# COURSE DESCRIPTION CARD - SYLLABUS

Course name
Computer analysis of mechatronic structures [S1Mech2>KAKM]

## Course

| | |
|---|---|
| Field of study<br>Mechatronics | Year/Semester<br>3/6 |
| Area of study (specialization)<br>– | Profile of study<br>general academic |
| Level of study<br>first-cycle | Course offered in<br>Polish |
| Form of study<br>full-time | Requirements<br>elective |

## Number of hours

| Lecture | Laboratory classes | Other |
|---|---|---|
| 15 | 30 | 0 |
| **Tutorials** | **Projects/seminars** | |
| 0 | 0 | |

## Number of credit points

4,00

## Coordinators

dr inż. Dominik Wojtkowiak
dominik.wojtkowiak@put.poznan.pl

## Lecturers

## Prerequisites

Knowledge of technical mechanics, material strength, basics of machine construction and drive systems, basics of the finite element method. Single-handedly formulation of technical problems, operation of 2D drawing and 3D modeling programs, reading technical documentation of constructions, strength calculations of machine elements, shaping design features of machine components. Understanding the necessity of expanding one's competencies, readiness to engage in discussions about the analyzed construction.

## Course objective

The aim of the subject is to acquire the skills of building simulation models enabling strength, stiffness, and functional analysis of designed mechanical and mechatronic constructions and their implementation in the ABAQUS program.

## Course-related learning outcomes

Knowledge:
Knows the structure of the FEM simulation model and the influence of individual model elements on the results of computer analysis of constructions.

Knows various types of computer analyses and their application in engineering practice.

Skills:
Can build a simulation model reflecting the actual conditions acting on the construction or occurring during the technological process.
Can implement the model in a chosen simulation environment.
Can analyze the results obtained from computer simulations and apply them in the design process, as well as prepare a report on the conducted simulation studies.

Social competences:
Understands the need for lifelong learning; can inspire and organize the learning process for others.

## Methods for verifying learning outcomes and assessment criteria

Learning outcomes presented above are verified as follows:

Lectures:
Written assessment verifying theoretical skills in building a simulation model of a given structural example. The assessment takes place in the last lecture and lasts 45 minutes.
Assessment criteria: the correctness of the simulation model construction is assessed (75% of the grade) and knowledge of basic notions used in construction analysis (25% of the grade).
Grading scale: below 50% 2.0, from 50% 3.0, from 60% 3.5, from 70% 4.0, from 80% 4.5, from 90% 5.0.
Laboratories:
Assessment in the form of verification of practical skills in building a simulation model of a given structural example and operating the ABAQUS program. The assessment takes place during the last laboratory classes (15) and lasts 90 minutes. Ongoing verification of acquired skills during independent analysis of selected design solutions (laboratory classes 13-14).
Assessment criteria: the correctness of the construction of the simulation model is assessed (75% of the grade) and its implementation in the ABAQUS program (25% of the grade).
Grading scale: below 50% 2.0, from 50% 3.0, from 60% 3.5, from 70% 4.0, from 80% 4.5, from 90% 5.0.

## Programme content

Construction of the FEM simulation model, simplifying assumptions in computer analysis of structures, modeling material properties, defining boundary conditions and loads, discretization of parts in FEM, static strength and stiffness analysis, dynamic analysis, temperature-displacement analysis, analysis of natural frequency vibrations, material failure analysis, simulation of phenomena in mechatronic devices.

## Course topics

Lectures:
Lecture 1 (2) - Introduction to CAE
Familiarization with the interface and capabilities of the ABAQUS/CAE preprocessor module. Visualization of obtained data in the ABAQUS/Viewer postprocessor. Creating 2D and 3D models in the ABAQUS program. Importing geometry from other 3D CAD software. Setting simulation parameters.
Lecture 2 (2) - Construction of the FEM simulation model
Construction of the FEM simulation model. Modeling engineering materials in computer simulation. Defining the geometry of the structure. Defining boundary conditions, loads, and contacts. Selection of the type of finite elements. Mesh control.
Lecture 3 (2) - Strength and stiffness analysis of the mechatronic systems parts
Static strength analysis of individual machine elements and assemblies (assemblies) of mechatronic structures. Static stiffness analysis of measurement system structures.
Lecture 4 (2) - Contact in FEM analysis
Defining contacts between machine elements in the ABAQUS program. Stress analysis in typical shape connections (threaded, key, spline, and pin connections) and in bearings.
Lecture 5 (2) - Dynamic analysis ABAQUS/Explicit
Dynamic analysis of mechatronic structures. Impact analysis on a metal rod wave phenomena.
Lecture 6 (2) - Modeling composite structures using the finite element method
Lecture 7 (2) - Modeling of piezoelectric and electromagnetic components
Lecture 8 (1) - Written assessment of theoretical verification of skills in building a given simulation model construction example.

Laboratories:
Laboratory 1-2 (4) - Modeling methods in FEM analysis of 3D elements
Comparison of solid models (task 1), beam models (task 2) and shell models (task 3) on the example of analysis static cantilever beam.
Laboratory 3 (2) - Simplifications used in structure analysis
Plane state of stress and symmetry in the model - analysis of a strain gauge beam (task 4).
Analysis of axisymmetric elements - analysis of a compressed sleeve (task 5).
Laboratory 4 (2) - Static analysis of assemblies
Static analysis of a latch made of plastic (task 7)
Laboratory 5-6 (4)- Analysis of the operation of drive system components
Static analysis of the Cardan shaft (task 8).
Laboratory 7-8 (4) - Dynamic analysis of device operation
Testing the behavior of a car joint (task 9)
Laboratory 9 (2) - Temperature and displacement analysis
Braking analysis of a car disc (task 10)
Laboratory 10 (2)- Vibration analysis
Testing the natural frequencies of the mass-spring-damper system (task 11)
Laboratory 11 (2) - Analysis of technological processes to determine the workforce
Testing the crushing process of an aluminum can (task 12).
Laboratory 12 (2)- Analysis of material failure
Simulation of material cracking using the XFEM method (task 13).
Simulation of material puncture using finite element removal (task 14).
Laboratory 13-14 - Analysis of selected design solutions
Application of knowledge and skills acquired during laboratory classes 1-12 to analyze selected design solutions. Preparing a report on the analyzes performed.
Laboratory 15 - Assessment
Assessment in the form of verification of practical skills in building a simulation model of a given construction example and using the ABAQUS program.

## Teaching methods

Lecture: Lecture with a multimedia presentation, using the case study method - analysis of solutions to real structural problems.
Laboratory: Workshop methods of practical laboratory classes at computer stations - laboratory classes 1-12. Case study analysis of real structural solutions (independent student work with support from the instructor) - classes 13-14.

## Bibliography

Basic:
1. Dębski P., Ponieważ G., Różyło P., Wójcik A.: Podstawy metody elementów skończonych - przykłady obliczeń numerycznych w programie Abaqus. Wyd. Politechniki Lubelskiej, Lublin 2015.
2. Różyło P., Dębski H.: Metoda elementów skończonych. Praktyczne przykłady zagadnień statycznych i dynamicznych w programie Abaqus. Część 1. Wyd. Politechniki Lubelskiej, Lublin 2020.
3. Różyło P., Dębski H.: Metoda elementów skończonych. Praktyczne przykłady zagadnień statycznych i dynamicznych w programie Abaqus. Część 2. Wyd. Politechniki Lubelskiej, Lublin 2020.
4. Różyło P.: Modelowanie struktur kompozytowych z wykorzystaniem metody elementów skończonych w programie Abaqus, Wyd. Politechniki Lubelskiej, Lublin 2023.

Additional:
1. Szturmowski B.: Inżynierskie zastosowanie MES w problemach mechaniki ciała stałego na przykładzie programu ABAQUS. Wyd. Akademii Marynarki Wojennej, 2013
2. Skrzat A.: Modelowanie liniowych i nieliniowych problemów mechaniki ciała stałego i przepływów ciepła w programie ABAQUS. Wyd. Politechniki Rzeszowskiej, 2014
3. Chróścielewski J., Burzyński S., Daszkiewicz K., Sobczyk B., Witkowski W.: Wprowadzenie do modelowania MES w programie ABAQUS. Wyd. Politechniki Gdańskiej, Gdańsk 2014.

## Breakdown of average student's workload

| | Hours | ECTS |
|---|---|---|
| Total workload | 100 | 4,00 |
| Classes requiring direct contact with the teacher | 45 | 2,00 |
| Student's own work (literature studies, preparation for laboratory classes/ tutorials, preparation for tests/exam, project preparation) | 55 | 2,00 |