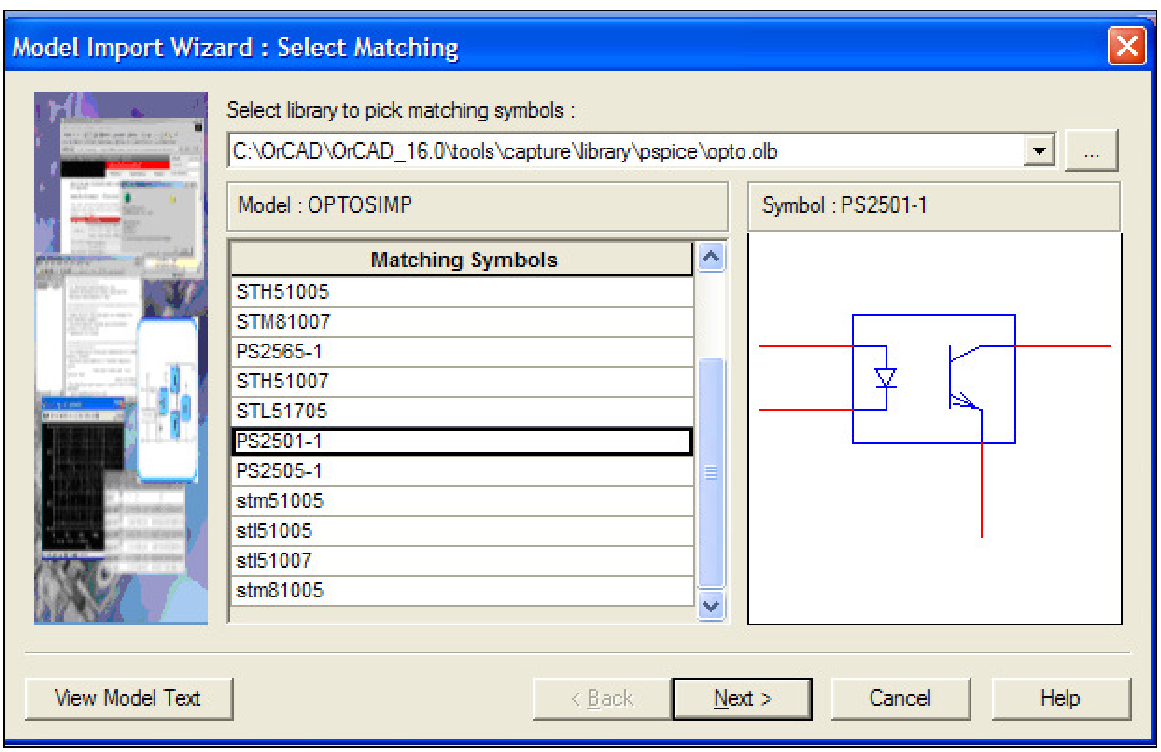
Go to '*Start Menu > Orcad > Pspice Accessories > Model Editor*'
Then either open up the library you wish to add the model to, or create a new one for the model. For this one, I made a new library, and called it '*MyLibrary*', cus I may want to add more models to it, but it was just for this model, I'd probably call it '*OptoSimp*' or something like that. I then go to '*Model > Import*' and select the saved text file with the SPICE code. On mine when its looking for files, it only looks for .mod filetypes by default, so I just selected all files so the text file shows up.
Add the 'PARAMS:' bit to the first line as shown.

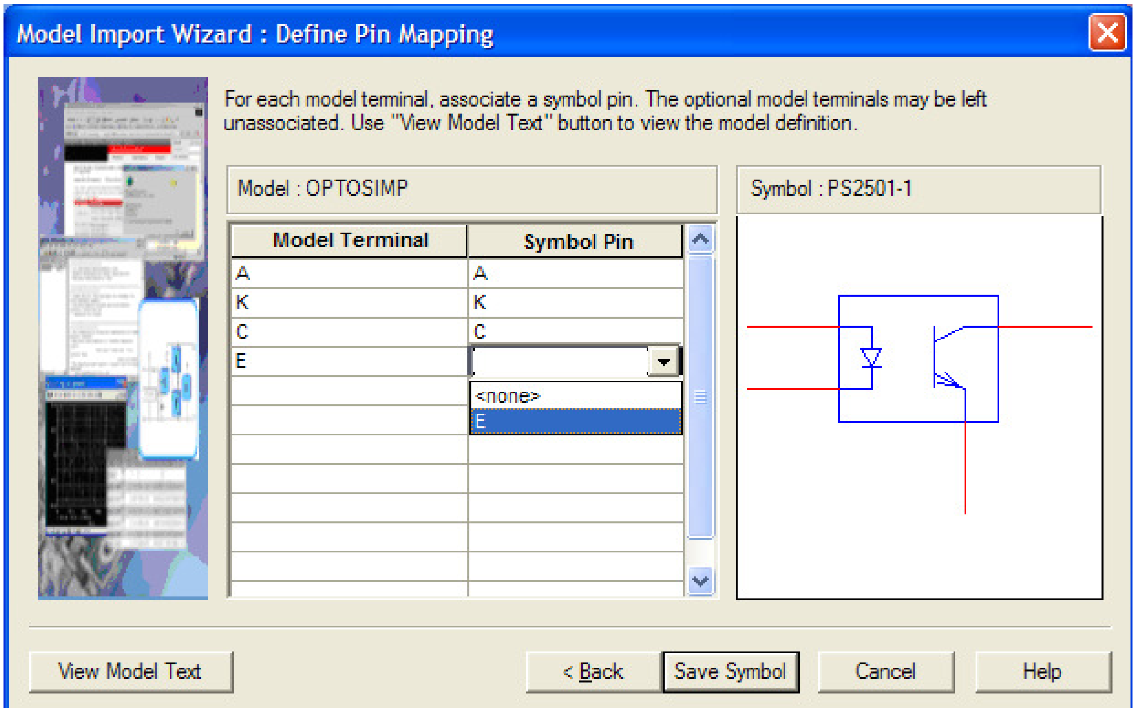


When done editing or adding the SPICE code, save it.
Then select '*File >Export to Capture Part Library…*' or '*File > Model Import Wizard…*' to create a symbol for the new SPICE model. The '*Export*' option I think creates a square box with just the ports for a symbol that you can either keep or add on to, and the 'Model Import' is what I used cus it lets you select a symbol from Orcad's parts libraries, saving some drawing time. It creates a '*MyLibrary.olb*' symbol file that becomes associated with the '*MyLibrary.lib*' file. Where symbols for the models that are in the new library are stored.

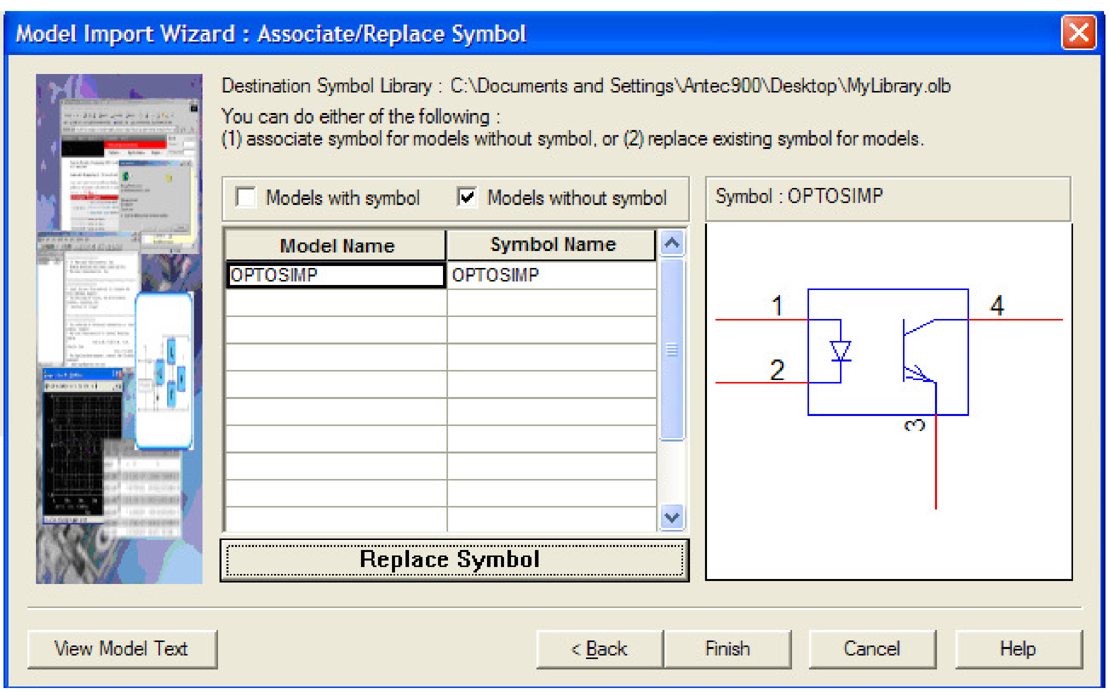


Then it takes you to the symbol menu, showing you you've got no symbol name for the OPTOSIMP model you just made. Select 'Associate Symbol' and then you can browse through the orcad libraries to select a symbol that's right on, or even closely resembles what you've got in mind for the symbol. If you're SPICE model only has four ports, it will only show you four port symbols in the libraries you can choose from. Click Next.

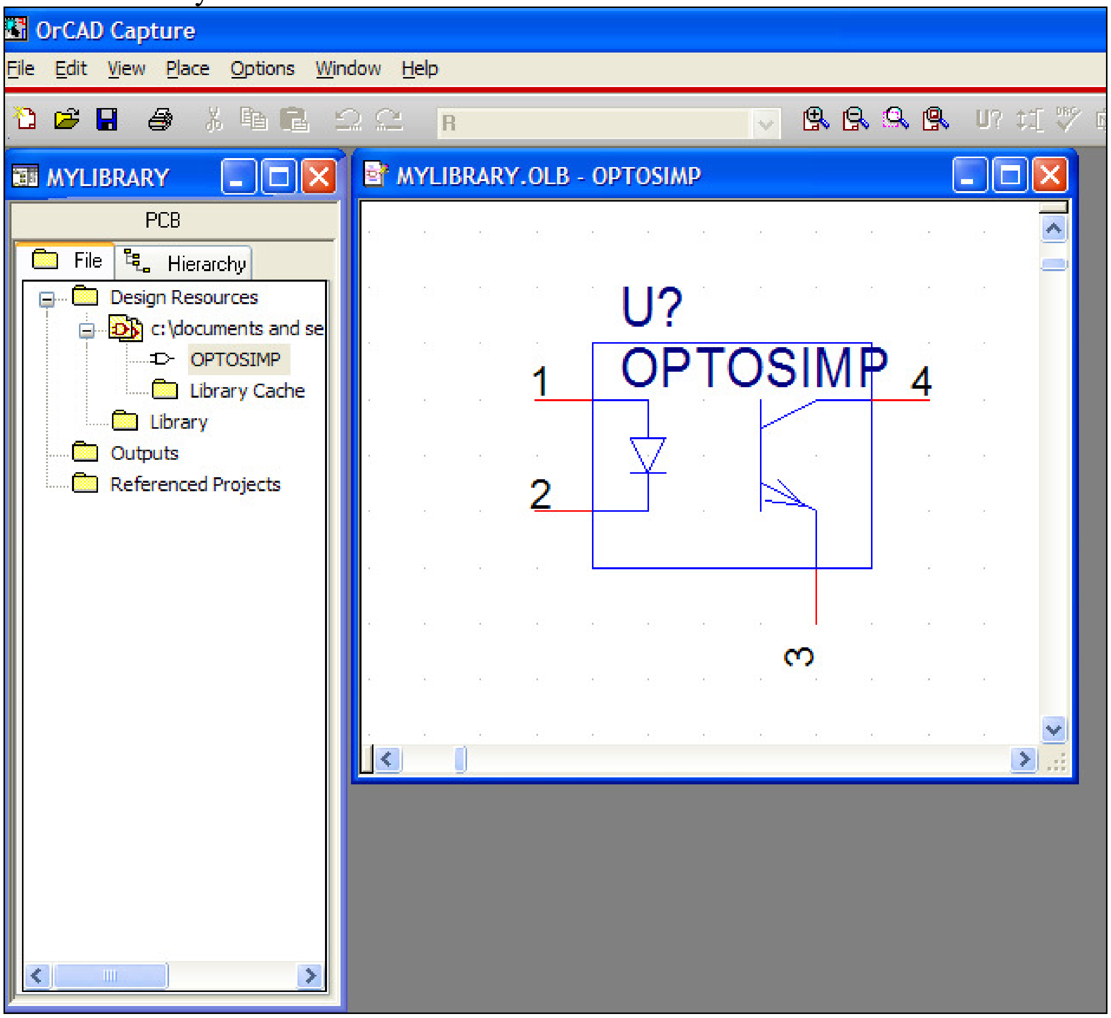


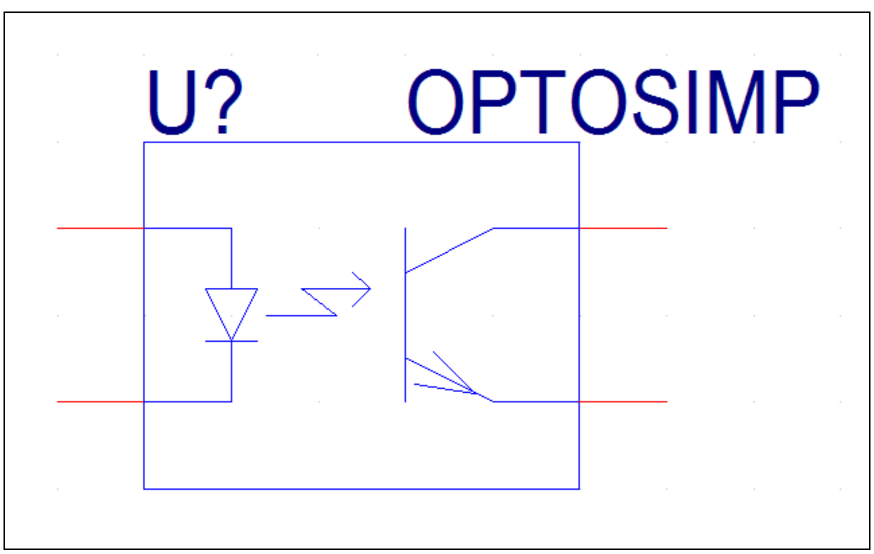


Then it takes you to match your SPICE model's port pins to the symbol's port pins. Sometimes the letters or numbers match like this case, and sometimes they don't. So it may take some examining of the symbol or the SPICE code (yuck!) to try and figure out which is which, cus its important to have matching ports or you might have connected the diode's cathode (K) to the output collector (C), and have some funky results in the simulation. Click Finish and Save and Exit the library.

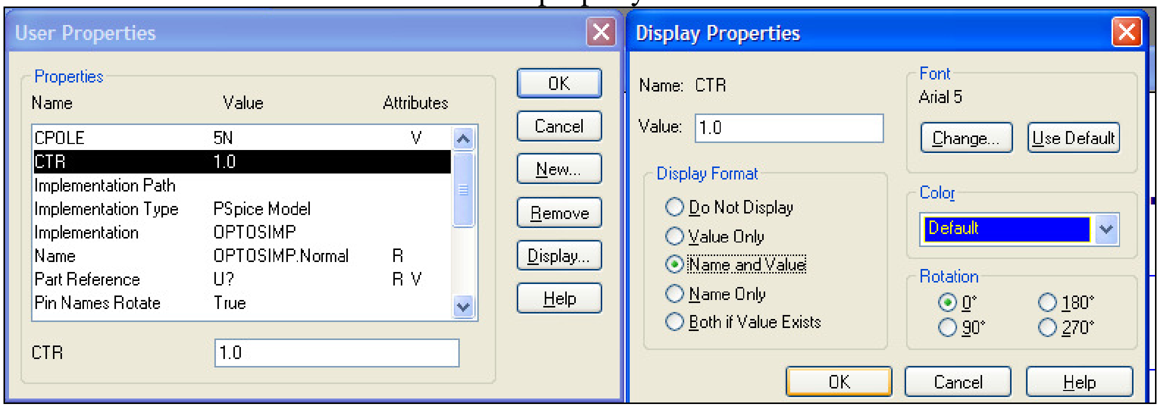


Now open up Orcad Capture, and and select '*File > Open > Library*' and select the newly edited '*MyLibrary.olb*' file. The symbol is shown, where it can now be left alone or modified to your heart's desire.

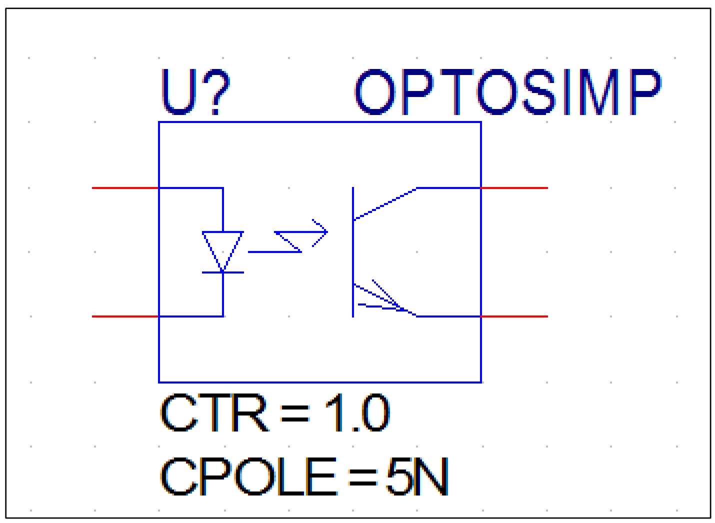




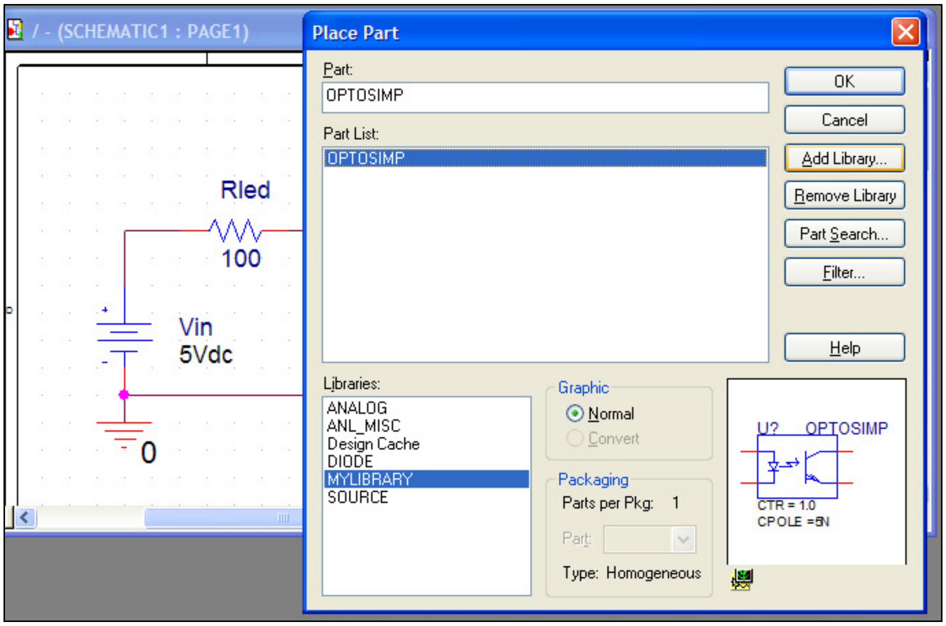
Go to '*Options* > *Part Properties*', and highlight the two parameters. Click '*Display*', and selecte '*Name and Value*' and OK. In the User Properties box, a '*V*' gets placed in their attributes column indicated that that property are now visible.



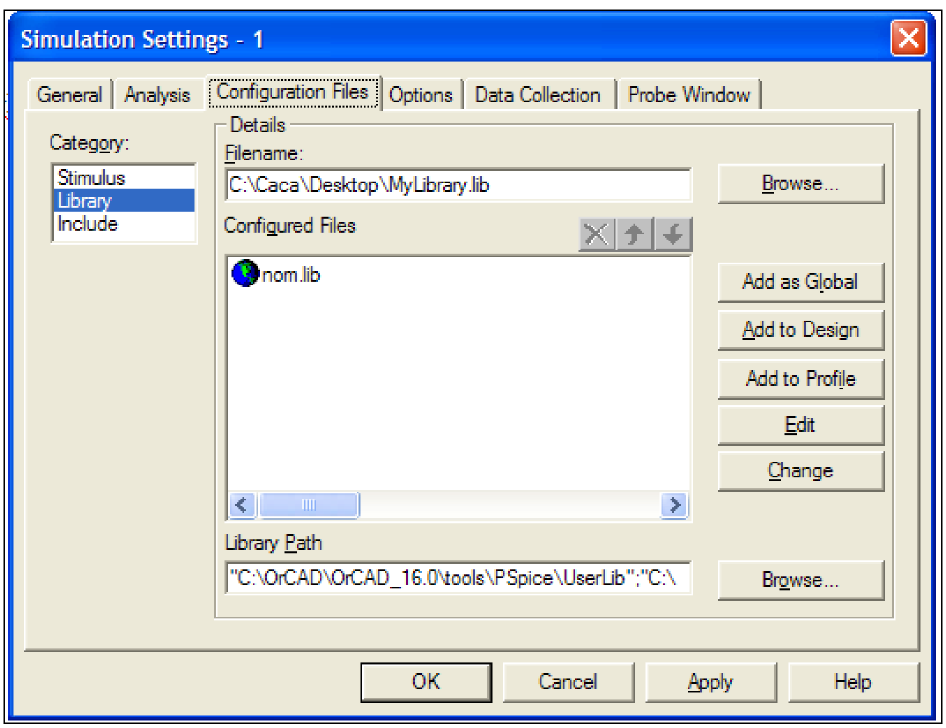
Now that the symbol is all finished, and the two parameters are shown, with their default values. When placed in a schematic for simulation, the user can easily edit the CTR and Cpole as they wish. Save and close the library, but leave Capture open.



In your simulation circuit, add the library to your parts list if not already done so, and then select the '*OPTOSIMP*' model to place into the schematic.



In order for the simulation to work, BE SURE to add the '*MyLibrary.lib*' file to the configuration library in the simulation settings. Either globally, so it's in there for all future projects, or to design, so it's only in the current project simulation.



Now doing some bias point simulations to test the model, shows that its working, and the results are very close to expected, given the changing CTR parameter. The voltage drop for the LED (2V) and the saturation voltage of the transistor (100mV) is also accounted for from the original SPICE code. If wanted, those could also be changed to parameters, by changing the code a bit and repeating these steps again.