Introduction to Computational Fluid Dynamics
AE 427

CHAPTER 0
–
Course Objectives and Syllabus

by
Asst. Prof. Dr. Emre Kara , Univ. of Gaziantep, TURKEY

Instructor

Asst. Prof. Dr. Emre Kara

Office: Z04

Email: emrekara@gantep.edu.tr

Lecture webpage:

http://www1.gantep.edu.tr/~emrekara/index.php/ae427/

FIRST OF ALL
TO ATTEND AE 427 INTRO TO CFD

Minimum requirement to become an AE 427 student is:

- To take and pass AE204 Fluid Mechanics and AE209 Thermodynamics in the previous semesters.
- **AE204 and AE209 GRADES OF FF/FD ARE NOT ACCEPTABLE. IF SO, PLEASE DROP THE COURSE.**

FIRST OF ALL
TO ATTEND AE 427 INTRO TO CFD

Minimum requirement to become an AE 427 student is:

- To take and pass AE204 Fluid Mechanics and AE209 Thermodynamics in the previous semesters.
- **AE204 and AE209 GRADES OF FF/FD ARE NOT ACCEPTABLE. IF SO, PLEASE DROP THE COURSE.**

Figure source: twitter @Jousefm2

FIRST OF ALL
TO ATTEND AE 427 INTRO TO CFD

Minimum requirement to become an AE 427 student is:

- To take and pass AE204 Fluid Mechanics and AE209 Thermodynamics in the previous semesters.
- **AE204 and AE209 GRADES OF FF/FD ARE NOT ACCEPTABLE. IF SO, PLEASE DROP THE COURSE.**

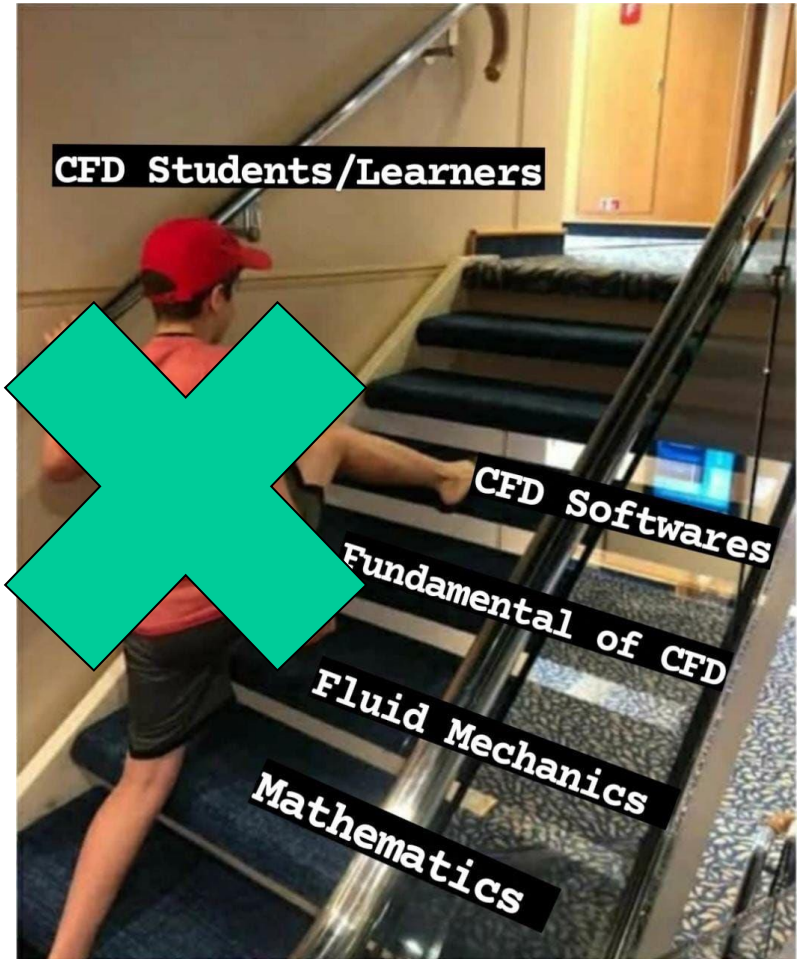


Figure source: twitter @Jousefm2

FIRST OF ALL
TO ATTEND AE 427 INTRO TO CFD

Minimum requirement to become an AE 427 student is:

- To take and pass AE204 Fluid Mechanics and AE209 Thermodynamics in the previous semesters.
- **AE204 and AE209 GRADES OF FF/FD ARE NOT ACCEPTABLE. IF SO, PLEASE DROP THE COURSE.**

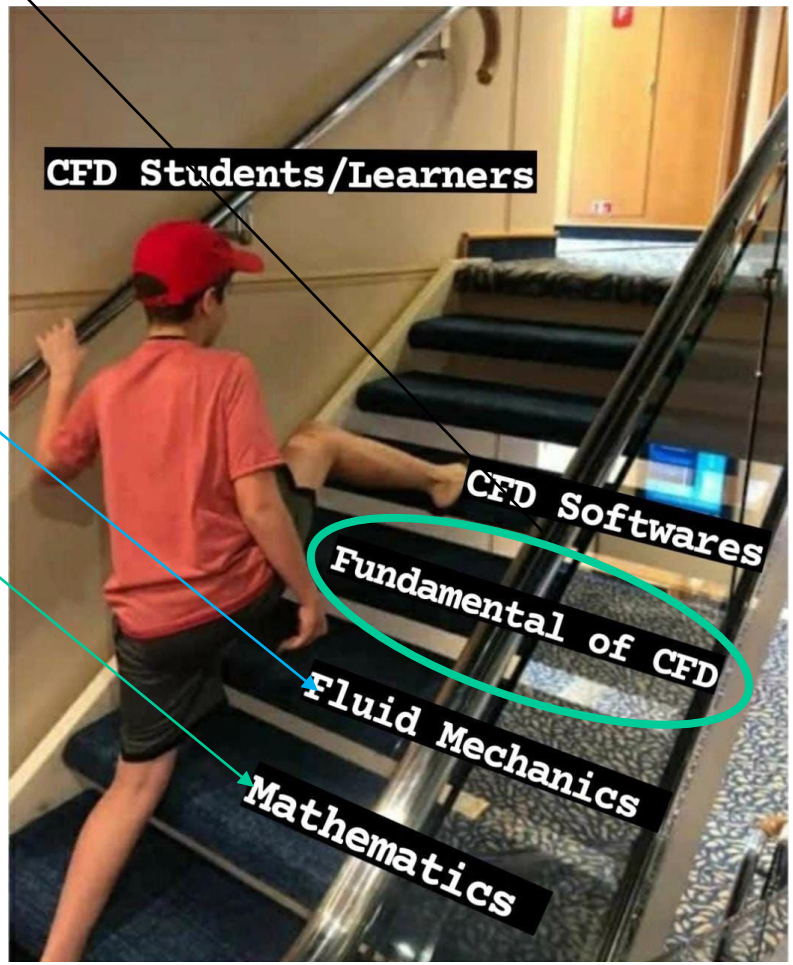


Figure source: twitter @Jousefm2

Course Information

Class Hours

Mondays

8:30-11:05

(COMLAB)

3-0 Credit

Course Information

During the course, Computational Fluid Dynamics (CFD) fundamentals will be covered and a commercial CFD software (licensed by the department) will be used to solve engineering problems. This lecture is designed for aerospace engineering students new to CFD. Combining an appropriate level of mathematical background, worked examples and step-by-step processes, students will walk through modeling and computing, as well as interpreting CFD results.

Objective

- The main objective of the course is to give the students a thorough knowledge of CFD from the theoretical and practical points of view.
- Other objectives of the course are to provide junior and senior year students with,
  - 1. basic CFD design skills,
  - 2. experience on CFD applications.
- During the course, we will cover CFD fundamentals and will use commercial CFD software to solve engineering problems parallel to the lecture. The class will be taught using Tu et al.'s textbook, Computational Fluid Dynamics A Practical Approach.

Teaching Policy

- The course teaching policy involves lectures, simulation examples (tutorials), two midterms and a final exam.

Compulsory Textbooks

Computational Fluid Dynamics

A Practical Approach

3rd Edition

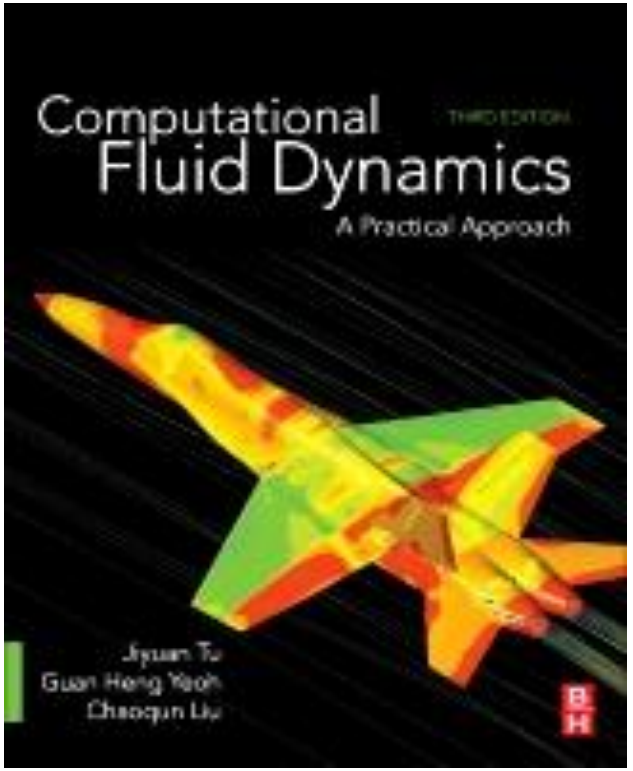
Jiyuan Tu

Guan Heng Yeoh

Chaoqun Liu

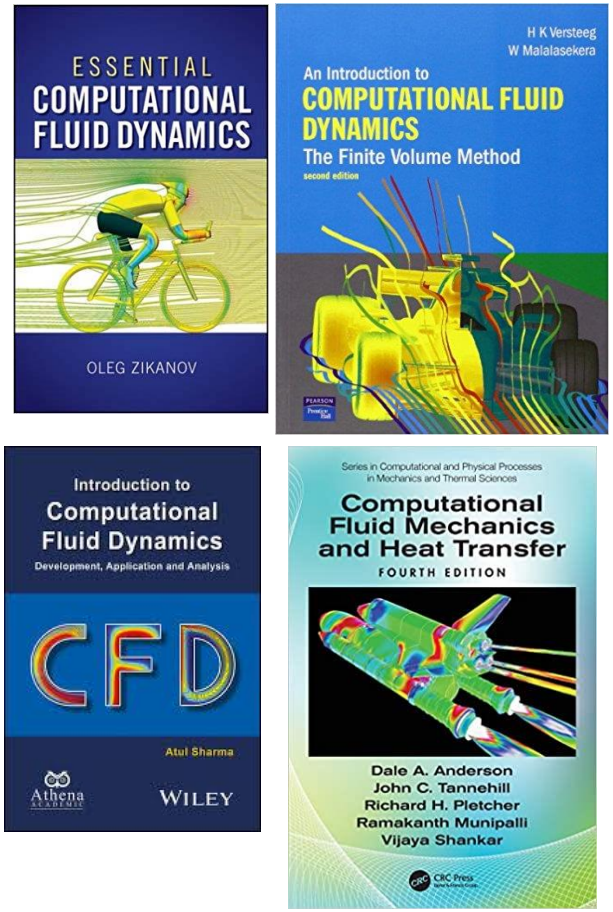
You can find the previous edition (2008) of textbook in our university library. New version is also available in bookstores.

Library number: TA 357 T835 2008



Recommended Texts

- An introduction to computational fluid dynamics: the finite volume method by Versteeg, and Malalasekera
- Computational Methods for Fluid Dynamics, Ferziger and Peric
- Computational Fluid Mechanics and Heat Transfer, by Pletcher, Tannehill and Anderson
- Introduction to Computational Fluid Dynamics: Development, Application and Analysis, by Sharma.
- Essential Computational Fluid Dynamics, by Zikanov



Internet Sources

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.

5. https://cfd2012.com/index.html

   Useful source site by Dr. Ahmed Al Makky of Cardiff University.

6. https://www.learncax.com/

   Useful source site for FREE CFD Training : Courses, Projects, Career ...

Software and Resources

- CFD software are built upon physics, modeling, numerics.
- Three types of available software
  - Commercial (e.g., ANSYS, Star-CCM, COMSOL)
  - Open source (OpenFOAM, Su2)
  - Research (e.g., CFDSHIP-IOWA, U^{2}RANS)
- More information on CFD can be got on the following website:
  - CFD Online: http://www.cfd-online.com/
  - Some CFD workbenchs (Each include preprocessor, grid generator, flow solver and visualization softwares in them) :
    - ANSYS: http://www.ansys.com/
    - COMSOL http://www.comsol.com/
    - Star-CCM https://www.plm.automation.siemens.com/global/en/products/simcenter/STAR-CCM.html
  - Some stand-alone grid generation softwares:
    - Cadence - Gridgen: http://www.pointwise.com
    - GridPro: http://www.gridpro.com/
  - Some stand-alone visualization softwares:
    - Tecplot: http://www.tecplot.com/
    - Paraview: https://www.paraview.org/

Software and Resources

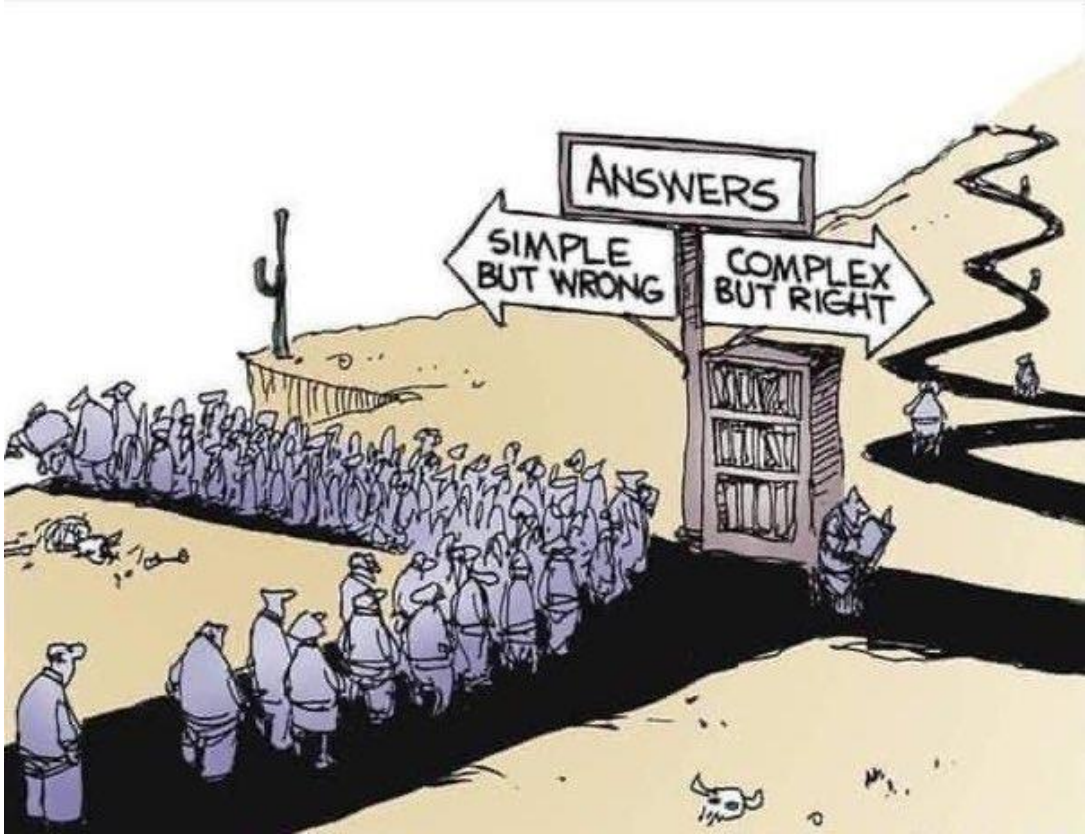
ANSYS Fluent: Students will use it in the COMLAB.

- You can download and install ANSYS Fluent Student 2024 R1 edition to your computer to practice in your free time. We will use ANSYS Fluent pre-installed in COMLAB.

https://www.ansys.com/academic/students/ansys-student (Built-in license valid until 01/31/2025)

Assessment (LOOK OUT !)

- Attendance
- Labs
- Homeworks
- Examinations
- Final Grades

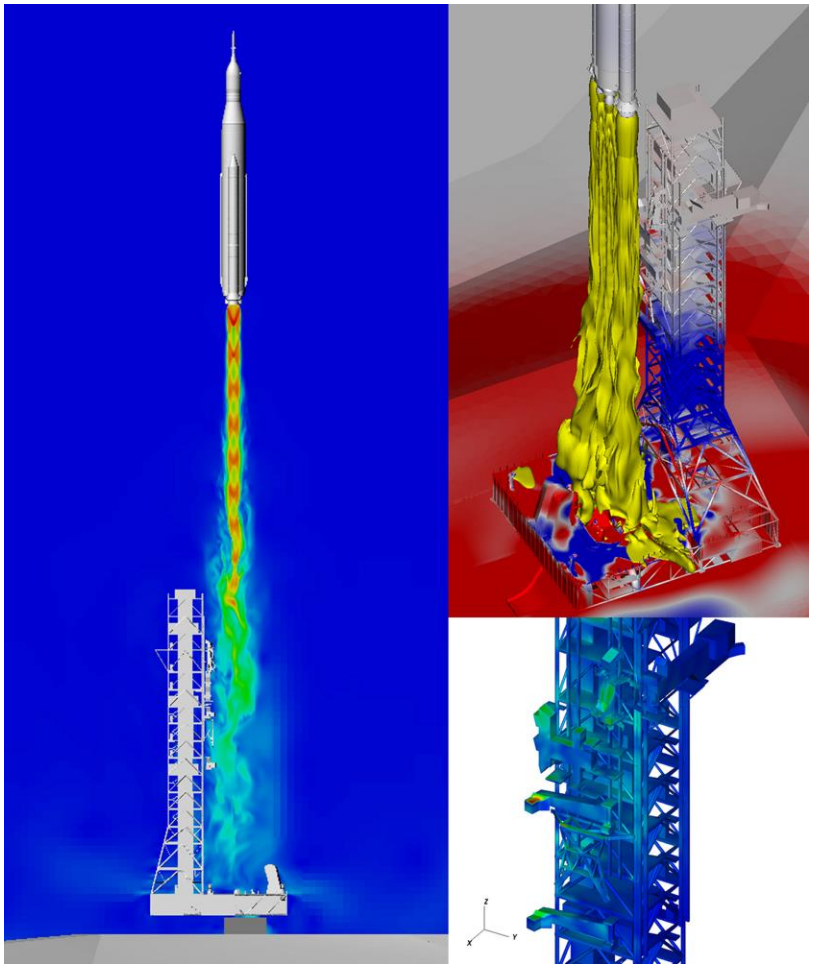


Assessment (LOOK OUT !)

- Attendance:

In case you have to miss a class, **you are responsible for keeping up** with the class work and being informed of all announcements made in the class concerning tests, etc.

If you encounter difficulties of any kind, feel free to come and see me in my office.



Assessment (LOOK OUT !)

- Labs:

There will be CFD sessions in which ANSYS DesignModeler, SpaceClaim, ANSYS Meshing and ANSYS Fluent will be used for applications. All students should keep the same pace up during the tutorials, there will be eight tutorials and students can try them after class at home, to get prepared for the midterms.

There will be also two midterm practice sessions prior week to the midterms.

Assessment (LOOK OUT !)

- Homeworks:

There will be <u>five</u> homeworks related to lecture. The homeworks are always due next lecture, no exception.

Assessment (LOOK OUT !)

- Examinations:
- One week before the midterms, there will be «midterm practice» sessions in which the instructor will tutor a similar study to midterm.

Assessment (LOOK OUT !)

- Examinations:
- **Midterm-1**:

  Correct geometry definition and mesh generation of the problem (geometry will be given for the mesh generation). The exam will be box-filling type, no partials. Students will fill the blanks on the question sheet, no USB submission.

Assessment (LOOK OUT !)

- Examinations:
- **Midterm-2** will be two parts:

1. A simple computational problem (**HAND WRITTEN solution**) connected to thermodynamics and/or fluid mechanics

2. CFD solution of the problem (Mesh will be given) This part of the exam will be box-filling type, no partials. Students will fill the blanks on the question sheet, no USB submission.

Assessment (LOOK OUT !)

- Examinations:

Final exam will be about CFD concepts, definitions (**multiple choice**) explained in class (and from textbook).

Assessment (LOOK OUT !)

- Final Grades:

Homeworks	10	%
Midterm 1	25	%
Midterm 2	25	%
Final	40	%
Total	100	%

Tentative Schedule

Lecture	Lecture Title	Class
Week-1	Chapter-0: Course Objectives and Syllabus	COMLAB
Week-2	Chapter 1: Introduction	COMLAB HW1 will be given.
Week-3	Chapter 2: CFD Solution Procedure – A Beginning LAB TUTORIAL 1: Creation of the geometry – Part 1	COMLAB
Week-4	Chapter 2: CFD Solution Procedure – A Beginning (Continues) LAB TUTORIAL 2: Creation of the geometry – Part 2	COMLAB HW2 will be given.
Week-5	Chapter 4: CFD Mesh Generation – A Practical Guideline LAB TUTORIAL 3: Ansys / Fluent Meshing	COMLAB
Week-6	Chapter 4: CFD Mesh Generation – A Practical Guideline (Continues) LAB TUTORIAL 4: Ansys / Fluent Meshing	COMLAB
Week-7	Practice session for midterm-1	COMLAB
Week-8	Midterm-1: Apr, 1st 2024, Monday at 8.30	COMLAB
Week-9	**RAMADAN HOLIDAY**	
Week-10	Chapter 3: Governing Equations for CFD – Fundamentals LAB TUTORIAL 5: Ansys Fluent	COMLAB HW3 will be given.
Week-11	Chapter 3: Governing Equations for CFD – Fundamentals (Continues) LAB TUTORIAL 6: Ansys Fluent	COMLAB
Week-12	Chapter 3: Governing Equations for CFD – Fundamentals (Continues) LAB TUTORIAL 7: Ansys Fluent	COMLAB HW4 will be given.
Week-13	Practice session for midterm-2	COMLAB
Week-14	Midterm-2: May, 13th 2024, Monday at 8.30	
Week-15	Chapter 5: CFD Techniques - The Basics LAB TUTORIAL 8: Ansys Fluent	COMLAB
Week-16	Chapter 5: CFD Techniques - The Basics (Continues)	COMLAB HW5 will be given.
	* During lab sessions, the topics will be covered in conjection with department licenced ANSYS in COMLAB.	

We will continue with Chapter-1: Introduction in the next lecture.