

1. Record Nr.	UNINA9910627259503321
Autore	Asadi Farzin
Titolo	Essential Circuit Analysis using LTspice® / / by Farzin Asadi
Pubbl/distr/stampa	Cham : , : Springer International Publishing : , : Imprint : Springer, , 2023
ISBN	9783031098536 9783031098529
Edizione	[1st ed. 2023.]
Descrizione fisica	1 online resource (554 pages)
Disciplina	621.3815
Soggetti	Electronic circuits Electronics Electric power production Electronic Circuits and Systems Electronics and Microelectronics, Instrumentation Electrical Power Engineering
Lingua di pubblicazione	Inglese
Formato	Materiale a stampa
Livello bibliografico	Monografia
Nota di bibliografia	Includes bibliographical references and index.
Nota di contenuto	Simulation of electric circuits with Ltspice -- Simulation of electronic circuits with Ltspice -- Simulation of digital circuits with Ltspice -- Simulation of power electronics circuits with Ltspice.
Sommario/riassunto	This textbook provides a compact but comprehensive treatment that guides students through the analysis of circuits, using LTspice®. Ideal as a hands-on source for courses in Circuits, Electronics, Digital Logic and Power Electronics this text focuses on solving problems using market-standard software, corresponding to all key concepts covered in the classroom. The author uses his extensive classroom experience to guide students toward deeper understanding of key concepts, while they gain facility with software they will need to master for later studies and practical use in their engineering careers. Serves as a hands-on complement to texts for Electric Circuits I/II, Electronics I/II, Digital Logic and Power Electronics; Integrates use of LTspice® throughout; Filled with examples that students will see throughout the typical course, solved with market-standard software.

