

VIPER31 PSPICE Model Usage Instructions

The PSPICE model has been developed on OrCAD Capture v. 17.2. We recommend to use this version or higher to avoid convergence problems and speed up the simulation. The zip package contains two files:

- “VIPER31.OLB”, the symbol to place in the schematic.
- “VIPER31.LIB”, the library containing the real implementation of the model.

In order to use the model, create a new OrCAD project, add the VIPER31.OLB library to your project and select the part you want to simulate. Figure 1 shows the list of available parts of VIPER31.

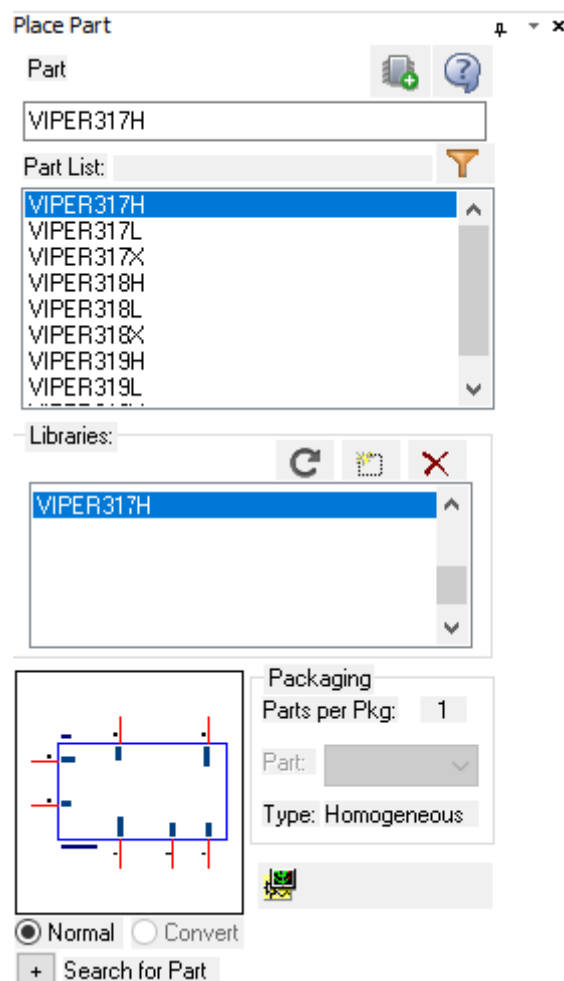


Figure 1 –VIPER31 models. Available parts.

Then add the library reference to your simulation profile, as shown in Figure 2.

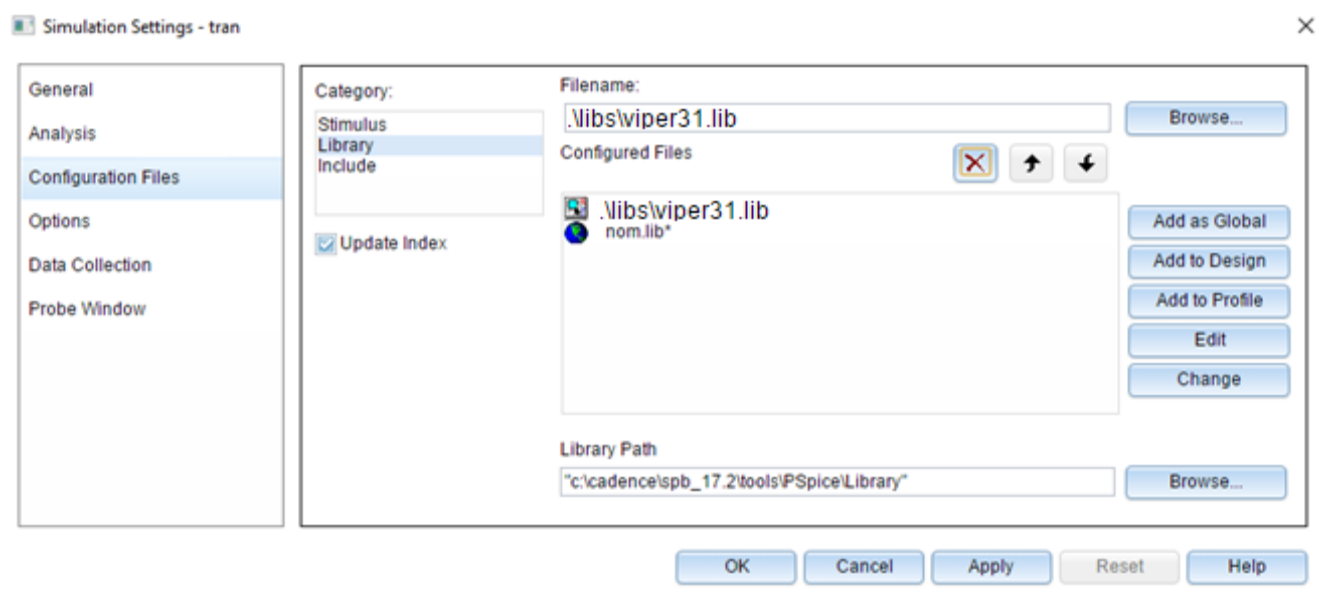
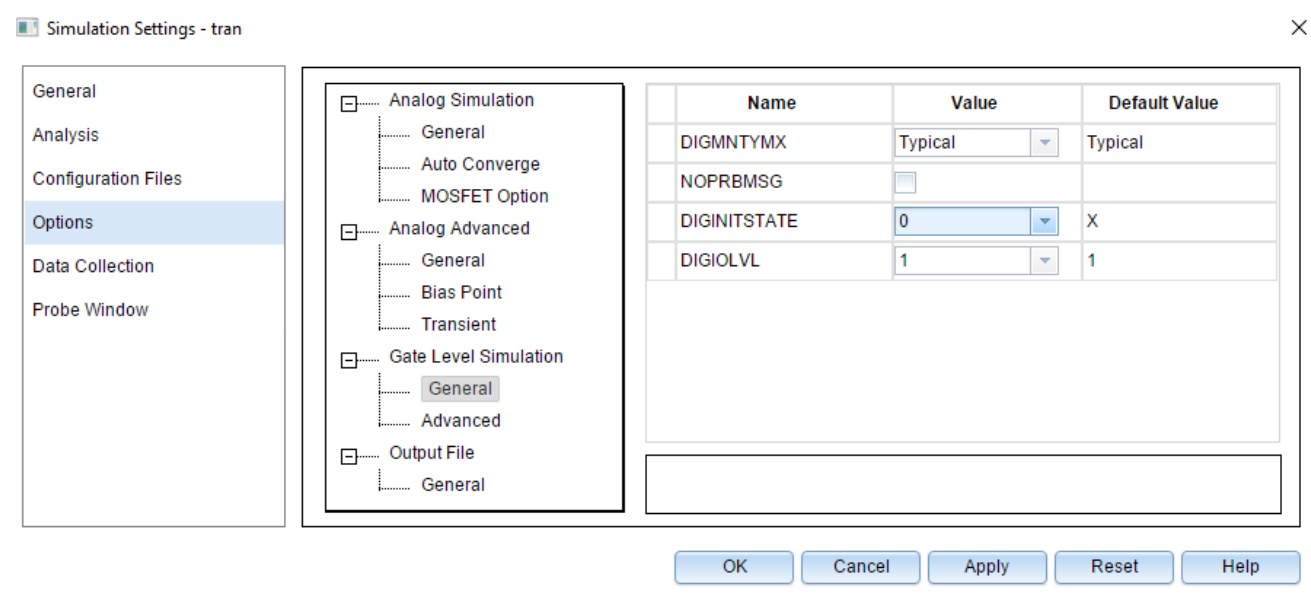


Figure 2 VIPER31 library added to the simulation profile

NOTE. Before starting the simulation, make sure that the option “DIGINITSTATE” is set to “0”.



In case of convergence problems, check the option “AutoConverge”.

Simulation Settings - tran
 ×

General
 Analysis
 Configuration Files
 Options
 Data Collection
 Probe Window

Analog Simulation
 General
 Auto Converge
 MOSFET Option

Analog Advanced
 General
 Bias Point
 Transient

Gate Level Simulation
 General
 Advanced

Output File
 General

	Name	Value	Default Value
☒	AutoConverge		
☒	ITL1	1000	1000
☒	ITL2	1000	1000
☒	ITL4	1000	1000
☒	RELTOL	0.05	0.05
☒	ABSTOL	1.0E-6	1.0E-6
☒	VNTOL	.001	.001
☒	PIVTOL	1.0E-10	1.0E-10
☒	Restart		

OK
 Cancel
 Apply
 Reset
 Help