Freie Universität Bozen
Libera Università di Bolzano
Università Liedia de Bulsan

Course description – ACADEMIC YEAR 2025/2026

Course title	**Advanced Methods for Fluid Machine Design**
Course code	42181
Scientific sector	ING-IND/08
Degree	Bachelor's in Industrial and Mechanical Engineering
Semester	II
Year	III
Academic Year	2025/26
Credits	6
Modular	No

Total lecturing hours	36
Total lab hours	
Total exercise hours	24
Attendance	Not compulsory, but strongly suggested
Prerequisites	-
Course page	

Specific educational objectives	The course of Advanced Methods for Fluid Machine Design is a compulsory course for the curriculum in Energy in the Bachelor of Industrial and Mechanical Engineering and it is an elective course for all the other curricula. It belongs to the scientific sector of Fluid Machines (ING-IND/08) and it consists of 36 hours of frontal lectures and 24 hours of practical exercises. The course can be intended as a container of fluid dynamic knowledge directly applicable in the field of mechanical engineering - therefore, Computational Fluid Dynamics (CFD) will be treated as a means by which to address engineering problems in the field of fluid machines design. The attempt that will be proposed here is to hold together as much as possible a purely knowledge-based approach to the basic subject matter - that is, CFD and the numerical methods involved - with an applied one - the use of programming, computational and simulation tools - whose aim is to develop all through the course typical case studies of turbomachines. The main specific educational objectives include: • understanding the theoretical global aspects underlying computational fluid dynamics (CFD) • understanding the basics of turbulence and its modeling in CFD • understanding the basic theoretical aspects of the finite volume method (FVM) • acquire the fundamental knowledge for a correct definition of a CFD problem • apply the fundamental aspects of CFD to fluid

	machines.

Lecturer	Alberizzi Jacopo C. – jacopocarlo.alberizzi@unibz.it
Scientific sector of the lecturer	ING-IND/08
Teaching language	English
Office hours	By appointment
Teaching assistant *(if any)*	Mohsen Fatehi - mohsen.fatehi@unibz.it
Office hours	-
List of topics covered	Frontal lectures have been structured according to the following modules: • Module 0 – Introduction to CFD, Partial Differential Equations and Vector Calculus. • Module 1 - Fundamental of Fluid Dynamics: basic concepts; the conservation concept; conservation of mass; conservation of momentum and forces in a fluid; conservation of energy; Navier-Stokes equations. • Module 2 - Introduction to Turbulence: Reynolds experiment; eddies and vorticity; boundary layers; scales of turbulence and energy cascade; turbulence in CFD. • Module 3 - The Finite Volume Method (FVM): the computational approach, FVM: main concepts; cells definition; discretization of the diffusive term; the convection-diffusion problem; properties of discretized equations; advanced discretization schemes; first order schemes; higher order schemes; summary of the discretization schemes; temporal discretization. • Module 4 - Numerical methods: gaussian elimination; Jacobi method; Gauss-Seidel method; poorly conditioned systems; pressure-velocity coupling. • Module 5 - Solving a CFD problem: a practical approach: geometry creation; meshing; physics and fluid properties; boundary conditions; solution procedure; initialization; convergence; post-processing. Practical exercises will include some basic CFD examples developed in ANSYS environments. Afterwards the design of turbomachinery equipment such as pumps, compressors, and turbines will be extensively treated using the ANSYS environment, according to the following sections: 1) Introduction in turbomachinery: categorizing of machines, applications of basic thermodynamic

	and fluid mechanics laws, how to use engineering standards and diagrams. 2) Cycle design of turbomachinery: Mean line design, throughflow, blade terminology, blade geometry generation, application of CFD. 3) Introduction of ANSYS turbomachinery modules: Vista modules for mean-line design, BladeGen module, Blade Editor module, Turbo Grid, and CFX. 4) Application of the module for designing: Using the modules, different kinds of examples will be designed from one dimensional design to CFD simulation and analyse the results.
Teaching format	The course consists of classroom lectures in which the topics are presented by the lecturer; digital presentations will be used. The practical exercises will be carried out using PCs - if needed, PC classroom will be booked.

Learning outcomes	**Intended Learning Outcomes (ILO)** Knowledge and understanding 1. Fundamental understanding of the Finite Volume Method and its use in CFD 2. Fundamental knowledge on the computational approach used in CFD for solving fluid machines related problems Applying knowledge and understanding 3. Ability to qualitatively and quantitatively define the stages required to solve a fluid dynamic problem according to the dictates of CFD Making judgements 4. Ability to evaluate discretization methods and major flow models (laminar and turbulent) 5. Critical approach to computational solutions, consciously questioning elements such as computational domain, computational mesh, and flow modeling parameters. Communication skills 6. Ability to structure and communicate a typical study-case in applied CFD for fluid machines Ability to learn 7. Ability to autonomously extend the knowledge

	acquired during the study course by reading and understanding

Assessment	**Summative assessment** **Form** / **%** / **Length /duration** / **ILOs assessed** Written exam / 50 / The written exam consists of four open-ended questions on the topics covered in the five modules. / 1,2,3,4,5,6 Project work / 50 / The project work consists of writing a technical report on an assigned case study. The case study will be based on the practical exercises developed with ANSYS software during the course. / 3,4,5,6
Assessment language	English
Evaluation criteria and criteria for awarding marks	Students regularly enrolled in the 3rd year of the bachelor's in industrial and mechanical engineering are eligible for the attendance of the lessons and the exam. Other exceptional cases must be discussed with the Professor. **Written exam** The written exam assesses the knowledge and understanding of the course topics as well as the ability to apply them to case studies and to make judgments. The following criteria will be considered: • Theoretical knowledge (both fundamental and applied) • Ability to provide examples/applications of the theoretical concepts • Ability to address a CFD problem considering the practical key aspects highlighted during the exercises • Communication skills and master of the technical language

Summative assessment table (Assessment):

Form	%	Length /duration	ILOs assessed
Written exam	50	The written exam consists of four open-ended questions on the topics covered in the five modules.	1,2,3,4,5,6
Project work	50	The project work consists of writing a technical report on an assigned case study. The case study will be based on the practical exercises developed with ANSYS software during the course.	3,4,5,6

	Project work (technical report) The work project aims to assess the most purely applicative skills in terms of analysis of the physics of a fluid dynamic problem, decision-making skills on the choice of simulation features, expository and argumentative clearness of results. The project will also be carried out during the exercise hours; therefore, participation and personal involvement will be part of the final evaluation. The exam will be weighted as follows: written part (15/30), project work (15/30).

Required readings	Lecture slides and official course notebook.
Supplementary readings	"Notes on Computational Fluid Dynamics: General Principles", C. Greenshields and H. Weller "Computational Fluid Dynamics - Principles and Applications", J. Blazek "A Guide to Fluid Mechanics" - H. Wang, Beihang University, Beijing, translated by Y. Zhang, School of Computer and Software Engineering, Nanyang Institute of Technology – Cambridge university Press "The Finite Volume Method in Computational Fluid Dynamics" – F. Moukalled, L. Magnani, M. Darwish