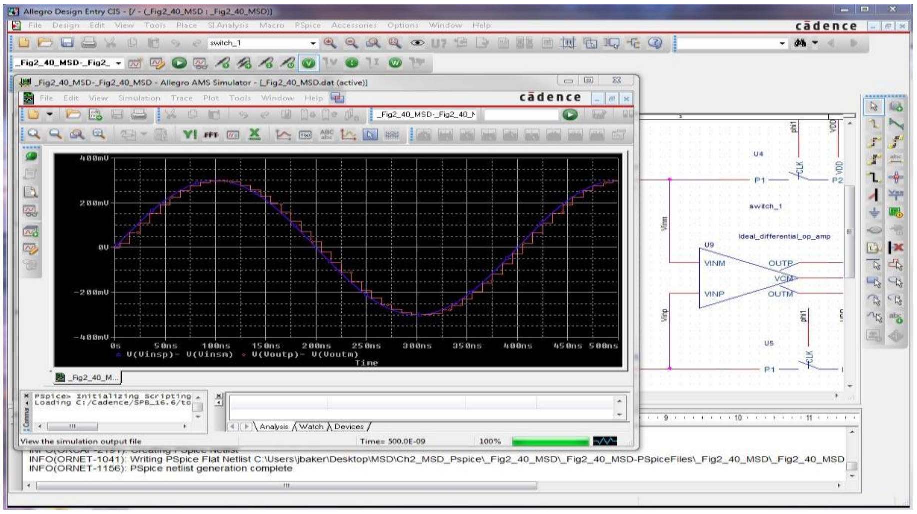
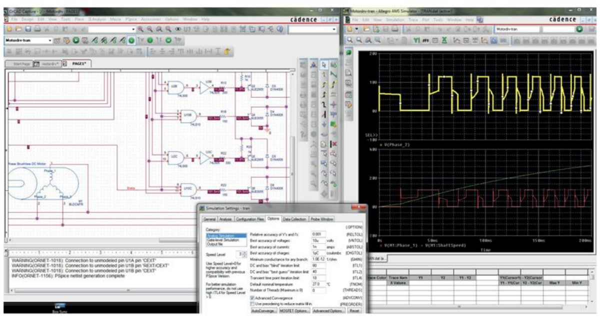


PSpice Session with Cadence



PSpice SPICE simulation technology combines industry-leading, native analog and mixed-signal engines to deliver a complete circuit simulation and verification solution. PSpice is used across a range of industries like Automotive, Power Supply and IoT –. In this session we will be covering topics including PSpice Modelling (SPICE Models and Model Types), Advanced Analysis (PSpice Advanced Analysis, SMOKE Analysis, PSpice Optimizer and Curvefit), System Option (PSpice System Option, MATLAB Visualization, MATLAB Modelling), PSpice and Allegro System Capture.

Please join us in a ½ day session covering PSpice where we will be using a combination of presentation and demonstrations on the day, and would welcome your participation either in person at Cadence, Velizy or via Webex.

cadence™

AGENDA

Introduction to PSpice and Orcad Capture

PSpice Modelling

Advanced Analysis

PSpice Systems Option

Integration of PSpice with Allegro Design Capture

Questions and Answers