BIRLA INSTITUTE OF TECHNOLOGY AND SCIENCE, Pilani
Pilani Campus
AUGS/ AGSR Division

SECOND SEMESTER 2020-21
COURSE HANDOUT

Date: 15.01.2021

In addition to part I (General Handout for all courses appended to the Time table) this portion gives further specific details regarding the course.

Course No	:	**ME G515**
Course Title	:	**COMPUTATIONAL FLUID DYNAMICS**
Instructor-in-Charge	:	**Dr. Shyam Sunder Yadav**
Instructor(s)	:	----
Tutorial/Practical Instructors:		----

1. Course Description: This course is an introduction to the various computational techniques for fluid dynamics. Following topics will be considered: Philosophy of Computational Fluid Dynamics, Governing equations of fluid dynamics, Mathematical behaviour of partial differential equations of fluid dynamics and their impact on CFD, Finite difference and finite volume discretization techniques on structured and unstructured grids, Numerical solution of steady and transient convection-diffusion equation, Solution techniques for compressible and incompressible Navier-Stokes equations, Various methods for solving system of linear equations, Techniques for turbulent flows.

2. Scope and Objective of the Course: This course is intended to develop the numerical skills for solving the equations governing fluid flow, heat & mass transfer and related transport phenomena. The students first learn how to derive the governing equation using fundamental laws of mass, momentum and energy conservation. Once the equations are derived, the student will learn how to simplify and classify the equations mathematically and how to interpret their physical and mathematical behavior? Subsequently the course focuses on the different discretization techniques for the governing equations like the finite difference and finite volume method. The student will learn how to select the right numerical scheme which correctly mimics the behaviour of partial differential equation they want to solve. They will also learn how to ensure that the numerical scheme is stable and the data from the numerical solution procedure is a valid representation of reality? Finally, the students will learn different CFD techniques for compressible and incompressible flows and will apply these for solving simple flow problems. As part of this course, the students have to complete a project of practical relevance using any licensed commercial / open source flow solvers. Most students prefer to use Ansys Fluent and OpenFOAM flow solver for their projects.

3. Text Books:

T1: J.D.Anderson , Computational Fluid Dynamics, Mcgraw Hill Inc., 1995, 6th Edition.
T2: Versteeg and Malajasekra: An introduction to computational fluid dynamics: The finite volume method, 2nd edition, Pearson

4. Reference Books:

R1: JH Ferziger, M Peric, Computational methods for Fluid Dynamics, Springer, 2002, 3rd Edition.
R2: Jiri Blazek, Computational Fluid Dynamics: Principles and Applications, Elsevier, 2015, 3rd Edition

R3: Charles Hirsch, Numerical computing of internal and external flows, Vol. 1, Elsevier, 2nd Edition

R4: Pletcher, Tannehill, Anderson, Computational fluid mechanics and heat transfer, CRC press, 3rd edition,

5. Course Plan:

Module No.	Lecture Session	Reference	Learning outcomes
1	**1a: CFD as a Research Tool, CFD as a Design Tool, The Impact of CFD, Applications of CFD**	**T1, chapter 1**	Why CFD is important? Where it has been applied? General CFD work flow procedure.
2	**2a-2b: Derivation of Governing equations of fluid dynamics** **2c-2d: Classification and characteristics of governing equations** **2e: Forms of equation for compressible flow**	**T1, chapter 2, chapter 3**	What are the equation which govern fluid dynamics? How to derive these? What is their physical significance? How to classify equations into hyperbolic, parabolic and elliptic types? Characteristics of different equation types, Forms of governing equations suitable for compressible flows
3	**3a-3c: Introduction to Finite difference method, Finite difference schemes for model equations, Explicit and Implicit approaches** **3d-3f: Consistency, stability and convergence of numerical schemes, Numerical diffusion and dispersion, Artificial viscosity**	**T1 chapter 4, R3 chapter 4 and chapter 7, R4 chapter 4**	Introduction to Finite difference method, Finite difference schemes for model hyperbolic, parabolic and elliptic equations, Explicit and Implicit approaches, Concepts of consistency, stability and convergence, Aspects of Numerical dissipation and dispersion; Artificial viscosity
4	**4a-4c: Solution techniques for system of linear equations**	**T2 chapter 7, R1 chapter 5, R3 chapter 10**	Solution techniques for system of linear equations, Various Direct and Iterative techniques
5	**5a: Introduction to finite volume method** **5b: FVM for diffusion equation on unstructured and structured grids** **5c-5d: FVM for convection-diffusion equation on structured and unstructured grids** **5e: FVM for transient equations**	**T2 chapter 4, chapter 5**	Introduction to finite volume method, Finite volume method for steady and unsteady convection diffusion equations on structured and unstructured grids
6	**6a: Lax-wendroff and Maccormack scheme** **6b: Application of Maccormack scheme to quasi 1D nozzl flow** **6c : Application of Maccormack scheme to 2D Prandtl-Meyer expansion wave**	**T1 chapter 6,7 and 8**	Simple CFD techniques 1: Lax-Wendroff scheme and MacCormack's scheme for compressible Navier-Stokes equations, Numerical solution of quasi-1D nozzle flow, 2D Prandtl-Meyer expansion wave
7	**7a: Pressure velocity coupling in incompressible flows**	**T1 chapter 9, T2 chapter 6**	Simple CFD techniques 2: Pressure correction technique for incompressible Navier

	7b-7c: Projection method for incompressible flows **7d-7f: SIMPLE family of algorithms for incompressible flows** **7g: Numerical solution of the lid driven cavity problem**		Stokes equations like Projection method, SIMPLE family of algorithms for pressure velocity coupling, Application to lid driven cavity problem
8	**Computational models for Turbulent flows**	**T2 chapter 3, R1 chapter 9, R2 chapter 7**	Governing equations for Turbulent flows, RANS models, LES, DNS

6. Evaluation Scheme:

Component	**Duration**	**Weightage (%)**	**Date & Time**	**Nature of component (Close Book/ Open Book)**
Mid-Semester Test	90 Min.	30	As per AUGSD	Closed book
Comprehensive Examination	3 hour	40	As per AUGSD	Partially closed and open book
CFD Project	-	30	Will be announced	Open resources

7. Online Consultation Hour: Will be decided in the class.

8. Notices: Personal emails will be sent

9. Make-up Policy: No makeup for any component

10. Note (if any): You should have a laptop which you will use to complete the project. You have to give a live demonstration of the procedure you followed while completing the project. You have to submit a project report consisting of the following: Abstract summary, Introduction to the problem including the previous works / literature on the problem, Governing equations, Procedure followed, Results and discussions and finally, Conclusion.

Instructor-in-charge
ME G515 CFD