Computational Fluid Dynamics
AE 433

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/ae433/

FIRST OF ALL
TO ATTEND AE 433 CFD Lecture

Minimum requirements to become a AE433 student are:

- To take the following courses:
  - AE204 Fluid Mechanics
  - AE256 Differential Equations

in the previous semesters.

Course Information

Class Hour:

ONLINE: Friday– 08:30-09:15 (GAUZEM-BBB)

IN CLASS: Friday– 08:30-11:05 (Comp. Lab)

3-0 Credit

ME and CFD

Master's Thesis: Design of an Alternative Glaucoma Drainage Device Using CFD Tools (USE OF GAMBIT & FLUENT SOFTWARES)

Papers, proceedings and projects from master's thesis:

1. CFD analysis of the Ahmed Glaucoma Valve and design of an alternative device", Computer Methods in Biomechanics and Biomedical Engineering, 13:6, 655-662, (2010).
2. "CFD Analysis of Ahmed Glaucoma Valve and Design of an Alternative Device", Uluslararası Katılımlı 4. Biyomekanik Kongresi Bildiri Kitabı, 16-17 Ekim, Erzurum/Türkiye, (2008).
3. "HAD Araçları Kullanılarak Alternatif Bir Glokom Drenaj Cihazı Tasarımı", 1. Makine ve Aksamları AR-GE Proje Pazarı Yarışması Etkinlik Projeler Kitabı, 13 Nisan, İstanbul/Türkiye, (2012).
4. "Design of an Alternative Glaucoma Drainage Device Using CFD Tools", Special Session in the von Karman Institute (VKI) for Fluid Dynamics, 11 May, Brussels/Belgium, (2012).
5. "HAD Araçları Kullanılarak Alternatif Bir Glokom Drenaj Cihazı Tasarımı", Türkiye İnovasyon Haftası - AR-GE Proje Pazarı Finalist Sergisi (Poster), 6-8 Aralık, İstanbul/Türkiye, (2012).
6. "HAD Araçları Kullanılarak Alternatif Bir Glokom Drenaj Cihazı Tasarımı", 2. Makine ve Aksamları AR-GE Proje Pazarı Yarışması (Poster), 26 Ekim, İstanbul/Türkiye, (2013).

ME and CFD

Doctoral Thesis: Development of a Navier Stokes Solver for Compressible Flows on Cartesian Grids with Aerodynamics Applications (MY OWN CODES WRITTEN IN VISUAL FORTRAN !)

Papers, proceedings and projects from doctoral thesis (PART-1):

1. “An octree-based solution-adaptive Cartesian grid generator and Euler solver for the simulation of three-dimensional inviscid compressible flows”, Progress in Computational Fluid Dynamics: An International Journal, 16:3, 131-145, (2016). DOI: 10.1504/PCFD.2016.076247
2. “A Navier Stokes solver for compressible turbulent flows on quadtree and octree based Cartesian grids”, Journal of Applied Fluid Mechanics,12:3,539-549, (2019).DOI: 10.29252/jafm.12.02.29156
3. “Shock Wave Capturing with Multi-Grid Accelerated, Solution Adaptive, Cartesian Grid Based Navier Stokes Solver”, Journal of Aeronautics and Space Technologies, 9:2, 63-73, (2016).
4. “Lift Coefficient Calculation using a Geometric/Solution Adaptive Navier Stokes Solver On Two-Dimensional Cartesian Grids For Compressible And Turbulent Flows”, AIP Conference Proceedings, 1889:1, 1-5, (2017). DOI: 10.1063/1.5004352
5. “Quad-Tree Based Geometric-Adapted Cartesian Grid Generation”, Proceedings of the 8th International Conference on Continuum Mechanics (CM '13), 16-19 July, Series No. 14, Rhodes Island/Greece, (2013).
6. “A Quad-Tree Based Automatic Adaptive Cartesian Grid Generator with Applications on Multi-Element Airfoils”, 7th Ankara International Aerospace Conference (AIAC’13), 11-13 September, Ankara/Turkey, (2013).
7. “A Solution Adaptive Multi-grid Euler Solver on Two-dimensional Cartesian Grids”, 8th Ankara International Aerospace Conference (AIAC’15), 10-12 September, Ankara/Turkey, (2015).
8. “Object-Oriented Programming Application to a CFD Code on Cartesian Grid Techniques”, International Conference on Computer Science and Engineering / Uluslararası Bilgisayar Bilimleri ve Mühendisliği Konferansı (UBMK 2016), 20-23 Ekim, Tekirdağ, (2016).

ME and CFD

Doctoral Thesis: Development of a Navier Stokes Solver for Compressible Flows on Cartesian Grids with Aerodynamics Applications (MY OWN CODES WRITTEN IN VISUAL FORTRAN !)

Papers, proceedings and projects from doctoral thesis (PART-2):

9. “Solution Refinement Effectiveness of Multi-Grid Accelerated, Cartesian Grid Based Navier Stokes Solver on Compressible and Laminar Flows”, 8th International Academic Conference of Young Scientists "Mechanical Engineering, Materials Science, Transport 2016" (MEMST-2016), November 24-26, Lviv, Ukraine, (2016).
10. “Lift Coefficient Calculation using a Geometric/Solution Adaptive Navier Stokes Solver On Two-Dimensional Cartesian Grids For Compressible And Turbulent Flows”, 16th conference on Power System Engineering, Thermodynamics & Fluid Flow (PSE17), June 13-15, Plzen, Czech Republic, (2017).
11. “Determination of Minimum Distance from a Cell Centroid to a Triangulated Surface: A Mesh Generation Implementation Technique”, International Advanced Researches and Engineering Congress (IAREC 2017), 16-18 Kasım, Osmaniye, (2017).
12. “A Solution Adaptive Cartesian Grid Based Euler Solution for Compressible Flow around BOEING TR-1322 Multi-element Airfoil”, Nevşehir Bilim ve Teknoloji Dergisi, 4:1, 69-80, (2015). DOI: 10.17100/nevbiltek.66399
13. “A Navier Stokes solver for compressible turbulent flows on quadtree and octree based Cartesian grids”, Journal of Applied Fluid Mechanics,12:3,539-549, (2019).DOI: 10.29252/jafm.12.02.29156 (SCI-E)
14. “Çift elips yapısı etrafında çözüm uyarlamalı Navier-Stokes çözücüsü kullanarak yüksek Reynolds sayılı akış analizi”, Dicle Üniversitesi Mühendislik Fakültesi Mühendislik Dergisi, 11:2, 563-573, (2020). DOI: 10.24012/dumf.536200 (ULAKBİM – TR DİZİN)

ME and CFD

Doctoral Thesis: Development of a Navier Stokes Solver for Compressible Flows on Cartesian Grids with Aerodynamics Applications (MY OWN CODES WRITTEN IN VISUAL FORTRAN !)

Papers, proceedings and projects from doctoral thesis (PART-2):

9. "Solution Refinement Effectiveness of Multi-Grid Accelerated, Cartesian Grid Based Navier Stokes Solver on Compressible and Laminar Flows", 8th International Academic Conference of Young Scientists "Mechanical Engineering, Materials Science, Transport 2016" (MEMST-2016), November 24-26, Lviv, Ukraine, (2016).
10. "Lift Coefficient Calculation using a Geometric/Solution Adaptive Navier Stokes Solver On Two-Dimensional Cartesian Grids For Compressible And Turbulent Flows", 16th conference on Power System Engineering, Thermodynamics & Fluid Flow (PSE17), June 13-15, Plzen, Czech Republic, (2017).
11. "Determination of Minimum Distance from a Cell Centroid to a Triangulated Surface: A Mesh Generation Implementation Technique", International Advanced Researches and Engineering Congress (IAREC 2017), 16-18 Kasım, Osmaniye, (2017).
12. "A Solution Adaptive Cartesian Grid Based Euler Solution for Compressible Flow around BOEING TR-1322 Multi-element Airfoil", Nevşehir Bilim ve Teknoloji Dergisi, 4:1, 69-80, (2015). DOI: 10.17100/nevbiltek.66399
13. "A Navier Stokes solver for compressible turbulent flows on quadtree and octree based Cartesian grids", Journal of Applied Fluid Mechanics,12:3,539-549, (2019).DOI: 10.29252/jafm.12.02.29156 (SCI-E)
14. "Çift elips yapısı etrafında çözüm uyarlamalı Navier-Stokes çözücüsü kullanarak yüksek Reynolds sayılı akış analizi", Dicle Üniversitesi Mühendislik Fakültesi Mühendislik Dergisi, 11:2, 563-573, (2020). DOI: 10.24012/dumf.536200 (ULAKBİM – TR DİZİN)

14 studies (doctoral thesis) if you write own codes,
6 studies (master thesis) if you use a commercial program such as ANSYS FLUENT !

ME and CFD

NEW STUDIES about CFD

Papers, proceedings and projects AFTER doctoral thesis:

1. "Determination of the Wall Function for Navier-Stokes Solutions on Cartesian Grids", 2nd Workshop on Nonlinear PDEs in Applied Mathematics, August 8 - 10, IZTECH, İzmir, Turkey (2017).
2. "Numerical Simulation of Hypersonic Flow over Double Ellipse Configuration with Multi-grid Accelerated and Cartesian Based Flow Solver". In Proceedings of the First International Conference on Applied Mathematics in Engineering (ICAME'18), June 27-29, Balıkesir, (2018).
3. "Numerical Investigation of Slant Angle Effect on a Simplified Car Model with Solution Adaptive Cartesian Grid Method", The IVth International Congress of Automotive and Transport Engineering, October 17 - 19, Technical University of Cluj-Napoca, Cluj, Romania (2018).
4. "CFD Simulation of Turbulent Flow Around a Shrouded Spur Gear for Predicting Load-Independent Windage Power Losses", The IVth International Congress of Automotive and Transport Engineering, October 17 - 19, Technical University of Cluj-Napoca, Cluj, Romania (2018).
5. "A Short Review of CFD Based System Identification in Aerodynamics Applications". In Proceedings of the International Conference on Applied Mathematics in Engineering (ICAME'18), June 27-29, Balıkesir, (2018).
6. "Numerical Investigation of Slant Angle Effect on a Simplified Car Model with Solution Adaptive Cartesian Grid Method", Proceedings of the 4th International Congress of Automotive and Transport Engineering: Chapter 4, Springer Nature Switzerland AG 2019, N. Burnete and B. O. Varga (Eds.): AMMA 2018, PAE, pp. 1–7, 2019. DOI: 10.1007/978-3-319-94409-8_4

ME and CFD

NEW STUDIES about CFD

Papers, proceedings and projects AFTER doctoral thesis:

7. "CFD Simulation of Turbulent Flow Around a Shrouded Spur Gear for Predicting Load-Independent Windage Power Losses", Proceedings of the 4th International Congress of Automotive and Transport Engineering: Chapter 3, Springer Nature Switzerland AG 2019, N. Burnete and B. O. Varga (Eds.): AMMA 2018, PAE, pp. 1–8, 2019. DOI: 10.1007/978-3-319-94409-8_3
8. "CFD Analysis and Optimal Sizing of Finned Surface on a Novel Combined Turbine-Peltier System", International Symposium On Automotive Science And Technology (ISASTECH2019), September 5 - 6, Ankara, (2019).
9. "Thermal analysis of an anti-icing system for a NACA 4412 airfoil", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
10. "Numerical investigation of the aerodynamic performance of a low Reynolds number S809 wind turbine airfoil", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
11. "Numerical investigation of jet orientation using co-flow thrust vectoring with Coanda effect", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
12. "A Navier Stokes solver for compressible turbulent flows on quadtree and octree based Cartesian grids", Journal of Applied Fluid Mechanics,12:3,539-549, (2019).DOI: 10.29252/jafm.12.02.29156 (SCI-E)
13. " Experimental investigation and numerical verification of Coanda effect on curved surfaces using co-flow thrust vectoring", IAREJ Journal – 15-04-2021 (TR-Dizin)

ME and CFD

NEW STUDIES about CFD

Papers, proceedings and projects AFTER doctoral thesis:

7. "CFD Simulation of Turbulent Flow Around a Shrouded Spur Gear for Predicting Load-Independent Windage Power Losses", Proceedings of the 4th International Congress of Automotive and Transport Engineering: Chapter 3, Springer Nature Switzerland AG 2019, N. Burnete and B. O. Varga (Eds.): AMMA 2018, PAE, pp. 1–8, 2019. DOI: 10.1007/978-3-319-94409-8_3
8. "CFD Analysis and Optimal Sizing of Finned Surface on a Novel Combined Turbine-Peltier System", International Symposium On Automotive Science And Technology (ISASTECH2019), September 5 - 6, Ankara, (2019).
9. "Thermal analysis of an anti-icing system for a NACA 4412 airfoil", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
10. "Numerical investigation of the aerodynamic performance of a low Reynolds number S809 wind turbine airfoil", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
11. "Numerical investigation of jet orientation using co-flow thrust vectoring with Coanda effect", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
12. "A Navier Stokes solver for compressible turbulent flows on quadtree and octree based Cartesian grids", Journal of Applied Fluid Mechanics,12:3,539-549, (2019).DOI: 10.29252/jafm.12.02.29156 (SCI-E)
13. " Experimental investigation and numerical verification of Coanda effect on curved surfaces using co-flow thrust vectoring", IAREJ Journal – 15-04-2021 (TR-Dizin)
14. Kara, E., Turhan, S., Kutlar, A. İ., & Güngör, K. (2021). Computational and Experimental Analysis of an In Vitro Microfluidic Experimental Setup on Testing Molteno, Ahmed Valve and Ex-Press Implants and Their Critical Comparisons. Current Eye Research, (just-accepted).

14 CFD studies (Solely experimental ones are not shared here!) after doctoral thesis in 6 years !

ME and CFD

NEW STUDIES about CFD

Papers, proceedings and projects AFTER doctoral thesis:

7. "CFD Simulation of Turbulent Flow Around a Shrouded Spur Gear for Predicting Load-Independent Windage Power Losses", Proceedings of the 4th International Congress of Automotive and Transport Engineering: Chapter 3, Springer Nature Switzerland AG 2019, N. Burnete and B. O. Varga (Eds.): AMMA 2018, PAE, pp. 1–8, 2019. DOI: 10.1007/978-3-319-94409-8_3
8. "CFD Analysis and Optimal Sizing of Finned Surface on a Novel Combined Turbine-Peltier System", International Symposium On Automotive Science And Technology (ISASTECH2019), September 5 - 6, Ankara, (2019).
9. "Thermal analysis of an anti-icing system for a NACA 4412 airfoil", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
10. "Numerical investigation of the aerodynamic performance of a low Reynolds number S809 wind turbine airfoil", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
11. "Numerical investigation of jet orientation using co-flow thrust vectoring with Coanda effect", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
12. "A Navier Stokes solver for compressible turbulent flows on quadtree and octree based Cartesian grids", Journal of Applied Fluid Mechanics,12:3,539-549, (2019).DOI: 10.29252/jafm.12.02.29156 (SCI-E)
13. " Experimental investigation and numerical verification of Coanda effect on curved surfaces using co-flow thrust vectoring", IAREJ Journal – 15-04-2021 (TR-Dizin)
14. Kara, E., Turhan, S., Kutlar, A. İ., & Güngör, K. (2021). Cor Vitro Microfluidic Experimental Setup on Testing Molteno, Critical Comparisons. Current Eye Research, (just-accepted).

14 CFD studies (Solely experimental ones are not given here!) after doctoral thesis in 6 years !

This study will be given in a Master Thesis (**vacant**)

ME and CFD

NEW STUDIES about CFD

Papers, proceedings and projects AFTER doctoral thesis:

7. "CFD Simulation of Turbulent Flow Around a Shrouded Spur Gear for Predicting Load-Independent Windage Power Losses", Proceedings of the 4th International Congress of Automotive and Transport Engineering: Chapter 3, Springer Nature Switzerland AG 2019, N. Burnete and B. O. Varga (Eds.): AMMA 2018, PAE, pp. 1–8, 2019. DOI: 10.1007/978-3-319-94409-8_3
8. "CFD Analysis and Optimal Sizing of Finned Surface on a Novel Combined Turbine-Peltier System", International Symposium On Automotive Science And Technology (ISASTECH2019), September 5 - 6, Ankara, (2019).
9. "Thermal analysis of an anti-icing system for a NACA 4412 airfoil", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
10. "Numerical investigation of the aerodynamic performance of a low Reynolds number S809 wind turbine airfoil", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
11. "Numerical investigation of jet orientation using co-flow thrust vectoring with Coanda effect", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
12. "A Navier Stokes solver for compressible turbulent flows on quadtree and octree based Cartesian grids", Journal of Applied Fluid Mechanics,12:3,539-549, (2019).DOI: 10.29252/jafm.12.02.29156 (SCI-E)
13. " Experimental investigation and numerical verification of Coanda effect on curved surfaces using co-flow thrust vectoring", IAREJ Journal – 15-04-2021 (TR-Dizin)
14. Kara, E., Turhan, S., Kutlar, A. İ., & Güngör, K. (2021). Cor Vitro Microfluidic Experimental Setup on Testing Molteno, Critical Comparisons. Current Eye Research, (just-accepted)

14 CFD studies (Solely experimental ones are not given here!) after doctoral thesis in 6 years !

This study will be verified using System Identification in a Doctoral Thesis (**vacant**)

ME and CFD

NEW STUDIES about CFD

Papers, proceedings and projects AFTER doctoral thesis:

7. "CFD Simulation of Turbulent Flow Around a Shrouded Spur Gear for Predicting Load-Independent Windage Power Losses", Proceedings of the 4th International Congress of Automotive and Transport Engineering: Chapter 3, Springer Nature Switzerland AG 2019, N. Burnete and B. O. Varga (Eds.): AMMA 2018, PAE, pp. 1–8, 2019. DOI: 10.1007/978-3-319-94409-8_3
8. "CFD Analysis and Optimal Sizing of Finned Surface on a Novel Combined Turbine-Peltier System", International Symposium On Automotive Science And Technology (ISASTECH2019), September 5 - 6, Ankara, (2019).
9. "Thermal analysis of an anti-icing system for a NACA 4412 airfoil", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
10. "Numerical investigation of the aerodynamic performance of a low Reynolds number S809 wind turbine airfoil", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
11. "Numerical investigation of jet orientation using co-flow thrust vectoring with Coanda effect", Fifth International Conference on Advances in Mechanical engineering (ICAME 2019), December 17 - 19, İstanbul, (2019).
12. "A Navier Stokes solver for compressible turbulent flows on quadtree and octree based Cartesian grids", Journal of Applied Fluid Mechanics,12:3,539-549, (2019).DOI: 10.29252/jafm.12.02.29156 (SCI-E)
13. " Experimental investigation and numerical verification of Coanda effect on curved surfaces using co-flow thrust vectoring", IAREJ Journal – 15-04-2021 (TR-Dizin)
14. Kara, E., Turhan, S., Kutlar, A. İ., & Güngör, K. (2021). Cor Vitro Microfluidic Experimental Setup on Testing Molteno, Critical Comparisons. Current Eye Research, (just-accepted)

14 CFD studies (Solely experimental ones are not given here!) after doctoral thesis in 6 years !

This study will be validated experimentally/numerically in two Master Theses (one **student selected, one is vacant**).

YOU and CFD

- Two successful AE433 students will be supervised next semester in two AE499 bachelor theses.
- One will work with ANSYS, other will work with OpenFOAM.
- The topics are not determined yet.

- Next year, one student will be supervised in a master thesis about a new CFD study by using AVL CFD package.

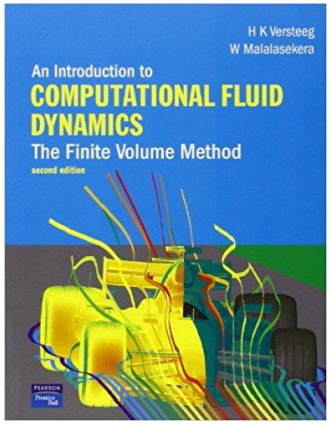
Objective

- The main objective of the course is to give the students a thorough knowledge of CFD from the theoretical and practical points of view.
- During the course, we will cover RANS models and scale-resolving simulations (DES and LES). We will also address accuracy and reliability of CFD simulations, as well as discretization techniques, solution strategies, and best standard practices when conducting CFD simulations.
- The class will be taught using textbooks, lecture notes, and the commercial CFD package and its modules.

Compulsory Text

«An introduction to computational fluid dynamics: the finite volume method.»

Versteeg, H. K., & Malalasekera, W. (2007). Pearson Education.

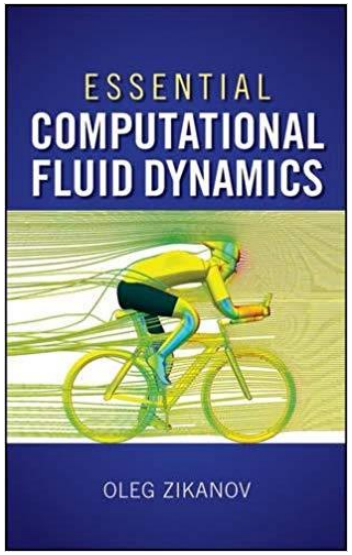


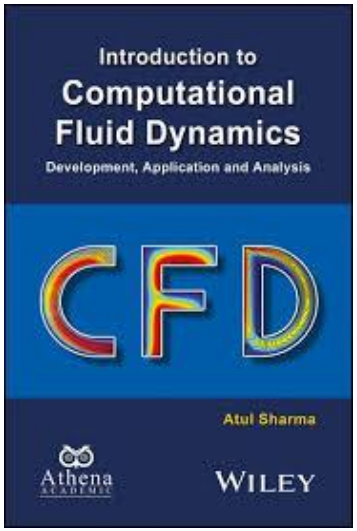
Chapters to be included to our syllabus:

- Chapter 1
- Chapter 2
- Chapter 3
- Chapter 10

Recommended Texts

- Introduction to Computational Fluid Dynamics: Development, Application and Analysis, by Sharma.
- Essential Computational Fluid Dynamics, by Zikanov
- Computational Fluid Dynamics - A Practical Approach, by Tu, Yeoh and Liu
- Computational Methods for Fluid Dynamics, Ferziger and Peric
- Computational Fluid Mechanics and Heat Transfer, by Pletcher, Tannehill and Anderson





Recommended Texts for Turbulence Modeling

- D. Wilcox. Turbulence Modeling for CFD. DCW Industries Inc., 2010.
- S. Pope. Turbulent Flows. Cambridge University Press, 2000.
- J. Mathieu and J. Scott. An Introduction to Turbulent Flow. Cambridge University Press, 2000.
- Lars Davidson. Turbulence modeling notes.
- NASA Turbulence Modeling Resources.
- The documentation of Ansys Fluent (user guide and theory guide).

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., FLUENT, CFX, 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/
  - CFD software
    - ANSYS: http://www.ansys.com/
    - COMSOL http://www.comsol.com/
    - Star-CCM - CD-adapco: http://www.cd-adapco.com/
  - Grid generation software
    - Cadence - Gridgen: http://www.pointwise.com
    - GridPro: http://www.gridpro.com/
  - Visualization software
    - Tecplot: http://www.tecplot.com/
    - Fieldview: http://www.ilight.com/

Software and Resources

- Our faculty has two licenses to use:

- 1. Undergraduate (ANSYS Fluent 17.0): Students can use it in the COMLAB.

Software and Resources

- Our faculty has two licenses to use:

- 1. Undergraduate (ANSYS Fluent 17.0): Students can use it in the COMLAB.

ANSYS® FLUENT®

- 2. Graduate (AVL Fire and Fire M – 5 students limited – 3 students left): Graduate students can contact me about the information.

Simulation Challenging Reality
AVL FIRE™ M
CFD Simulation in a Multi-Domain Environment

Software and Resources

- Our faculty has two licenses to use:

- 1. Undergraduate (ANSYS Fluent 17.0): Students can use it in the COMLAB.

ANSYS® FLUENT®

- 2. Graduate (AVL Fire and Fire M – 5 students limited – 3 students left): Graduate students can contact me about the information.

Simulation Challenging Reality
AVL FIRE™ M
CFD Simulation in a Multi-Domain Environment

- We will use ANSYS Fluent Student edition in AE433 class and if possible (on-site lectures) ANSYS Fluent 17.0 in COMLAB. If graduate students select a CFD-based study, then they can use AVL Fire & Fire M.

Assessment (LOOK OUT !)

- Attendance
- Labs
- 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 homework, quizzes, 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 Lab sessions in the second half of the semester, after the midterm. In each lab, there will be one demonstration, one homework.

Assessment (LOOK OUT !)

- Examinations:

Midterm will be about CFD concepts, definitions (**multiple choice**), and short exercises plus section with short computational problems (**HAND WRITTEN**).

There will be 4 homeworks related to the tutorials given in class.

Final exam will be the reports/presentations in PC lab (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):

  It consists of a report submission to the instructor and project presentation by each student to the whole class.

  Each CFD project will be performed by one student, **no group studies are allowed for this semester**.

  CFD problem to be analyzed will be chosen by the students based upon their own interests and with advice from the instructor.

Assessment (LOOK OUT !)

- FINAL (Term Project):

PROJECT OUTLINE AND RULES WILL BE GIVEN AT LAST LECTURE.

Gradings and due dates:

Presentation (10 pts)	final exam date
Successful CFD execution (15 pts)	final exam date
Report (15 pts)	last day of the finals

Final project report will include:

- Literature survey
- Project realization
- Selecting experimental study under the instructor's supervision
- Computational study
- Comparison of numerical results with experimental study

Assessment (LOOK OUT !)

- Final Grades:

Midterm	30	%
Homeworks	30	%
Final (Term Project)	40	%
Total	100	%

Letter grades will be given relative to the average of the class !

Tentative Schedule

Lecture	Lecture Title	Class	Homework
Week-1	Introduction	GAUZEM	-
Week-2	Basic aspects of CFD	GAUZEM	Reading: Ch.1
Week-3	Conservation laws of fluid motion & boundary conditions	GAUZEM	Reading: Ch.2 & derivations for some eqns
Week-4	Turbulence and its modelling – Part-1	GAUZEM	Download ANSYS Student Edition
Week-5	**Turkish Republic Anniversary (Oct 29th)**	**Holiday**	**Holiday**
Week-6	Turbulence and its modelling – Part-2	COMLAB	Watch and apply 11all supplementary videos till Week-10
Week-7	Turbulence and its modelling – Part-3	COMLAB	Reading: Ch.3 and some derivations
Week-8	Errors and uncertainty in CFD modelling	COMLAB	Reading: Ch.10.1-10.7
Week-9	Midterm 26.11.2021 9.00	COMLAB	-
Week-10	Introduction to COM LAB and Tutorial-1	COMLAB	Homework-1 - Submit just before next lecture
Week-11	Tutorial-2	COMLAB	Homework-2 - Submit just before next lecture
Week-12	Tutorial-3	COMLAB	Homework-3 - Submit just before next lecture
Week-13	Tutorial-4	COMLAB	Homework-4 - Submit just before next lecture
Week-14	Tutorial-5	COMLAB	Select your Project for the final