

Workshop Series for the Winter Semester 2021/2022

The IEEE Student Branch presents a series of workshops on topics, which are important for studies in any subject close to electrical engineering and thus, can be very useful. The tutorials and workshops will be held every two weeks on

Thursday 9.15 - 10.45 am (even Week)

G03-112

How to Write a Good Protocol

Dr.-Ing. Mathias Magdowski

Not only for laboratory work, but also for research projects and in technical jobs in general, a correct and clear documentation of experiments and measurements in protocols is crucial. The workshop will show how frequent mistakes can be avoided. Important rules and suitable software for writing protocols will also be shown.

21.10.

M. Sc. Hannes Schreiber

LaTeX for Beginners

The workshop serves as an introduction to the popular typesetting software to produce i.a. quality reports. The basics of working with LaTeX will be presented interactively. It is shown how a document is developed from scratch and how a consistent scientific document can be created by following simple rules of formatting.

4.11.

MATLAB - Crash Course

M. Sc. Eric Glende

In this workshop, a basic overview of the wide capabilities of the program MATLAB will be given. This includes examples and demonstrations to the topics "desktop basics", "mathematical operations and functions", "arrays and matrices", "programming in MATLAB" and "text and characters".

18.11.

M. Sc. Benjamin Hoepfner

Proper Plots with LaTeX

In scientific work, the presentation of results with the help of graphics is often decisive for the impression of the entire work. LaTeX offers numerous possibilities for the creation of high-quality vector graphics which are presented in this workshop.

2.12.

First Steps in Python

Dipl.-Ing. Max Rosenthal

Proficiency in a programming language is required in almost every engineering activity. In this workshop, first steps in dealing with Python as well as the available mathematical and graphic libraries are presented by using simple examples.

16.12

Dr.-Ing. Mathias Magdowski

Electric Circuit Simulation with LTspice

How to use a circuit simulator is something that every electrical engineer should be aware of, even if there is no specific university lecture about this. In this workshop, you will get used to LTspice, a freeware circuit simulator that is small and simple to use, yet very powerful in its applications. We will look at DC circuits, AC analyses and transient simulations in order to investigate different plain and complex circuits.

13.01.