Module descriptor
– summer term 2020 –

Hands-on project to finite element analysis

PJ 0530 L 164 6 ECTS

Informations related to the Corona crisis:

Please note that the start of the lectures is postponed until 04/20/2020 and that TU Berlin has switched to an essential-only presence mode of operation on 03/20/2020. At the time of writing, it is not clear how and when the lectures will be held in the summer term 2020.

In order to establish a contact between the lecturers and the students, it is required that everyone registers on the ISIS webpage of this module. However, a password is required for the registration. Please send an e-mail to Sebastian Glane to obtain the password.

Target audience:

This course is part of the module „Mechanische Eigenschaften der Werkstoffe – MEW.“ Furthermore, this course addresses students majoring in mechanical engineering, aero space engineering, material sciences, mathematics, transport systems, physics and engineering sciences.

Team:

lecturer Prof. Dr. rer. nat. Wolfgang H. Müller MS 09 ph.: 314-27682
secretary S. King MS 08a ph.: 314-22332
teaching assistants Sebastian Glane, M.Sc. MS 311 ph.: 314-21151
Gregor Ganzosch, M.Sc. MS 308 ph.: 314-23457
Lecture time:

lectures, tutorials  
Monday, 4 to 6 p.m. (c.t.), MS 107, starting 04/20/2020  
Friday, 2 to 4 p.m. (c.t.), MS 107, starting 04/24/2020

Learning outcomes:

• Obtaining background information on advanced strength of materials theory,
• handling commercial finite element software,
• solving a complex stress analysis problem,
• soft skills: solving engineering problems collaboratively in teams, presenting and documenting results.

Course procedure and exam regulations:

In this course, a stress and fatigue analysis of microelectronic components shall be performed using the finite element software Abaqus. In the first 4 to 5 weeks, conventional lectures and tutorials teach the following topics:

• Introduction to manufacturing technology in modern microelectronics (Surface Mount Technology, SMT),
• introduction to materials used in SMT,
• basic mechanics of elastoplasticly deformable bodies,
• description directionally depend materials,
• basic concepts of plasticity and creep,
• fatigue and lifetime analysis.

The lectures are accompanied by homework exercises. In these homework exercises, the finite element program Abaqus is learned and used. At the end of the lecture series, a midterm screening is performed. A maximum of 20 points can be obtained. For a qualified participation in a project, it is necessary to obtain a minimum of 10 points in the midterm screening and to complete the homework exercises. Otherwise, we strongly advise to participate in the project in a later semester.

During the following 5 to 6 weeks, the students solve a given stress and fatigue analysis problem for a SMT-component using Abaqus. This task is performed in groups of 5 persons maximum. The groups are formed after the midterm screening. Advices regarding the project task are given during consultation hours. The groups themselves have to ensure that the work load is evenly distributed among the group members.

It is mandatory to submit a project report in the form of a scientific paper in order to take the oral exam at the end of the lecture period. The oral exam consists of a 15 minutes presentation on the project’s results and a subsequent 15 minutes interview.

The assessment of the student’s performance and grades is entirely based on the result of the oral exam.