THE DOCTORAL SCHOOL OF IPPT PAN

COURSE OFFERED IN THE DOCTORAL SCHOOL OF IPPT PAN

Name of the course	Polish	**Mechanika ciał odkształcalnych z wykorzystaniem programu Abaqus**				
	English	**Applied Continuum Mechanics with Abaqus**				
Type of the course	**Specialized course**					
Course coordinator	**Ph.D. Mech. Eng. Marcin Nowak**		Course teacher	**Ph.D. Mech. Eng. Marcin Nowak**		
Implementing unit	**ZMM**		Scientific discipline / disciplines	**Mechanical engineering**		
Level of education	**doctoral studies**		Semester	**summer or winter**		
Language of the course	**English or Polish**					
Type of assessment	**project**		Number of hours in a semester	**30**	ECTS credits	**3**

Type of classes		Lecture	Auditory classes	Project classes	Laboratory	Seminar
Number of hours	in a week	**1**	**0**	**0**	**1**	**0**
	in a semester	**15**	**0**	**0**	**15**	**0**

1. Prerequisites

Basic knowledge of Mechanical/Civil engineering. Understanding the concepts of stress and strain, the continuum hypothesis and fundamental equations of continuum mechanics. Basic knowledge of Finite Element Method. Abaqus Learning Edition program (available free of charge to anyone).

2. Course objectives

During the lectures, we will explore the practical application of acquired knowledge in the field of continuum mechanics through numerical simulations. The primary objective of this course is to introduce students to the Abaqus program, utilizing a series of computational examples as learning tools. Abaqus is a powerful finite element analysis software program used for simulating various types of engineering and scientific problems. It is widely used in industry and academia for solving complex simulations. Abaqus includes a range of capabilities for modelling and analysing structures, fluids, thermal effects, and electromagnetic fields, among other physics.

3. Course content (separate for each type of classes)

Lecture

Main topics:

- Introduction: syntax and conventions, model definition, job execution, documentation, a quick review of the finite element method
- Material models: linear and nonlinear elastic models, plasticity models, viscoelasticity models, progressive damage and failure, heat transfer properties
- Linear dynamics: modal-based solutions, extracting real eigenvalues, damping
- Nonlinearity: sources of nonlinearity, the solution of nonlinear problems
- Contact: features of general contact, defining contact, limitations of general contact, contact pairs,
- Rigid body definition, analytical rigid surface, rigid elements, rigid body constrains

• Finite elements: continuum elements, structural elements and rigid elements • Running simulation:, use of Grafen computing cluster in IPPT PAN • Analysis techniques: Eulerian analysis, particle methods, design sensitivity analysis, parametric studies • Viewing the output from analysis: Reading the output database, customizing a model plot, displaying the deformed model shape, animating a contour plot • Python scripting: Python language basics, automation of the model creation process • Extension of the Abaqus: Abaqus User Subroutines, creating your own mechanical constitutive behaviour of a material
Laboratory
Main topics: • Hello World in Abaqus: Introduction to Abaqus GUI, scripting, and running a basic simulation. • Defining and implementing linear, nonlinear, plasticity, and viscoelastic material models. • Extracting eigenvalues, mode shapes, and performing modal-based dynamic analysis. • Understanding and solving geometric, material, and contact nonlinear problems. • Defining general contact, contact pairs, friction models, and diagnosing contact issues. • Creating rigid body constraints, analytical rigid surfaces, and using rigid elements in simulations. • Understanding and choosing appropriate continuum, structural, and rigid element types. • Running Abaqus simulations locally and on the Grafen computing cluster. • Extracting, visualizing, animating, and exporting simulation results. • Automating model creation, result analysis, and implementing user subroutines in Abaqus.

4. Learning outcomes			
Number of the learning outcome	Learning outcomes description	Reference to the learning outcomes according to the 8th level of PRK	Learning outcomes verification methods*
Knowledge			
1	The graduate acquires fundamental knowledge in performing numerical simulations in the Abaqus environment.	P8S_WG	project evaluation
2	The graduate acquires knowledge about basic types of numerical simulations and the influence of adopted assumptions on the calculation results.	P8S_WG	project evaluation
3	The graduate knows how to transfer acquired knowledge to the industrial sphere and disseminate the results of his research.	P8S_WK	assessment of activity during classes
Skills			
1	The graduate is able to use the Abaqus program to perform basic simulations in the field of deformable body mechanics.	P8S_UW	project evaluation
2	The graduate is able to determine the influence of simulation parameters on the final results.	P8S_UW	assessment of activity during classes

3	The graduate knows how to transfer the acquired knowledge to the industrial sphere and disseminate the results of his research.	P8S_UW	assessment of activity during classes
Communication			
1	The graduate is able to communicate on specialized topics relevant to the represented scientific discipline.	P8S_UW	assessment of activity during classes
2	The graduate is able to plan and carry out a research project in a methodologically correct manner.	P8S_UO	assessment of activity during classes
Social competences			
1	The graduate is ready to think and act in a creative and entrepreneurial manner.	P8S_KO	assessment of activity during classes
2	The graduate is ready to critically evaluate the achievements of the represented scientific discipline, including their own contribution to its development.	P8S_KK	assessment of activity during classes

*Allowed learning outcomes verification methods: exam; oral exam; written test; oral test; project evaluation; report evaluation; presentation evaluation; active participation during classes; homework; tests

5. Assessment criteria
assessment of activity during classes, project evaluation

6. Literature

Primary references:

[1] **Michał Kleiber, Piotr Kowalczyk**, Introduction to Nonlinear Thermomechanics of Solids, Spring 2018

[2] **Thomas Mase,** Continuum Mechanics for Engineers (1999)

[3] Abaqus User Manual 2024

Secondary references:

[1] **Niels Saabye Ottosen, Matti Ristinmaa** - The Mechanics of Constitutive Modeling-Elsevier Science (2005)

7. PhD student's workload necessary to achieve the learning outcomes**		
No.	Description	Number of hours
1	Hours of scheduled instruction given by the lecturer in the classroom	**30**
2	Hours of consultations with the lecturer, exams, tests, etc.	**15**
3	Amount of time devoted to the preparation for classes, preparation of presentations, reports, projects, homework	**25**
4	Amount of time devoted to the preparation for exams, test, assessments	**15**
	Total number of hours	**85**
	ECTS credits	**3**

** 1 ECTS = 25–30 hours of the PhD students work (2 ECTS ≈ 60 hours; 4 ECTS ≈ 110 hours, etc.)