Ansys + Coburg University of Applied Sciences

“Through hands-on experience with Ansys Discovery software, Ansys Fluent simulation, and the Ansys CFX solution in both introductory and advanced CFD classes for bachelor’s and master’s students, we bridge theoretical learning with practical application. This approach not only enhances course satisfaction, but also deepens understanding of fundamental concepts such as boundary conditions, grid studies, accuracy, time-step selection, and turbulence models. Ultimately, students gain proficiency in solving real-world computational fluid dynamics (CFD) challenges, preparing them for successful careers as engineers in the industry.”

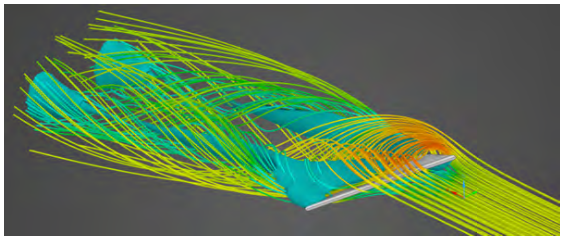
— **Philipp Epple**
Prof. Dr. -Ing., / Coburg University of Applied Sciences

Mastering Computational Fluid Dynamics With Ansys: Introductory and Advanced Class Offerings

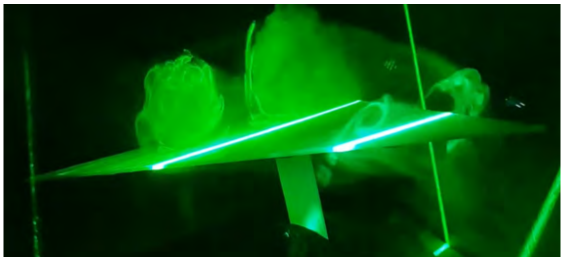
Today, theoretical classes in CFD must be complemented with practical exercises using a Navier-Stokes solver. Coburg University of Applied Sciences in Coburg, Germany, offers CFD classes for bachelor's and master's students, along with advanced master's classes. In the introductory classes, the fundamental theory covers the derivation of Navier-Stokes equations and a thorough understanding of each of its terms, the analytical solutions of fundamental flows (as in the flow in pipes or over a flat plate), the derivation of RANS equations, and the discussion of turbulent stresses and turbulence models.

This theory is accompanied with hands-on CFD classes that involve learning workflows with Ansys Workbench™ simulation integration platform and Ansys CFX® CFD software, including geometry generation, mesh creation, boundary condition setting, grid refinement, and turbulence model selection.

Coburg University of Applied Sciences



Simulation of the flow around a delta wing with Discovery software



Flow visualization around a delta wing

/ Challenges

To learn CFD, hands-on experience with Navier-Stokes solvers like Ansys Discovery™ 3D product simulation platform, Ansys Fluent® fluid simulation software, and CFX software is crucial.

The challenge of CFD lectures is to teach the theory (i.e., derive the fundamental equations, continuity, momentum, and energy equations); practice with these equations; teach discretization methods, finite differences and finite volumes to solve these equations numerically; and enable the students to solve real-world CFD problems.

In mechanical engineering, students must learn how to solve complex flow problems and to perform real-world product development and optimization in fluid mechanics. This can only be accomplished with professional CFD software. Discovery software, Fluent software, and CFX software are used for this purpose in teaching, as these are state-of-the art software packages used in industry and academia.

/ Technology Used

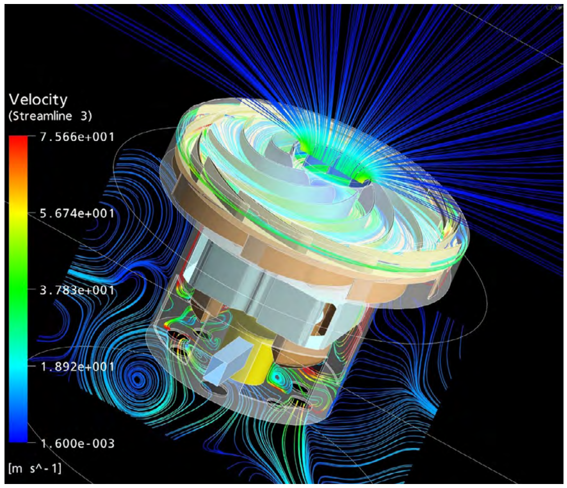
- Ansys Workbench software
- Ansys CFX CFD solver
- Ansys ICEM™ CFD mesh generation capability
- Ansys Fluent fluid simulation solution
- Ansys Discovery 3D product simulation platform

/ Engineering Solution

The CFD classes are divided into theoretical classes and practical classes using Ansys. In the practical classes, it is important that students learn the whole workflow: generating or impoting geometry, generating and studying grids, setting up boundary conditions and monitor points, selecting the time step and turbulence model, and solving and post-processing the solution using visualization features like contour plots, stream lines, and vector plots. In addition to this, students learn how to perform quantitative post-processing and export the quantitative results to CSV files. With Discovery software, the students learn:

- How to explore different flows, like the flow around a cylinder, airfoil, or pipe
- How to change and immediately see the effects of the boundary conditions and geometry modifications

Additionally, students gain hands-on experience with Ansys Meshing™ 2D/3D mesh generation and analysis capability , Ansys TurboGrid™ turbine blade meshing software, and the ICEM capability for generating various grid types and refining grids based on theoretical concepts. Practical applications include uncertainty estimation via Richardson extrapolation in CFX software, studying turbulence model impacts with the backward-facing step, and simulating rotating frames of reference with axial and radial fans.



Simulation of the flow through a vacuum cleaner fan with CFX software

/ Benefits

When solving complex and real-world simulation problems, CFD classes have a theoretical part that is necessary in understanding CFD and writing code. However, to meet industry and academia requirements, engineering simulation is needed. Therefore engineering simulation using Discovery software, Fluent simulation, and the CFX tool is included in classes. The software helps students develop an understanding in fundamental and complex flows. For example, students learn how to understand the differences between incompressible and compressible flows at subsonic and supersonic velocities. Additionally, students learn how to solve real-world problems in fluid mechanics, such as the flow over airfoils, cars, or an entire aircraft, as well as the flow through turbomachines.

/ Company Description:

Prof. Dr.-Ing. Philipp Epple is a fuild mechanics professor at Coburg University of Applied Sciences in Coburg, Northern Bavaria, Germany, teaching introductory and advanced courses in CFD. He also teaches classes in fluid mechanics, advanced fluid mechanics, heat transfer, and thermodynamics for bachelor's and master's students.

Prof. Dr.-Ing. Philipp Epple teaches CFD at Coburg University of Applied Sciences.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724-746-3304
ansysinfo@ansys.com

When visionary companies need to know how their world-changing ideas will perform, they close the gap between design and reality with Ansys simulation. For more than 50 years, Ansys software has enabled innovators across industries to push boundaries by using the predictive power of simulation. From sustainable transportation to advanced semiconductors, from satellite systems to life-saving medical devices, the next great leaps in human advancement will be powered by Ansys.

Visit www.ansys.com for more information.