Univ.-Prof. Dr. rer. nat. Wolfgang H. Müller
Technische Universität Berlin
Fakultät V – Institut für Mechanik
FG Kontinuumsmechanik und Materialtheorie
Sekretariat MS 2
Einsteinufer 5
10587 Berlin

Technische Universität Berlin
LKM

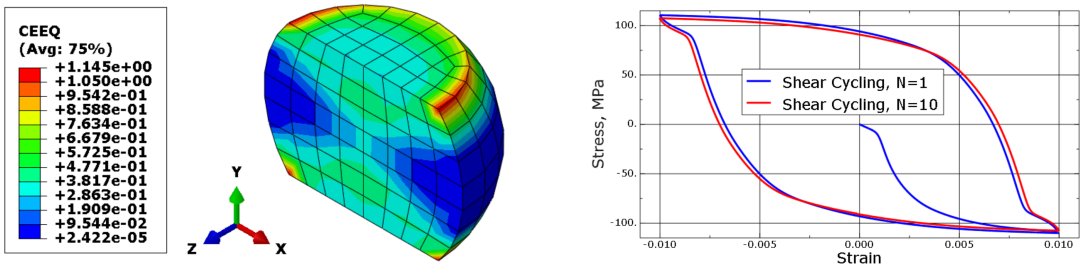
Module descriptor

– Summer term 2024 –

Kontinuumsphysikalische Simulationen

(**Numerical simulations of continuum mechanics problems**)

PJ 50527 L 046, 6 LP



Target audience:

This course – **held in English language** – is aimed at advanced students who are interested in the modeling of complex technical problems within the framework of continuum mechanics. This includes students of physical engineering science, mechanical engineering, aerospace, materials science, physics, and civil engineering at the end of their Bachelor studies or at the Master level. Students interested in the event should have good knowledge of mechanics from the lectures Mechanics I–III of the basic studies as well as some knowledge of tensor calculus. An advantage would be if you passed the following modules: Hands-on project to finite element analysis and Grundlagen der Kontinuumstheorie I-II. Furthermore, basic programming skills and basics knowledge of the FE-method is required.

Team

Lecturer	Dr. Aleksandr Morozov	MS 311	morozov@tu-berlin.de
Sekretariat	Grit Lamprecht	MS 08a	grit.lamprecht@tu-berlin.de

Several lectures will be given by an invited lecturer Assist. Prof. Sergei Khakalo from Aalto University, Espoo, Finland.

Lecture time

Mondays,	14:00 - 16:00,	MS 107,	starting 15.04.2024
Tuesdays,	10:00 - 12:00,	MS 210,	starting 16.04.2024

Course procedure and course content

Three blocks of lectures are separated by practical parts of the course. The lectures cover background information on continuum mechanics utilizing tensor calculus, constitutive models of creep with damage, basic concepts of finite element method, and an introduction to Abaqus user subroutines. This information is sufficient for successful processing of the projects and should bring all participants to a similar level of knowledge. The practical part of the course consists of regular homework assignments and an individual project. These will be done in groups of max. five persons. Advice will be given to the groups during consultation hours by the lecturers. The final presentation and subsequent oral exam takes place at the end of the course. Depending on the amount of students and, therefore, projects, some intermediate presentations might be arranged to exchange results between the project groups. More on the content and schedule can be found on the ISIS-webpage[1] of the course. To get access to the ISIS webpage and to enroll in the course the following password is required: **KontiSim24**

Learning Outcomes:

- ability to formulate a boundary value problem in strong and weak forms.
- ability to carry out analytical and numerical solutions for mechanical problems.
- hands-on experience to develop basic programming scripts in, *e.g.*, Python or Matlab for scientific and engineering needs.
- skills in using advanced modelling capabilities of Abaqus FEA software, such as Python scripting for Abaqus and writing user subroutines.
- Soft skills: work in teams, writing technical reports in LaTeX, creating presentation with LaTeX-beamer or MS Powerpoint.

Exam regulations

It is mandatory to pass the homework assignments and to submit the final report to participate in the oral examination.

The oral exam consists of a 15-minute presentation on the project's results and a subsequent 30-minute interview.

The final assessment of the group performance is based on the oral presentation, subsequent interview session, and the report in a ratio of 30:40:30. An overall performance of 50% is given a grade of 4.0. 95% of the maximum possible performance results in a grade of 1.0. In between is scaled linearly.

[1] https://isis.tu-berlin.de/course/view.php?id=37065