- **Course: Applied Computational Fluid Dynamics**
- **Sessions: 3 hours+ 1.5 hours /week**
- **Instructor: Araz Sarchami**
- **Contact: Sarchami@mie.utoronto.ca**
- **Office: MB117**

Course Description:

The course is designed for Students with no or little CFD (Computational Fluid Dynamics) knowledge who want to learn CFD application to solve engineering problems. The course will provide a general perspective to the CFD and its application to fluid flow and heat transfer and it will teach the use of some of the popular CFD packages and provides Students will learn basics of CFD and will use that basic knowledge to learn Fluent Ansys CFD software. However it should be emphasized that software learning is not the objective of this class and it only will be used as a tool to teach student the concepts in a practical framework.

Most CFD packages have a variety of modules to deal with a specific type of flow. Students will be introduced to different modules and their specific applications. They will then be able to utilize the CFD package to simulate any particular problem. Ansys software will be the commercial package that will be used in this course. Ansys Fluent is the most common commercial CFD code available and most of the engineering companies use this code for their research & development and product analysis.

Course Objectives:

Students will be familiar with the following at the end of this class:

- Basics of computational fluid dynamics
- Governing Equations in computational fluid dynamics
- Discretization methods and numerical solutions:
    - FDM :Finite Difference Method
    - FEM: Finite Element method
    - FVM: Finite Volume Method
- General CFD simulation process

- Geometry modeling
- Mesh generation
- Various solution methods and their suitability for different engineering applications
- CFD Simulation of:
    - Basic fluid flows
    - 2D modeling
    - 3D Modeling
    - Heat transfer
    - Heat exchanger modeling
    - Turbulent modeling
    - Transient flows
    - User Defined Functions
    - Post Processing

Course Structure:

The course outline is designed in a way to give students the opportunity to learn the basic science of Computational fluid dynamics and at the same time get familiar with Ansys Fluent software. They will be able to apply their knowledge of the science in the format of a popular commercial CFD code to solve practical and industrial problems.

The class will be a combination of on-board teaching and in-lab training. The Basic CFD and its theoretical aspects will be taught on board while the Ansys Fluent will be taught in lab environment and with the actual software.

In each session students will focus on the theoretical aspects and science, and then they will learn how to apply them in Ansys fluent environment. Small educational problems will be chosen for each session and students will try to solve the problem using their knowledge of CFD and Ansys fluent software.

Course Outline:

The course is divided into 13 weeks as follows:

Week 1 &2

- Basic CFD

  - Introduction
    - What is CFD
    - Real world Application examples
  - Governing equations
  - Finite volume
  - Steady\transient
  - Inviscid\Viscous
  - Laminar\Turbulent
  - Boundary condition and its types
  - Initialization
  - CFD Procedure
    - Pre-processing
    - Solution
    - Post-processing

Week 3

- Geometrical Modeling:
  - Optimum geometry\Negative volume
  - 2D Geometry
    - Sketches and Planes
    - Transforms
    - Constraints
  - Axi-symmetrical
  - 3D
    - Bodies: states, types, status, inheritance
    - Booleans
    - 3D features

Week 4

- Mesh Generation
  - Intro
  - Various mesh types
    - Assembly level versus part/body level
    - Meshing by element shape
  - Meshing Strategy

        - What is a good mesh?
            - Mesh selection
        - CFD/Fluids meshing strategy
        - Mesh metrics
            - quality
    - Global and local mesh controls
        - Physics, solver, and relevance preference
        - Sizing
            - Smoothing
            - Transition
            - Span angle
            - Inflation algorithm
    - Boundary condition selection

Week 5

- General solution process – Ansys Fluent
    - Physical properties
    - Solution methods
    - Boundary and zone
    - Solvers
    - Monitors
- Boundary conditions and cell zone conditions
    - Boundary conditions
        - Wall conditions
        - Inlet/outlet
        - Pressure
        - Symmetry
    - Cell zone conditions
        - Fluid conditions
        - Sources in momentum, energy equations
    - Profiles
        - Boundary/Initial profiles

Week 6 & 7

- Physical properties
  - Density
    - general methods
    - Boussinesq approximation
    - Incomp. Ideal gas
  - Viscosity
    - General methods
    - Non-Newtonian fluids
  - Operating pressure
  - Reference parameters
- Solution methods
  - Overview
  - Spatial discretization methods
    - 1^{st}, 2^{nd} orders
    - Other schemes
  - Pressure based solver
    - Pressure-velocity coupling
    - Under relaxation factors
  - Solution limits
  - Hybrid and constant initialization
  - Steady, time dependent
  - Monitoring
    - Residuals
    - Surface, volume monitors
    - Flux and force
- Modeling basic flows
  - General setup
  - Steady/unsteady
  - Inviscid flows
  - Viscous laminar flow

Week 8 & 9

- Heat transfer
  - Intro
  - Solution strategies
  - Conduction, convection
  - Natural convection and buoyancy driven flows
  - Intro to heat exchanger modeling

Week 10

- Turbulence modeling
    - Various models
        - Choosing a turbulence model
        - Spallart alarms, K-e, k-w
    - Setup options: viscous heating,…
    - Turbulence BCs
    - Near wall treatments
        - Standard model
        - Scalable model
        - Enhanced wall model
        - Non-equilibrium model

Week 11 & 12

- Transient flows
    - 2D transient
    - 3D transient
    - initialization
    - Time marching
    - Numerical methods for transient flow
    - Transient flow post processing
- UDF
    - Basics of user defined functions
    - Programming
- Two benchmark cases as extra in class projects

Week 13

- Multiphase Flow
    - DPM
    - Eulerian
    - VOF

Resources

- "Computational Fluid Dynamics", T. J. Chung, Cambridge University Press
- "The Handbook of Fluid Dynamics", R. W. Johnson, CRC Press
- Ansys Fluent Guide
- Ansys User Portal: https://support.ansys.com/portal/site/AnsysCustomerPortal
- CFD Forum: http://www.cfd-online.com/

Evaluation

- Mid-term Project: 35% (Individual)
- Final Project: 65% (Individual)