ME 6189: Computational Fluid Dynamics

- 3.00 credit hours (Saturday: 5:00-6:30 PM; Wednesday: 6:30-8:00 PM)
- **Course content**

  Equations of motion, Discretisation, Solution algorithm,

  Parabolic and parabolic-elliptic flows,

  Turbulent flows calculation,

  Handling of irregular geometry.

- **Course teacher**

Dr. M Zakir Hossain
Assistant Professor
Dept. of Mechanical Engineering
BUET, Dhaka-1000.
Room: M602 (EME Building)
Email: zakir (at) me.buet.ac.bd, zakir92 (at) yahoo.com
Web: http://teacher.buet.ac.bd/zakir

ME 6189: Computational Fluid Dynamics

- **Assessment**

  1. Assignment 30%
  2. Presentation* 20%
  3. Final Exam: 50%

  * Presentation of a journal paper. The paper should be from (i) ASME journals, (ii) Computers & Fluids, (iii) Numerical Heat Transfer (iv) International Journal of CFD, (v) Journal of Computational Physics (vi) International Journal for Numerical Methods in Fluids.

- **ACADEMIC OFFENCES**

  Students must write their assignments in their own words.

  If students take an idea, or a passage of text from book, journal, web etc, they must acknowledge this by proper referencing such as footnotes or citations.

  Plagiarism is a major academic offence.

ME 6189: Computational Fluid Dynamics

- **References**

1. An Introduction to Computational Fluid Dynamics (The finite Volume Method) by H K Versteeg & W Malalasekera, Publisher:Prentice Hall.

2. Numerical Heat Transfer and Fluid Flow by Suhas V. Patankar, Publisher: McGraw-Hill.

3. Computational Fluid Dynamics, The Basics with Applications by John D. Anderson Jr., Publisher: McGraw-Hill.

Fluid Dynamics: an introduction

- The science of fluid dynamics describes the motion of liquids and gases and their interaction with solid bodies.

- It is a broad, interdisciplinary field that touches almost every aspect of our daily lives, and it is central to much of science and engineering.

- Fluid dynamics impacts defense, homeland security, transportation, manufacturing, medicine, biology, energy and the environment.

- Predicting the flow of blood in the human body, the behavior of microfluidic devices, the aero-dynamic performance of airplanes, cars, and ships, the cooling of electronic components, or the hazards of weather and climate, all require a detailed understanding of fluid dynamics, and therefore substantial research.

Fluid Dynamics: an introduction

- The governing equations describing the fluid motions in a physical system are the Navier-Stokes equations.
  - have no general analytical solution, and
  - computational solutions are challenging.

What is CFD ?

- Computational Fluid Dynamics or CFD is the analysis of systems involving
  - fluid flow,
  - heat transfer and
  - associated phenomena (such as chemical reactions)

  by means of computer-based simulation.

- CFD uses powerful computers and applied mathematics to model fluid flow situations.

Applications of CFD

Industrial applications	Aerospace
	Architecture
	Automotive
	Biomedical
	Chemical and Process
	Combustion
	Electronics and computers
	Glass manufacturing
	HVAC (heat, ventilation and cooling)
	Petroleum
	Power
	Marine
	Mechanical
	Metallurgical
	Nuclear
	Train design
	Turbo machinery
	Water
Environmental applications	Atmospheric pollution
	Climate calculations
	Fire in buildings
	Oceanic flows
	Pollution of natural waters
	Safety
Physiological applications	Cadiovascular flows (heart, major vessels)
	Flow in lungs and breathing passages

Advantages of CFD: Wanot,1996

- It provides a detailed understanding of flow distribution, weight losses, mass and heat transfer, particulate separation, etc. Consequently, all these will give plant managers a much better and deeper understanding of what is happening in a particular process or system.
- It makes it possible to evaluate geometric changes with much less time and cost than would be involved in laboratory testing.
- It can answer many 'what if' questions in a short time.
- It is able to reduce scale-up problems because the models are based on fundamental physics and are scale independent.
- It is particularly useful in simulating conditions where it is not possible to take detailed measurements such as high temperature or dangerous environment in an oven.
- Since it is a pro-active analysis and design tool, it can highlight the root cause not just the effect when evaluating plant problems.

Common CFD codes

CFD code	Company	Web site
CFX	AEA Technology	http://www.software.aeat.com/cfx/
FLUENT	Fluent Inc	http://www.fluent.com/
PHOENICS	Concentration Heat & Momentum Ltd (CHAM)	http://www.cham.co.uk/
STAR-CD	Computational Dynamics Ltd	http://www.cd.co.uk
FLOW3D	Flow Science, Inc	http://www.flow3d.com
CFD-ACE	CFD Research Corporation	http://www.cfdrc.com
ICEM CFD	ICEM Technologies	http://icemcfd.com/icepak.html
AMI-VSAERO	Analytical Methods, Inc (AMI)	http://www.am-inc.com
STORESIM/TETMESH	Computational Mechanics Company, Inc	http://www.comco.com/
IGG™	NUMECA International SA	http://www.numeca.com/
TECPLOT	Amtec Engineering, Inc	http://www.amtec.com/
PAM-FLOW	Engineering Systems International SA	http://www.esi.com.au/
FLOVENT	Flomerics Inc (FLOVENT)	http://www.flomerics.com/
AVS/EXPRESS	Advanced Visual Systems, Inc	http://www.avs.com/
FLO++	Softflo	http://www.softflo.com/
CFD++	Metacomp Technologies, Inc	http://www.metacomptech.com/

References

- Bin Xia, Da-Wen Sun, “Applications of computational fluid dynamics (CFD) in the food industry: a review”, Computers and electronics in Agriculture, 34 (2002) 5–24.