Advanced Computational Fluid Dynamics AE 508

Week-1

by

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

Advanced Computational Fluid Dynamics AE 508

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

FIRST OF ALL
AE 508 Advanced CFD Lecture

- Background needed:
  - **AE433 or AE427 CFD BACKGROUND OR SIMILAR LECTURE**

(I SHOULD APPROVE IT. OTHERWISE YOU SHOULD **DROP** YOUR REGISTRATION.)

FIRST OF ALL
AE 508 Advanced CFD Lecture

- Background needed:
  - **AE433 or AE427 CFD BACKGROUND OR SIMILAR LECTURE**

  (I SHOULD APPROVE IT. OTHERWISE YOU SHOULD **DROP** YOUR REGISTRATION.)

  - Undergraduate Numerical Analysis
  - Fluid Mechanics, Heat Transfer
  - Aerodynamics I-II
  - Basic computer skills in MATLAB
  - Basic skills in any of the prestigous CFD softwares such as ANSYS Fluent, AVL Fire M, Star CCM+, OpenFOAM, SimScale, COMSOL Multiphysics, MSC Cradle

Class Information

Class: Thursday – 13:30-16:05

(S15-COMLAB) 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
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).

ME and CFD

NEW STUDIES about CFD

Papers, proceedings and projects AFTER doctoral thesis:

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. Kara, Emre, Erpulat, Hüdai, "Experimental investigation and numerical verification of Coanda effect on curved surfaces using co-flow thrust vectoring", International Advanced Researches and Engineering Journal, 5:1, 72-78, (2021). DOI: 10.35860/iarej.758397 (ULAKBİM – TR DİZİN)
14. Kara, Emre, Turhan, Sinan, Kutlar, A. İhsan and Güngör, Kıvanç, "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,47:1, 69–78, (2022). DOI: 10.1080/02713683.2021.1951298 (SCI-E)
15. Çanlıoğlu, İbrahim Efdal, Kara, Emre, "Computational Fluid Dynamics Study of Lift Enhancement on a NACA 0012 Airfoil Using A Synthetic Jet Actuator", 23rd Congress on Thermal Science and Technology, September 08 - 10, Gaziantep, (2021).
16. Çanlıoğlu, İ. E. & Kara, E. " Sentetik Jet Eyleyici Kullanarak NACA0012 Kanat Profilinde Kaldırma Kuvveti İyileştirmesinin Hesaplamalı Akışkanlar Dinamiği Çalışması ", Gazi Üniversitesi Mühendislik Mimarlık Fakültesi Dergisi , XX (X) , X-XX (202X) - Accepted

ME and CFD

NEW STUDIES about CFD

Papers, proceedings and projects AFTER doctoral thesis:

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. Kara, Emre, Erpulat, Hüdai, "Experimental investigation and numerical verification of Coanda effect on curved surfaces using co-flow thrust vectoring", International Advanced Researches and Engineering Journal, 5:1, 72-78, (2021). DOI: 10.35860/iarej.758397 (ULAKBİM – TR DİZİN)
14. Kara, Emre, Turhan, Sinan, Kutlar, A. İhsan and Güngör, Kıvanç, "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,47:1, 69–78, (2022). DOI: 10.1080/02713683.2021.1951298 (SCI-E)
15. Çanlıoğlu, İbrahim Efdal, Kara, Emre, "Computational Fluid Dynamics Study of Lift Enhancement on a NACA 0012 Airfoil Using A Synthetic Jet Actuator", 23rd Congress on Thermal Science and Technology, September 08 - 10, Gaziantep, (2021).
16. Çanlıoğlu, İ. E. & Kara, E. " Sentetik Jet Eyleyici Kullanarak NACA0012 Kanat Profilinde Kaldırma Kuvveti İyileştirmesinin Hesaplamalı Akışkanlar Dinamiği Çalışması ", Gazi Üniversitesi Mühendislik Mimarlık Fakültesi Dergisi , XX (X) , X-XX (202X) - Accepted

16 CFD studies (Solely experimental ones are not shared here!) after doctoral thesis in 7 years !

My Lab's Projects

- Anti-Icing on Wing Leading-Edges
- Coanda Effect for Thrust Vectoring
- Hybrid Rocket Engine Optimization
- Synthetic Jet Actuators on Wings

My Lab's Projects in Progress

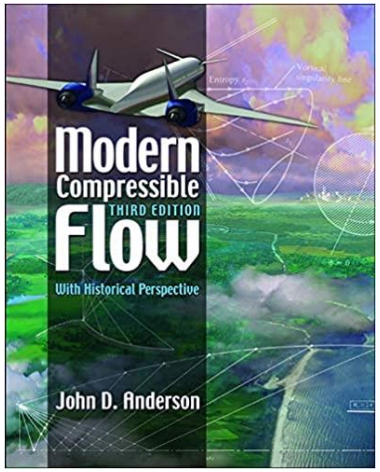
- Anti-Icing on Wing Leading-Edges (system identification study is **vacant**, **PhD**)
- Coanda Effect for Thrust Vectoring (**experimental** study is **vacant**, **MSc**)
- ~~Hybrid Rocket Engine Optimization (no vacancy, with MSc student)~~
- Synthetic Jet Actuators on Wings (3D CFD optimization study part is **vacant**, **MSc**)

All projects are/will be BAP or TÜBİTAK projects depending on the student.

Compulsory Textbooks

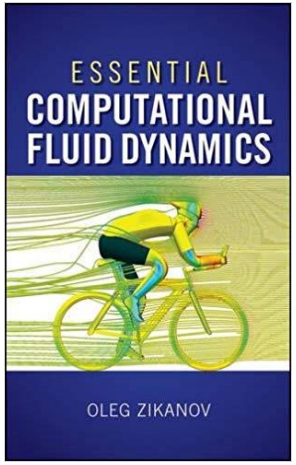
We will use textbooks (given in the next slide) and different sources for the lecture notes.

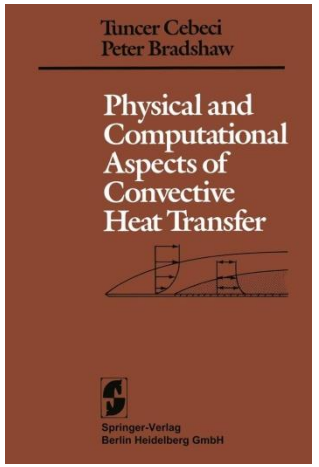
We will use «Modern Compressible Flow: With Historical Perspective» book (3rd edition) for the assignments.

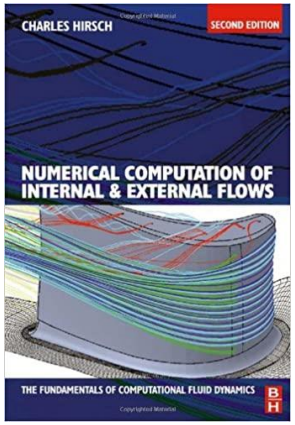


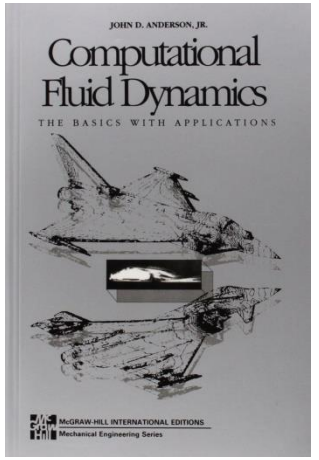
Reference Textbooks

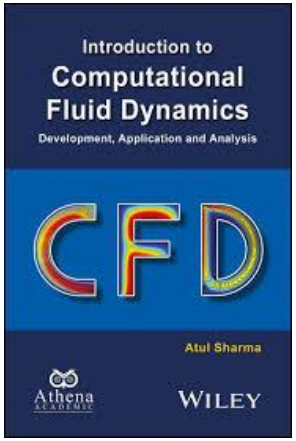
- Computational Fluid Dynamics: The Basics with Applications by Anderson
- Numerical Computation of Internal and External Flows, I and II by Hirsch
- Essential Computational Fluid Dynamics, by Zikanov
- Physical and Computational Aspects of Convective Heat Transfer by Cebeci et al.
- Introduction to Computational Fluid Dynamics: Development, Application and Analysis, by Sharma.
- Computational Methods for Fluid Dynamics, Ferziger and Peric
- 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., ANSYS, Star-CCM, COMSOL, OpenFOAM)
  - Research (e.g., SU2, CFDSHIP-IOWA, U^{2}RANS)
- 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. http://cfd2012.com/index.html
   - Useful source site by Dr. Ahmed Al Makky of Cardiff University. HE HAS MATLAB CODES FOR CFD!
6. https://www.learncax.com/
   - Useful source site for FREE CFD Training : Courses, Projects, Career .
7. https://www3.nd.edu/~gtryggva/ Conceptual part of this course is following the footsteps of Prof. Dr. Tryggvason!
   - Tryggvason is widely recognized for his contributions to computational fluid dynamics, including to the development of methods for computations of multiphase flows, as well as for pioneering direct numerical simulations of such flows.

AE 508 Advanced CFD Lecture

- Course goals are:
  - To learn how to solve the Euler and Navier-Stokes equations for engineering problems,
  - To know what the current state of CFD in literature is.

AE 508 Advanced CFD Lecture

- Course goals are:
  - To learn how to solve the Euler and Navier-Stokes equations for engineering problems,
  - To know what the current state of CFD in literature is.
  - NOT to learn how to use ANSYS
  - NOT to learn how to mesh
  - NOT to learn how to select turbulence models
  - NOT to learn how to apply tutorials

  So, do NOT see the lecture as «ANSYS lecture» or CFD beginner lecture. By the way, they (commercial CFD package use, meshing, turbulence model selection, etc) all are needed to be known prior to the AE508 class. They are given in AE427 and AE428 lectures.

AE 508 Advanced CFD Lecture

- Course goals are:
  - To learn how to solve the Euler and Navier-Stokes equations for engineering problems,
  - To know what the current state of CFD in literature is.

  HOW?

AE 508 Advanced CFD Lecture

- Course goals are:
  - To learn how to solve the Euler and Navier-Stokes equations for engineering problems,
  - To know what the current state of CFD in literature is.

  HOW?
  - Reading in detail the selected topics from reference textbooks (such as finite difference methods, accuracy considerations of CFD, stability of the convergence, etc)

AE 508 Advanced CFD Lecture

- Course goals are:
  - To learn how to solve the Euler and Navier-Stokes equations for engineering problems,
  - To know what the current state of CFD in literature is.

  HOW?
  - Reading in detail the selected topics from reference textbooks (such as finite difference methods, accuracy considerations of CFD, stability of the convergence, etc)
  - Practicing on the engineering problems (particularly compressible flow) using both one customized code (in MATLAB) and one CFD tool (ANSYS Fluent or OpenFOAM depending on the student's selection)

Assessment (LOOK OUT !)

- Assignments
- Midterm Exams
- Final Exam Project

Assessment (LOOK OUT !)

- Assignments (4 x 7.5 %):

  There will be four homeworks (two classic CFD, two computer-based) throughout the semester. Computer-based homeworks will be related tutorial given in COMLAB. You can use any CFD software but in class we will use the preinstalled ANSYS v.19.2 in COMLAB. Each assignment will be performed by one student, **no group studies are allowed**.

Assessment (LOOK OUT !)

- Midterms (15 % +15 %):
  - There will be two midterm exams considering in-lecture presentations and textbook. First midterm will be multiple-choice type, second midterm will be a CFD application using ANSYS Fluent.

- Final Examination (20 % + 20 %):
  - There will be a multiple-choice type final exam considering in-lab studies (20 %)
  - A final project type examination (details will be shared later) will be given according to the school schedules (20 %)

Assessment (LOOK OUT !)

- Final Grades:

Assignments	4 × 7.5	%
Midterm Exams	15+15	%
Final Exam+Project	20+20	%
Total	100	%

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

Lecture	Lecture Title	Homework
Week-1	Course Objectives and Syllabus	-
Week-2	Introduction, Mesh Generation and Solution Techniques, Basic Concepts: The Advection-Diffusion Equation, Finite Difference	-
Week-3	Accuracy, Stability, The Modified Equation, Multi-Dimensional Eqns	Read chapters 1,2 and 3 in the “Modern Compressible Flow with Historical Perspective by John D. Andersson”.
Week-4	Conservation Eqns, Solving the Navier-Stokes eqns using the streamfunction-vorticity formulation, Solving Partial Differential Equations: Characteristics and classification of PDEs: Theory,	Homework-1 (submit at the beginning of lecture on Week-5)
Week-5	The Wave Equation and Advection, Hyperbolic Equations with Solution Methods, Flux Vector Splitting	Read Godunov Theorem, The Sweby Diagram Limiters, ENO, WENO
Week-6	Higher order methods, Solving Partial Differential Equations: Artificial Viscosity, The Euler Equations	Read chapters 5, 6 and 7 in the “Modern Compressible Flow with Historical Perspective.
Week-7	Solving Partial Differential Equations: 2D Euler equations	-
Week-8	Solving Navier Stokes Equations	Homework-2 (submit at the beginning of lecture on Week-10)
Week-9	Midterm 1: Nov, 24th 2022 at 13.30 (COMLAB)	
Week-10	Tutorial-1	Homework-3 - (submit at the beginning of lecture on Week-12)
Week-11	Submit HW-3. Tutorial-2	-
Week-12	Midterm 2: Dec, 15th 2022 at 13.30 (COMLAB)	-
Week-13	Tutorial-3	Homework-4 - (submit at the beginning of lecture on Week-14). Presentation and Report rules will be revealed.
Week-14	Submit HW-4. Tutorial-4	Finalize Project selection for final & begin preparation

BEFORE WE START
LET'S CLEAR SOME SPACE.

- Background needed:
  - **AE427 CFD BACKGROUND OR SIMILAR LECTURE**

(I SHOULD APPROVE IT. OTHERWISE YOU SHOULD **DROP** YOUR REGISTRATION.)

So?

I am waiting for your responses.