module title: **COMPUTATIONAL FLUID DYNAMICS**		
field of study: **Mechanical Engineering**	type of study: **full-time**	course code: **S6_1-3**
course: **Modelling & Simulation in Mechanics**	degree: **Master (MSc)**	year: **I** semester: **II**
type of classes: **lecture, project, EXAM**	hours per week: **3L, 3P**	No of ECTS credits: **5**

MODULE DESCRIPTION

TARGETS

T1. Provide theory of numerical methods and algorithms used in CFD.
T2. Provide theory of fluid flow modelling.
T3. To acquire capabilities to perform numerical analysis of the flow field.

PREREQUISITES & ADDITIONAL REQUIREMENTS

R1. Fundamentals of mechanics, thermodynamics and fluid mechanics.
R2. Knowledge on algebra calculus and basics of numerical methods.
R3. Safety rules during the use of laboratory equipment.
R4. Capability of using source literature.
R5. Capability of individual work and collaboration in a group.
R6. Data analysis and presentation of results.

LEARNING OUTCOMES

LO1. Knowledge on simulation strategies for incompressible and compressible flows.
LO2. Knowledge on advanced discretization methods.
LO3. Knowledge on mesh generation methods.
LO4. Knowledge on time integration approaches and stability analysis methods.
LO5. Knowledge on usage ANSYS software (Workbench, ICEM CFD, Fluent).
LO6. Ability to perform complete flow simulation: from geometry plots to flow visualisation.
LO7. Ability to report, analyse, interpret and present the simultion results.

TEACHERS

module coordinator: dr Andrzej Bogusławski, assoc. prof. - abogus@imc.pcz.czest.pl
academic teachers:
- dr Andrzej Bogusławski, assoc. prof. - abogus@imc.pcz.czest.pl
- dr Artur Tyliszczak, assoc. prof. - atyl@imc.pcz.czest.pl
- dr Maciej Marek - marekm@imc.pcz.czest.pl

MODULE CONTENT

LECTURE	hours
L 1-6 - Governing equations for fluid flow: Euler and Navier-Stokes equations, energy equation and equations for scalar fields. Vorticity-stream function formulation. Cylindrical and spherical co-ordinate systems.	**6**
L 7-9 - Specification of the boundary conditions for incompressible and compressible flows. Compatibility relations.	**3**
L 10-12 - Methodology / strategy of solving flow problems.	**3**
L 13-18 - Collocated, staggered and half-staggered meshes.	**6**
L 19-21 - Structured and unstructured meshes.	**3**
L 22-27 - Finite volume / finite difference methods.	**6**
L 28-33 - High-order and spectral/pseudo-spectral methods.	**6**
L 34-39 - Time integration methods. Stability analysis for time dependent problems.	**6**
L 40-42 - SIMPLE type methods, projection method, auxiliary potential method for pressure calculation.	**3**
L 43-45 - Solution efficiency, multigrid approach, parallel computations.	**3**

PROJECT	hours
P 1-15 - Using the ANSYS WORKBENCH software. Mesh generation in ANSYS ICEM CFD software. Assessment of the mesh quality. Validation of the solution methodology / accuracy using simple test cases (jet flows, channel flows, cavity flows). Comparison of the discretization methods.	**15**
P 16-24 - Using of the User Defined Functions (UDF). Elements of C programming for UDF's.	**9**
P 25-30 - Scripts and journals for automation of the solution procedures. Parallel computations on PC clusters.	**6**
P 31-45 - Flow modelling for selected turbulent flow problems: turbulent jets, turbulent flames, bluff-bodies, turbine blades, etc.	**15**

TEACHING TOOLS

1 - lecture with the use of multimedia presentations
2 - instructions to laboratory exercises
3 - computer laboratory, software for fluid flow simulation

SOURCE LITERATURE

1. Ferziger J.H.: Computational Methods for Fluid Dynamics, Springer, 1996
2. Fletcher C.A.J.: Computational Techniques for Fluid Dynamics, Springer-Verlag, 1991
3. Patankar S. V.: Numerical Heat Transfer and Fluid Flow, McGraw-Hill Book, 1980
4. User guide for ANSYS Fluent software
5. Mathieu J., Scott J.: An introduction to turbulent flows. Cambridge University Press, 2000
6. Kundu P., Cohen I.: Fluid mechanics. Academic Press, 2010
7. Pope S.: Turbulent flows. Cambridge University Press, 2000