t is critical to engage students with fundamental circuits topics in the classroom. With Multisim, students can access industry-standard SPICE simulation in an interactive, graphical learning environment to visualize and reinforce electronics theory. Multisim includes specialized teaching features, including more than 20 virtual measurement instruments to visualize circuit behavior, which students can

Multisim is an industry-standard, best-in-class SPICE simulation environment. It is the cornerstone of the NI circuits teaching solution to build expertise through practical application in designing, prototyping, and testing electrical circuits.

The Multisim design approach helps you save prototype iterations and optimize printed circuit board (PCB) designs earlier in the process.

exp