UNIVERSITÀ DI PISA

DESTEC
DIPARTIMENTO DI INGEGNERIA
DELL'ENERGIA, DEI SISTEMI, DEL TERRITORIO E DELLE COSTRUZIONI

The use of the CFD commercial code ANSYS CFX for single phase applications: modelling approach and strategies with a focus on hydrogen technologies

Held by

Daniele Melideo, PhD
Assistant Professor

Venue: TBD

Dates: (tentative, depending on the availability of computer rooms):

1. Tuesday, March 5th, 2024; 9:30 – 12:30
2. Tuesday, March 12th, 2024 9:30 – 12:30
3. Tuesday, March 19th, 2024 9:30 – 12:30
4. Tuesday, March 26th, 2024 9:30 – 12:30

Abstract

Numerical simulations addressing practical issues across various aspects of our daily lives have been conducted for several decades. When coupled with experimental data, simulations serve as a valuable means to gain insights into physical phenomena across a diverse range of industrial and non-industrial applications. With the integration of Computational Fluid Dynamics (CFD) into both research and industrial domains, it has emerged as a highly effective and promising tool capable of providing accurate predictions for a wide array of practical phenomena, even within intricate systems. The key benefits of employing CFD include its cost-effectiveness in comparison to experiments, the ability to explore numerous parameters for a given problem without incurring significant additional costs, and the capacity to conduct comprehensive investigations of simulated phenomena by evaluating variables that may be challenging to precisely measure in experimental settings. On the other hand, CFD simulations can be computationally intensive, requiring substantial computing power, memory, and storage resources. This can lead to high operational costs, especially for complex simulations or large-scale problems; in addition, interpreting and setting up CFD simulations demand expertise in both fluid dynamics and numerical methods, and skilled professionals are needed to ensure accurate simulation results.

This seminar is addressed to MS students, PhD students, post-doc researchers interested in delving into the application of ANSYS CFX, mainly in the context of hydrogen-related scenarios; particular attentions will be dedicated to the model creations, mesh strategies and the validation process. The seminar will include hands-on demonstrations and the practical implementation of case studies.

Agenda

Day 1

- Introduction of Computational Fluid Dynamics (CFD)
- Governing equations in fluid dynamics
- Initial and boundary conditions
- Turbulence modelling
- Validation and sensitivity analysis of CFD results
- Overview of ANSYS CFX Software

Day 2

- Application of CFD for hydrogen technologies
- Hydrogen dispersion in confined spaces
- Fast filling and emptying of hydrogen tank

Day 3

- Importance of meshing in CFD
- Techniques for mesh generation
- Creation of different types of mesh using Pointwise

Day 4

- Setting up a CFD simulation
- Hands-on demonstration of a CFD case study