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 – 13.30-16.00

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

ME and CFD

NEW STUDIES about CFD

Papers, proceedings and projects AFTER doctoral thesis:

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. "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)

ME and CFD

NEW STUDIES about CFD

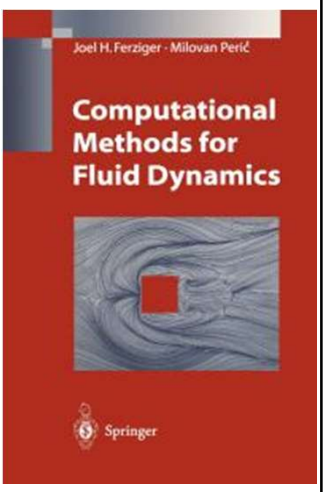
Papers, proceedings and projects AFTER doctoral thesis:

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. "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)

18 CFD studies (Experimental ones are not shared here!) after doctoral thesis in 9 years !

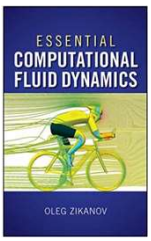
Compulsory Textbook

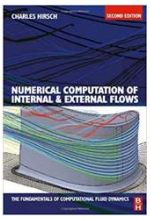
- Computational Methods for Fluid Dynamics, Ferziger and Peric
- You can download the online version of the first edition after logging in to proquest from Gaziantep University Server:
- https://ebookcentral.proquest.com/lib/gaziantep-ebooks/detail.action?docID=3097819

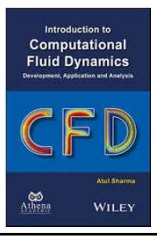


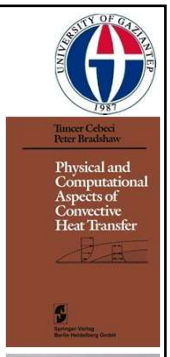
Reference Textbooks

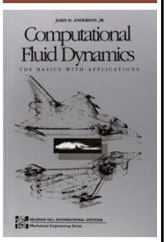
- Computational Fluid Dynamics: The Basics with Applications by Anderson
- Numerical Computation of Internal and External Flows, I and II by Hirsch
- Essential Computational Fluid Dynamics, by Zikanov
- Physical and Computational Aspects of Convective Heat Transfer by Cebeci et al.
- Introduction to 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.

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 This course is following the footsteps of Prof. Dr. Peric!
   - 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 is based on the book Computational Methods for Fluid Dynamics by Ferziger, Perić & Street and includes detailed analysis, examples, and practical advice for CFD applications.
- The course is designed for students who want to deepen their understanding of CFD.
- The material includes PowerPoint presentations, transcripts, sample codes, and simulation files for Simcenter STAR-CCM+.

AE 508 Advanced CFD Lecture

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

Assessment (LOOK OUT !)

- Assignments (Homeworks)
- Midterms
- Final Exam

Assessment (LOOK OUT !)

- Assignments:

There will be three homeworks throughout the semester. Each CFD homework should be performed by one student, **no group studies are allowed**. They will be solved using Star-CCM+ and compared with experimental and/or theoretical references from literature.

Assessment (LOOK OUT !)

- Midterms:

Two midterms will be given. Both will be take-home exams.

Assessment (LOOK OUT !)

- Final Examination:

A final exam (multiple choices) will be given according to the school schedules.

Assessment (LOOK OUT !)

- Final Grades:

Assignments (Homeworks)	15	%
Midterm-1 (Take home)	20	%
Midterm-2 (Take home)	25	%
Final Exam	40	%
Total	100	%

Tentative Schedule

Week-1: Introduction & Star-CCM+ Installation
Week-2: Lecture-1: Approaches to solving flow problems & Lecture-2: Introduction to CFD methods and error estimation
Week-3: Lecture-3: Governing equations and their properties & Lecture-4: Introduction to finite-difference methods
Week-4: Lecture-5: Advanced FD methods and error analysis & Lecture-6: Application examples for FD methods – **HW1**
Week-5: HOLIDAY (Oct 29th)
Week-6: Lecture-7: Introduction to finite-volume (FV) methods & Lecture-8: Advanced FV methods and error analysis
Week-7: Lecture-9: Application examples for FV methods & Lecture-10: Introduction to solution methods for algebraic equation systems (**TAKE HOME EXAM WILL BE GIVEN AT THE END OF THE LECTURE, take home exam will be finished by students in two weeks**) **12 Nov 2024 till 26 Nov 2024**
Week-8: Finish Midterm-1 till next week (No lecture this week)
Week-9: Submit your midterm-1 exam before lecture starts! - Lecture-11: Advanced solution methods for algebraic equation systems & Lecture-12: Application examples for solving algebraic equation systems – **HW2**
Week-10: Lecture-13: Introduction to time integration methods & Lecture-14: Advanced time integration methods
Week-11: Lecture-15: Application examples for solving unsteady scalar transport problems & Lecture-16: Introduction to solution methods for the Navier-Stokes equations
Week-12: Lecture-17: Fractional-step methods (FSM) for the solution of Navier-Stokes equations & Lecture-18: SIMPLE and related methods for the solution of Navier-Stokes equations (**TAKE HOME EXAM WILL BE GIVEN AT THE END OF THE LECTURE, take home exam will be finished by students in two weeks**) **17 Dec 2024 till 31 Dec 2024**
Week-13: Finish Midterm-2 till next week (No lecture this week)
Week-14: Submit your midterm-2 exam answer sheets before lecture starts! - Lecture 19: Examples of Solutions of Navier-Stokes Equations – Steady-State Flows – **HW3**
Week-15: Lecture 20: Examples of application of FSM and SIMPLE for computing unsteady 2D and 3D flows using Cartesian grids