Number of module: 1	Module: Numerical Simulation
Coordinator of module	**Prof. Dr. R. Stank**
Lecturer	**Prof. Dr. R. Stank**
Period	The lectures „Computational Simulation Techniques“ an the application orientated lectures for renewable energies „Wind Turbine Design with CFD“ and „CFD Simulation for Biogas Plants“ are hold once a year
Credits	5 CP
Workload	on-campus program 64 h, self-study 146 h, with one third for the lecture, one third for the utilization of the software and one third to compile an individual report on the base on a CFD computation for renewable energy applications
Status	optional
Prerequisites	
Max. number of participants	Ca. 25
Language	German/Englisch

Skills to be acquired / Learning objectives

Subject based and methodical skills

The students are able to apply numerical simulations and in particular CFD as well as to assess the quality of the numerical results by evaluating the mesh and the convergence behaviour. The students are competent to isolate the important geometry features for the flow out of CAD and based on that to generate the computational mesh. They generate converged and consistent numerical solutions.
By their deep physical understanding the students are able to analyze the flow and to improve the flow path and/or the technical application.

Personal and social skills

T The students have the ability to reach the educational objectives sure and independently. They learn creatively and in small teams and they analyse the numerical results together before including them in a report. The students are in a position to communicate and present convincingly and to document their results.

Contents

Lecture: Computational Simulation Techniques

This lecture contains the numerical techniques to solve coupled partial differential equations including explicit algorithms, boundary conditions and spatial discretisation. The commercial software package ANSYS CFD is introduced and used to simulate flow fields. Numerical solution parameters are treated and the convergence behaviour is explained and studied.
The physical flow phenomena laminar and turbulent flows and shock waves are introduced and the way how to handle them in a numerical simulation is explained.

Lecture: CFD Simulation for Biogas Plants

The lecture "CFD Simulation for Biogas Plants“ deals with multi-components flows as they occur in bio gas plants. Mixture consisting of gases or liquids and porous media are explained as well as drying processes. The programming of variable component properties and heat transfer mechanisms in the ANSYS CFD language CEL is covered.

Lecture: Windturbine Design with CFD The lecture "Windturbine Design with CFD" includes the airfoil section theory and dicusses the numerical investigation of lift and drag curve with the help of CFD. The two dimensional results are transferred to 3D wing section theory in order to determine the local chord length distribution of the rotor. Instationary computations are carried out to analyse the design parameter chord length, number of revolutions and nacelle design etc. and to determine the wind pressure force on the structure.	
Related courses: Computational Simulation Techniques (2 CP) Windturbine Design with CFD (3 PC) CFD Simulation for Biogas Plants (3 CP	
Teaching skills/Advanced Teaching and Learnig	Presentation with Beamern and Overhead Teamwork in small groups Supervision of using the commercial software package ANSYS CFD
Exam	Written exam and individual report
Literature/Teaching aids/Studying Material	• Script • Manuals and Tutorials of ANSYS