



# C EURO<sup>2</sup>

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.

- 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



This project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No 101101903. The JU receives support from the Digital Europe Programme and Germany, Bulgaria, Austria, Croatia, Cyprus, Czech Republic, Denmark, Estonia, Finland, Greece, Hungary, Ireland, Italy, Lithuania, Latvia, Poland, Portugal, Romania, Slovenia, Spain, Sweden, France, Netherlands, Belgium, Luxembourg, Slovakia, Norway, Türkiye, Republic of North Macedonia, Iceland, Montenegro, Serbia