IMPRS Magdeburg

International Max Planck Research School for Analysis, Design, and Optimization in Chemical and Biochemical Process Engineering

Module	Computational Fluid Dynamics
Responsibility	Dr.-Ing. G. Janiga, G. Fru
Goals	Students participating in this course will get both a solid theoretical knowledge of Computational Fluid Dynamics (CFD) as well as a practical experience of problem-solving on the computer. Best-practice guidelines for CFD are discussed extensively. CFD-code properties and structure are described and the students first realize the own, simple CFD-code, before considering different existing industrial codes with advantages and drawbacks. At the end of the module, the students are able to use CFD in an autonomous manner for solving a realistic test-case, including a critical check of the obtained solution.
Contents	1. Introduction and organization. Historical development of CFD. Importance of CFD. Main methods (finite-differences, -volumes, -elements) for discretization. 2. Vector- and parallel computing. Introduction to Linux, main instructions, account structuration, FTP transfer. 3. How to use supercomputers, optimal computing loop, validation procedure, Best Practice Guidelines. Detailed introduction to Matlab, presentation and practical use of all main instructions. 4. Linear systems of equations. Iterative solution methods. Examples and applications. Tridiagonal systems. ADI methods. Realization of a Matlab-Script for the solution of a simple flow in a cavity (Poisson equation), with Dirichlet-Neumann boundary conditions. 5. Practical solution of unsteady problems. Explicit and implicit methods. Stability considerations. CFL and Fourier criteria. Choice of convergence criteria and tests. Grid independency. Impact on the solution. 6. Introduction to finite elements on the basis of Femlab. Introduction to Femlab and practical use based on a simple example. 7. Carrying out CFD: CAD, grid generation and solution. Importance of gridding. Best Practice (ERCOFTAC). Introduction to Gambit, production of CAD-data and grids. Grid quality. Production of simple and complex (3D burner) grids. 8. Physical models available in Fluent. Importance of these models for obtaining a good solution. Introduction to Fluent. Practical solution using Fluent. Influence of grid and convergence criteria. First- and second-order discretization. Grid-dependency. 9. Properties and computation of turbulent flows. Turbulence modeling, k-ε models, Reynolds-Stress-models. Research methods (LES, DNS). Use of Fluent to compute a turbulent flow behind a backward-facing step, using best practice instructions. Comparison with experiments. Limits of CFD.

Module	Computational Fluid Dynamics
	10. Non-newtonian flows, importance and computation. Use of Fluent to compute a problem involving a non-newtonian flow (medical application), using best practice guidelines. Analysis of results. Limits of CFD. 11. Multi-phase flows, importance and computation. Lagrangian and Eulerian approaches. Modeling multi-phase flows. Use of Fluent to compute expansion of solid particles in an industrial furnace, using best practice guidelines. Comparison with experiments. Limits of CFD. 12. -14 Summary of the lectures. Short theoretical questionnaire. Dispatching subjects for the final CFD-project, begin of work under supervision. Students work on their project during the last weeks, using also free-time. In the second half of the last lecture, oral presentations by the students of the results they have obtained for their project, with intensive questions concerning methods and results.
Requirements	Bachelor, fluid mechanics
Teaching	Lecture and computer exercises
Work Load	Time of attendance Lecture / Exercises: 3 h / week Autonomous work: post processing of lectures, preparation of exercises, project and exam
Examination	written test and presentation of the project
Credits	5 Credit Points
Duration	one term