

Course title:	Basics of CFD simulation
Institute/Division:	FACULTY OF CHEMICAL ENGINEERING AND TECHNOLOGY
Number of contact hours:	15 hours (15 h computer laboratories)
Course duration:	1 semester (6 th semester of regular I cycle studies - spring)
ETCS credits:	2
Course description:	Basic information about CFD modelling, methodology, methods, accuracy, tools - How to run Ansys, Software overview, DesingModeler, Geometry, PLANES, SKETCHES, Sketching toolboxes, Mesh, Fluent solver, Boundary conditions, How to performed Heat transfer Steady calculations. Preparing to use more advanced software ANSYS with a focus on fluid flow and heat transfer modelling using FLUENT, Data analysis, Geometry generation, Mesh generation, Post-processing software and methods
Education effects :	<ul style="list-style-type: none"> - knowledge: Student knows basic numerical methods used in engineering calculations - skills: Students will get knowledge how to solve in the easy way complex problem using ANSYS software - social: student is able to work independently and in the group
Literature:	<p>[1] Ferziger J. H., Peric Milovan — Computational Methods for Fluid Dynamics, Berlin Heilderberg New York, 2001, Springer Verlag</p> <p>[2] Anderson J. D — Computational Fluid Dynamics: The Basics with Applications, Columbus, 1995, McGraw Hill</p>
Assessment method:	project
Prerequisites:	Knowledge of the basics of thermodynamics, fluid mechanics and heat exchange.
Primary target group:	Students from all specialties
Lecturer:	dr inż. Barbara Larwa
Contact person:	<u>bl@chemia.pk.edu.pl</u> , (12) 628 2739
Remarks:	Selectable course