School of Interdisciplinary Engineering and Sciences (SINES)
National University of Sciences & Technology (NUST)

Defining futures

TITLE: ***COMPUTATIONAL FLUID DYNAMICS***

Pre-requisite: Foundations of Linear Algebra; Single and Multivariable Calculus.
Rationale: The rationale for changing the course content of the Computational Fluid Dynamics (CFD) course is to focus solely on the Finite Volume Method (FVM). The Finite Volume Method is the most widely adopted numerical approach

for modeling fluid flow problems in both research and commercial CFD codes due to its ability to handle complex geometries and irregular meshes. Focusing the Computational Fluid Dynamics course exclusively on the Finite Volume Method will provide students with the opportunity to develop a strong foundation in this widely-used technique. This will equip the students with the necessary depth of knowledge and expertise required to excel in modern CFD applications, research, and industry settings.

Objectives: The course is aimed to provide students with the essential concepts of CFD solution algorithms so that they can either develop their own CFD code or use commercially available/open-source CFD software with confidence and can interpret the results after post- processing. A strong emphasis is given to the understanding and application of underlying methods for solving various CFD problems.

Outcomes: On successful completion of the course, the students should be able to:

- Develop a solid understanding of the fundamental principles, equations, and governing laws of fluid dynamics that form the basis for computational fluid dynamics (CFD) simulations.
- Understand the application of the finite volume method to numerically solve fluid flow problems.
- Acquire hands-on experience with industry-standard CFD software packages and develop the skills necessary to set up, solve, and analyze CFD simulations for a variety of fluid flow scenarios.
- Learn the techniques and tools required for generating appropriate computational meshes for CFD simulations
- Gain knowledge of different types of boundary conditions commonly encountered in CFD simulations and how to apply them accurately.
- Understand convergence criteria and strategies for achieving reliable and accurate CFD solutions.

Course Code: CSE-801

Credit Hours:3-0

Course Contents:

Week 1 - 2:
Introduction to CFD, Mathematical review
Week 3 - 4:
Governing Equations of Fluid Flow (Navier-Stokes Equations)
Week 5-6:
Finite volume method for diffusion problem, Mesh generation, Boundary conditions

Week 7-8:
Finite volume method convection-diffusion problem.
Week 9:
Mid Semester Exam
Week 10-11:
Solution methods for discretized equations
Week 12-13:
Solution algorithms for Pressure velocity couplings (Navier-Stokes Equations)
Week 14:
Residuals in CFD and convergence criteria
Week 15:
Errors and stability
Week 16:

Latest trends in CFD, e.g. FSI, Multi-Physics, AI integration, etc.

	Existing	**Proposed**
1.	Introduction to CFD	Retained
2.	Mathematical review	New content
3.	Governing Equations of Fluid Flow (Navier-Stokes Equations)	Retained
4.	Finite difference formulation for PDEs	Deleted
5.	Finite volume method for the diffusion problem	New content
6.	Mesh generation	Retained
7.	Boundary conditions	Retained
8.	Finite volume method convection-diffusion problem	New content
9.	Solution methods for discretized equations	Retained
10.	Solution algorithms for Pressure velocity couplings (Navier-Stokes Equations)	New Content
11.	Residuals in CFD and convergence criteria	New content
12.	Errors and stability	Retained
13.	Turbulence and turbulence modelling	Deleted
14.	Latest trends in CFD, e.g. FSI, Multi-Physics, AI integration, etc.	Retained

Week 17:
Course Review and Wrap-Up
Week 18:
End semester exam
<u>Course content deleted:</u>
Finite difference formulation for PDEs, Turbulence and turbulence modelling
Course Text / Reference Books:

- *An Introduction to Computational Fluid Dynamics*, H. K. Versteeg and W. Malalasekera, 2nd Edition, Pearson, 2007.
- *The finite volume method in computational fluid dynamics, An advanced introduction with OpenFoam and Matlab*, F. Moukalled, L. Mangani, M. Darwish, Springer, 2016.
- *Computational Fluid Dynamics, A practical Approach*, Jiyuan Tu et al., 3rd Edition, BH, 2018.
- *Computational Fluid Dynamics: The Basic with Applications*, J.D. Anderson, Jr., McGraw Hill, Inc., 1995.
- *An Introduction to ANSYS Fluent 2022*, John E. Matsson, SDC Publications, 2022
- *Notes on Computational Fluid Dynamics: General Principals*, Christopher J. Greenshields, Henry G. Weller, CFD Direct, 2022

<u>Nature of Assessments</u>
Assessment will be carried out as per NUST statutes.

Comparative Chart: