

ECTS Document

	ECTS	STUDY TIME (HOURS)			Peroid (semester)
		Theory	Exercises Guided	Exercises Non-guided	
Modeling of RF devices and circuits	3	48	32	10	1

Area of study: Advanced Electronic System Design and Integration Technologies

Course topic: Modeling of RF devices and circuits

Study Program: Post Graduate Course

Number of Credits: 3

Study time: 88 hours self study

Period (Semester): Semester 2

Course responsible: Elissaveta Gadjeva

Course lecturers: Elissaveta Gadjeva

Objectives:

The aims of this module are:

- to develop an in-depth understanding of RF device modeling approaches;
- to develop an appropriate familiarity with computer methods and tools for investigation of RF circuit characteristics;
- to provide the students with a knowledge for computer-aided parameter extraction procedures of RF device models.

Contents:

A. Modeling of RF circuits

- Determination of S-parameters using PSpice-like simulators. RF circuit stability investigation based on k-factor and μ -factor determination. Stability analysis using PSpice simulation and macro-definitions in Probe. Gain determination. Application of PSpice simulator and post-processing in Probe to calculate the characteristics MAG and MSG;

- Modeling of high-power RF circuits. Determination of total output power, power efficiency and power added efficiency using PSpice. Investigation of distortions in RF circuit. Determination of -1 dB compression point and third-order intercept point using PSpice.

B. Modeling of passive elements

- Development of parameterized PSpice models of the passive elements;
- Development of PSpice models of planar spiral inductors and transformers taking into account geometry parameters.

C. Modeling of active elements. Nonlinear models and small-signal models.

D. Noise modeling of RF elements

- Development of parameterized frequency-dependent noise sources and correlated noise sources with complex correlation coefficients using PSpice.

E. Parameter extraction of equivalent circuits for passive and active RF elements

- Realization of direct extraction procedures for planar spiral inductor and transformer models using PSpice and Probe. Realization of direct extraction procedures for small-signal HBT and MOSFET models using PSpice and Probe.
- Parameter extraction for models of RF elements based on optimization procedures.

Teaching and learning method: e-learning and self study.

Prerequisite knowledge: Microelectronics – VLSI Part I, Microelectronics – VLSI Part II

Teaching and learning material: 6 e-learning tutorials for theory, 4 e-learning exercises; additional recommended references.

Evaluation method: Solutions of exercises must be sent to lecturer for the guided exercises.

Examination method: Unseen examination (50%) and coursework exercises (50%).

Recommended references:

1. Gilmore, R., Practical RF Circuit Design for Modern Wireless Systems. v. I. Passive Circuits and Systems, Artech House, 2003.
2. Gilmore, R., Practical RF Circuit Design for Modern Wireless Systems. v.II. Active Circuits and Systems, Artech House, Inc., 2003.
3. Bahl, I., Lumped Elements for RF and Microwave Circuits, Artech House, Inc., 2003.
4. Maas, S., Nonlinear Microwave and RF Circuits, Artech House, Inc., 2003.
5. Carr, J., RF Components and Circuits, Newnes, Ltd., 2002.
6. Cadence PSpice Reference Guide, Cadence Design Systems, Inc., 2000 (pspcref.pdf).
7. Cadence PSpice User's Guide, Cadence Design Systems, Inc., 2000 (pspug.pdf).