1. Record Nr. UNINA9910812168603321

Autore	Asadi Farzin
Titolo	Electric and electronic circuit simulation using TINA-TI® / / Farzin Asadi
Pubbl/distr/stampa	Oxon, UK : , : River Publishers, , [2022] ©2022
ISBN	1-00-333279-X 1-000-77351-5 1-000-77346-9 1-003-33279-X 87-7022-685-7
Edizione	[1st ed.]
Descrizione fisica	1 online resource (458 pages)
Collana	River Publishers Series in Circuits and Systems
Disciplina	621.3815
Soggetti	Electronic circuits - Computer simulation Electronic circuits - Data processing
Lingua di pubblicazione	Inglese
Formato	Materiale a stampa
Livello bibliografico	Monografia
Nota di bibliografia	Includes bibliographical references and index.
Sommario/riassunto	A circuit simulator is a computer program that permits us to see circuit behavior, i.e. circuit voltages and currents, without making the circuit. Use of a circuit simulator is a cheap, efficient, and safe way to study the behavior of circuits. The Toolkit for Interactive Network Analysis (TINA) is a powerful yet affordable SPICE based circuit simulation and PCB design software package for analyzing, designing, and real time testing of analog, digital, VHDL, MCU, and mixed electronic circuits and their PCB layouts. This software was created by DesignSoft. TINA-TI is a spinoff software program that was designed by Texas Instruments (TI) in cooperation with DesignSoft which incorporates a library of pre-made TI components to for the user to utilize in their designs. This book shows how a circuit can be analyzed in the TINA-TI environment. Students of engineering (for instance, electrical, biomedical, mechatronics and robotics to name a few), engineers who work in industry and anyone who want to learn the art of circuit simulation with TINA-TI can benefit from this book.