Introduction to Computational Fluid Dynamics

Taygun Recep Güngör, PhD

İstanbul Teknik Üniversitesi

Welcome to the Course

Taygun Recep Güngör

Education

- B.Sc. Mechanical Engineering, ITU, Turkey, 2011

- M.Sc. Mechanical Engineering, ITU, Turkey, 2014

- M.Sc. Fluid Dynamics, The von Karman Institute, Belgium, 2015

- Ph.D. Aeronautics and Astronautics Engineering, ITU, Turkey, 2023

  Mechanical Engineering, Laval University, Canada, 2023
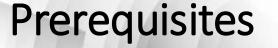
Welcome to the Course

Course level:

Introductory / Beginner Level

Target audience:

- Students, engineers, and professionals interested in the basics of computational fluid dynamics (CFD)

- Individuals with a background in engineering, physics, or mathematics who want an introductory understanding of CFD.

Goals

- Introduce the fundamental concepts and principles of computational fluid dynamics.

- Provide an overview of the steps involved in setting up and performing a basic CFD simulation.

- Demonstrate how CFD is applied to solve simple fluid flow problems.

- Familiarize participants with the terminology and key components of CFD analysis.

Prerequisites

- Basic understanding of calculus and differential equations.

- Fundamental knowledge of fluid mechanics principles.

What you will be able to do at the end

- Explain the basic concepts and significance of computational fluid dynamics.

- Understand the general workflow of a CFD simulation, including pre-processing, solving, and post-processing.

- Identify the key components involved in setting up a CFD simulation, such as boundary conditions and solver settings.

- Interpret basic CFD results and understand their implications in simple fluid flow scenarios.

What you will not be able to do at the end

- Detailed mathematical derivations of numerical methods used in CFD.

- Advanced CFD techniques like turbulence modeling, multi-phase flows, or chemical reactions.

- Proficiency in using specific commercial or open-source CFD software packages.

- How to set up and solve complex, real-world fluid dynamics problems using CFD.

Course overview

- Lecture 1: What is CFD?

- Lecture 2: Steps of a CFD simulation: Pre-processing

- Lecture 3: Steps of a CFD simulation: Solving and Post-processing

- Lecture 4: A test case: CFD of a turbulent boundary layer

Thanks