

## *Loading New Parts into Orcad Capture 16.6 version*

**Professor Angela Rasmussen**

### **Background:**

Not all parts are included in the Cadence Orcad simulator. You may need to load new parts into the schematic window in order to perform a simulation. To do that, a “SPICE Model” for the part is needed. A SPICE model tells PSPICE how the new part operates when voltages and currents are applied to the device. One can even create their own model for a device. Models can be created as described in the Cadence Orcad manual. Often, others offer the models online and they can be found by searching the internet. This tutorial discusses how to load a model into the schematic editor of Cadence Orcad and then how to simulate using the new part. A zener diode with part name 1N4740A will be used as an example in this tutorial.

### **Procedure:**

A spice model in Cadence Orcad is described in two files:

- a. A .lib file (this file contains the actual data information for simulating the part)
- b. A .olb file (this file contains the schematic part drawing only)

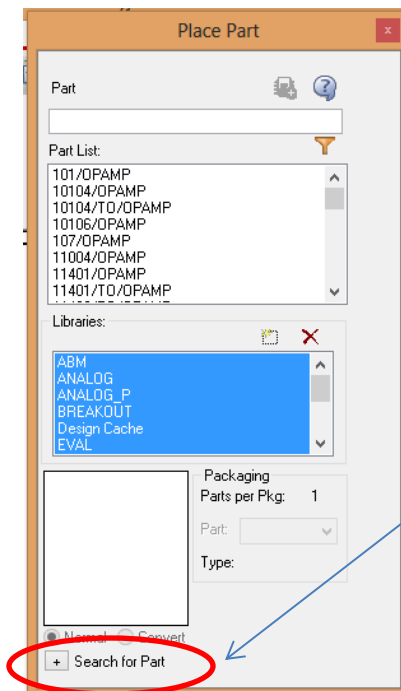
For this tutorial, I have posted the necessary files on the course website.

1) Download the SPICE model files: **Diode.OLB** and **Diode.LIB** from the ece2280 lab website:

<http://www.ece.utah.edu/~ece2280> under the lab link.

Create a new directory on your computer and place the downloaded files in that directory so that you know where they are located.

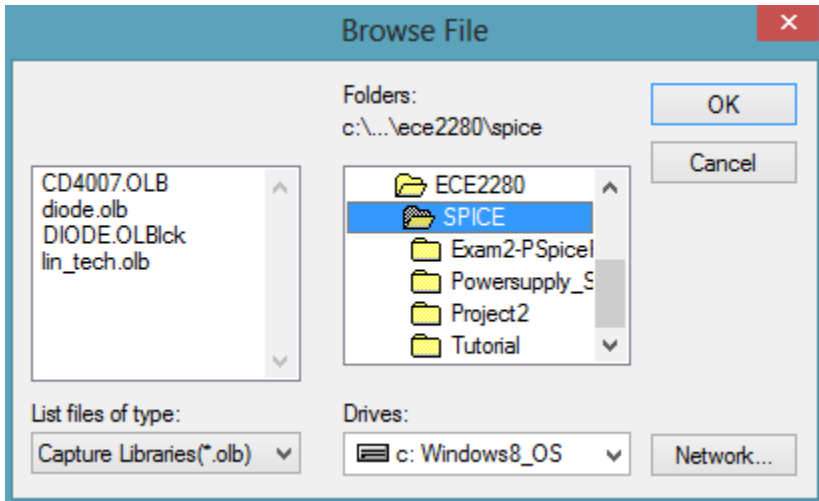
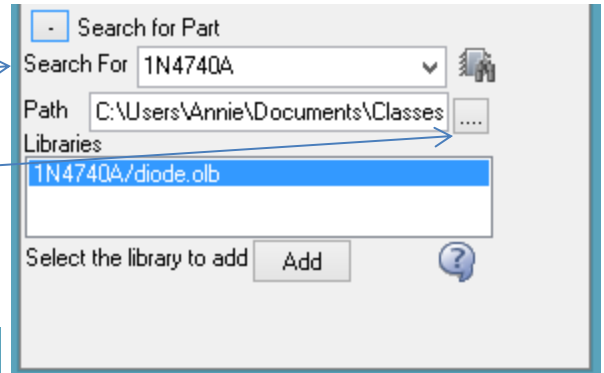
- 2) Open Capture, start a new blank project and follow the procedure for drawing your schematic.
- 3) When you want to draw the new part (in this case the zener diode), click on the Place Part symbol or use the pull down menu. At the bottom of the place part box there is a +“Search for Part” button. Click on this to expand it=>



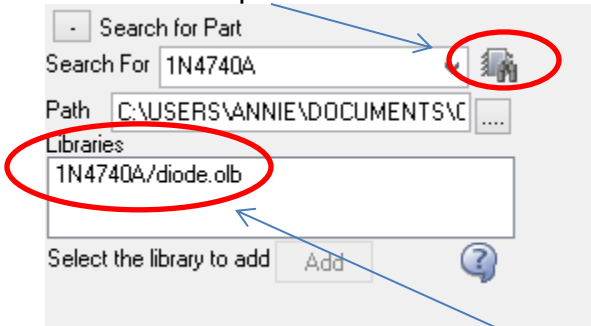
After this has expanded,  
Put in the part number => 1N4740A

Click on the .... box

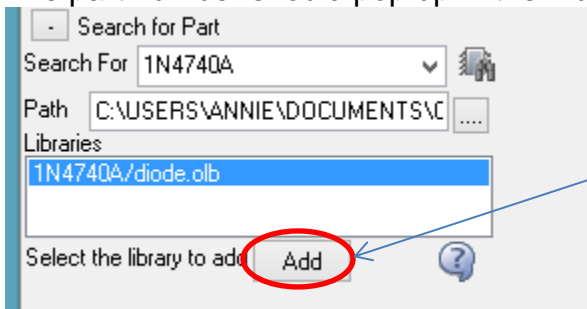
Browse to the folder where you saved the **diode.olb** file  
and select that folder.



Click on the part search button:

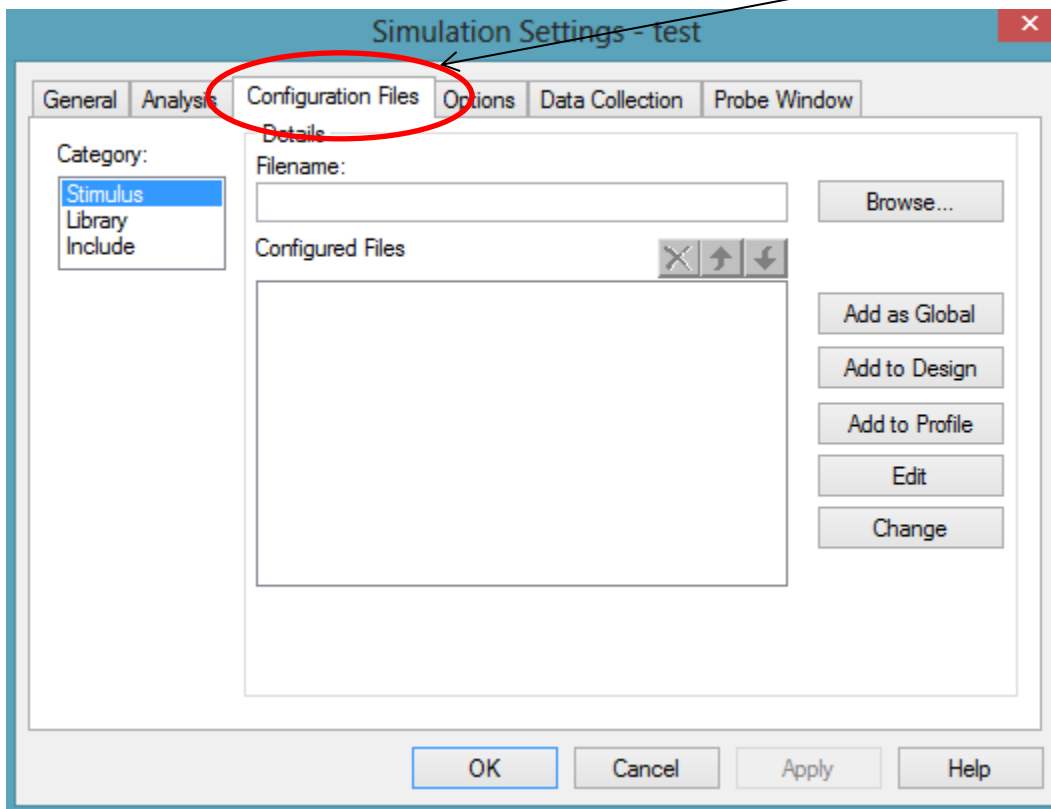


The part number should pop up in the Libraries area. Select it and click the Add button.

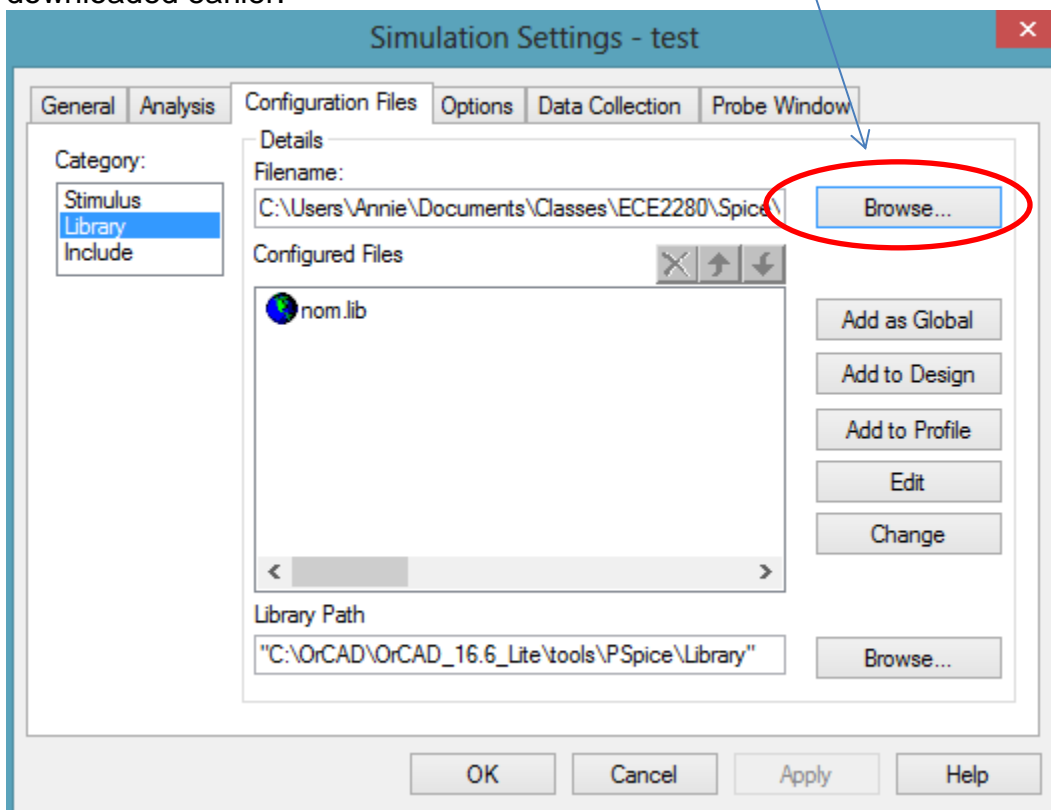


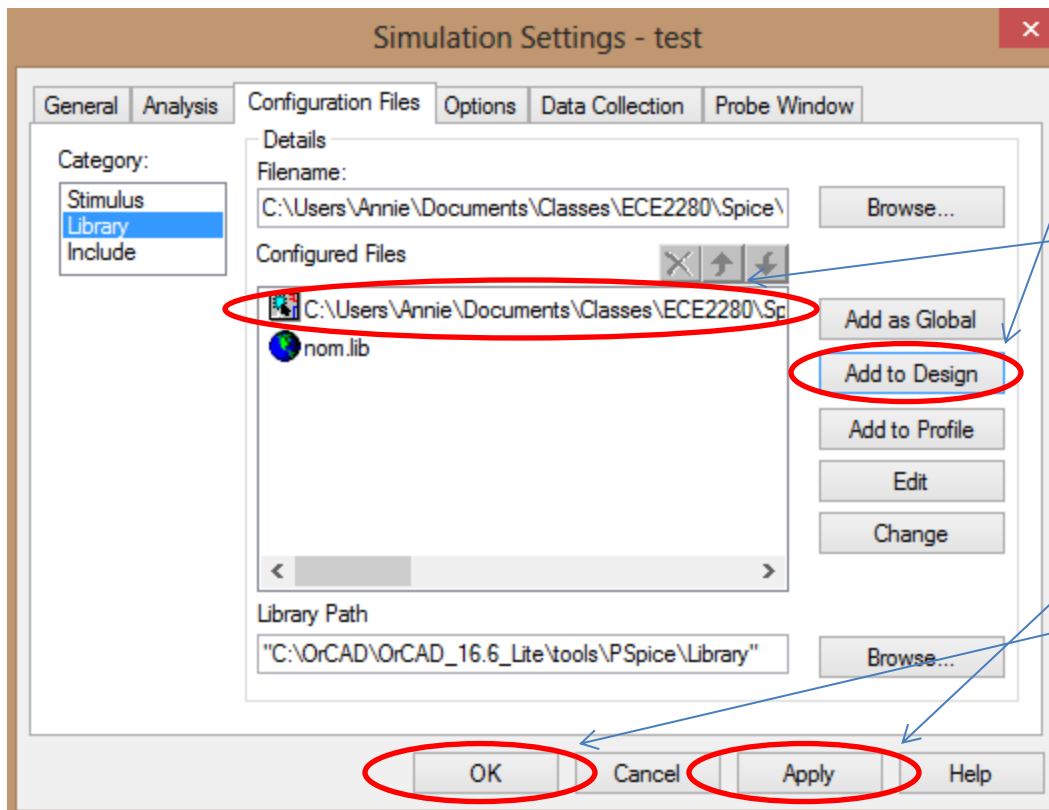
You can now place the part by double clicking on it and dragging it anywhere in the schematic it needs to be placed. Once you have finished drawing your schematic, you will also need to add the model to the part. This is done as follows:

4) Create a “new simulation” profile. The simulation settings window will then pop up. If it does not, click on the “edit simulation profile”. Click on the tab that says “Configuration Files”.



Under the Category area, select Library. Click on “Browse” button and locate the **diode.lib** file you downloaded earlier.





Click “Add to Design” button to add the pathway for the simulator to locate the file.

You should see the pathway now under the configured files area.

Then press **Apply** and then **OK** and run your simulation.

*Note: If you see an error that your part is “undefined,” you probably haven’t clicked on “Add to Design” as stated above.*

### **WARNINGS:**

You may find a library on the Internet that has your PART # within it. If you download and use this library, perform some simple IV simulations to ensure it actually matches your manufacturer’s specification sheet. If the library doesn’t produce similar IV-curves to your SPEC sheet, it may not be the right part. Be very careful of this! It is always a good idea to perform a simple simulation and ensure it matches your SPEC sheet before relying on it.