Master's Degree
Computational Fluid Mechanics

Master’s Degree
Computational Fluid Mechanics

» Modality: **online**
» Duration: **12 months**
» Certificate: **TECH Global University**
» Credits: **60 ECTS**
» Schedule: **at your own pace**
» Exams: **online**

Website: **www.techtitute.com/us/engineering/master-degree/master-degree-computational-fluid-mechanics**

Index

01
Introduction

Simulation has become one of the fundamental elements of science, proving to be very reliable for validating theories and replacing experiments that would normally be more costly and time-consuming to carry out, obtaining much more optimal designs. This is why Computational Fluid Dynamics (CFD) is so relevant and why engineers with extensive and specific knowledge of its development are in demand. And this is the reason why TECH has designed a program that equips the student with the necessary skills to be able to face a professional future in this area, with the maximum possible efficiency. All this, through a 100% online content that addresses topics such as advanced models, methods and advanced mathematics for CFD.

Become an expert in Computational Fluid Mechanics in only 12 months"

Simulation has become one of the pillars of science and Computational Fluid Dynamics (CFD) is a computational technique that seeks to simulate the motion of fluids. This tool offers multiple advantages over other types of Fluid Mechanics studies, such as time savings, cost reduction in experiments, the possibility of analyzing conditions that are very complicated to simulate experimentally and a practically unlimited level of detail.

In order to know this technique in depth, it is necessary to acquire highly technical and specialized skills and knowledge in algorithms, methods and the models that make up a simulator. This is the reason why TECH has designed a Master's Degree in Computational Fluid Mechanics, to enable the student to work in this sector as a CFD developer or as an advanced user, through a global and specialized vision of the entire development environment.

Thus, throughout the syllabus, topics such as the origin of turbulence, GPU environments, iterative methods, finite volume methods or advanced methods for CFD, among many other highly relevant aspects, are addressed in depth. All this, in a comfortable 100% online modality that seeks to give students total freedom to organize their studies and schedules.

This program is comprised of multimedia content designed by the best experts in the field and updated information based on the most rigorous sources, in addition to the most innovative teaching technologies. All materials are available to the student from the first day, being able to access them with any device with internet connection, whether Tablet, mobile or computer.

This **Master's Degree in Computational Fluid Mechanics** contains the most complete and up-to-date program on the market. The most important features include:

- The development of case studies presented by experts in the Master's Degree program in Computational Fluid Mechanics
- The graphic, schematic and eminently practical contents of the system provide advanced and practical information on those disciplines that are essential for professional practice
- Practical exercises where self-assessment can be used to improve learning
- Its special emphasis on innovative methodologies
- Theoretical lessons, questions to the expert, debate forums on controversial topics, and individual reflection assignments
- Content that is accessible from any fixed or portable device with an Internet connection

Enhance your professional profile with new knowledge in CFD and stand out in a booming sector"

Deepen your knowledge and acquire new skills in compressible fluids and multiphase flow"

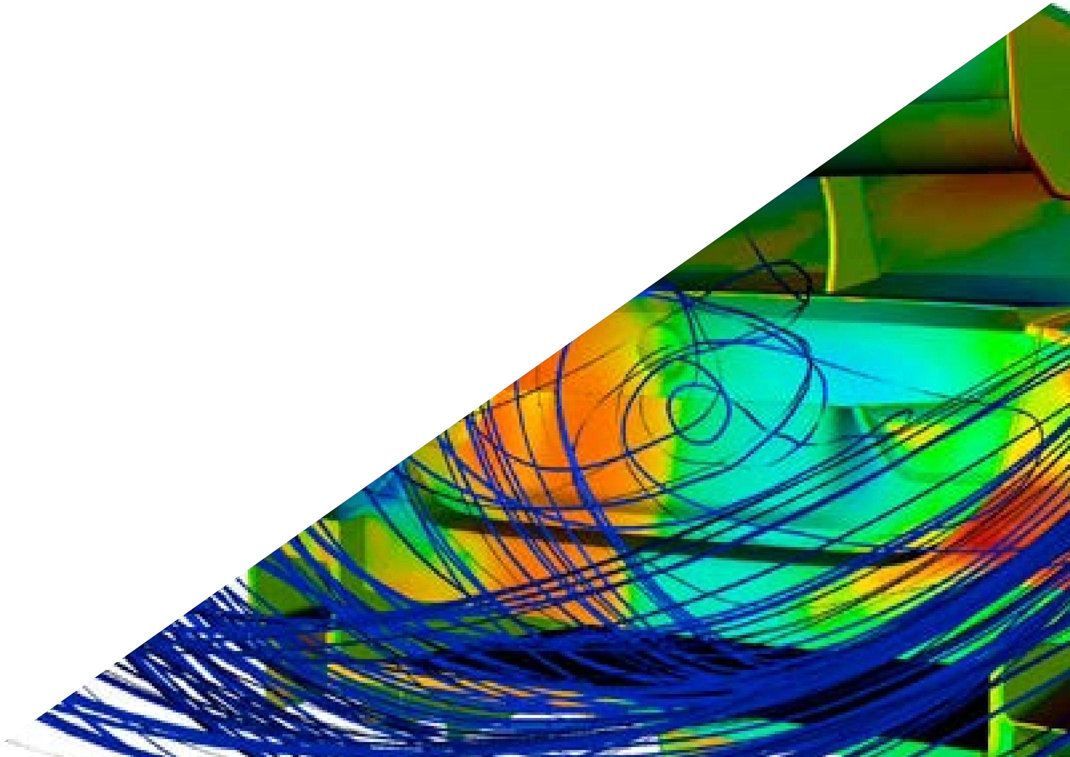
Learn all about advanced CFD models, thanks to the most complete theoretical and practical material.

Enroll now and get access to all the content on fluid turbulence modeling.

The program's teaching staff includes professionals from sector who contribute their work experience to this educational program, as well as renowned specialists from leading societies and prestigious universities.

Its multimedia content, developed with the latest educational technology, will provide the professional with situated and contextual learning, i.e., a simulated environment that will provide an immersive education programmed to learn in real situations.

The design of this program focuses on Problem-Based Learning, by means of which the professional must try to solve the different professional practice situations that are presented throughout the academic course. For this purpose, the student will be assisted by an innovative interactive video system created by renowned experts.



02

Objectives

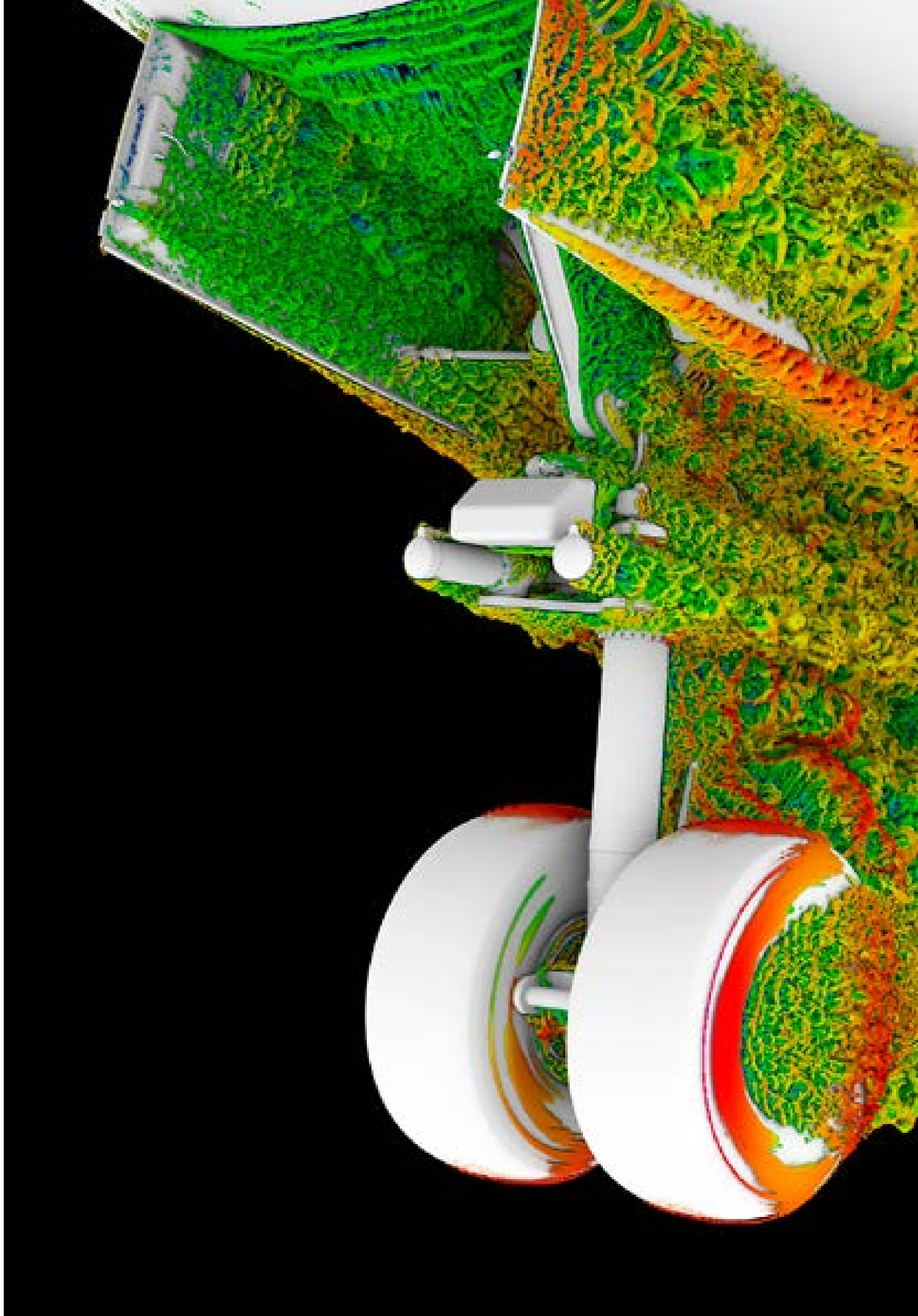
The objective of this Master's Degree in Computational Fluid Mechanics is to provide the student with specialized theoretical and practical knowledge on the development of Computational Fluid Mechanics simulators, including the complete ecosystem. In this way, the student will be able to face a professional future in this area, with total guarantee of success.

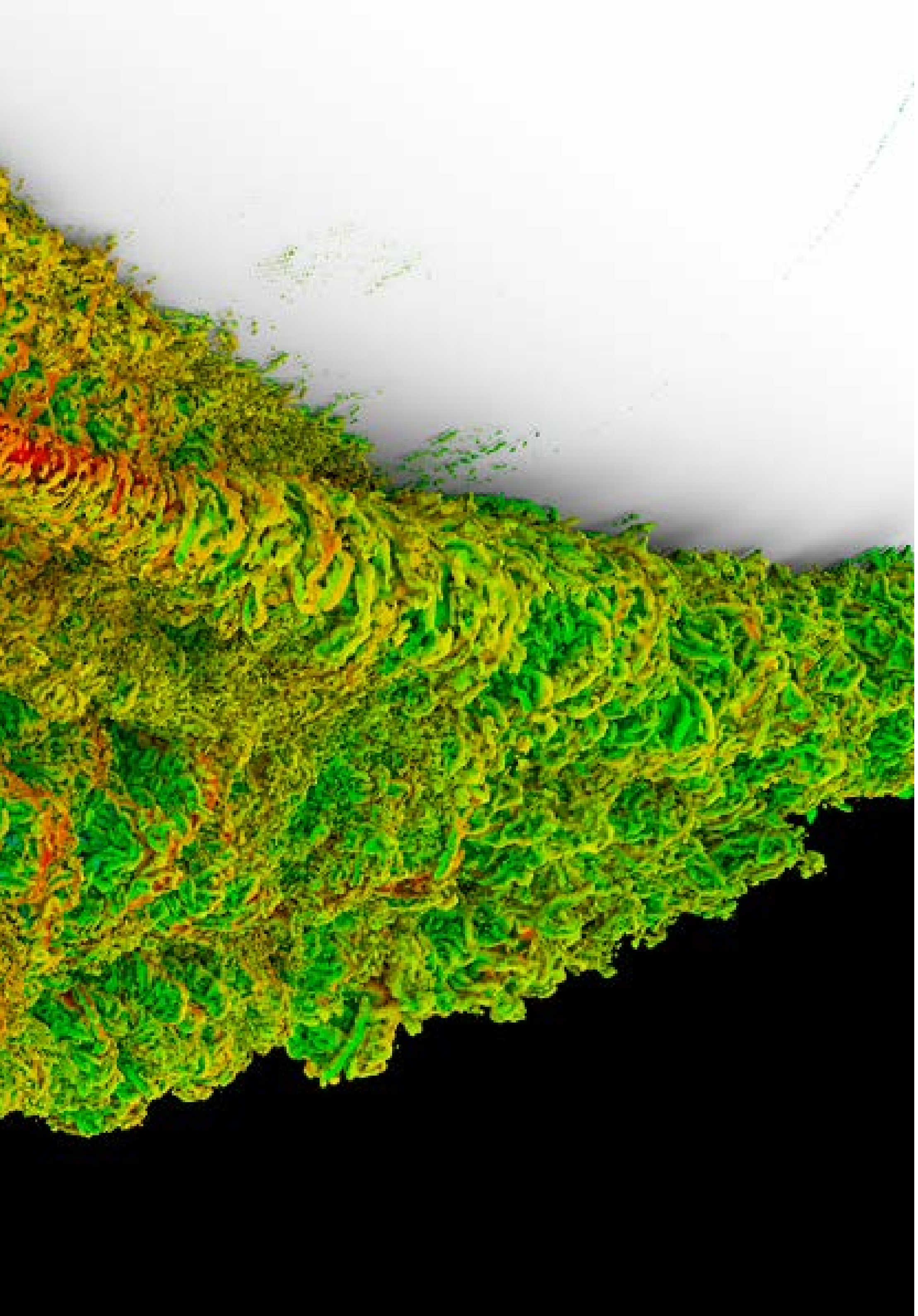
Give your career the boost it needs and specialize in one of the most promising areas of engineering"

General Objectives

- Establish the basis for the study of turbulence
- Develop CFD statistical concepts
- Determine the main computational techniques from turbulence research
- Generate specialized knowledge in the method of Finite Volumes
- Acquire specialized knowledge in fluid mechanics calculation techniques
- Examine the wall units and the different regions of a turbulent wall flow
- Determine the characteristics of compressible flows
- Examine multiple models and multiphase methods
- Develop expertise on multiple models and methods in multiphysics and thermal analysis
- Interpret the results obtained by correct post-processing

"You will achieve your most demanding objectives thanks to the most innovative tools in the field of CFD simulation"





Specific Objectives

Module 1. Fluid Mechanics and High-Performance Computing

- Identify the equations of turbulent flows
- Examine the closure problem
- Establish the dimensionless numbers needed for modeling
- Analyze the main CFD techniques
- Examine the main experimental techniques
- Developing the different types of supercomputers
- Show the future: GPU

Module 2. Advanced mathematics for CFD

- Develop the mathematical concepts of turbulence
- Generate specialized knowledge on the application of statistics to turbulent flows
- Fundamental method of solving CFD equations
- Demonstrate methods of solving algebraic problems
- Analyze the multigrid method
- Examining the use of eigenvalues and eigenvectors in CFD problems
- Determine methods for solving non-linear problems

Module 3. CFD in Research and Modeling Environments

- Analyzing the future of artificial intelligence in turbulence
- Apply classical discretization methods to Fluid Mechanics problems
- Determine the different turbulent structures and their importance
- Show the method of characteristics
- To present the effect of the evolution of supercomputing on CFD problems
- Examine the main open problems in turbulence

Module 4. CFD in Application Environments: Finite Volumes Methods

- Analyze the FEM or MVF environment
- Specify what, where and how the boundary conditions can be defined
- Determine possible time steps
- Concretizing and designing Upwind schemes
- Develop high order schemes
- Examine convergence loops and in which cases to use each one
- Expose the imperfections of CFD results

Module 5. Advanced Methods for CFD

- Develop the Finite Element Method and the Smoothed Particle Hydrodynamics Method
- Analyze the advantages of Lagrangian versus Eulerian methods, in particular, SPH vs. FVM
- Analyze the Monte Direct Simulation method and the Lattice-Boltzmann Method
- Evaluate and interpret spatial aerodynamics and microfluid dynamics simulations
- Establish the advantages and disadvantages of LBM versus the traditional FVM method

Module 6. Modeling of turbulence in Fluid

- Applying the concept of orders of magnitude
- Present the problem of closure of the Navier-Stokes equations
- Examine energy budget equations
- Develop the concept of turbulent viscosity
- Substantiate the different types of RANS and LES
- Present the regions of a turbulent flow
- Model the energy equation

Module 7. Compressible Fluids

- Develop the main differences between compressible and incompressible flow
- Examine typical examples of the occurrence of compressible fluids
- Identify the peculiarities in the solution of hyperbolic differential equations
- Establish the basic methodology for solving the Riemann problem
- Compile different resolution strategies
- Analyze the pros and cons of the different methods
- Present the applicability of these methodologies to the Euler / Navier-Stokes equations showing classical examples

Module 8. Multiphase flow

- Distinguish what type of multiphase flow is to be simulated: continuous phases, such as simulating a ship at sea, a continuous medium; discrete phases, such as simulating specific droplet trajectories and use statistical populations when the number of particles, droplets or bubbles is too large to be simulated
- Establish the difference between Lagrangian, Eulerian and mixed methods
- Determine the tools best suited to the type of flow to be simulated
- Modeling the effects of surface tension and phase changes such as evaporation, condensation or capitation
- Develop boundary conditions for wave simulation, learn about the different wave models and apply the so-called numerical beach, a region of the domain located at the exit whose objective is to avoid wave reflection

Module 9. Advanced CFD Models

- Distinguish what type of physical interactions are to be simulated: fluid-structure, such as a wing subject to aerodynamic forces, fluid coupled with rigid body dynamics, such as simulating the motion of a buoy floating in the sea, or thermofluid, such as simulating the distribution of temperatures in a solid subject to air currents
- Distinguish the most common data exchange schemes between different simulation software and when one or the other can or is best to be applied
- Examine the various heat transfer models and how they can affect a fluid
- Model convection, radiation and diffusion phenomena from a fluid point of view, model sound creation by a fluid, model simulations with advection-diffusion terms to simulate continuous or particulate media and model reactive flows

Module 10. Post-processing, validation and application in CFD

- Determine the types of post-processing according to the results to be analyzed: purely numerical, visual or a mixture of both
- Analyzing the convergence of a CFD simulation
- Establish the need for CFD validation and know basic examples of CFD validation
- Examine the different tools available on the market
- To provide a foundation for the current context of CFD simulation

03

Skills

The structure of this Master's Degree in Fluid Mechanics has been designed by experts in the field, who seek to provide students with the necessary skills and knowledge to be able to face the professional reality of this sector, with optimum quality in the work and the maximum ability to resolve the different difficulties that may arise. TECH offers complete, up-to-date and accurate content to meet the highest demands.

With the new skills you will acquire after completing the program, you will position yourself as an expert engineer in Computational Fluid Mechanics"

General Skills

- To know the main supercomputing techniques
- Identify and define the concept of residual
- Differentiate the different turbulent structures
- Optimal configuration of each simulation
- Obtain specialized knowledge in Fluid Mechanics calculation techniques
- Model the energy equation
- Identify the main numerical methods in solving the Riemann problem
- Choose the type of simulation or model to be applied that best suits the context and identify the pros and cons of each method
- Combining multiple strategies to get the best results where they are most needed
- Interpret the results obtained by correct post-processing

Specific Skills

- Developing the different types of supercomputers
- Determine methods for solving non-linear problems
- Apply classical discretization methods to Fluid Mechanics problems
- Specify what, where and how the boundary conditions can be defined
- Evaluate and interpret spatial aerodynamics and microfluid dynamics simulations
- Present the problem of closure of the Navier-Stokes equations
- Compile different resolution strategies
- Establish the difference between Lagrangian, Eulerian and mixed methods
- Distinguish the most common data exchange schemes between different simulation software and when one or the other can or is best to be applied
- Know the different tools available in the market

Improve your knowledge and skills with all the material available in the online Campus on post-processing, validation and application in CFD"

04

Course Management

In its maxim of offering an elite education for all, TECH has the best specialists to develop programs and curricula that meet the highest needs and demands. The professionals that make up the team of experts in Computational Fluid Mechanics have designed the contents based on their outstanding experience and extensive professional career, with the aim of providing the student with the best skills and the most complete knowledge on the subject.

An outstanding team of experts in the field, who have designed this program in CFD to give you a unique opportunity to face a successful future career in this sector"

Management

Dr. García Galache, José Pedro

- XFlow Development Engineer at Dassault Systèmes
- PhD in Aeronautical Engineering from the Polytechnic University of Valencia
- Degree in Aeronautical Engineering from the Polytechnic University of Valencia
- Master's Degree in Research in Fluid Mechanics from the Von Kármán Institute for Fluid Dynamics
- Short Training Program in the Von Kármán Institute for Fluid Dynamics

Professors

Dr. Espinoza Vásquez, Daniel

- Consultant Aeronautical Engineer at Alten SAU
- Freelance CFD and Programming Consultant
- CFD Specialist at Particle Analytics Ltd
- Research Assistant at the University of Strathclyde
- Teaching Assistant in Fluid Mechanics, University of Strathclyde
- Dr. in Aeronautical Engineering from the University of Strathclyde
- Master's Degree in Computational Fluid Mechanics, Cranfield University
- Degree in Aeronautical Engineering from Universidad Politécnica de Madrid

Mr. Mata Bueso, Enrique

- Senior Engineer of Thermal Conditioning and Aerodynamics at Siemens Gamesa
- Application Engineer and CFD R&D Manager in Dassault Systèmes
- Thermal Conditioning & Aerodynamics Engineer at Gamesa-Altran
- Fatigue and Damage Tolerance Engineer at Airbus-Atos
- R&D CFD Engineer at UPM
- Aeronautical Technical Engineer specializing in Aircraft at the UPM
- Master's Degree in Aerospace Engineering from the Royal Institute of Technology in Stockholm

Ms. Pérez Tainta, Maider

- Cement fluidization engineer at Kemex Ingesoa
- Process Engineer at J.M. Jauregui
- Researcher in hydrogen combustion at Ikerlan
- Mechanical Engineer at Idom
- Graduate in Mechanical Engineering from the University of the Basque Country (UPV)
- Master's Degree in Mechanical Engineering
- Interuniversity Master's Degree in Fluid Mechanics
- Python programming program program program

05

Structure and Content

The structure and content of this program have been designed by the reputed professionals that make up TECH's team of experts in Computational Fluid Mechanics. The syllabus has been created under the most efficient pedagogical methodology, Relearning, which guarantees the optimal assimilation of the contents by the students, in a natural, agile and precise way. All this has resulted in the most complete and innovative theoretical and practical materials on the academic market.

Quality content designed by CFD experts to help you achieve your most ambitious career goals as an engineer"

Module 1. Fluid Mechanics and High-Performance Computing

1.1. Dynamics of computational fluid mechanics
- 1.1.1. The origin of the turbulence
- 1.1.2. The need for modeling
- 1.1.3. CFD work process

1.2. The Equations of Fluid Mechanics
- 1.2.1. The continuity equation
- 1.2.2. The Navier-Stokes equation
- 1.2.3. The energy equation
- 1.2.4. The Reynolds averaged equations

1.3. The problem of closing equations
- 1.3.1. The Bousinesq hypothesis
- 1.3.2. Turbulent viscosity in a spray
- 1.3.3. CFD Modeling

1.4. Dimensionless numbers and dynamic similarity
- 1.4.1. Dimensionless numbers in fluid mechanics
- 1.4.2. The principle of dynamic similarity
- 1.4.3. Practical example: wind tunnel modeling

1.5. Turbulence Modeling
- 1.5.1. Direct numerical simulations
- 1.5.2. Simulations of large eddies
- 1.5.3. RANS Methods
- 1.5.4. Other Methods

1.6. Experimental Techniques
- 1.6.1. PIV
- 1.6.2. Hot wire
- 1.6.3. Wind and water tunnels

1.7. Supercomputing environments
- 1.7.1. Supercomputing of the future
- 1.7.2. Supercomputer operation
- 1.7.3. Tools for use

1.8. Software in parallel architectures
- 1.8.1. Distributed environments: MPI
- 1.8.2. Shared memory: GPU
- 1.8.3. Data engraving: HDF5

1.9. *Grid computing*
- 1.9.1. Description of computer farms
- 1.9.2. Parametric problems
- 1.9.3. Queuing systems in *grid computing*

1.10. GPU, the future of CFD
- 1.10.1. GPU Environments
- 1.10.2. GPU Programming
- 1.10.3. Practical example: artificial intelligence in fluids using GPUs

Module 2. Advanced mathematics for CFD

2.1. Fundamentals of Mathematics
- 2.1.1. Gradients, divergences and rotations. Total derivative
- 2.1.2. Ordinary Differential Equations
- 2.1.3. Partial derivative equations

2.2. Statistics
- 2.2.1. Averages and moments
- 2.2.2. Probability density functions
- 2.2.3. Correlation and energy spectra

2.3. Strong and weak solutions of a differential equation
- 2.3.1. Function bases. Strong and weak solutions
- 2.3.2. The finite volume method. The heat equation
- 2.3.3. The finite volume method. Navier-Stokes

2.4. Taylor's Theorem and Discretization in time and space
- 2.4.1. Finite differences in 1 dimension. Error order
- 2.4.2. Finite differences in 2 dimensions
- 2.4.3. From continuous equations to algebraic equations

2.5. Algebraic problem solving, LU method
- 2.5.1. Algebraic problem solving methods
- 2.5.2. The LU method on full matrices
- 2.5.3. The LU method in sparse matrices

2.6. Algebraic Problem Solving, w Methods I
   2.6.1. Iterative methods. Waste
   2.6.2. Jacobi's method
   2.6.3. Generalization of Jacobi's method
2.7. Algebraic problem solving, iterative methods II
   2.7.1. Multi-grid methods: V-cycle: interpolation
   2.7.2. Multi-grid methods: V-cycle: extrapolation
   2.7.3. Multi-grid methods: W-cycle
   2.7.4. Error estimation
2.8. Eigenvalues and eigenvectors
   2.8.1. The algebraic problem
   2.8.2. Application to the heat equation
   2.8.3. Stability of differential equations
2.9. Non-linear evolution equations
   2.9.1. Heat equation: explicit methods
   2.9.2. Heat equation: implicit methods
   2.9.3. Heat equation: Runge-Kutta methods
2.10. Stationary non-linear equations
   2.10.1. The Newton-Raphson method
   2.10.2. 1D Applications
   2.10.3. 2D Applications

Module 3. CFD in Research and Modeling Environments

3.1. Research in Computational Fluid Dynamics (CFD)
   3.1.1. Challenges in turbulence
   3.1.2. Advances in Chronic Obstructive Pulmonary Disease
   3.1.3. Artificial Intelligence
3.2. Finite differences
   3.2.1. Presentation and application to a 1D problem. Taylor's Theorem
   3.2.2. 2D Applications
   3.2.3. Boundary Conditions
3.3. Compact finite differences
   3.3.1. Objective SK Lele's article
   3.3.2. Obtaining coefficients
   3.3.3. Application to a 1D problem
3.4. The Fourier Transform
   3.4.1. The Fourier transform. From Fourier to the present day
   3.4.2. The FFTW package
   3.4.3. Cosine transform: Tchebycheff
3.5. Spectral methods
   3.5.1. Application to a fluid problem
   3.5.2. Pseudospectral methods: Fourier + CFD
   3.5.3. Placement methods
3.6. Advanced time discretization methods
   3.6.1. The Adams-Bamsford method
   3.6.2. The Crack-Nicholson method
   3.6.3. Runge-Kutta
3.7. Structures in turbulence
   3.7.1. The vortex
   3.7.2. The life cycle of a turbulent structure
   3.7.3. Visualization Techniques
3.8. The Characteristics Method
   3.8.1. Compressible Fluids
   3.8.2. Application A breaking wave
   3.8.3. Application: Burguers equation
3.9. CFD and supercomputing
   3.9.1. The memory problem and the evolution of computers
   3.9.2. Parallelization techniques
   3.9.3. Domain decomposition
3.10. Open problems in turbulence
   3.10.1. Modeling and the VonKarma constant
   3.10.2. Aerodynamics: boundary layers
   3.10.3. Noise in CFD problems

Module 4. CFD in Application Environments: Finite Volumes Methods

4.1. Finite Volume Methods
  4.1.1. Definitions in FVM
  4.1.2. Historical Background
  4.1.3. MVF in Structures
4.2. Source Terms
  4.2.1. External volumetric forces
    4.2.1.1. Gravity and centrifugal force
  4.2.2. Volumetric (mass) and pressure source term (evaporation, cavitation and chemical)
  4.2.3. Scalar source term
    4.2.3.1. Temperature and species
4.3. Applications of boundary conditions
  4.3.1. Input and Output
  4.3.2. Symmetry condition
  4.3.3. Wall condition
    4.3.3.1. Tax values
    4.3.3.2. Values to be solved by parallel calculation
    4.3.3.3. Wall models
4.4. Boundary Conditions
  4.4.1. Known boundary conditions: Dirichlet
    4.4.1.1. Scalars
    4.4.1.2. Diseases
  4.4.2. Boundary conditions with known derivative: Neumann
    4.4.2.1. Zero gradient
    4.4.2.2. Finite gradient
  4.4.3. Cyclic boundary conditions: Born-von Kármán
  4.4.4. Other boundary conditions: Robin
4.5. Temporary integration
  4.5.1. Explicit and implicit Euler
  4.5.2. Lax-Wendroff time step and variants (Richtmyer and MacCormack)
  4.5.3. from Runge-Kutta multi-stage time step
4.6. Upwind Schematics
  4.6.1. Riemann's Problem
  4.6.2. Main upwind schemes: MUSCL, Van Leer, Roe, AUSM
  4.6.3. Design of an upwind spatial scheme
4.7. High order schemes
  4.7.1. High-order discontinuous Galerkin
  4.7.2. ENO and WENO
  4.7.3. High order schemes. Advantages and Disadvantages
4.8. Pressure-velocity convergence loop
  4.8.1. PISO
  4.8.2. SIMPLE, SIMPLER and SIMPLEC
  4.8.3. PIMPLE
  4.8.4. Transient loops
4.9. Moving contours
  4.9.1. Overlocking techniques
  4.9.2. Mapping: mobile reference system
  4.9.3. *Immersed boundary method*
  4.9.4. Overlapping meshes
4.10. Errors and uncertainties in CFD modeling
  4.10.1. Precision and accuracy
  4.10.2. Numerical errors
  4.10.3. Input and physical model uncertainties

Module 5. Advanced Methods for CFD

5.1. Finite Element Method (FEM)
  5.1.1. Domain discretization. Finite Elements
  5.1.2. Form functions. Reconstruction of the continuous field
  5.1.3. Assembly of the coefficient matrix and boundary conditions
  5.1.4. Solving Systems of Equations
5.2. FEM: case study. Development of a FEM simulator
  5.2.1. Form functions
  5.2.2. Assembling the coefficient matrix and applying boundary conditions
  5.2.3. Solving Systems of Equations
  5.2.4. Post-Process

5.3. Smoothed Particle Hydrodynamics (SPH)
  5.3.1. Fluid field mapping from particle values
  5.3.2. Evaluation of derivatives and particle interaction
  5.3.3. The smoothing function. The kernel
  5.3.4. Boundary Conditions
5.4. SPH: development of a simulator based on SPH
  5.4.1. The kernel
  5.4.2. Storage and sorting of particles in voxels
  5.4.3. Development of boundary conditions
  5.4.4. Post-Process
5.5. Direct Simulation Monte Carlo (DSMC)
  5.5.1. Kinetic-molecular theory
  5.5.2. Statistical mechanics
  5.5.3. Molecular equilibrium
5.6. DSMC: methodology
  5.6.1. Applicability of the DSMC method
  5.6.2. Modeling
  5.6.3. Considerations for the applicability of the method
5.7. DSMC: applications
  5.7.1. Example in 0-D: thermal relaxation
  5.7.2. 1-D example: normal shock wave
  5.7.3. 2-D example: supersonic cylinder
  5.7.4. 3-D example: supersonic corner
  5.7.5. Complex example: Space Shuttle
5.8. Lattice-Boltzmann Method (LBM)
  5.8.1. Boltzmann equation and equilibrium distribution
  5.8.2. De Boltzmann a Navier-Stokes. Chapman-Enskog Expansion
  5.8.3. From probabilistic distribution to physical magnitude
  5.8.4. Conversion of units. From physical quantities to lattice quantities
5.9. LBM: numerical approximation
  5.9.1. The LBM algorithm. Transfer step and collision step
  5.9.2. Collision operators and momentum normalization
  5.9.3. Boundary Conditions
5.10. LBM: case study
  5.10.1. Development of a simulator based on LBM
  5.10.2. Experimentation with various collision operators
  5.10.3. Experimentation with various turbulence models

Module 6. Modeling of turbulence in Fluid

6.1. Turbulence. Key features
  6.1.1. Dissipation and diffusivity
  6.1.2. Characteristic scales. Orders of magnitude
  6.1.3. Reynolds Numbers
6.2. Definitions of Turbulence. From Reynolds to the present day
  6.2.1. The Reynolds problem. The boundary layer
  6.2.2. Meteorology, Richardson and Smagorinsky
  6.2.3. The problem of chaos
6.3. The energy cascade
  6.3.1. Smaller scales of turbulence
  6.3.2. Kolmogorov's hypothesis
  6.3.3. The cascade exponent
6.4. The closure problem revisited
  6.4.1. 10 unknowns and 4 equations
  6.4.2. The turbulent kinetic energy equation
  6.4.3. The turbulence cycle
6.5. Turbulent viscosity
  6.5.1. Historical background and parallels
  6.5.2. Initiation problem: jets
  6.5.3. Turbulent viscosity in CFD problems
6.6. RANS methods
  6.6.1. The turbulent viscosity hypothesis
  6.6.2. The RANS equations
  6.6.3. RANS methods. Examples of use
6.7. The evolution of SLE
  6.7.1. Historical Background
  6.7.2. Spectral filters
  6.7.3. Spatial filters. The problem in the wall

6.8. Wall turbulence I
- 6.8.1. Characteristic scales
- 6.8.2. The momentum equations
- 6.8.3. The regions of a turbulent wall flow

6.9. Wall turbulence II
- 6.9.1. Boundary layers
- 6.9.2. Dimensionless numbers of a boundary layer
- 6.9.3. The Blasius solution

6.10. The energy equation
- 6.10.1. Passive scalars
- 6.10.2. Active scalars. The Bousinesq approach
- 6.10.3. Fanno and Rayleigh flows

Module 7. Compressible Fluids

7.1. Compressible Fluids
- 7.1.1. Compressible and incompressible fluids. Differences
- 7.1.2. Equation of State
- 7.1.3. Differential equations of compressible fluids

7.2. Practical examples of the compressible regime
- 7.2.1. Shock Waves
- 7.2.2. Prandtl-Meyer Expansion
- 7.2.3. Nozzles

7.3. Riemann's Problem
- 7.3.1. Riemann's problem
- 7.3.2. Solution of the Riemann problem by characteristics
- 7.3.3. Non-linear systems: shock waves. Rankine-Hugoniot condition
- 7.3.4. Nonlinear systems: waves and expansion fans. Entropy condition
- 7.3.5. Riemannian Invariants

7.4. Euler Equations
- 7.4.1. Invariants of the Euler equations
- 7.4.2. Conservative Variables vs. Primitive variables
- 7.4.3. Solution Strategies

7.5. Solutions to the Riemann problem
- 7.5.1. Exact solution
- 7.5.2. Conservative numerical methods
- 7.5.3. Godunov's method
- 7.5.4. Flux Vector Splitting

7.6. Approximate Riemann solvers
- 7.6.1. HLLC
- 7.6.2. Roe
- 7.6.3. AUSM

7.7. Higher order methods
- 7.7.1. Problems of higher order methods
- 7.7.2. *Limiters* and TVD methods
- 7.7.3. Practical Examples

7.8. Additional aspects of the Riemann Problem
- 7.8.1. Non-homogeneous equations
- 7.8.2. Splitting dimensional
- 7.8.3. Applications from the Navier-Stokes equations

7.9. Regions with high gradients and discontinuities
- 7.9.1. Importance of meshing
- 7.9.2. Automatic mesh adaptation (AMR)
- 7.9.3. *Shock Fitting* Methods

7.10. Compressible flow applications
- 7.10.1. Sod problem
- 7.10.2. Supersonic wedge
- 7.10.3. Convergent-divergent nozzle

Module 8. Multiphase flow

8.1. Flow regimes
- 8.1.1. Continuous phase
- 8.1.2. Discrete phase
- 8.1.3. Discrete phase populations

8.2. Continuous phase
- 8.2.1. Properties of the liquid-gas interface
- 8.2.2. Each phase a domain
  - 8.2.2.1. Phase resolution independently
- 8.2.3. Coupled solution
  - 8.2.3.1. Fluid fraction as a descriptive phase scalar
- 8.2.4. Reconstruction of the - gas-liquid interface

8.3. Marine simulation
- 8.3.1. Wave regimes. Wave height vs.. Depth
- 8.3.2. Input boundary condition. Wave simulation
- 8.3.3. Non-reflective output boundary condition. Numerical beach
- 8.3.4. Lateral boundary conditions. Lateral wind and drift

8.4. Surface Tension
- 8.4.1. Physical Phenomenon of the Surface Tension
- 8.4.2. Modeling
- 8.4.3. Interaction with surfaces. Angle of wetting

8.5. Phase shift
- 8.5.1. Source and sink terms associated with phase change
- 8.5.2. Evaporation models
- 8.5.3. Condensation and precipitation models. Nucleation of droplets
- 8.5.4. Cavitation

8.6. Discrete phase: particles, droplets and bubbles
- 8.6.1. Resistance strength
- 8.6.2. The buoyancy force
- 8.6.3. Inertia
- 8.6.4. Brownian motion and turbulence effects
- 8.6.5. Other forces

8.7. Interaction with the surrounding fluid
- 8.7.1. Generation from continuous phase
- 8.7.2. Aerodynamic drag
- 8.7.3. Interaction with other entities, coalescence and rupture
- 8.7.4. Boundary Conditions

8.8. Statistical description of particle populations. Packages
- 8.8.1. Transportation of stocks
- 8.8.2. Stock boundary conditions
- 8.8.3. Stock interactions
- 8.8.4. Extending the discrete phase to populations

8.9. Water film
- 8.9.1. Water Sheet Hypothesis
- 8.9.2. Equations and modeling
- 8.9.3. Source term from particles

8.10. Example of an application with OpenFOAM
- 8.10.1. Description of an industrial problem
- 8.10.2. *Setup* and simulation
- 8.10.3. Visualization and interpretation of results

Module 9. Advanced CFD Models

9.1. Multiphysics
- 9.1.1. Multiphysics Simulations
- 9.1.2. System Types
- 9.1.3. Application Examples

9.2. Unidirectional Cosimulation
- 9.2.1. Unidirectional Cosimulation. Advanced Aspects
- 9.2.2. Information exchange schemes
- 9.2.3. Applications

9.3. Bidirectional Cosimulation
- 9.3.1. Bidirectional Cosimulation. Advanced Aspects
- 9.3.2. Information exchange schemes
- 9.3.3. Applications

9.4. Convection Heat Transfer
- 9.4.1. Heat Transfer by Convection. Advanced Aspects
- 9.4.2. Convective heat transfer equations
- 9.4.3. Methods for solving convection problems

9.5. Conduction Heat Transfer
- 9.5.1. Conduction Heat Transfer. Advanced Aspects
- 9.5.2. Conductive heat transfer equations
- 9.5.3. Methods of solving driving problems

9.6. Radiative Heat Transfer
- 9.6.1. Radiative Heat Transfer. Advanced Aspects
- 9.6.2. Radiation heat transfer equations
- 9.6.3. Radiation troubleshooting methods

9.7. Solid-fluid-heat coupling
- 9.7.1. Solid-fluid-heat coupling
- 9.7.2. Solid-fluid thermal coupling
- 9.7.3. CFD and FEM

9.8. Aeroacoustics
- 9.8.1. Computational aeroacoustics
- 9.8.2. Acoustic analogies
- 9.8.3. Resolution methods

9.9. Advection-diffusion problems
- 9.9.1. Advection-diffusion problems
- 9.9.2. Scalar Fields
- 9.9.3. Particle methods

9.10. Coupling models with reactive flow
- 9.10.1. Coupling models with reactive flow. Applications
- 9.10.2. System of differential equations. Solving the chemical reaction
- 9.10.3. CHEMKIN
- 9.10.4. Combustion: flame, spark, Wobee
- 9.10.5. Reactive flows in a non-stationary regime: quasi-stationary system hypothesis
- 9.10.6. Reactive flows in turbulent flows
- 9.10.7. Catalysts

Module 10. Post-processing, validation and application in CFD

10.1. Postprocessing in CFD I
- 10.1.1. Postprocessing on Plane and Surfaces
  - 10.1.1.1. Post-processing in the plane
  - 10.1.1.2. Post-processing on surfaces

10.2. Postprocessing in CFD II
- 10.2.1. Volumetric Postprocessing
  - 10.2.1.1. Volumetric post-processing I
  - 10.2.1.2. Volumetric post-processing II

10.3. Free CFD post-processing software
- 10.3.1. Free Postprocessing Software
- 10.3.2. Paraview
- 10.3.3. Paraview usage example

10.4. Convergence of simulations
- 10.4.1. Convergence
- 10.4.2. Mesh convergence
- 10.4.3. Numerical convergence

10.5. Classification of methods
- 10.5.1. Applications
- 10.5.2. Types of Fluid
- 10.5.3. Scales
- 10.5.4. Calculation machines

10.6. Model validation
- 10.6.1. Need for Validation
- 10.6.2. Simulation vs. Experiments
- 10.6.3. Validation examples

10.7. Simulation methods. Advantages and Disadvantages
- 10.7.1. RANS
- 10.7.2. LES, DES and DNS
- 10.7.3. Other Methods
- 10.7.4. Advantages and Disadvantages

10.8. Examples of methods and applications
- 10.8.1. Case of a body subjected to aerodynamic forces
- 10.8.2. Thermal case
- 10.8.3. Multiphase case

10.9. Good Simulation Practices
- 10.9.1. Importance of Good Practices
- 10.9.2. Good Practices
- 10.9.3. Simulation errors

10.10. Free and commercial software
- 10.10.1. FVM Software
- 10.10.2. Software for other methods
- 10.10.3. Advantages and Disadvantages
- 10.10.4. CFD La Simulation Futures

A curriculum designed to guarantee your success as a CFD expert, quickly and easily”

06

Methodology

This academic program offers students a different way of learning. Our methodology uses a cyclical learning approach: **Relearning.**
This teaching system is used, for example, in the most prestigious medical schools in the world, and major publications such as the **New England Journal of Medicine** have considered it to be one of the most effective.

Discover Relearning, a system that abandons conventional linear learning, to take you through cyclical teaching systems: a way of learning that has proven to be extremely effective, especially in subjects that require memorization"

Case Study to contextualize all content

Our program offers a revolutionary approach to developing skills and knowledge. Our goal is to strengthen skills in a changing, competitive, and highly demanding environment.

"At TECH, you will experience a learning methodology that is shaking the foundations of traditional universities around the world"

You will have access to a learning system based on repetition, with natural and progressive teaching throughout the entire syllabus.

The student will learn to solve complex situations in real business environments through collaborative activities and real cases.

A learning method that is different and innovative

This TECH program is an intensive educational program, created from scratch, which presents the most demanding challenges and decisions in this field, both nationally and internationally. This methodology promotes personal and professional growth, representing a significant step towards success. The case method, a technique that lays the foundation for this content, ensures that the most current economic, social and professional reality is taken into account.

> *Our program prepares you to face new challenges in uncertain environments and achieve success in your career"*

The case method is the most widely used learning system in the best faculties in the world. The case method was developed in 1912 so that law students would not only learn the law based on theoretical content. It consisted of presenting students with real-life, complex situations for them to make informed decisions and value judgments on how to resolve them. In 1924, Harvard adopted it as a standard teaching method.

What should a professional do in a given situation? This is the question that you are presented with in the case method, an action-oriented learning method. Throughout the program, the studies will be presented with multiple real cases. They will have to combine all their knowledge and research, and argue and defend their ideas and decisions.

Relearning Methodology

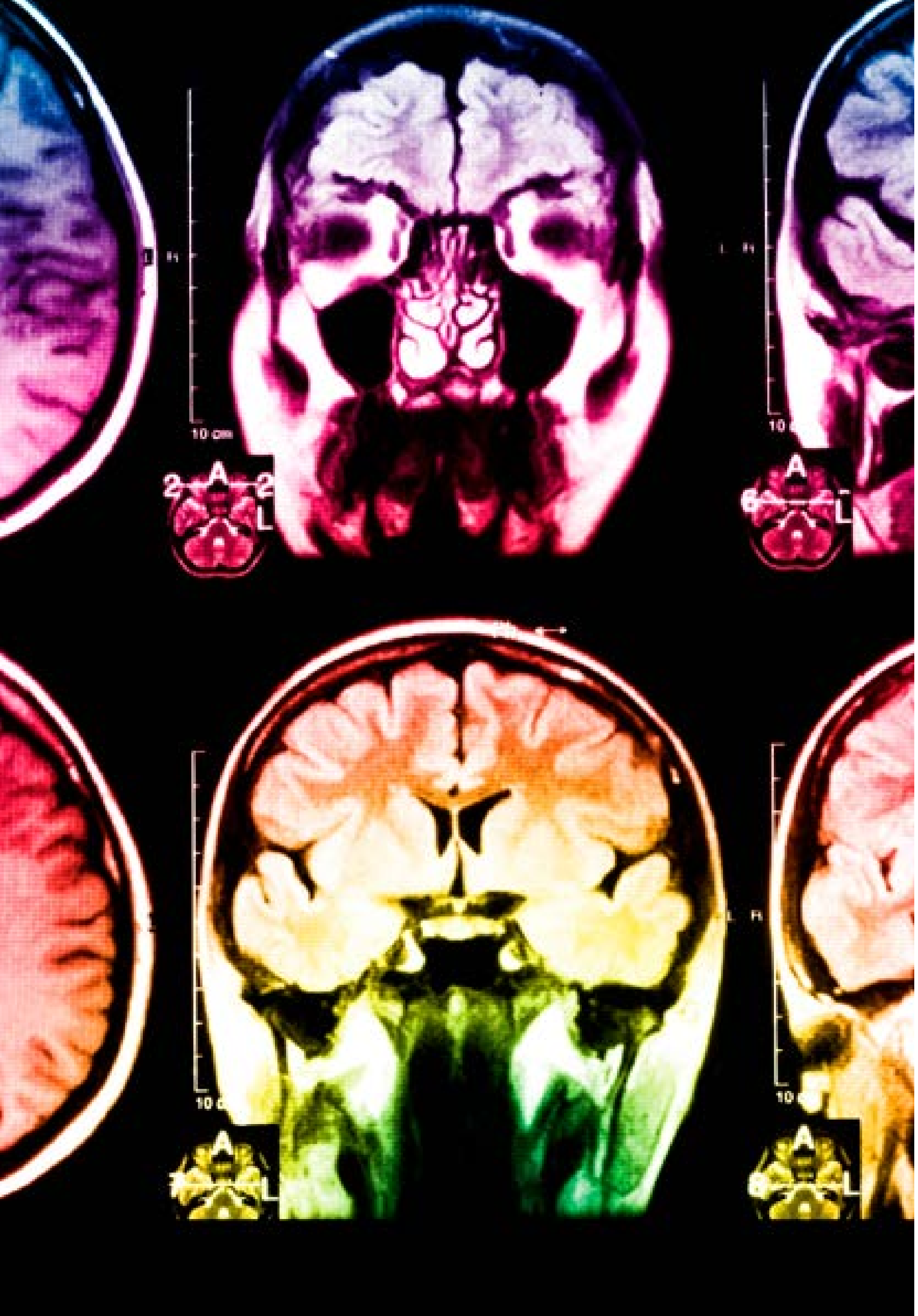
TECH effectively combines the Case Study methodology with a 100% online learning system based on repetition, which combines 8 different teaching elements in each lesson.

We enhance the Case Study with the best 100% online teaching method: Relearning.

In 2019, we obtained the best learning results of all online universities in the world.

At TECH, you will learn using a cutting-edge methodology designed to train the executives of the future. This method, at the forefront of international teaching, is called Relearning.

Our university is the only one in the world authorized to employ this successful method. In 2019, we managed to improve our students' overall satisfaction levels (teaching quality, quality of materials, course structure, objectives...) based on the best online university indicators.



In our program, learning is not a linear process, but rather a spiral (learn, unlearn, forget, and re-learn). Therefore, we combine each of these elements concentrically. This methodology has trained more than 650,000 university graduates with unprecedented success in fields as diverse as biochemistry, genetics, surgery, international law, management skills, sports science, philosophy, law, engineering, journalism, history, and financial markets and instruments. All this in a highly demanding environment, where the students have a strong socio-economic profile and an average age of 43.5 years.

Relearning will allow you to learn with less effort and better performance, involving you more in your training, developing a critical mindset, defending arguments, and contrasting opinions: a direct equation for success.

From the latest scientific evidence in the field of neuroscience, not only do we know how to organize information, ideas, images and memories, but we know that the place and context where we have learned something is fundamental for us to be able to remember it and store it in the hippocampus, to retain it in our long-term memory.

In this way, and in what is called neurocognitive context-dependent e-learning, the different elements in our program are connected to the context where the individual carries out their professional activity.

This program offers the best educational material, prepared with professionals in mind:

Study Material

All teaching material is produced by the specialists who teach the course, specifically for the course, so that the teaching content is highly specific and precise.

These contents are then applied to the audiovisual format, to create the TECH online working method. All this, with the latest techniques that offer high quality pieces in each and every one of the materials that are made available to the student.

Classes

There is scientific evidence suggesting that observing third-party experts can be useful.

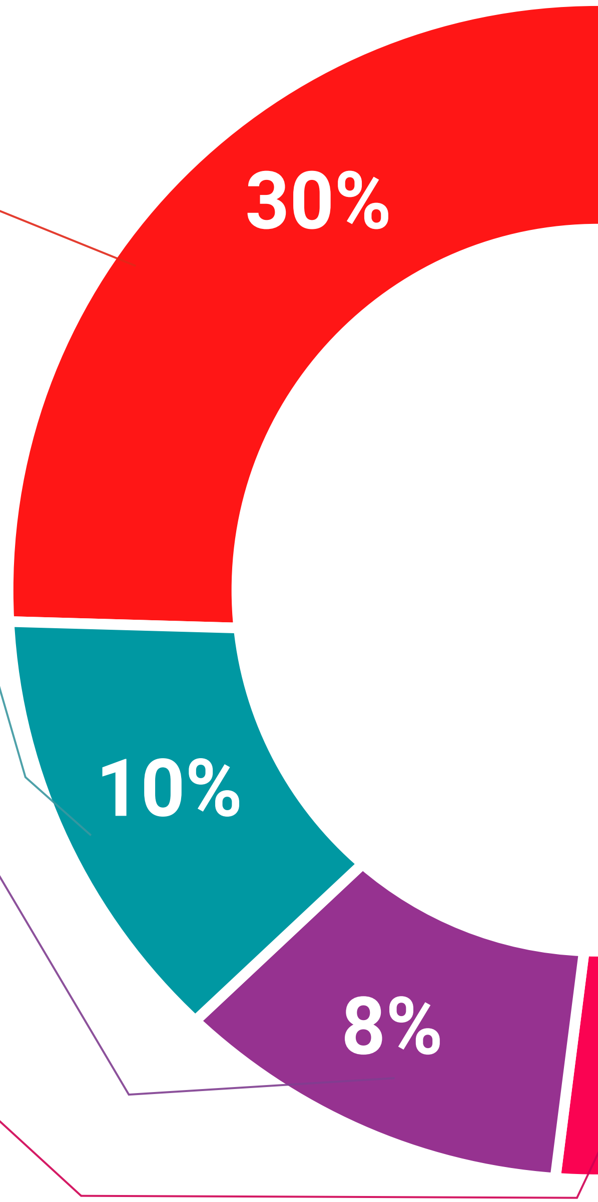
Learning from an Expert strengthens knowledge and memory, and generates confidence in future difficult decisions.

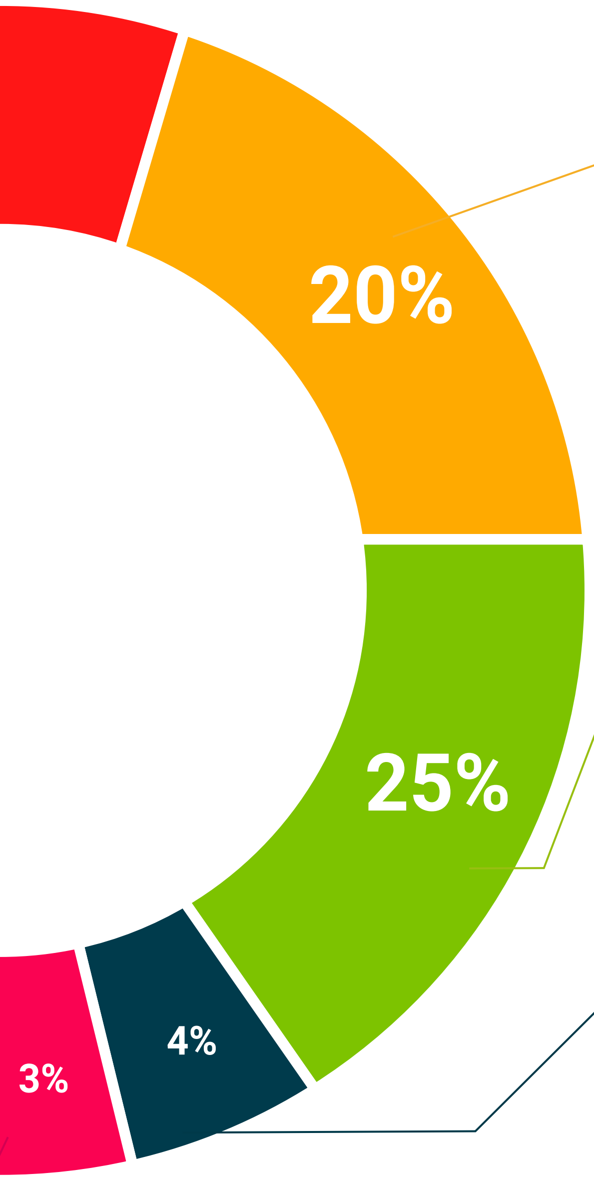
Practising Skills and Abilities

They will carry out activities to develop specific skills and abilities in each subject area. Exercises and activities to acquire and develop the skills and abilities that a specialist needs to develop in the context of the globalization that we are experiencing.

Additional Reading

Recent articles, consensus documents and international guidelines, among others. In TECH's virtual library, students will have access to everything they need to complete their course.





Case Studies

Students will complete a selection of the best case studies chosen specifically for this program. Cases that are presented, analyzed, and supervised by the best specialists in the world.

Interactive Summaries

The TECH team presents the contents attractively and dynamically in multimedia lessons that include audio, videos, images, diagrams, and concept maps in order to reinforce knowledge.

This exclusive educational system for presenting multimedia content was awarded by Microsoft as a "European Success Story".

Testing & Retesting

We periodically evaluate and re-evaluate students' knowledge throughout the program, through assessment and self-assessment activities and exercises, so that they can see how they are achieving their goals.

07

Certificate

The Master’s Degree in Computational Fluid Mechanics guarantees students, in addition to the most rigorous and up-to-date education, access to a Master’s Degree diploma issued by TECH Global University.

Successfully complete this program and receive your university qualification without having to travel or fill out laborious paperwork"

This program will allow you to obtain your **Master's Degree diploma in Computational Fluid Mechanics** endorsed by **TECH Global University**, the world's largest online university.

TECH Global University is an official European University publicly recognized by the Government of Andorra (***official bulletin***). Andorra is part of the European Higher Education Area (EHEA) since 2003. The EHEA is an initiative promoted by the European Union that aims to organize the international training framework and harmonize the higher education systems of the member countries of this space. The project promotes common values, the implementation of collaborative tools and strengthening its quality assurance mechanisms to enhance collaboration and mobility among students, researchers and academics.

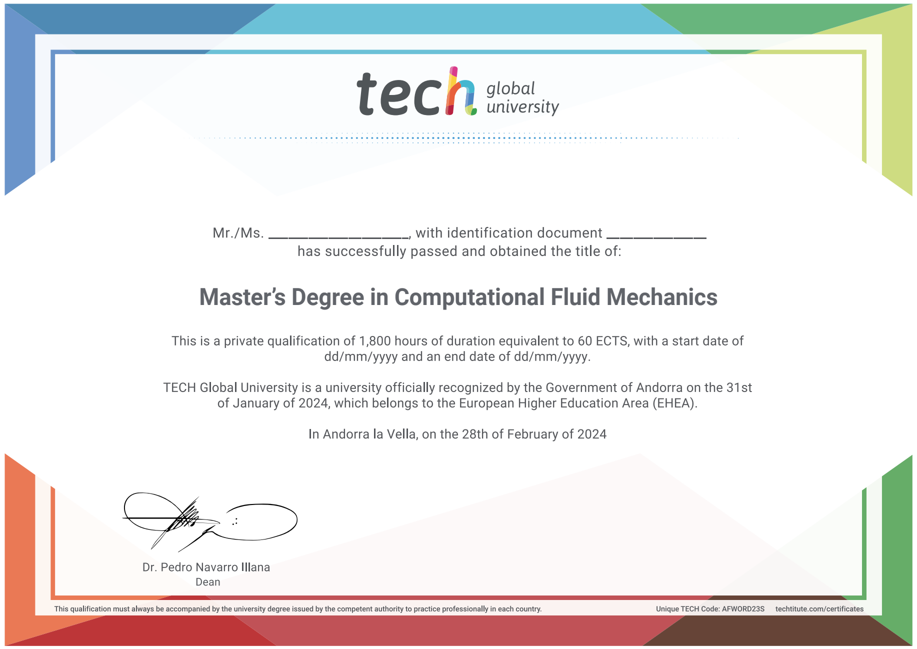
This **TECH Global University** title is a European program of continuing education and professional updating that guarantees the acquisition of competencies in its area of knowledge, providing a high curricular value to the student who completes the program.

Title: **Master's Degree in Computational Fluid Mechanics**

Modality: **online**

Duration: **12 months**

Accreditation: **60 ECTS**


tech global university

Mr./Ms. ________________, with identification document ____________
has successfully passed and obtained the title of:

Master's Degree in Computational Fluid Mechanics

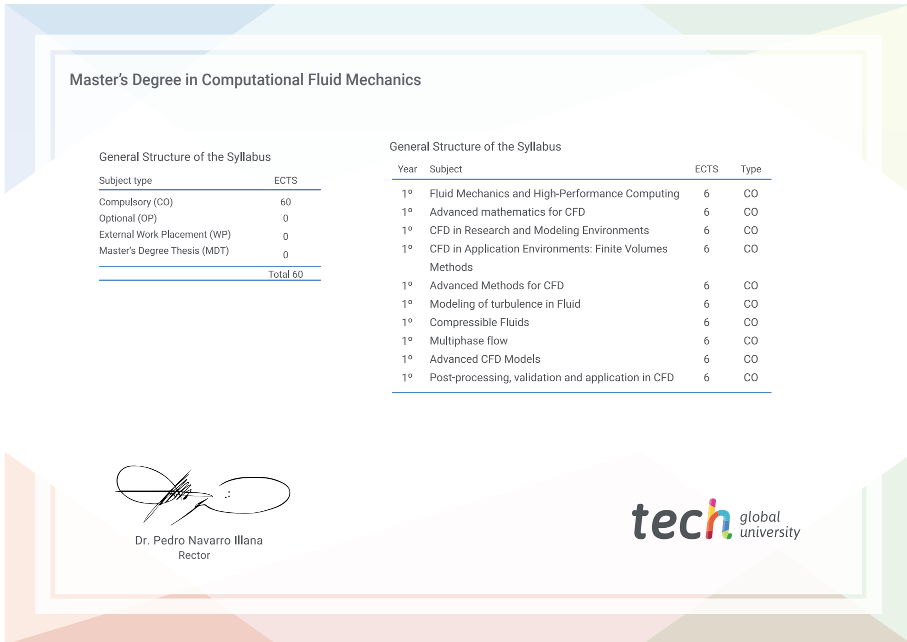
This is a private qualification of 1,800 hours of duration equivalent to 60 ECTS, with a start date of dd/mm/yyyy and an end date of dd/mm/yyyy.

TECH Global University is a university officially recognized by the Government of Andorra on the 31st of January of 2024, which belongs to the European Higher Education Area (EHEA).

In Andorra la Vella, on the 28th of February of 2024

Dr. Pedro Navarro Illana
Dean

This qualification must always be accompanied by the university degree issued by the competent authority to practice professionally in each country. Unique TECH Code: AFWORD23S techtitute.com/certificates


Master's Degree in Computational Fluid Mechanics

General Structure of the Syllabus

Subject type	ECTS
Compulsory (CO)	60
Optional (OP)	0
External Work Placement (WP)	0
Master's Degree Thesis (MDT)	0
	Total 60

General Structure of the Syllabus

Year	Subject	ECTS	Type
1º	Fluid Mechanics and High-Performance Computing	6	CO
1º	Advanced mathematics for CFD	6	CO
1º	CFD in Research and Modeling Environments	6	CO
1º	CFD in Application Environments: Finite Volumes Methods	6	CO
1º	Advanced Methods for CFD	6	CO
1º	Modeling of turbulence in Fluid	6	CO
1º	Compressible Fluids	6	CO
1º	Multiphase flow	6	CO
1º	Advanced CFD Models	6	CO
1º	Post-processing, validation and application in CFD	6	CO

Dr. Pedro Navarro Illana
Rector

tech global university

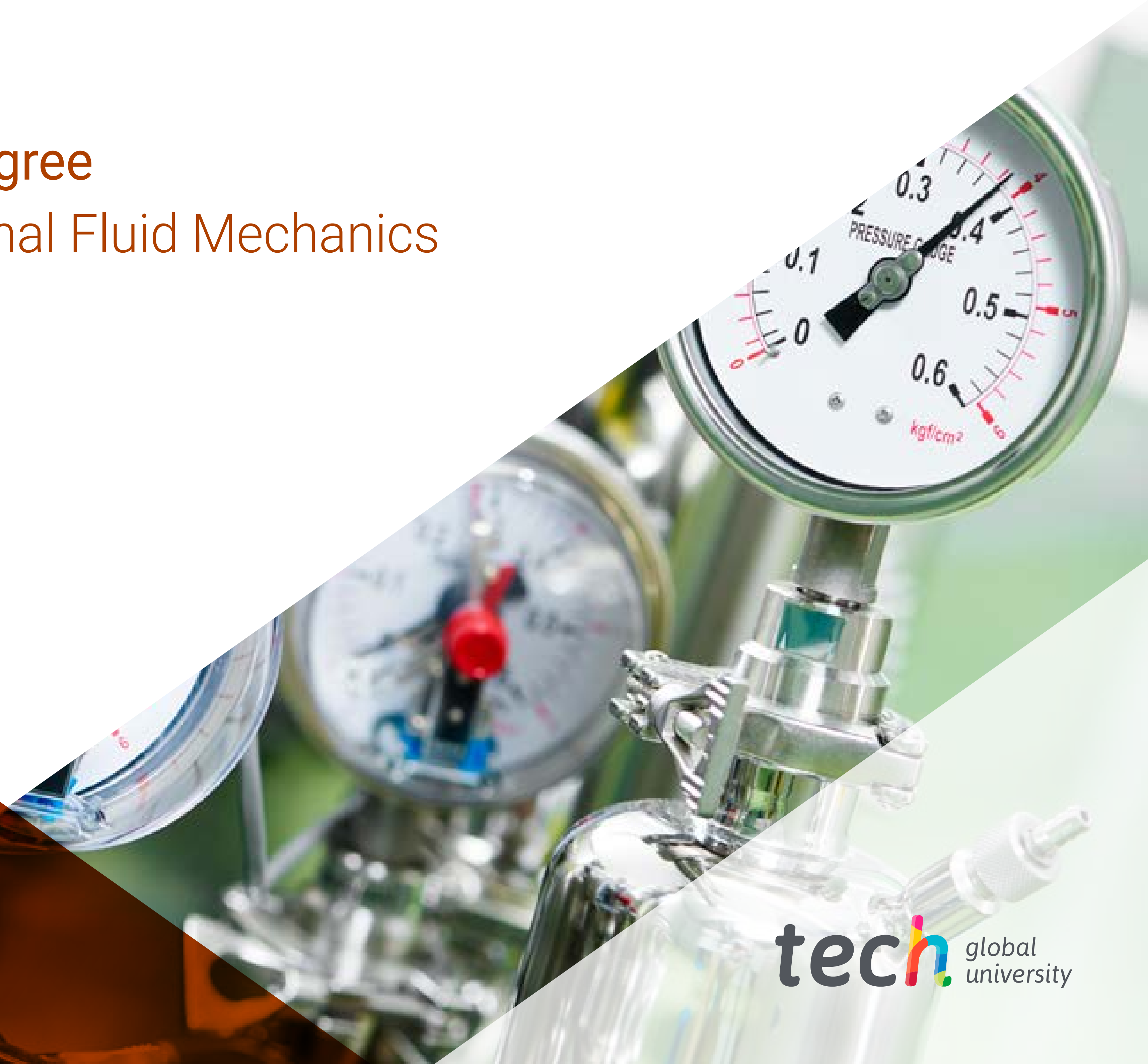
*Apostille Convention. In the event that the student wishes to have their paper diploma issued with an apostille, TECH Global University will make the necessary arrangements to obtain it, at an additional cost.

future confidence people health education information tutors guarantee accreditation teaching institutions technology learning community commitment personalized service innovation knowledge present quality online training development language virtual classroom

Master's Degree
Computational Fluid Mechanics

» Modality: **online**
» Duration: **12 months**
» Certificate: **TECH Global University**
» Credits: **60 ECTS**
» Schedule: **at your own pace**
» Exams: **online**

Master’s Degree
Computational Fluid Mechanics



tech global university