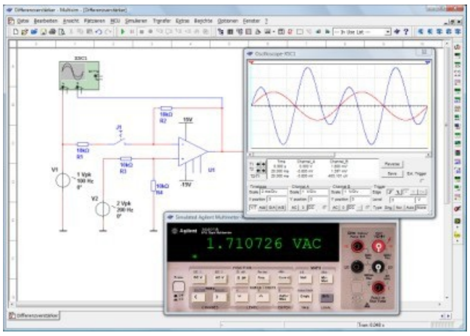
Item No.: SO2002-1A

Software, NI Multisim 12, single licence(D, GB)



Multisim from National Instruments is an intuitive software used to model circuitry with functions designed especially for training and educational purposes. Students can use the SPICE-simulation, now reputed to be industry standard for the interactive simulation of electronic circuitry, without having to master the ins and outs of SPICE-syntax and network lists. As such you can focus utterly and completely on the investigation of the circuitry and its behavior. With the assistance of interactive virtual instruments, including oscilloscopes, multimeters and current sensors, students are able to rapidly and easily project and measure circuit properties in a software environment. Multisim contains a comprehensive component database with over 10,000 elements that permits users to experiment on the most varied of circuit topologies in order to become more familiar with the operation of circuits

- Intuitive circuit modeling and powerful SPICE-simulation
- Expanded analysis like, e.g. Monte Carlo, temperature sweep, parameter sweep
- Interactive work with virtual instruments, including oscilloscopes, DMMs, function generators, characteristic recorders, etc.
- Database with thousands of models for the virtual realisation of analog and digital circuits
- Virtual plug-in assembly on pin board in 3D-environment prior to implementation in the lab

National Instruments Multisim software is subject to United States export control regulations.