



Advanced Computational Fluid Dynamics AE 508

CHAPTER 0

—

Course Objectives and Syllabus

by

Assoc. Prof. Dr. Emre Kara , Univ. of Gaziantep, TURKEY

Instructor

Assoc. Prof. Dr. Emre Kara

Office: Z04

Email: emrekara@gantep.edu.tr





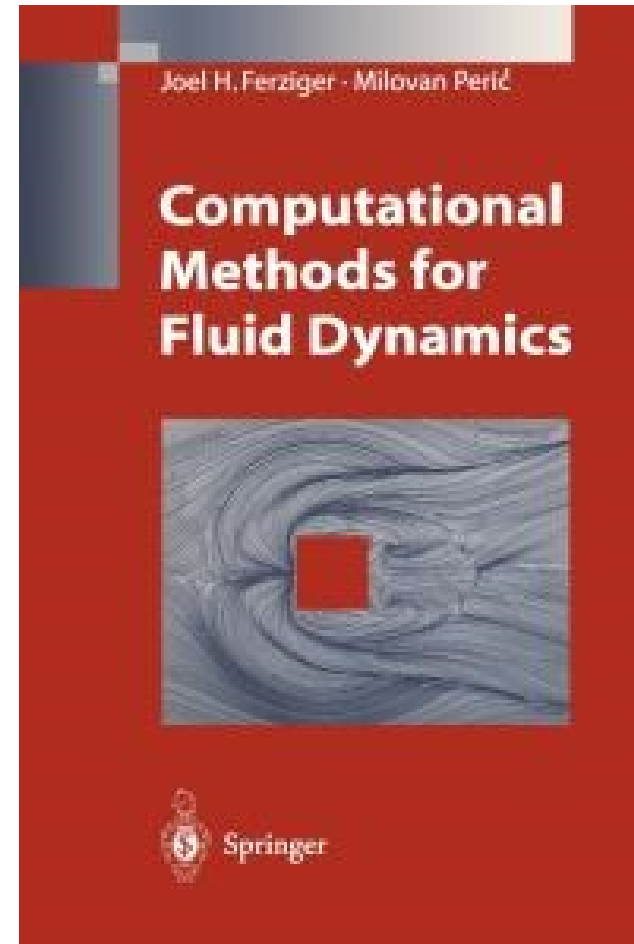
Class Information

Class Hours: Monday 08.30-11.00 (A01)

3-0 credit

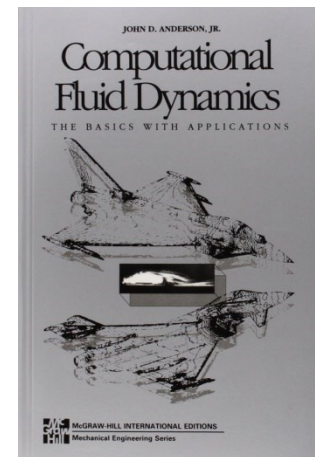
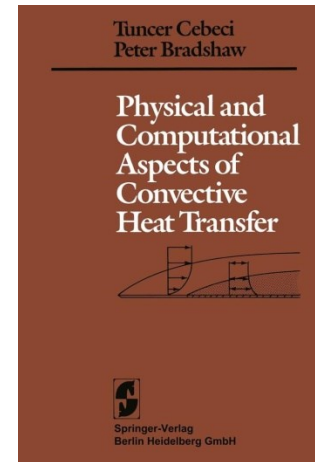
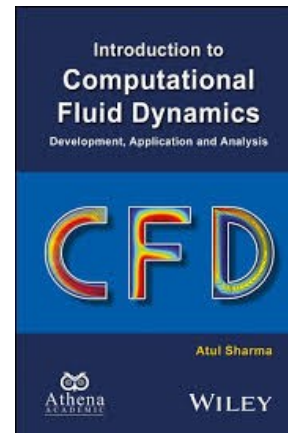
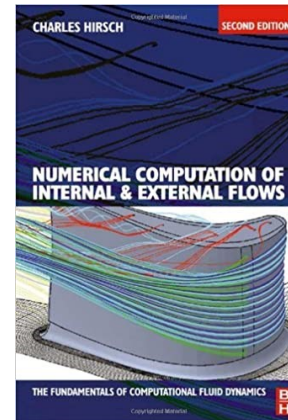
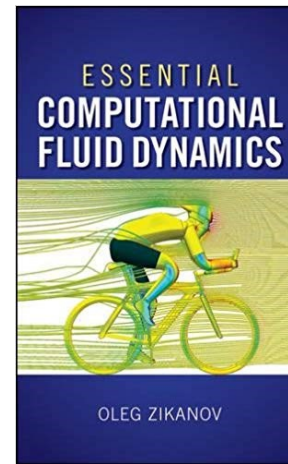
Compulsory Textbook

- Computational Methods for Fluid Dynamics, Ferziger and Peric
- You can download the online version of the first edition after logging in to proquest from Gaziantep University Server:
- <https://ebookcentral.proquest.com/lib/gaziantep-ebooks/detail.action?docID=3097819>



Reference Textbooks

- Computational Fluid Dynamics: The Basics with Applications by Anderson
- Numerical Computation of Internal and External Flows, I and II by Hirsch
- Essential Computational Fluid Dynamics, by Zikanov
- Physical and Computational Aspects of Convective Heat Transfer by Cebeci et al.
- Introduction to Computational Fluid Dynamics: Development, Application and Analysis, by Sharma.
- Computational Fluid Mechanics and Heat Transfer, by Pletcher, Tannehill and Anderson
- Computational Fluid Dynamics - A Practical Approach, by Tu, Yeoh and Liu



Software and Resources

- CFD software was built upon physics, modeling, numerics.
- Two types of available software
 - Commercial (e.g., FLUENT, CFX, Star-CCM, COMSOL, OpenFOAM)
 - Research (e.g., SU2, CFDSHIP-IOWA, U²RANS)
- More information on CFD can be got on the following website:
 1. <https://www.cfd-online.com/>
 - An online center for Computational Fluid Dynamics, largest CFD site on the net with services like discussion forums, jobs, links and a wiki
 2. <https://cfd.ninja/>
 - Various free tutorials using ANSYS, OpenFOAM and more, with the goal learn, disseminate and share knowledge that is useful for the solution of current problems in engineering using tools of CFD
 3. <https://www.fetchcfd.com>
 - Platform to publish, share, collaborate, discover & download simulations also in 3D/VR/AR.
 4. <https://www.youtube.com>
 - Many video are available beginning with examples. Just search computational fluid dynamics!
 5. <https://www.learncax.com/>
 - Useful source site for FREE CFD Training : Courses, Projects, Career .
 6. <https://community.sw.siemens.com/s/topic/0TOVb0000000GETOA2/the-peri%C4%87-lectures-on-cfd>
 - The Perić Lectures on CFD cover a wide range of essential topics, from the fundamentals of fluid dynamics to advanced numerical methods and their implementation.

AE 508 Advanced CFD Lecture



- This course includes detailed analysis, proofs, and practical advices for CFD applications.
- The course is designed for students who want to deepen their understanding of CFD.

AE 508 Advanced CFD Lecture

- Background needed:
 - Undergraduate Numerical Analysis and Fluid Mechanics
 - Basic computer skills in ANSYS Fluent (preferably AE427 course or equivalent taken)
 - ANSYS 2022 R2 version will be used.

Assessment (LOOK OUT !)

- Homeworks
- Midterms
- Final Exam

Assessment (LOOK OUT !)

- Homeworks
 - 8 applications will be given. All details will be on the given PDF file. The students will submit a report every week after 2nd week (except midterm and final weeks).

Assessment (LOOK OUT !)

- Midterms:

One midterm will be given. It will be take-home exam including presentations of Part-2 the next week by each student. They will be two parts:

-Part-1: Hand-written solution of the subject related to lecture notes (5%).

-Part-2: CFD applications solved in ANSYS Fluent 2022 R2 version. The resulting images and tables will be asked for. The students will have a limited time for powerpoint presentations of their results.

The students' report, PPT presentation and the CFD application shown in their laptop will be equally evaluated as 5% each (a total of 15%).

Assessment (LOOK OUT !)

- Final Examination:

A final exam will be given according to the school schedules. There will be two parts:

- 1) Multiple-choice type. Lecture notes will be the base of the exam (10%).
- 2) CFD Application. Student should bring their laptops for the second part of the final. They will use their laptop in the second part of the final, CFD cases similar to midterm. They will submit the report as PDF file and CFD application in ANSYS Fluent 2022 R2 version (15% each, 30 % in total).

Assessment (LOOK OUT !)

- Final Grades:

Homeworks	30	%
Midterm (Take home)	30	%
Final Exam	40	%

Total	100	%
-------	-----	---

Tentative Schedule

Week-1 (21st Sep): Introduction

Week-2 (28th Sep): Chapter 1 - Introduction to CFD – HW1 will be given, report to be submitted next week.

Week-3 (5th Oct): Chapter 2 - Basic Equations of Fluid Dynamics and Heat Transfer & Chapter 3 - Classification of PDEs – HW2 will be given, report to be submitted next week.

Week-4 (12th Oct): Chapter 4 – Finite Difference Methods – HW3 will be given, report to be submitted next week.

Week-5 (19th Oct): Chapter 5 - Elliptic Equations – HW4 will be given, report to be submitted next week.

Week-6 (26th Oct): Chapter 6 - Parabolic Equations (**TAKE HOME EXAM WILL BE GIVEN. Take home exam will be finished by students till next week before presentations.**)

Week-7 (2nd Nov): Midterm (Presentations)

Week-8 (9th Nov): No lecture this week

Week-9 (16th Nov): Chapter 7 - Hyperbolic Equations – HW5 will be given, report to be submitted next week.

Week-10 (23rd Nov): Chapter 8 - Stability Analysis – HW6 will be given, report to be submitted next week.

Week-11 (30th Nov): Chapter 9 - Vorticity-Stream Function Formulations & Chapter 10 - Solutions of Navier Stokes Equations by FDM – HW7 will be given, report to be submitted next week.

Week-12 (7th Dec): Chapter 11-Finite Volume Method – Part 1

Week-13 (14th Dec): Chapter 11-Finite Volume Method – Part 2 – HW8 will be given, report to be submitted next week.

Week-14 (21st Dec): Review