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.**

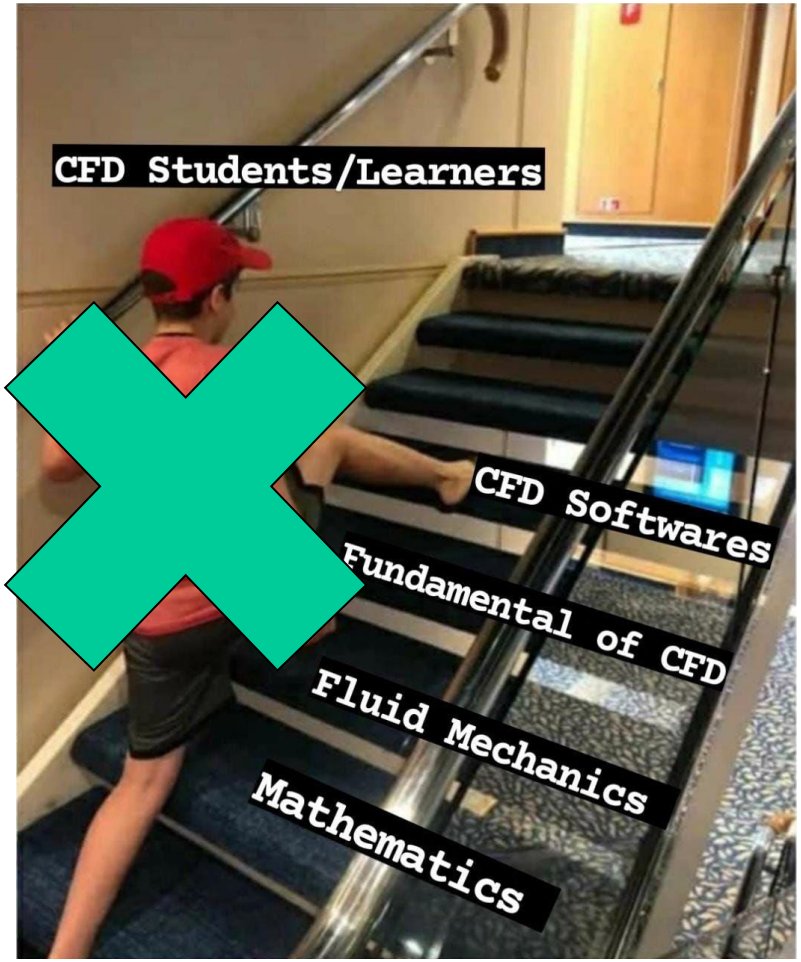


Figure source: twitter @Jousefm2

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.**

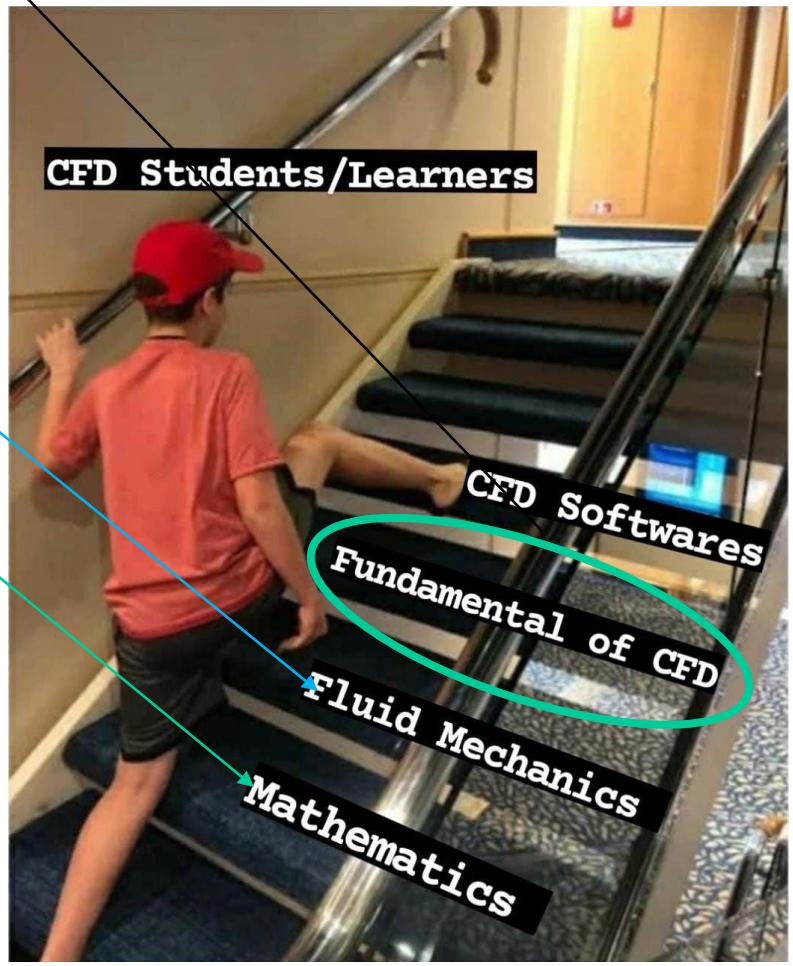


Figure source: twitter @Jousefm2

Course Information

Class Hours

FRIDAY

13:30-16:05

(A13 and 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

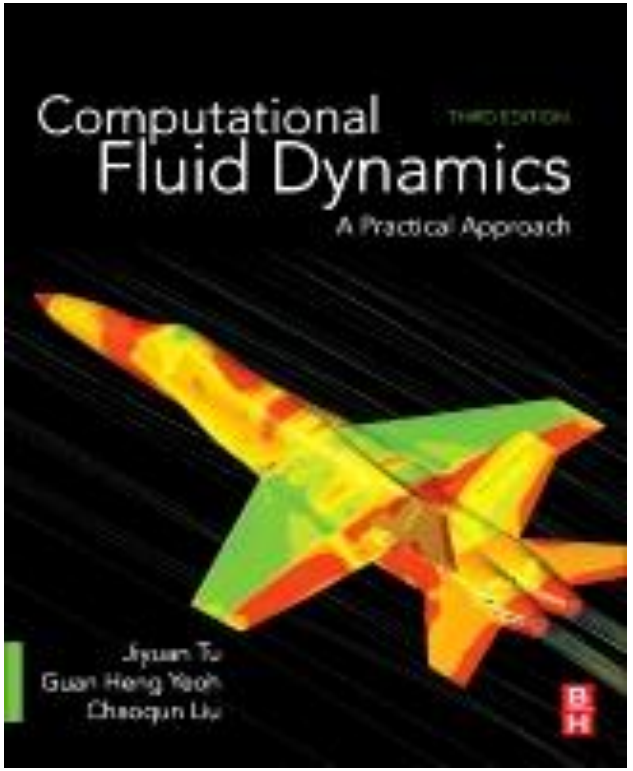
Jiyuan Tu

Guan Heng Yeoh

Chaoqun Liu

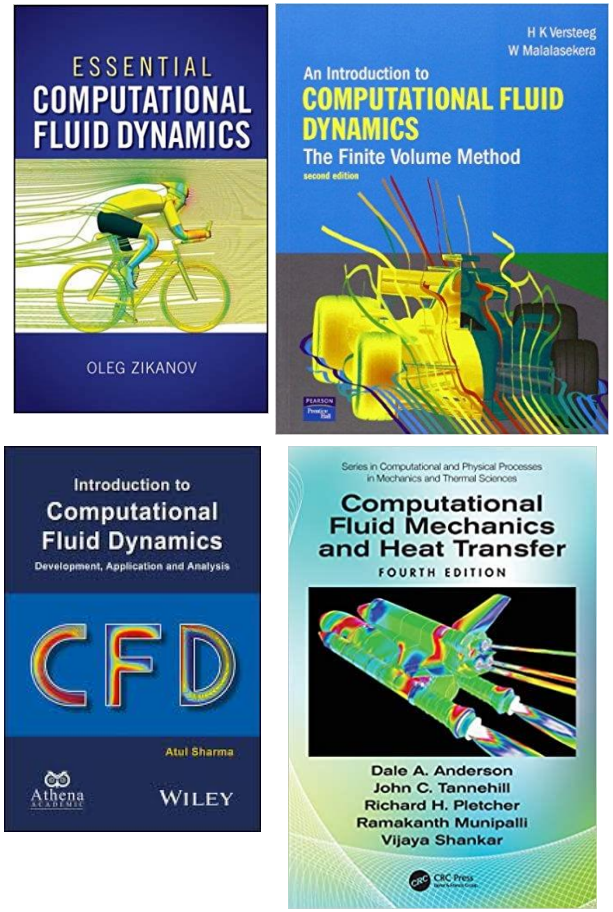
You can find the previous edition (2008) of textbook in our university library. New version is also available in bookstores.

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

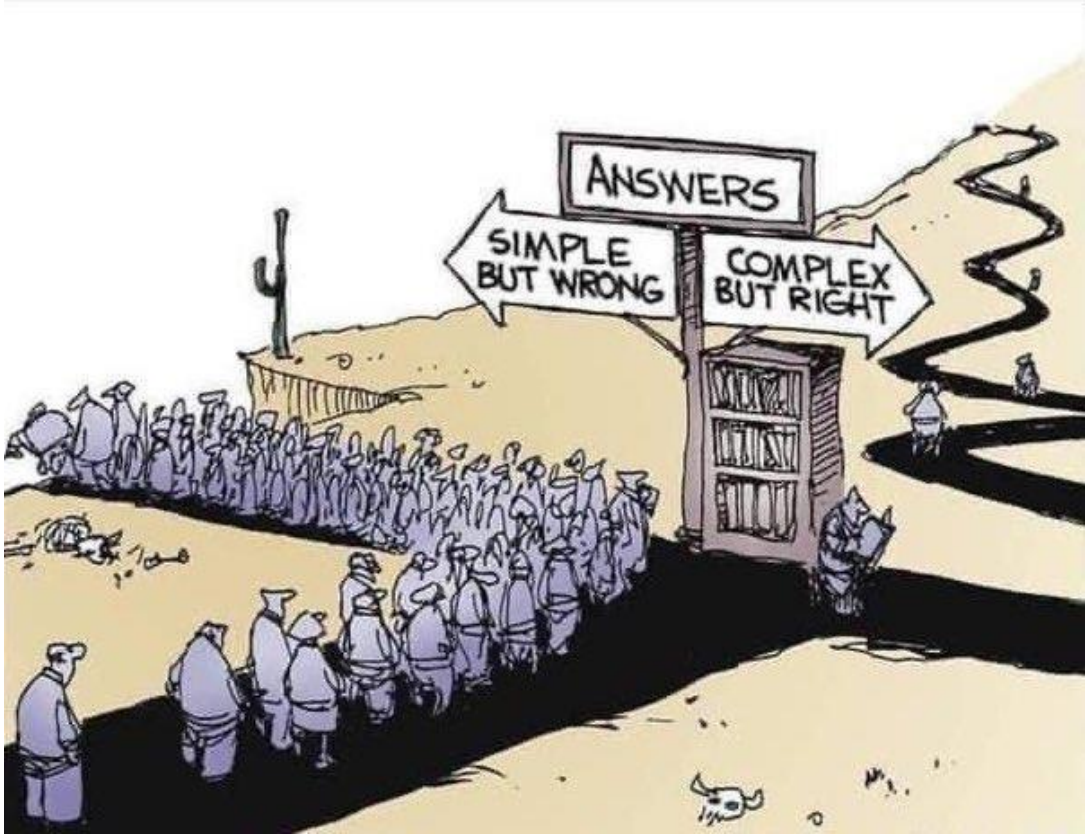
ANSYS Fluent: Students will use it in the COMLAB.

- You can download and install ANSYS Fluent Student 2023 R2 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 07/31/2024)

Assessment (LOOK OUT !)

- Attendance
- Labs
- Homeworks
- Examinations
- Final Grades

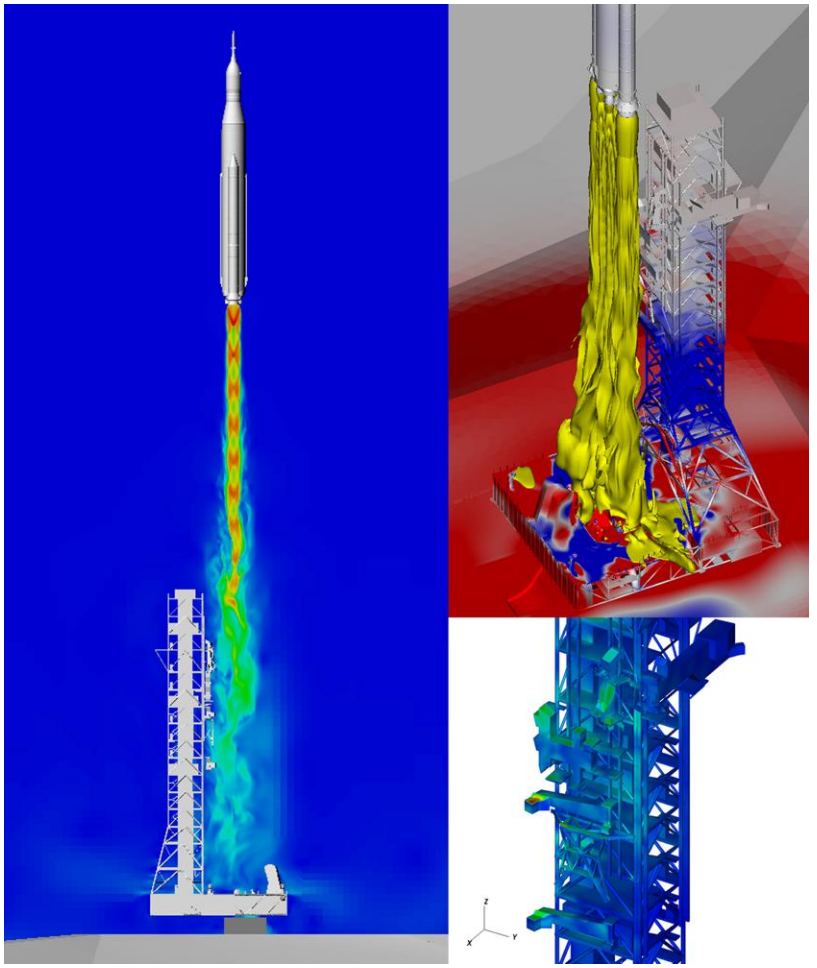


Assessment (LOOK OUT !)

- 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 meshing and ANSYS Fluent will be used for applications. All students should keep the same pace up during the tutorials, there will be ten tutorials and students can try them after class at home, to get prepared for 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:

Midterm-1:

Correct geometry definition and mesh generation of the problem.

Midterm-2 will be two parts:

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

2. Mesh generation and CFD solution of the problem (geometry will be given)

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

Make-up exams may be given for legitimate excuses if you contact the instructor as soon as you return to the school. It will be given for excused absences only and must be scheduled immediately upon returning to class. Excused absences require a medical excuse or notice of official school business by the V.P. of Academic Affairs.

Assessment (LOOK OUT !)

- Final Grades:

Homeworks	10	%
Midterm 1	20	%
Midterm 2	30	%
Final	40	%
Total	100	%

Tentative Schedule

Lecture	Lecture Title	Class
Week-1	Chapter-0: Course Objectives and Syllabus & Chapter 1: Introduction	A13
Week-2	Chapter 1: Introduction (Continues) - Chapter 2: CFD Solution Procedure – A Beginning	A13 & COMLAB
Week-3	LAB TUTORIAL 1: Creation of the geometry – Part 1 Chapter 2: CFD Solution Procedure – A Beginning (Continues)	A13 & COMLAB
Week-4	LAB TUTORIAL 2: Creation of the geometry – Part 2 Chapter 4: CFD Mesh Generation – A Practical Guideline	A13 & COMLAB
Week-5	LAB TUTORIAL 3: Ansys / Fluent Meshing Chapter 4: CFD Mesh Generation – A Practical Guideline (Continues) - Chapter 3: Governing Equations for CFD – Fundamentals	A13 & COMLAB HW1 will be given.
Week-6	LAB TUTORIAL 4: Ansys / Fluent Meshing Chapter 3: Governing Equations for CFD – Fundamentals (Continues)	A13 & COMLAB HW2 will be given.
Week-7	Midterm-1: Nov, 17th 2023, Friday at 13.30	COMLAB
Week-8	LAB TUTORIAL 5: Ansys / Fluent Meshing Chapter 3: Governing Equations for CFD – Fundamentals (Continues)	A13 & COMLAB HW3 will be given.
Week-9	LAB TUTORIAL 6: Ansys Fluent Chapter 3: Governing Equations for CFD – Fundamentals (Continues)	A13 & COMLAB
Week-10	LAB TUTORIAL 7: Ansys Fluent Chapter 5: CFD Techniques - The Basics	A13 & COMLAB HW4 will be given.
Week-11	LAB TUTORIAL 8: Ansys Fluent Chapter 5: CFD Techniques - The Basics – Continues	A13 & COMLAB
Week-12	Midterm-2: Dec, 22nd 2023, Friday at 13.30	A13 & COMLAB
Week-13	LAB TUTORIAL 9: Ansys Fluent Chapter 5: CFD Techniques - The Basics – Continues	A13 & COMLAB HW5 will be given.
Week-14	Chapter 5: CFD Techniques - The Basics – Continues LAB TUTORIAL 10: A Case study	A13 & COMLAB

Now, we will continue with an introduction.