

OrCAD PSpice Agenda. (2 Day Course)

1Day

- Schematic Design for PSpice Simulation
- PSpice Simulation Option Parameters
- New Simulation & Simulation Setting
- PSpice Analysis Type
 - (Bias Point, DC sweep, AC sweep, Transient)
- PSpice Optional Simulation
 - (Parametric, Monte-Carlo, Temperature)

2Day

- PSpice Model Editor
- Subcircuit Model Creation
- PSpice Advanced Analysis
 - (Sensitivity, Monte-Carlo, Optimizer, Smoke)

The training content and sequence may vary slightly for course efficiency.