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.**

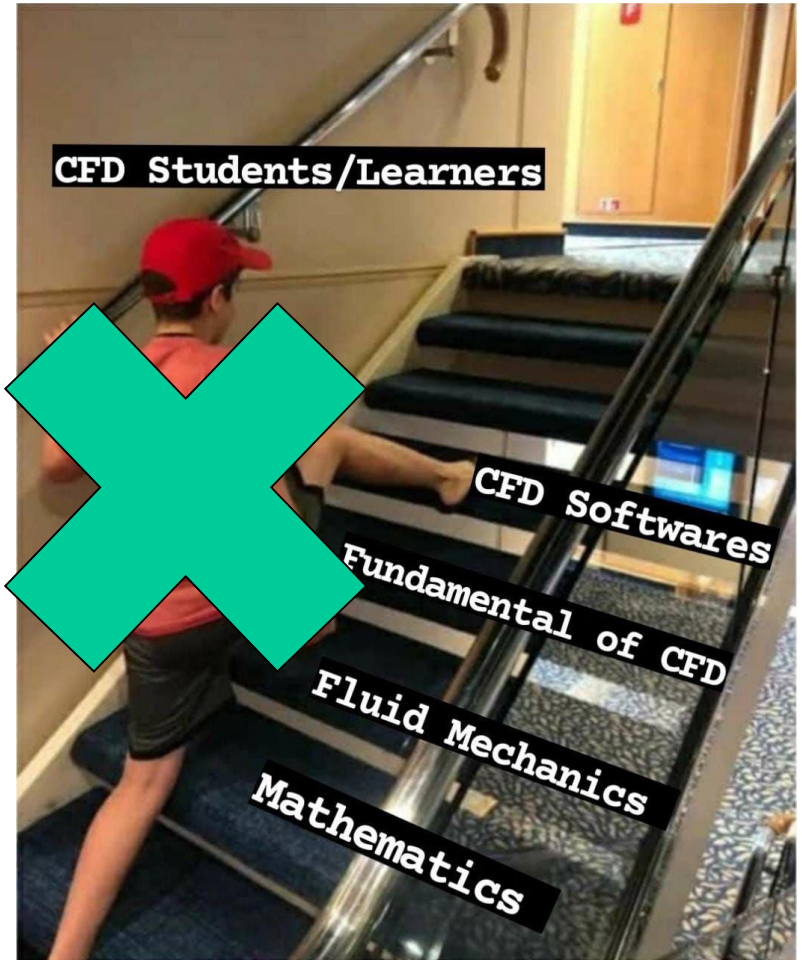


Figure source: twitter @Jousefm2

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.**

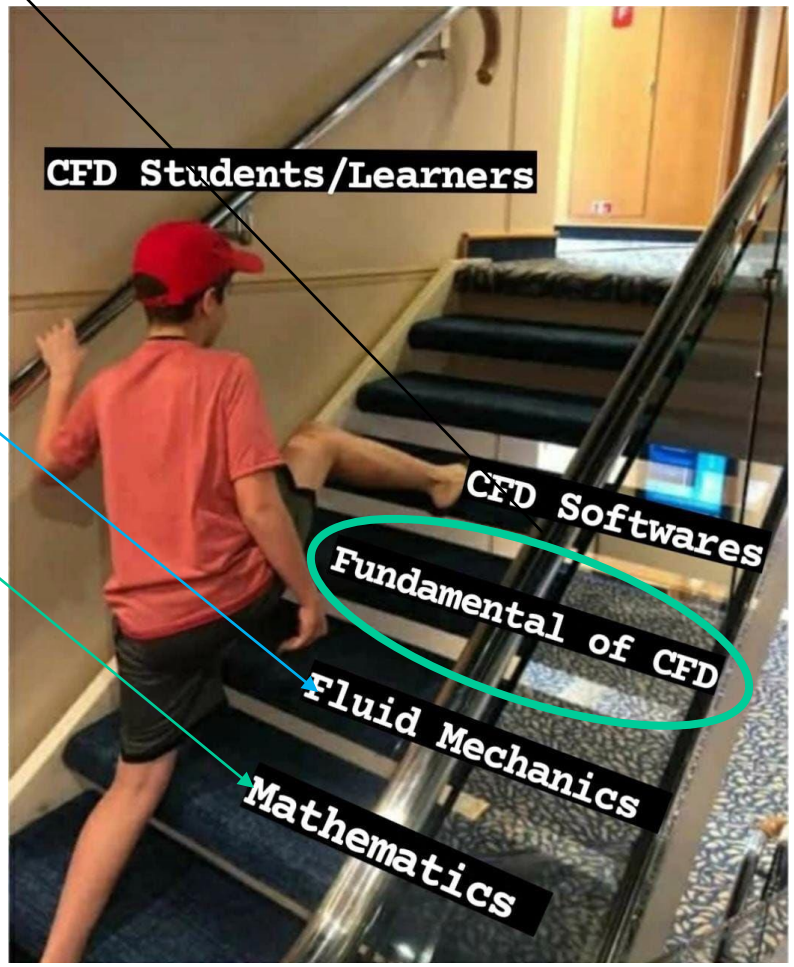


Figure source: twitter @Jousefm2

FIRST OF ALL
TO ATTEND AE 427 INTRO TO CFD

- Please send your transkripts till next lecture to emrekara@gantep.edu.tr for NUMESYS submission

Course Information

Class Hour:

THURSDAY–
08:30-11:05
(S15–COMLAB)

3-0 Credit

Course Information

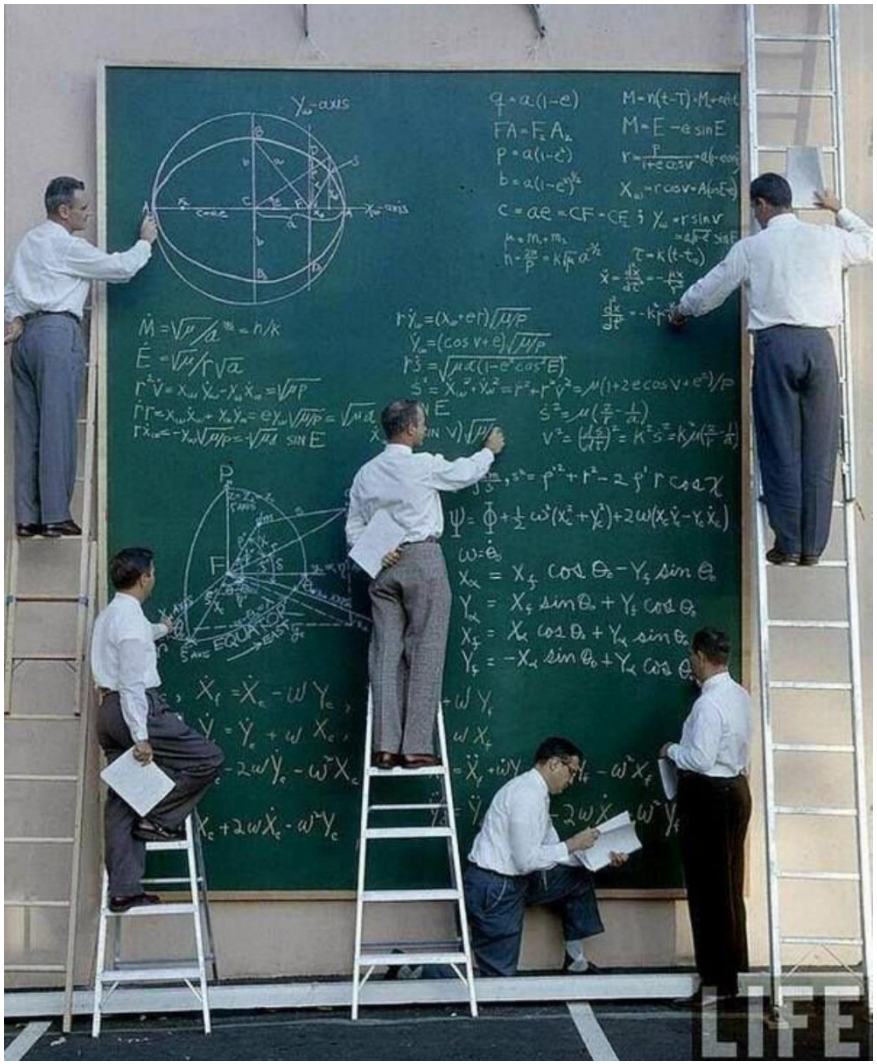
During the course, Computational Fluid Dynamics (CFD) fundamentals will be covered in the first half and a commercial CFD software (licensed by the department) will be used to solve engineering problems at the second half of the lecture. This lecture is designed for aerospace engineering students new 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 of collaboration in final project.
- During the course, we will cover CFD fundamentals and will use commercial CFD software to solve engineering problems at the second half of 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), homeworks, a midterm and a final exam team project.



Compulsory Textbooks

Computational Fluid Dynamics

A Practical Approach

3rd Edition

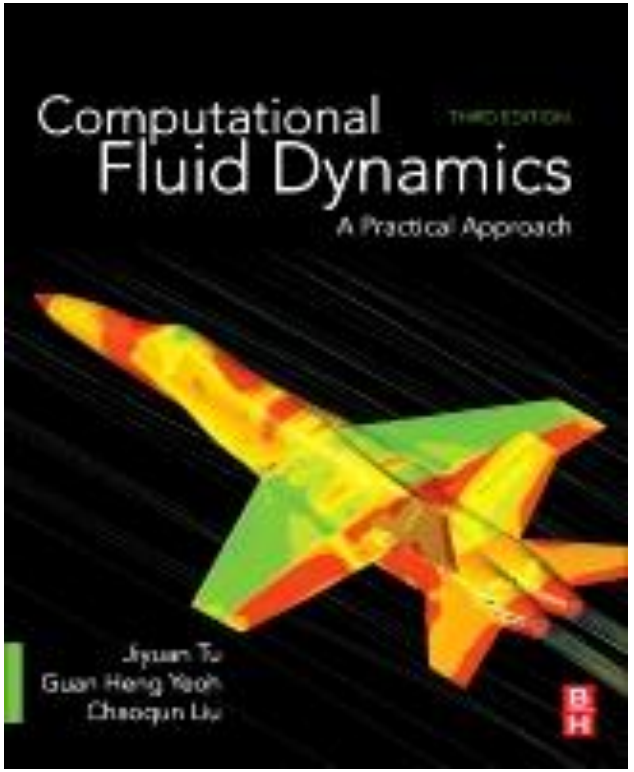
Jiyuan Tu

Guan Heng Yeoh

Chaoqun Liu

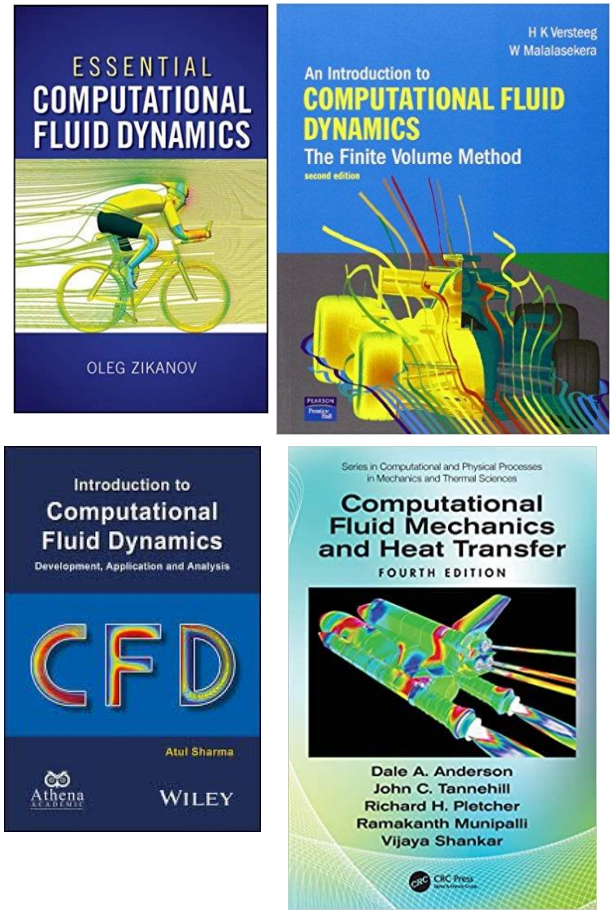
You can find the previous edition (2008) of textbook in our university library. New version is also available in bookstores.

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

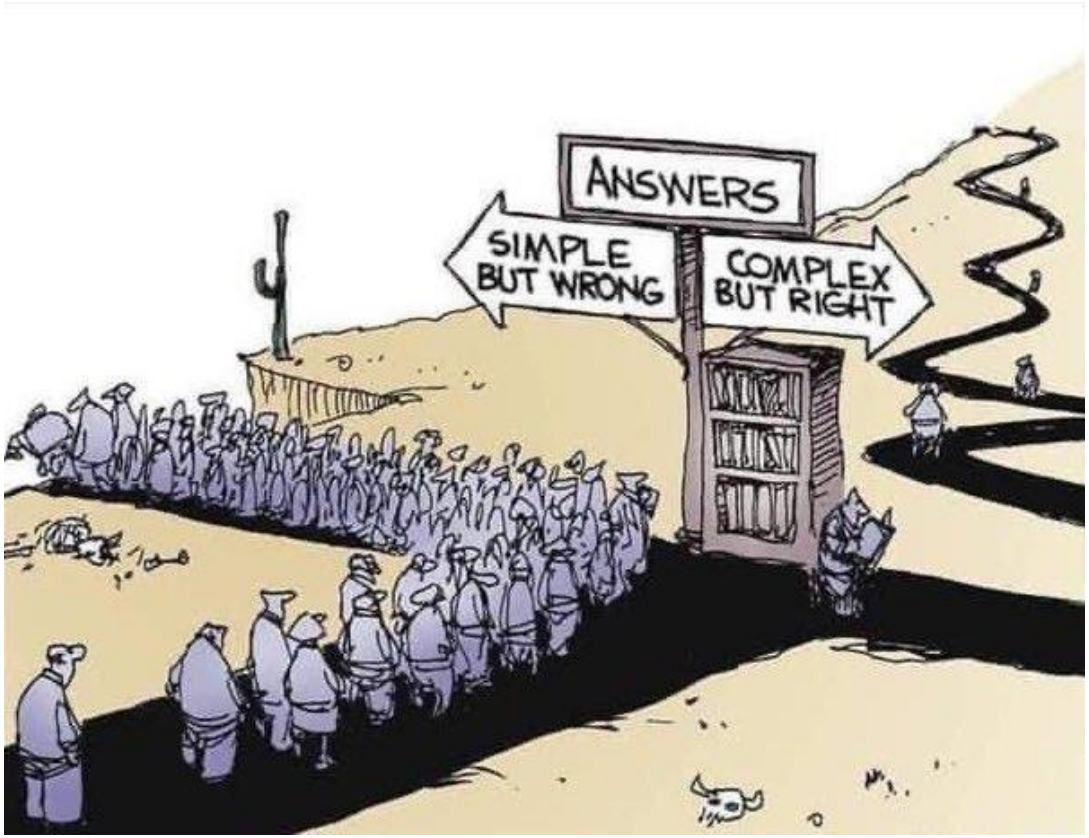
ANSYS Fluent: Students can use it in the COMLAB.

- You can use ANSYS Fluent Student 2022 R2 edition for AE 427 final project. We will use ANSYS Fluent 19.2 pre-installed in COMLAB. They are not quite different.

https://www.ansys.com/academic/students/ansys-student (Built-in license valid until 07/31/2023)

Assessment (LOOK OUT !)

- Attendance
- Labs and Homeworks
- Examinations
- Final Grades

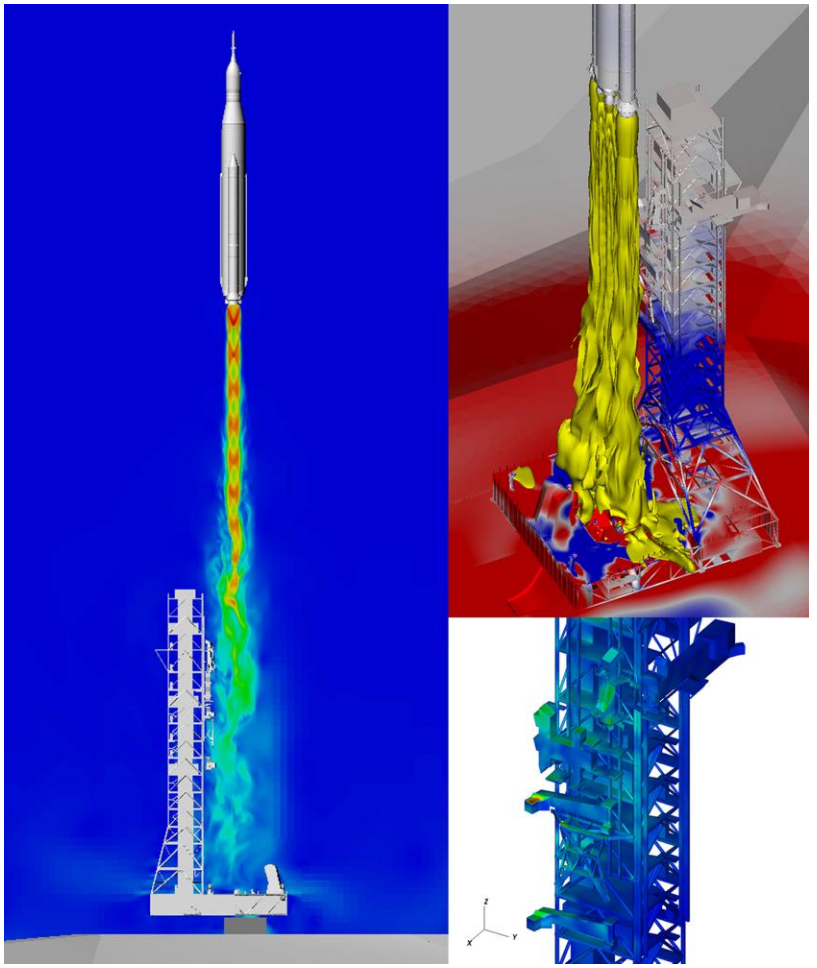


Assessment (LOOK OUT !)

- 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 and Homeworks:

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 five homeworks and students can try the tutorials at home. The homeworks are always due next lecture, no exception.

Assessment (LOOK OUT !)

- Examinations:

Midterm will be about CFD concepts, definitions (**multiple choice**), and computational problems (**HAND WRITTEN**).

Computational problems part of the midterm will be open book.

Final exam will be the reports/presentations in COMLAB (next slide).

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 (Term Project):

  Each CFD project will be performed by a student.

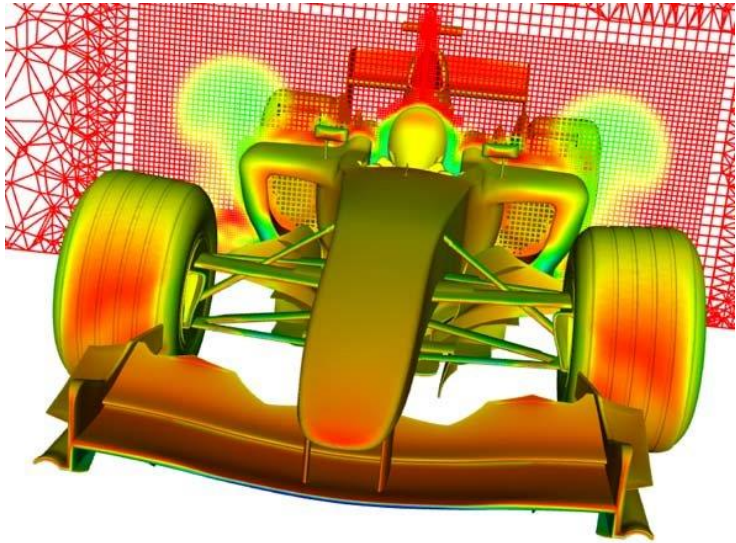
  Project topic will be selected by each student from the suggested studies by instructor and will approved by the instructor before the 14th week.

Figure source: twitter @Jousefm2

Assessment (LOOK OUT !)

- FINAL (Term Project):

AN EXAMPLE REPORT AND OTHER RULES WILL BE ANNOUNCED AT THE 14TH WEEK.



Gradings:

Presentation (15 pts)	final exam date
Report (10 pts)	final exam date
Successful CFD execution (15 pts)	final exam date

Final project report will include:

- Abstract
- Introduction with Literature survey.
  (Selecting experimental study correctly from literature is important)
- Methodology (Project realization)
- Results and Discussion (Computational study including comparison of numerical results with experimental study)
- Conclusion
- References

Assessment (LOOK OUT !)

- Final Grades:

Homeworks (5 x 6 %)	30	%
Midterm	30	%
Final	40	%
Total	100	%

Tentative Schedule

Lecture	Lecture Title	Class
Week-1	Chapter-0: Course Objectives and Syllabus & Chapter 1: Introduction	COMLAB
Week-2	Chapter 2: CFD Solution Procedure – A Beginning & Chapter 3: Governing Equations for CFD – Fundamentals	COMLAB
Week-3	Chapter 3:Governing Equations for CFD – Fundamentals – Continues	COMLAB
Week-4	Chapter 3:Governing Equations for CFD – Fundamentals – Continues & Chapter 4: CFD Mesh Generation – A Practical Guideline	COMLAB
Week-5	Chapter 5: CFD Techniques - The Basics	COMLAB
Week-6	Chapter 5: CFD Techniques - The Basics – Continues	COMLAB
Week-7	Chapter 5: CFD Techniques - The Basics – Continues	COMLAB
Week-8	Midterm Nov, 17th 2022, Thurs. at 08.30 EXAMPLE HOMEWORK WILL BE SHARED AFTER THE MIDTERM.	COMLAB
Week-9	Introduction to fluid dynamics, properties of fluids, flow classifications, simulation examples, homework-1	COMLAB
Week-10	Introduction to fluid kinematics, description of fluid motion, fluid flow rotation, fluid flow visualization, fluid flow measurement, simulation examples, homework-2	COMLAB
Week-11	Introduction to fluid statics, hydrostatic equilibrium and buoyancy, barometers/manometers, simulation examples, homework-3	COMLAB
Week-12	Viscous laminar flows, external flows, internal flows, simulation examples, homework-4	COMLAB
Week-13	Introduction to laminar boundary layer theory, integral analysis of boundary layers, theory of laminar boundary layers, flat plate Blasius solution, other boundary layer solutions and 3D layers, simulation examples, homework-5	COMLAB
Week-14	Basics of turbulent flows, stability of laminar flows and transition to turbulence description of turbulence, governing equations of turbulent flows, closure RANS equations, turbulent boundary layers, simulation examples. EXAMPLE FINAL REPORT WILL BE SHARED.	COMLAB
	* During the lab sessions, the topics will be covered in conjection with department licenced ANSYS Fluent in COMLAB	

Course Information of AE428 Applied CFD Lecture (next semester)

If you become successful in AE427 (this lecture), AE301 Heat Transfer and AE305 Aerodynamics-I lectures, you can take the continuing lecture AE428 Applied Computational Fluid Dynamics lecture at the Spring semester. It will be executed in our COMLAB using ANSYS Fluent for Aerospace Applications.