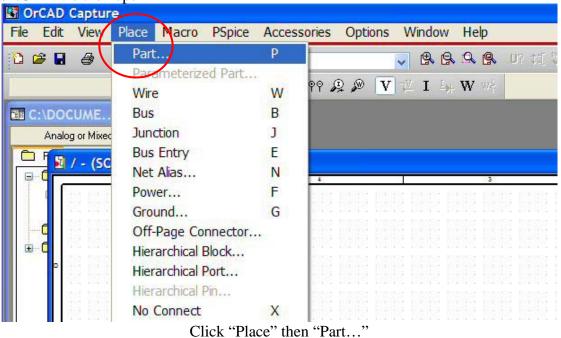


PSpice Library Usage in OrCAD

Step I - add PSpice library to schematic:

Start a new project => open schematic page => Place Part => Add Library => browse to the

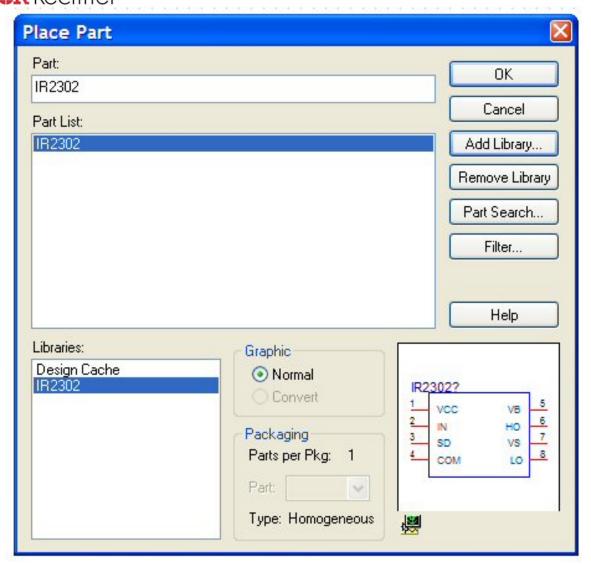
IR2302.OLB file then open



Browse File Part: Look in: 🗀 IR-lib G Ø 📂 🖽 * OK IR2302.OLB Part List Add Library. My Recent Documents Part Search... Desktop Filter. Help My Documents Libraries: Graphic 3 Design Cache Convert My Computer IR2302.OLB Packaging File name Open Parts per Pkg: Files of type Cancel Capture Library(*.olb) Open as read-only Type: Homogeneous

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

International Rectifier

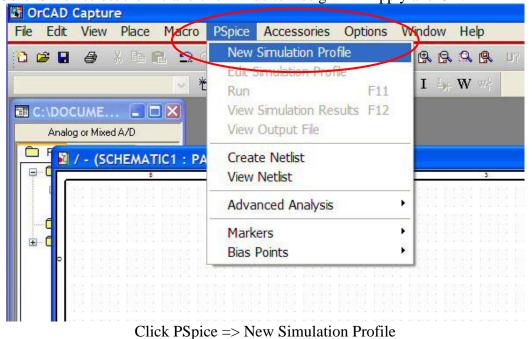


IR2302 will be added to library for use in schematic.

International **IOR** Rectifier

Step II – add library to simulation profile:

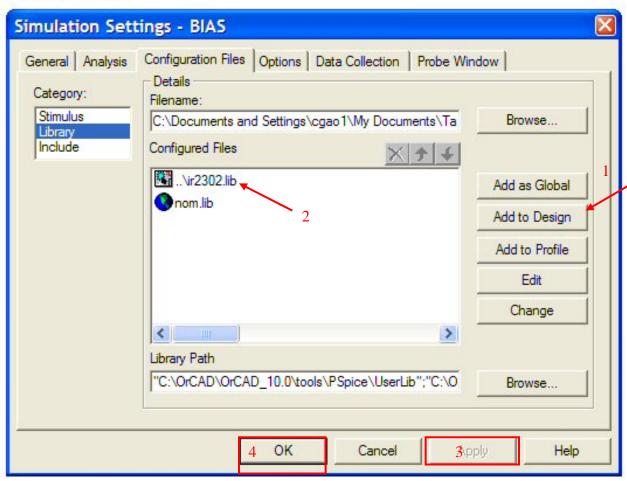
When the schematic is completed and ready to run simulation, go to PSpice menu => New Simulation Profile => Configuration Files tab => select "Library" from "Category" => Browse to the **IR2302.lib** then select the file and click "Add to Design" => Apply and OK.



\DOCUME... □□X Analog or Mixed A/D ?× **→ 1 1 1 1 1 1 1 1** Look in: | IR-lib Configuration Files Options | Data Collection | Probe Window | 🖬 ir2302.lib Category Filename: Stimulus Library Include My Recent Documents Configured Files X + 4 4 6 oil.mon Add as Global 2 Desktop Add to Design Add to Profile Edit My Documents Change 5 Library Path ly Compute "C:\OrCAD\OrCAD_16.0\tools\PSpice\UserLib";"C:\O My Network Places ir2302.lib • Open Help Files of type Library Files (*.lib) -Cancel

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

International Rectifier



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

The process is now complete.