Advanced Computational Fluid Dynamics AE 508

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/ae508/

Class Information

Class Hours: Tuesday – 09.25-12.00 (A14)

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).
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).

ME and CFD

NEW STUDIES about CFD

Papers, proceedings and projects AFTER doctoral thesis:

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. “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)
13. “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. “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).
15. “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)
16. “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)
17. «Determination of Optimum Parameter Space of a Fluidic Thrust Vectoring System based on Coanda Effect Using Gradient-Based Optimization Technique». Journal of Applied Fluid Mechanics, 16(10), 1974-1988, (2023). DOI: 10.47176/JAFM.16.10.1855 (SCI-E)
18. «Sentetik jet eyleyici kullanarak NACA0012 kanat profilinde kaldırma kuvveti iyileştirmesinin hesaplamalı akışkanlar dinamiği çalışması». Gazi Üniversitesi Mühendislik Mimarlık Fakültesi Dergisi, 38(3), 1821-1838, (2023). DOI: 10.17341/gazimmfd.1132881 (SCI-E)
19. "Design optimization of a fluidic thrust vectoring system based on coanda effect using meta-models" International Journal of Turbo & Jet-Engines, 2024. https://doi.org/10.1515/tjj-2024-0090 (SCI-E) - published online.
20. Parametric Analysis and Design Optimization of a Loudspeaker-Driven Synthetic Jet Actuator Using Response Surface Model in Quiescent Conditions. International Journal of Computational Methods, 2450068. (SCI-E) - published online.

ME and CFD

NEW STUDIES about CFD

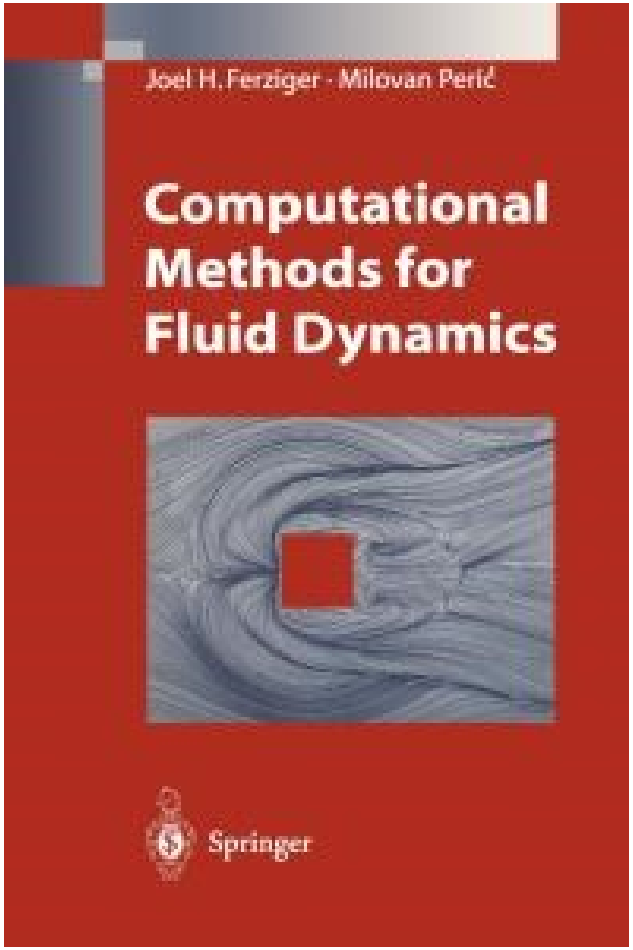
Papers, proceedings and projects AFTER doctoral thesis:

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. "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)
13. "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. "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).
15. "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)
16. "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)
17. «Determination of Optimum Parameter Space of a Fluidic Thrust Vectoring System based on Coanda Effect Using Gradient-Based Optimization Technique». Journal of Applied Fluid Mechanics, 16(10), 1974-1988, (2023). DOI: 10.47176/JAFM.16.10.1855 (SCI-E)
18. «Sentetik jet eyleyici kullanarak NACA0012 kanat profilinde kaldırma kuvveti iyileştirmesinin hesaplamalı akışkanlar dinamiği çalışması». Gazi Üniversitesi Mühendislik Mimarlık Fakültesi Dergisi, 38(3), 1821-1838, (2023). DOI: 10.17341/gazimmfd.1132881 (SCI-E)
19. "Design optimization of a fluidic thrust vectoring system based on coanda effect using meta-models" International Journal of Turbo & Jet-Engines, 2024. https://doi.org/10.1515/tjj-2024-0090 (SCI-E) - published online.
20. Parametric Analysis and Design Optimization of a Loudspeaker-Driven Synthetic Jet Actuator Using Response Surface Model in Quiescent Conditions. International Journal of Computational Methods, 2450068. (SCI-E) - published online.

20 CFD studies (Experimental ones are not shared here!) after doctoral thesis in 10 years !

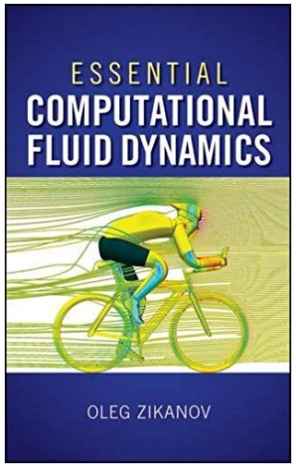
Compulsory Textbook

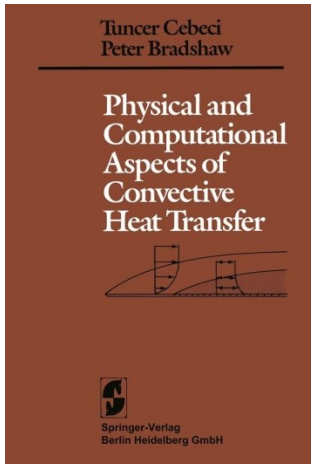
- Computational Methods for Fluid Dynamics, Ferziger and Peric
- You can download the online version of the first edition after logging in to proquest from Gaziantep University Server:
- https://ebookcentral.proquest.com/lib/gaziantep-ebooks/detail.action?docID=3097819

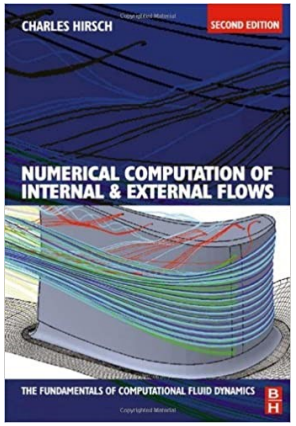


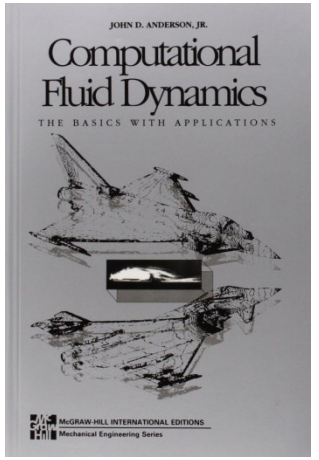
Reference Textbooks

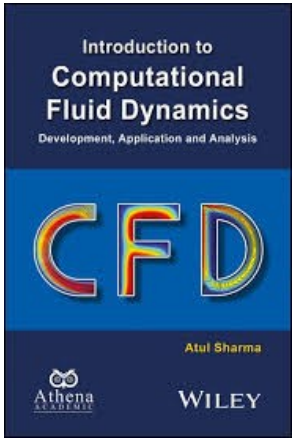
- Computational Fluid Dynamics: The Basics with Applications by Anderson
- Numerical Computation of Internal and External Flows, I and II by Hirsch
- Essential Computational Fluid Dynamics, by Zikanov
- Physical and Computational Aspects of Convective Heat Transfer by Cebeci et al.
- Introduction to Computational Fluid Dynamics: Development, Application and Analysis, by Sharma.
- Computational 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., FLUENT, CFX, 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. https://www.learncax.com/
   - Useful source site for FREE CFD Training : Courses, Projects, Career .
6. https://community.sw.siemens.com/s/topic/0TOVb0000000GETOA2/the-peri%C4%87-lectures-on-cfd
   - The Perić Lectures on CFD cover a wide range of essential topics, from the fundamentals of fluid dynamics to advanced numerical methods and their implementation.

AE 508 Advanced CFD Lecture

- This course includes detailed analysis, proofs, and practical advices for CFD applications.
- The course is designed for students who want to deepen their understanding of CFD.

AE 508 Advanced CFD Lecture

- Background needed:
  - Undergraduate Numerical Analysis and Fluid Mechanics
  - Basic computer skills in coding (such as Fortran 77/90/2003, Python, C++, MATLAB programs)

Assessment (LOOK OUT !)

- Midterms
- Final Exam

Assessment (LOOK OUT !)

- Midterms:

Two midterms will be given. Both will be take-home exams including presentations of them the next week by each student. They will be CFD applications written, compiled and solved in a computer programming language selected by the student (Fortran, C++, Python, MATLAB, etc). For both, the resulting images and tables will be asked for. The students will have a limited time for powerpoint presentations of their results.

Assessment (LOOK OUT !)

- Final Examination:

A final exam will be given according to the school schedules. There will be two parts:

1) Lecture notes will be the base of the final exam including midterms given. This part will be closed book – closed notes type.

2) Student can bring their laptops for the second part of the final. They can use their laptop in the second part of the final.

Assessment (LOOK OUT !)

- Final Grades:

Midterm-1 (Take home)	25	%
Midterm-2 (Take home)	25	%
Final Exam	50	%
Total	100	%

Tentative Schedule

Week-1: Introduction
Week-2: Chapter 1 - Introduction to CFD
Week-3: Chapter 2 - Basic Equations of Fluid Dynamics and Heat Transfer
Week-4: Chapter 3 - Classification of PDEs and Introduction to FD Methods
Week-5: Chapter 4 - FD Methods
Week-6: Chapter 5 - Elliptic Equations (**TAKE HOME EXAM WILL BE GIVEN, students can ask questions. Take home exam will be finished by students till next week before presentations.**)
Week-7: Midterm-1 (Presentations)
Week-8: Chapter 6 - Parabolic Equations
Week-9: Chapter 7 - Hyperbolic Equations
Week-10: Chapter 8 - Stability Analysis
Week-11: Chapter 9 - Vorticity-Stream Function Formulations & Chapter 10 - Solutions of Navier Stokes Equations by FDM
(**TAKE HOME EXAM WILL BE GIVEN, students can ask questions. Take home exam will be finished by students till next week before presentations.**)
Week-12: Midterm-2 (Presentations)
Week-13: Chapter 11-Finite Volume Method - Part 1
Week-14: Chapter 12-Finite Volume Method – Part 2