1. Record Nr.   UNINA9910583378303321

   Autore   Fitzpatrick Dennis

   Titolo   Analog design and simulation using OrCAD Capture and PSpice / /
                      Dennis Fitzpatrick

   Pubbl/distr/stampa Oxford, England ; ; Cambridge, Massachusetts : , : Newnes, , 2018
                      ©2018

   ISBN   0-08-102506-8
                      0-08-102505-X

   Edizione   [Second edition.]

   Descrizione fisica 1 online resource (1 volume) : illustrations

   Disciplina   621.381502855369

   Soggetti   Electronic circuit design
                      Electronic circuits - Computer simulation

   Lingua di pubblicazione   Inglese

   Formato   Materiale a stampa

   Livello bibliografico   Monografia

   Note generali   Includes index.

   Sommario/riassunto New to this edition: Updated to using OrCAD Release 17.2 and its new
                      features; Coverage of PSPICE extra features: PSpice Designer, PSpice
                      Designer Plus, Modelling Application, PSpice Part Search Symbol Viewer,
                      PSpice Report, Associate PSpice model, New delay functions for
                      Behavioural Simulation Models, New Models, Support for negative
                      values in hysteresis voltage and threshold voltage; A new chapter on
                      PSpice Advanced Analysis Analog Design and Simulation Using OrCAD
                      Capture and PSpice, Second Edition provides step-by-step instructions
                      on how to use the Cadence/OrCAD family of Electronic Design
                      Automation software for analog design and simulation. The book
                      explains how to enter schematics in Capture, set up project types,
                      project libraries and prepare circuits for PSpice simulation. There are
                      chapters on the different analysis types for DC Bias point, DC sweep,
                      AC frequency sweep, Parametric analysis, Temperature analysis,
                      Performance Analysis, Noise analysis, Sensitivity and Monte Carlo
                      simulation. Subsequent chapters explain how the Stimulus Editor is
                      used to define custom analog and digital signals, how the Model Editor
                      is used to view and create new PSpice models and Capture parts and
                      how the Magnetic Parts Editor is used to design transformers and

inductors. Other chapters include Analog Behavioral models, Test Benches as well as how to create hierarchical designs. The book includes the latest features in the OrCAD 17.2 release and there are exercises with step by step instructions at the end of each chapter that enables the reader to progress based upon their experience and knowledge gained from previous chapters. The author worked for Cadence for over eight years and supported and delivered OrCAD PSpice training courses all over Europe. This book has been endorsed by Cadence. In addition, there are new chapters on the PSpice Advanced Analysis suite of tools: Sensitivity Analysis, Optimizer, Monte Carlo, and Smoke Analysis.The chapters show how circuit performance can effectively be maximised and optimised for variations in component tolerances, temperature effects, manufacturing yields and component stress. Provides both a comprehensive user guide and a detailed overview of simulation using OrCAD Capture and PSpice Includes worked and ready to try sample designs and a wide range of to-do exercises Covers Capture and PSpice together