Contribution ID: **87**

Type: **Poster Presentation**

Modeling Power Factor Correction Circuits with LT-Spice

Wednesday 6 June 2018 10:15 (15 minutes)

LT-Spice is a powerful simulation language that is specifically optimized for modeling Switch Mode Power conversion. It is not limited to small numbers of nodes and freely available. We are presenting several examples of simulations for popular electronic power factor correction circuits that improve the input power factor of AC Power Supplies by active wave-shaping of the AC input current and the associated avoidance of harmonics.

Authors: Prof. GIESSELMANN, Michael (Texas Tech University); ROY, Vishwajit (Texas Tech University)

Presenter: Prof. GIESSELMANN, Michael (Texas Tech University)

Session Classification: Oral 10 - Power Electronics