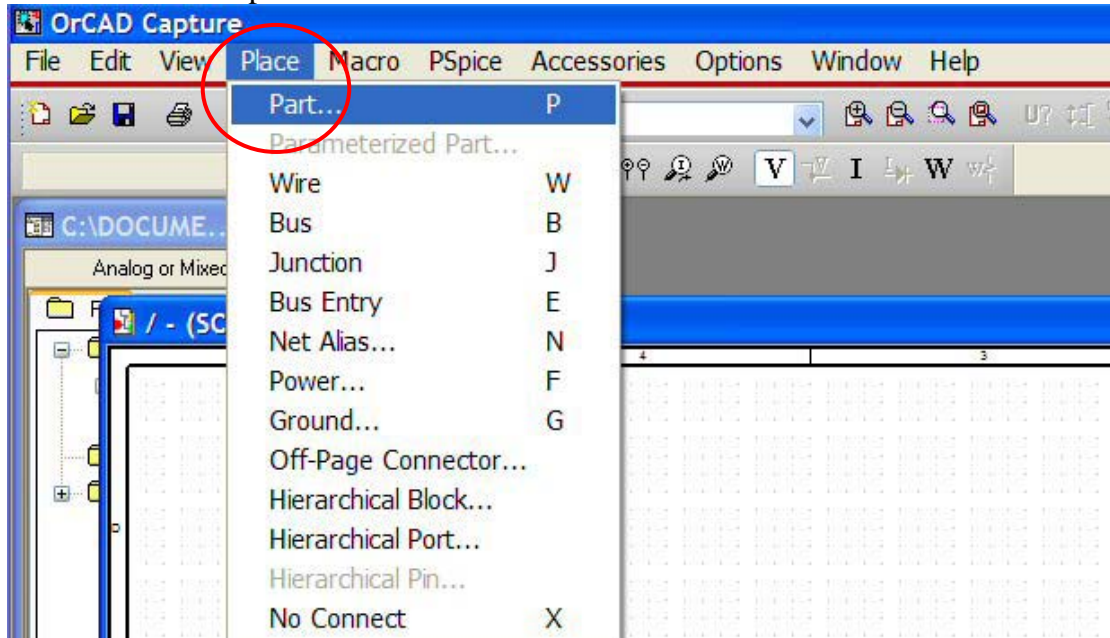


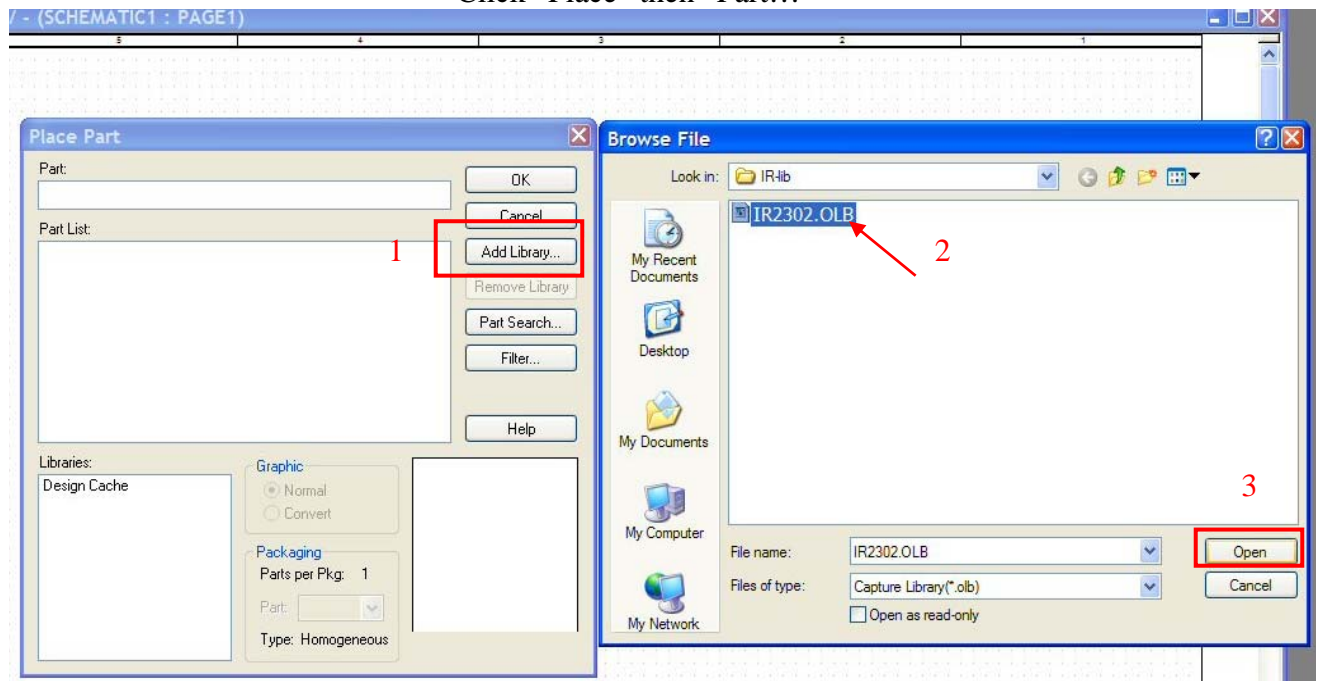
## PSpice Library Usage in OrCAD

Step I - add PSpice library to schematic:

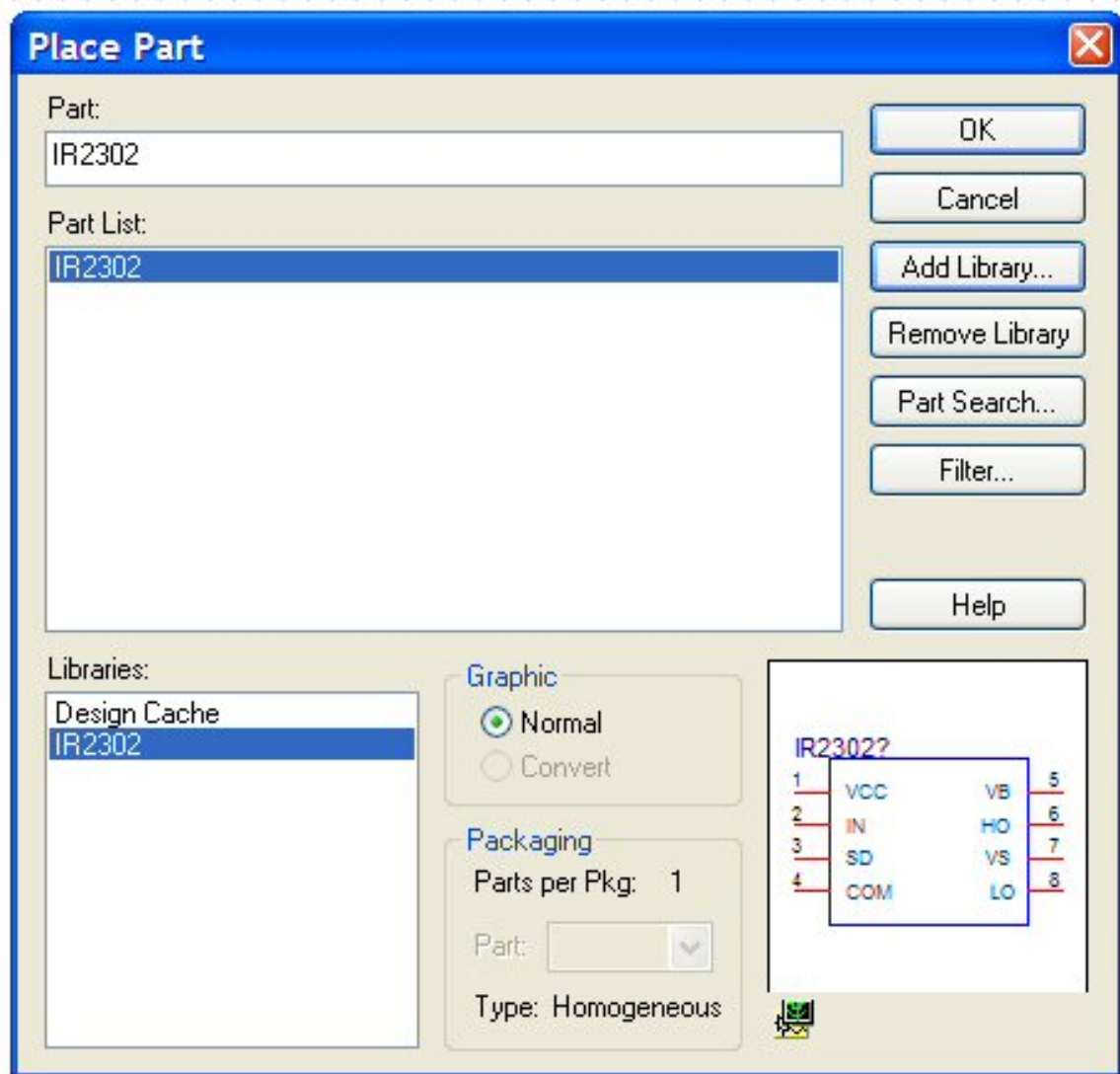
Start a new project => open schematic page => Place Part => Add Library => browse to the **IR2302.OLB** file then open



Click "Place" then "Part..."



Click "Add Library..." then browse to the IR2302.OLB file, select it then click "Open"

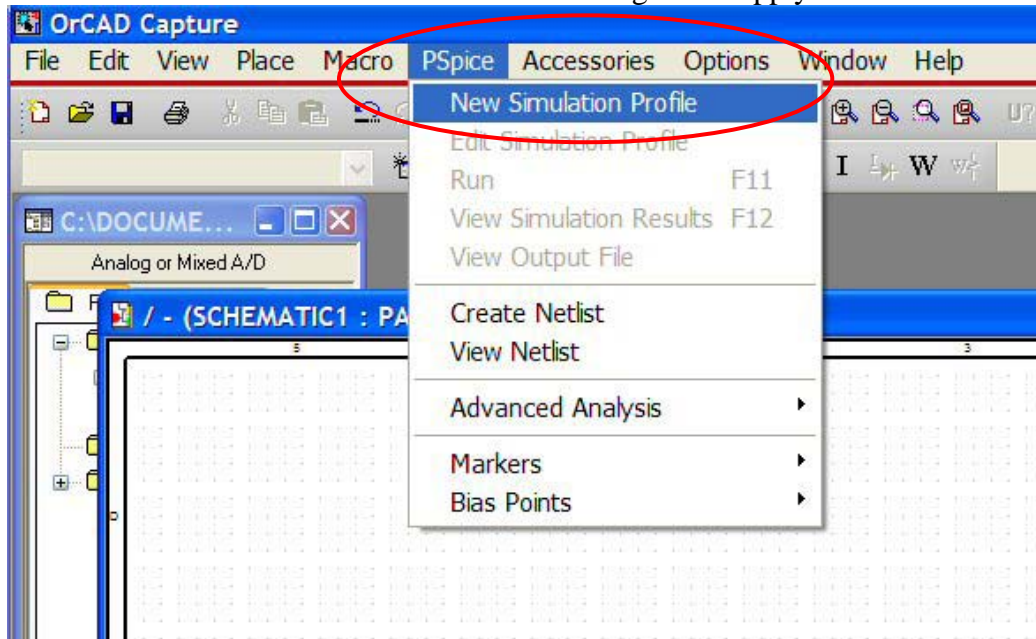


IR2302 will be added to library for use in schematic.

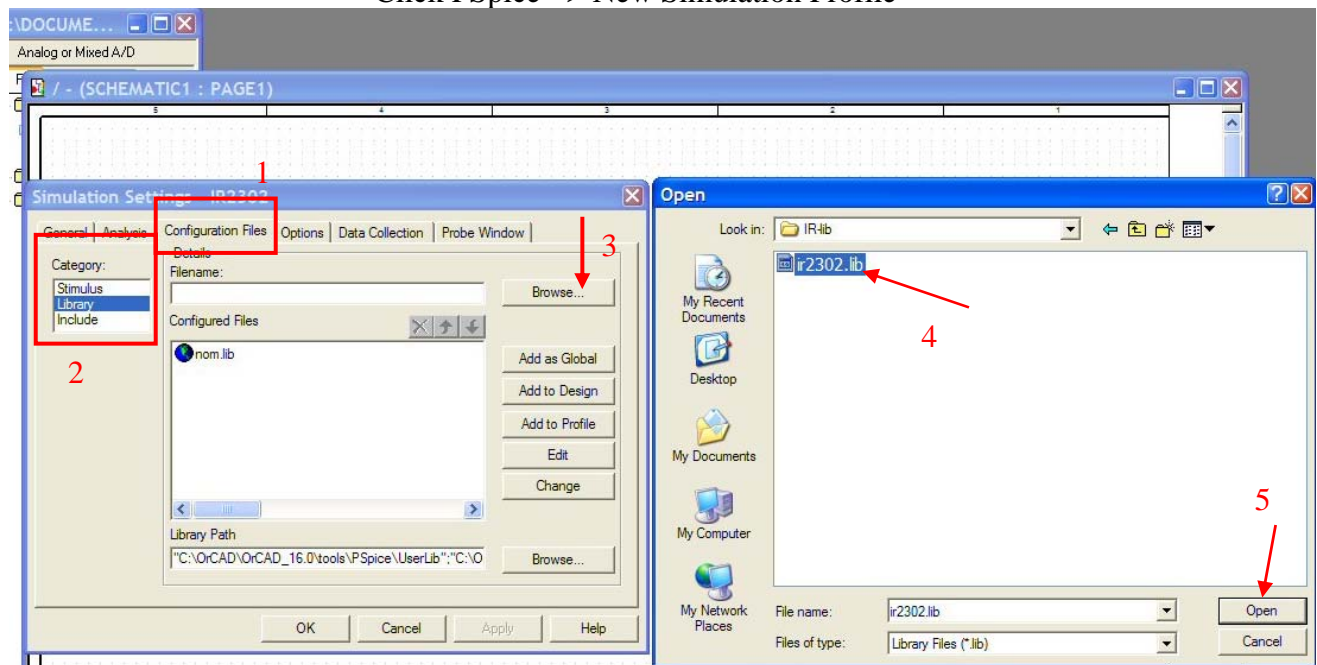
# International IOR Rectifier

## Step II – add library to simulation profile:

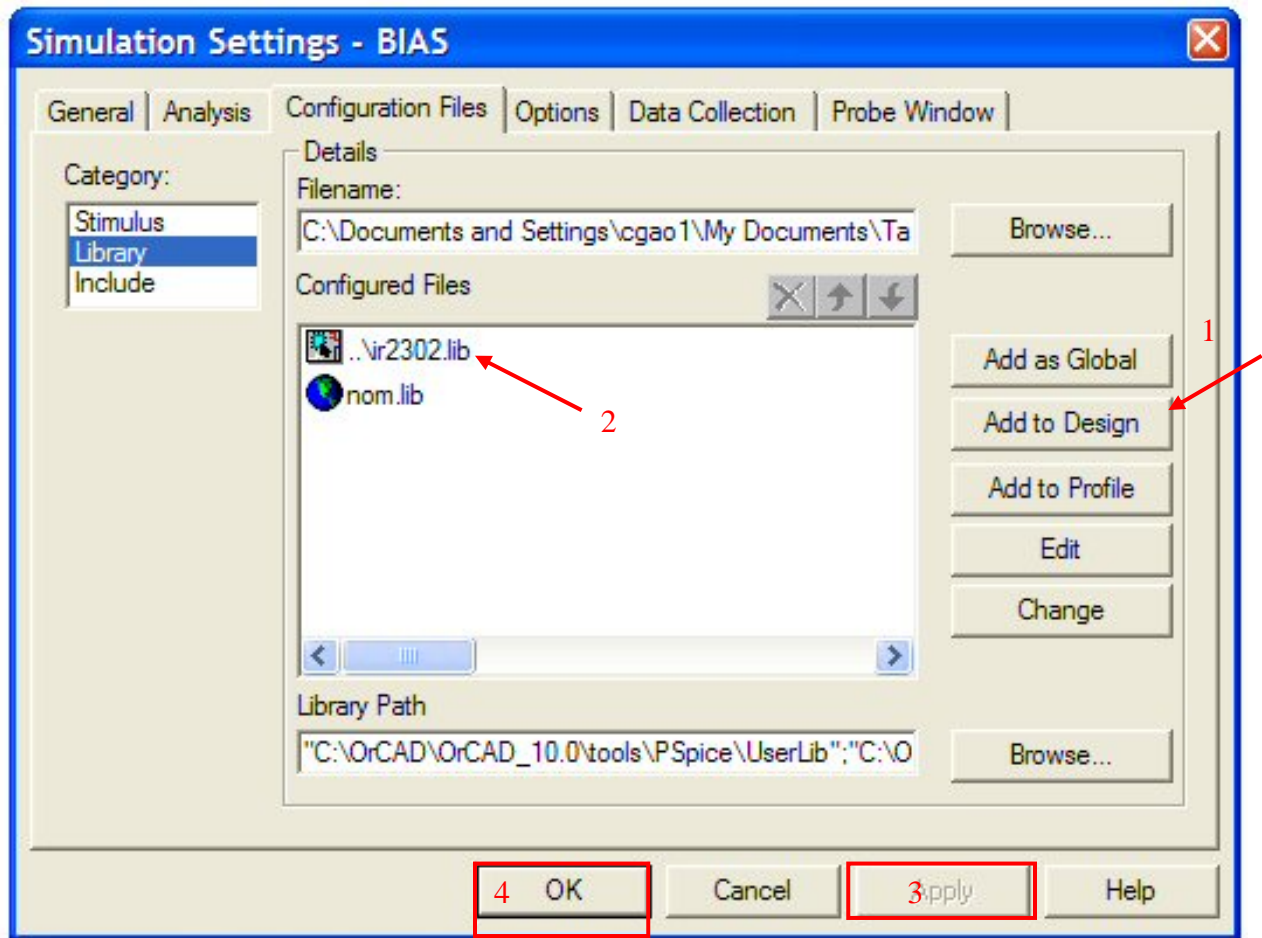
When the schematic is completed and ready to run simulation, go to PSpice menu => New Simulation Profile => Configuration Files tab => select “Library” from “Category” => Browse to the **IR2302.lib** then select the file and click “Add to Design” => Apply and OK.



Click PSpice => New Simulation Profile



Select Configuration Files and Library in Category, and then browse to the **IR2302.lib**, select the file and click Open.



Click Add to Design, you should see **IR2302.lib** is added and then click Apply and OK.

The process is now complete.