Advanced numerical modeling using OpenFOAM

Faculty of	**Mechanical and Power Engineering**
Name in English	**Advanced numerical modeling using OpenFOAM**
Name in Polish	**Zaawansowane modelowanie numeryczne w środowisku OpenFOAM**
Main field of study	**Power Engineering**
Specialization	**-**
Level of studies	**II level**
Form of studies	**full-time**
Kind of subject	**optional-specialization**
Subject code	**W09ENG-SM2343**
Group of courses	**NO**

	Lecture	Classes	Laboratory	Project	Seminar
Number of hours of organized classes in University (ZZU)	15		30		
Number of hours of total student workload (CNPS)	50		50		
Form of crediting	Zaliczenie		Zaliczenie		
For group of courses mark final course with (X)					
Number of ECTS points	2		2		
including number of ECTS points for practical (P) classes			2		
including number of ECTS points for direct teacher-student contact (BU) classes	0,68		1,36		

PREREQUISITES RELATING TO KNOWLEDGE, SKILLS AND OTHER COMPETENCES

1.	Preliminary knowledge related to numerical modelling including basic discretization schemes.

SUBJECT OBJECTIVES

C1	Introduction to the OpenFoam numerical toolbox and OpenFoam programming.
C2	Introduction to basic and advanced numerical models implemented in OpenFoam.
C3	Developing skills to define new mathematical and numerical models.
C4	Developing skills to implement new numerical and mathematical models in OpenFoam.
C5	

SUBJECT LEARNING OUTCOMES

relating to knowledge:	
PEU_W01	knows and understands basics of finite volume discretization and its specifics in the Computational Fluid Dynamics
PEU_W02	knows and understands the structure of OpenFoam numerical toolbox and basics of OpenFoam programming
PEU_W03	knows and understands advanced numerical models including: conjugate heat transfer, flow with mixing and reactions
PEU_W04	knows and understands dynamics mesh concepts including Arbitrary Mesh Interface
PEU_W05	knows and understands basic and more complex boundary conditions
relating to skills:	
PEU_U01	is able to: use the basic and advanced numerical models offered by the OpenFOAM

PEU_U02	is able to: use the basic and advanced preprocessing and postprocessing utilities offered by the OpenFoam environment
PEU_U03	is able to: use the ParaView software to visualise the numerical data
PEU_U04	is able to: implement new equations and to develop the exiting OpenFoam solvers
PEU_U05	is able to: implement new numerical models in the OpenFoam
PEU_U06	is able to: define numerical models with dynamic mesh
PEU_U07	is able to: define numerical multiphase numerical models

PROGRAMME CONTENT

Form of classes - lecture		Number of hours
Wy1	Summary of conservation equations and their representation by partial differential equations. Introduction to OpenFoam software.	2
Wy2	Introduction to OpenFoam software including: the OpenFoam structure, available numerical models and basic pre- and post-processing utilities.	2
Wy3	Thermodynamic and transport models available in OpenFoam. Introduction to conjugate heat transfer modelling and detailed presentation of a corresponding flow example.	2
Wy4	Introduction to modelling of flow with multiple spices and flow with chemical reactions. Detailed presentation of a corresponding numerical example based on lecturer own research.	2
Wy5	Introduction to Finite Volume Method and basic methods of discretization used in CFD.	2
Wy6	Introduction to OpenFoam programming #1: implementation of Burgers equation. Discussion of consequences of non-linear nature of the Burgers equation.	2
Wy7	OpenFoam programming #2: adding a new PDE equation to an existing solver, compilation and usage.	2
Wy8	OpenFoam programming #3: implementation of variable viscosity in an existing model, explanation of definition of new numerical model with the implemented viscosity model. Discussion of influence of the new model on the flow.	2
Wy9	Introduction to complex boundary condition.	2
Wy10	OpenFoam programming #4: creation of a new numerical library by definition and implementation of Casson viscosity model. Presentation of its usage and consequences in the flow.	2
Wy11	Introduction to turbulence and turbulence modelling: RANS equations, Eddy viscosuty.	2
Wy12	Introduction to the concept of wall functions and basics of their implementation.	2
Wy13	Modelling of wind power plant: linear and angular momentum theory. Introduction to usage of additional source terms in numerical modelling.	2
Wy14	OpenFoam programming #5: definition and implementation of heat transfer in superfluid helium	2
Wy15	Final test of the lecture	2
Suma godzin		30

laboratory		Number of hours
La1	Introduction to the OpenFoam, discussion of its structure and installation. First Example: Lid-driven cavity flow and flow in a channel.	2
La2	Usage of complex boundary conditions (derived from basic BCs) based on T-junction example.	2
La3	Usage and definition of conjugate heat transfer model. Running calculation in parallel mode.	2
La4	OpenFoam programming #1: adding a new PDE equation to existing solver, compilation and usage.	2
La5	OpenFoam programming #2: step-by-step implementation of a variable viscosity in an existing solver. Exercise: modelling of a flow of two miscible fluids with different viscosities.	2
La6	Definition of a numerical model with dynamic mesh based on a sloshing tank example and a rigid body motion.	2
La7	Dynamic mesh motion using Arbitrary Mesh Interface	2
La8	OpenFoam programming #3: step-by-step implementation of a new viscosity model (Casson model).	2
La9	Usage of the new viscosity model implemented during Lab 8 in a flow in channel and comparison of the results with different viscosity models.	2
La10	Definition and running of numerical model of a flow of reacting mixture including: combustion, reactions and mixing.	2
La11	Adapting the reactingFoam for a passive gases mixing.	2

La12	Definition of a numerical model of helium discharge and its propagation in a tunnel.	2
La13	Adding new physical models as source terms (using fvOptions). Modelling of wind turbine as Actuation Disc Source.	2
La14	Adding new physical models as source terms (using fvOptions). Modelling of porosity.	2
La15	Presentation of individual student's projects.	2
Suma godzin		30

TEACHING TOOLS USED	
N1	Traditional lecture with a use of slides
N2	Laboratories – computational exercises
N3	Laboratories – individual solution of numerical problems using OpenFoam
N4	Consultation
N5	Independent work – individual study , homework and final project

EVALUATION OF SUBJECT EDUCATIONAL EFFECTS ACHIEVEMENT

Evaluation (F– forming (during semester), C– concluding (at semester end)	Educational effect number	Way of evaluating educational effect achievement
F1	PEK_W01 -- PEK_W05	Final test
C1	PEK_W01 -- PEK_W05	Homework
C2	PEK_U01 -- PEK_U07	Exercises during laboratory classes

PRIMARY AND SECONDARY LITERATURE

Primary literature	
1	F Moukalled, L Mangani, M Darwish; The Finite Volume Method in Computational Fluid Dynamics An Advanced Introduction with OpenFOAM and Matlab
2	User Guide, Tutorial Guide, Programmers Guide, https://www.openfoam.com/documentation/
Secondary literaturę	
1	OpenFOAM wiki: http://openfoamwiki.net/index.php/Main_Page
2	J Ferziger, M Peric, R Street; Computational Methods for Fluid Dynamics

SUBJECT SUPERVISOR (NAME AND SURNAME, E-MAIL ADDRESS)

Imię i nazwisko:	Ziemowit Malecha
E-mail:	Ziemowit.malecha@pwr.edu.pl