

Title: Circuit Analysis and Optimization

Lecturer: Prof. Dr. Tadej Tuma

Assistant: Assoc. Prof. Dr. Arpad Buermen

Aim of the course:

This is a compulsory course in the 1. semester of the Master's degree curriculum "Electronics". The aim is to introduce students to the theoretical background of analog circuit simulation. The course also involves laboratory work in the advanced field of circuit simulation and optimization with SPICE OPUS.

Required (pre)knowledge:

Basics in mathematics and analog circuits.

Lectures:

In three-hour weekly lectures the following themes are covered:

- Numerical methods in modern circuit simulation tools: Modified nodal equations, LU decomposition, Newton Raphson iterations for nonlinear circuits, integration algorithms for dynamic circuits, frequency domain and pole-zero analysis.
- Practical use of parametric optimization tools: Parameter identification, matching, cost function definition, overview of constrained and unconstrained optimization methods, penalty functions, design corners, mismatch analysis.

Laboratory work:

There are two-hour weekly sessions of laboratory work, where the following is covered:

- Introduction to the circuit simulator SPICE OPUS (group work).
- A comprehensive tutorial of all analyses types and output post-processing (group work).
- Introduction to the circuit optimization GUI with simple case studies (group work).
- The analysis or optimization of a selected analog case (individual project).

Examination:

The students have to complete and present their individual laboratory projects. Then they apply for an oral examination covering the lecture topics.

Selected references:

- T. Tuma, A. Buermen, Circuit Simulation with SPICE OPUS, Theory and Practice, Springer, 2009
- Webpage of analog circuit simulator SPICE OPUS (www.spiceopus.si)