(*faculty stamp*)

Z1-PU7	WYDANIE N1	Strona 1 z 2

COURSE DESCRIPTION

1. Course title: Computational Fluid Dynamics	**2. Course code**
3. Validity of course description: 2012/2013	
4. Level of studies: 2nd cycle of higher education	
5. Mode of studies: intramural studies	
6. Field of study: Industrial and Engineering Chemistry	RCH
7. Profile of studies: Process Engineering for Green Chemical Technologies	
8. Programme: general	
9. Semester: 2	
10. Faculty unit teaching the course: Department of Chemical and Process Engineering	
11. Course instructor: Marcin Lemanowicz, Ph.D., assistant	
12. Course classification: field	
13. Course status: compulsory	
14. Language of instruction: English	
15. Pre-requisite qualifications: basic knowledge of Mathematics, Physics, Fluid Mechanics, Computer Science, Computer Aided Design	
16. Course objectives: The goal of this course is to present the basic applications of CFD in Chemical Engineering and to present the Ansys Workbench environment with a particular emphasis on DesignModeler, Mesh, Fluent and CFD-Post. After this course students should be able to prepare a numerical analysis of some of the Fluid Mechanics phenomena.	

17. Description of learning outcomes:

Nr	Learning outcomes description	Method of assessment	Teaching methods	Learning outcomes reference code
1.	student knows the basics of Ansys Workbench	Unaided analysis of an issue chosen by tutor	laboratory	K2A_W01 +++ K2A_W02 +
2.	student possess a basic knowledge concerning numerical simulations of different chemical engineering phenomena	Unaided analysis of an issue chosen by tutor	laboratory	K2A_W01 ++ K2A_W02 + K2A_W03 +
3.	student is able to create a geometrical model using Ansys Workbench	Unaided analysis of an issue chosen by tutor	laboratory	K2A_U07 ++ K2A_U09 +++
4.	Student is able to generate a valid numerical mesh using Ansys Workbench	Unaided analysis of an issue chosen by tutor	laboratory	K2A_U07 ++ K2A_U09 +++
5.	student is able to run a simulation using Ansys Fluent	Unaided analysis of an issue chosen by tutor	laboratory	K2A_U01 ++ K2A_U06 + K2A_U07 ++ K2A_U09 +++
6.	student can think in a creative way	Unaided analysis of an issue chosen by tutor	laboratory	K2A_K06 +++

18. Teaching modes and hours

Lecture / BA /MA Seminar / Class / Project / Laboratory

Laboratory sem. 2 - 45h

19. Syllabus description:

During the course students are introduced to:

- methods of creation of 2D sketches, 3D bodies as well as how to import a geometry from other CAD software using DesignModeler
- methods of generation of different numerical meshes depending on given geometry a the nature of analyzed system using Mesh

module

- solver settings depending on a given geometry and mesh
- calculation methods using Fluent
- methods of results generation as well as their presentation using CFD-Post

20. Examination: no

21. Primary sources:

Ansys Workbench, manual.

Blazek J., Computational Fluid Dynamics: Principles and Applications, Elsevier Science 2005.

22. Secondary sources:

Jaworski Z., Mechaniczna numeryka płynów w inżynierii chemicznej, Wydawnictwo Exit, Warszawa 2005.

Date A.W., Introduction to Computational Fluid Dynamics, Cambridge University Press 2009.

Peyret R. (ed.), Handbook of Computational Fluid Mechanics, Academic Press 1996.

23. Total workload required to achieve learning outcomes

Lp.	Teaching mode :	Contact hours / Student workload hours
1	Lecture	-/-
2	Classes	-/-
3	Laboratory	45/15
4	Project	-/-
5	BA/ MA Seminar	-/-
6	Other	15/15
	Total number of hours	60/30

24. Total hours:90

25. Number of ECTS credits: 3

26. Number of ECTS credits allocated for contact hours: 2

27. Number of ECTS credits allocated for in-practice hours (laboratory classes, projects): 2

26. Comments:-

Approved:

…… ……

(date, Instructor's signature) (*date , the Director of the Faculty Unit signature)*