CFD MODELLING WITH ANSYS FLUENT

3 ECTS (ELECTIVE)

AGH University of Science and Technology

Course responsible: prof. dr. Andrzej Nowak (SUT)

Course overview

The aim of the course is to provide students with fundamentals of the numerical modelling of heat and fluid flow problems using selected commercial code. Lectures and Computer laboratory projects aim to enhance students' problems solving skills using ANSYS Fluent software.

The course consists of lectures and computer laboratory.

Lectures cover: Introduction to CFD. Fundamental governing equations – continuity, momentum and energy. Methodology of solving CFD problems. Coupling of velocity and pressure fields. Modelling of turbulent flows. Modelling of radiation. Multiphase flows and their modelling.

Computer laboratory is carried out personally by each student. First he is guided by instructor through heat and fluid flow problems typical for energy sector. Then students are expected to undertake the number of individual projects covering the most important areas of computational fluid dynamics. Student has to get credits for all undertaken projects

Outcome of the course

After this course the student should be able to:

- Define properly geometry of the body.
- Discretize above considered object and generate appropriate mesh.
- Define materials of the object.
- Define boundary/initial conditions for fluid flow and energy transport.
- Select suitable solver and obtain convergent solution.
- Carry out postprocessing.

Course coordinator & teachers

Prof. dr. Andrzej.J.Nowak, , SUT, E-MAIL: andrzej.j.nowak@polsl.pl