OrCAD PSpice is Self Learning easy to understand series. This comes with LAB Files for almost all lessons, Software to Practice.

Lesson 1: Building a Design for Simulation

Lesson 2: DC Bias Point Analysis

Lesson 3: DC Sweep Analysis

Lesson 4: AC Sweep Analysis

Lesson 5: Simulating a Text Netlist

Lesson 6: Stimulus Editor

Lesson 7: Transient Analysis

Lesson 8: Resolving Simulation Errors

Lesson 9: Transformers

Lesson 10: Parametric Analysis

Lesson 11: Editing a Model

Lesson 12: Adding New Parts

Lesson 13: Temperature Analysis

Lesson 14: Monte Carlo Analysis

Lesson 15: Hierarchical Blocks and Symbols

Lesson 16: Analog Behavioral Modeling

Lesson 17: Simulating Digital Circuits

Lesson 18: Mixed Analog and Digital

Lesson 19: Measurements

Lesson 20: Worst Case Analysis

Lesson 21: Noise Analysis