Professional Master's Degree

Computational Fluid Dynamics

tech technological university

Professional Master's Degree
Computational Fluid Dynamics

- Modality: **online**
- Duration: **12 months**
- Certificate: **TECH Technological University**
- Schedule: **at your own pace**
- Exams: **online**

Website: www.techtitute.com/us/information-technology/professional-master-degree/master-computational-fluid-dynamics

Index

01

Introduction

The use of numerical methods and algorithms to analyze and solve problems involving fluid flows is becoming increasingly common. Computational Fluid Dynamics (CFD) is a technique that involves a wide variety of sciences and requires a broad and deep knowledge of the subject. This is the reason why professionals in this area are increasingly in demand and why TECH has designed this degree, which seeks to specialize students to be able to work in this sector successfully, all in a 100% online mode that addresses topics such as the modeling of fluid turbulence, multiphase flow and the future of CFD simulation, among others.

Become an expert in CFD simulation in only few months and with total freedom of organization"

Computational Fluid Dynamics encompasses a wide range of sciences, such as Mathematics, Computer Science, Engineering and Physics. This technique uses numerical methods and algorithms to study and solve the different difficulties that may arise in the simulation of fluid motion. For this reason, professionals working in this field require very advanced skills and knowledge in algorithms, methods and the models that make up a simulator, and are increasingly in demand.

This is the reason why TECH has designed a Professional Master's Degree in Computational Fluid Dynamics, to provide students with specialized skills and knowledge in CFD simulation with which to face a successful future career in this area. In this way, the teaching materials cover topics such as the origin of turbulence, CFD modeling, advanced mathematics for CFD, artificial intelligence, moving contours and multiphysics simulations, among many other sections.

All this, giving total freedom to the student to adapt their schedules and studies, combining them with their other work and personal obligations, thanks to a 100% online modality, in addition to the most dynamic multimedia materials, information extracted from the most rigorous and updated sources, as well as the most efficient teaching methodology.

This **Professional Master's Degree in Computational Fluid Dynamics** 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 Computational Fluid Dynamics
- The graphic, schematic, and practical contents with which they are created, provide practical information on the 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

> *Get the most comprehensive knowledge in CFD and boost your professional profile in one of the most promising sectors of the IT industry"*

> *Thanks to the most updated theoretical and practical material you will be able to know all the novelties of the Computational Fluid Dynamics sector"*

Enjoy all the specialized information on compressible fluids and multiphase flow to expand your knowledge in the subject.

Access all the content from day one and acquire new skills in fluid turbulence modeling.

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

The 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 immersive education programmed to learn in real situations.

This program is designed around Problem-Based Learning, whereby the professional must try to solve the different professional practice situations that arise during the academic year For this purpose, the students will be assisted by an innovative interactive video system created by renowned and experienced experts.

02

Objectives

The objective of this Professional Master's Degree in Computational Fluid Dynamics is to give the student the ability to work in the industry as an advanced user and developer of CFD tools. All this, thanks to the most complete, dynamic and updated contents in the academic market.

Specialize in one of the most promising areas of Computer Science and stand out for your new skills, thanks to TECH"

General Objectives

- Establish the basis for the study of turbulence
- Develop CFD statistical concepts
- Determine the main calculation techniques in Turbulence Research
- Generate specialized knowledge in the method of Finite Volumes
- Acquire specialized knowledge in fluid mechanic 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

> *Reach the most demanding goals thanks to the most innovative and practical CFD simulation tools"*

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
- Develop 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
- Substantiate the method of solving CFD equations
- Demonstrate methods of solving algebraic problems
- Analyze the multigrid method
- Examine the use of eigenvalues and eigenvectors in CFD problems
- Determine methods for solving non-linear problems

Module 3. CFD in Research and Modeling Environments

- Analyze 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
- 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
- Define and design 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 Carlo 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 Fluids

- Apply 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
- Model 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, 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
- Analyze 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
- Provide a foundation for the current context of CFD simulation

03

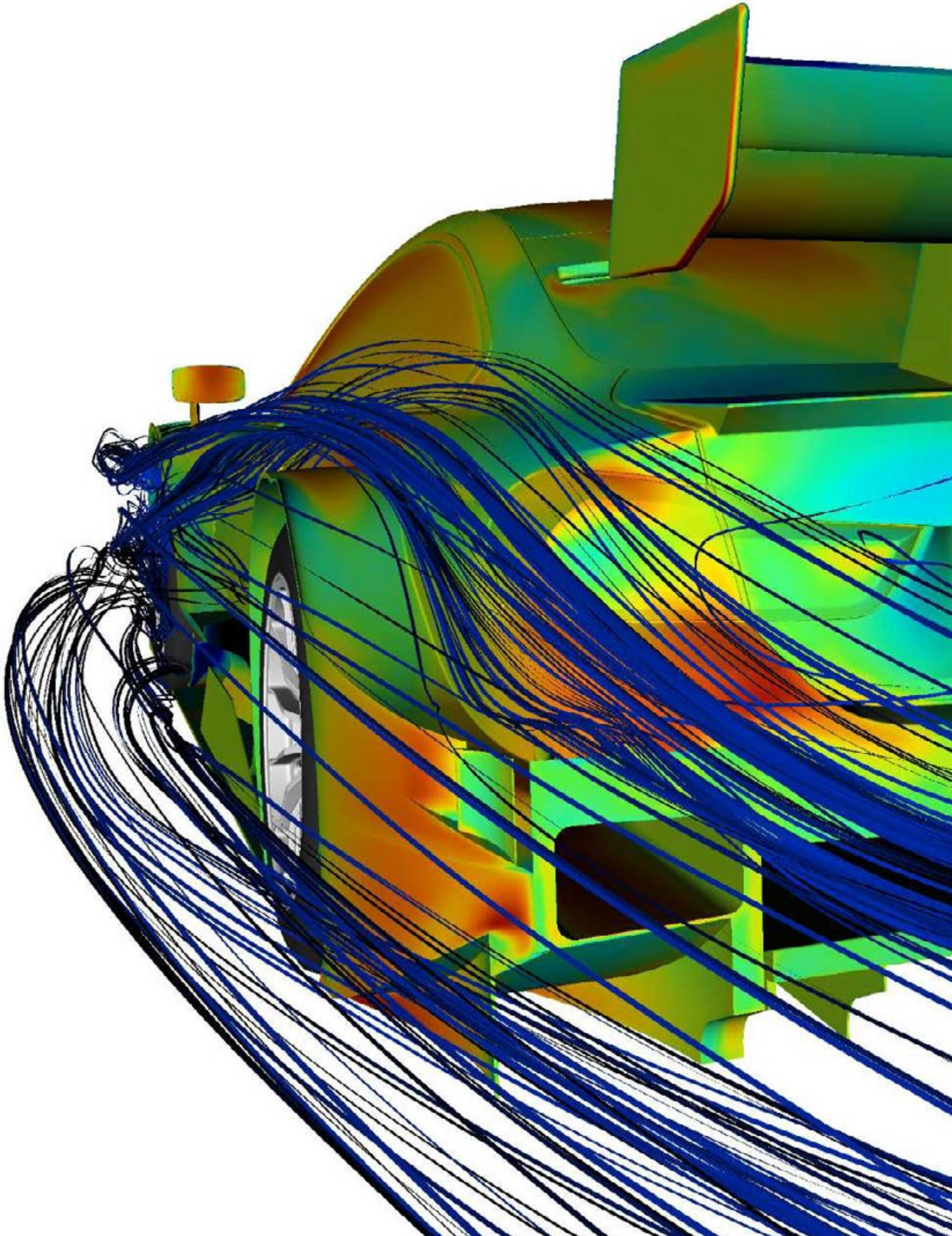
Skills

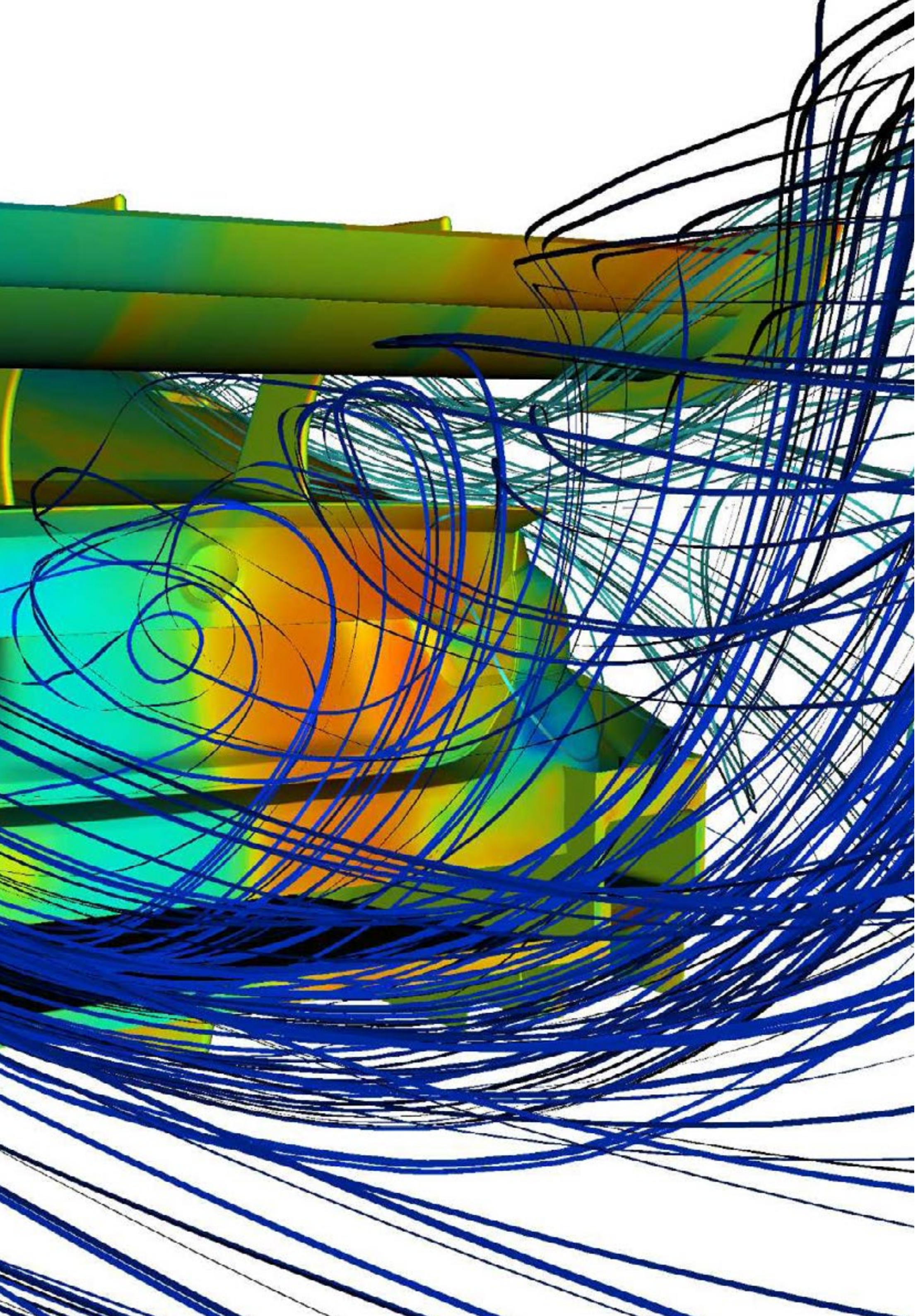
This Professional Master's Degree in Fluid Dynamics has been designed by experts in the field, who seek to provide students with skills with which to face a successful professional future in this sector. In this way, the student will be able to solve any situation or inconvenience they may face, thanks to the most complete and updated didactic material, as well as the availability of the latest teaching technologies.

> *You will be able to position yourself in the labor market as an expert in Computational Fluid Dynamics in just a few months"*

General Skills

- Know the main supercomputing techniques
- Identify and define the concept of residual
- Differentiate the different turbulent structures
- Optimally configure each simulation
- Obtain specialized knowledge in Fluid Dynamics computational techniques
- Model the energy equation
- Identify the main numerical methods in solving the Riemann problem
- Choose the type of simulation or model to apply that best suits the context, as well as identify the pros and cons of each method
- Combine multiple strategies to get the best results where they are most needed
- Interpret the results obtained by correct post-processing





Specific Skills

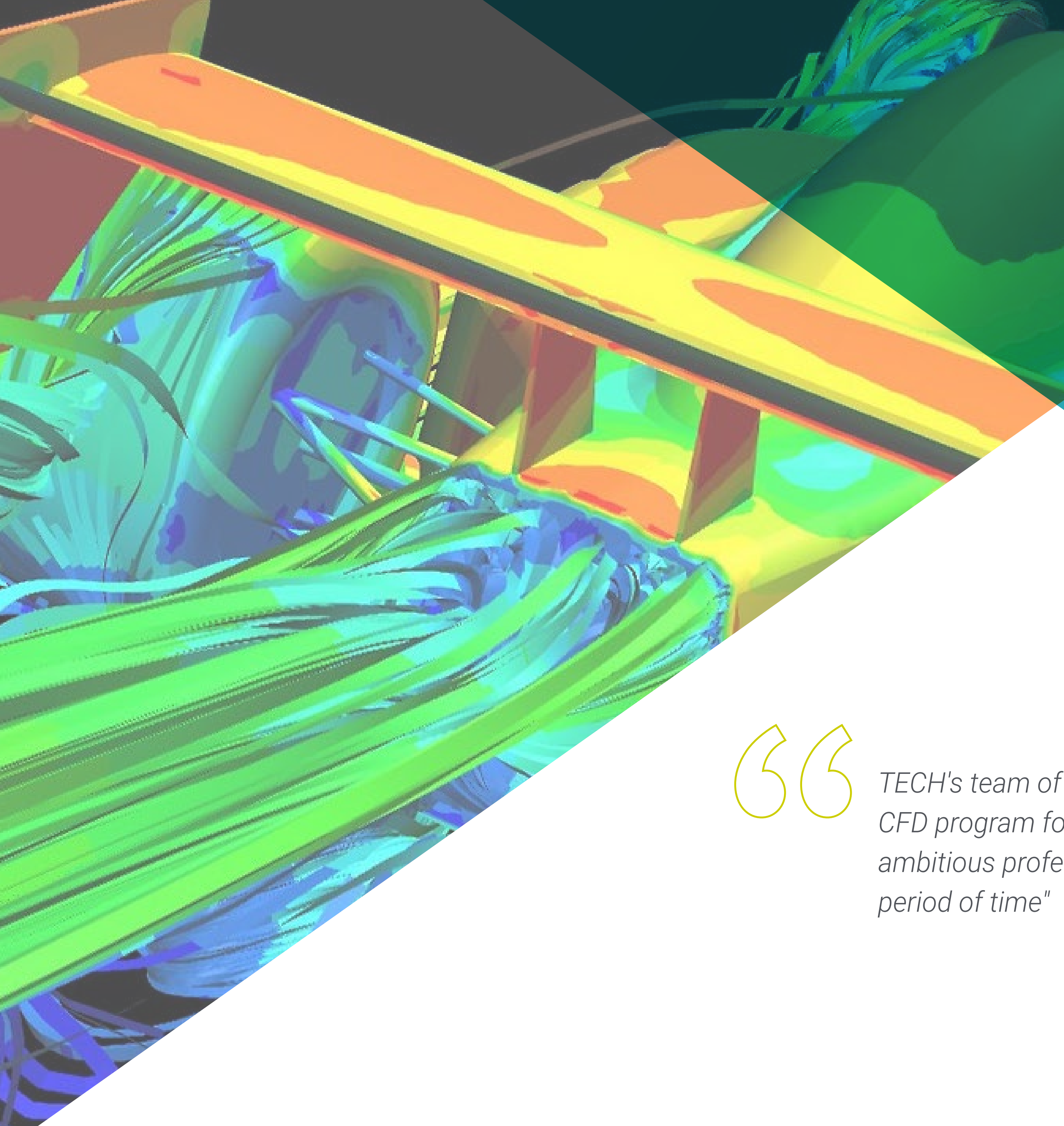
- Developing the different types of supercomputers
- Determine methods for solving non-linear problems
- Apply classical discretization methods to Fluid Mechanics problems
- Specify what, where and how the boundary conditions can be defined
- Evaluate and interpret spatial aerodynamics and microfluid 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 on the market

> *Delve into areas such as Postprocessing, Validation and CFD Application, thanks to all the material available on the Virtual Campus"*

04

Course Management

In In the search to offer elite education to its students, TECH has created a team with the best specialists in Computational Fluid Dynamics These experts have designed the contents based on their outstanding experience and extensive professional trajectory, with the objective of providing the student with the best skills and the most complete knowledge about CFD Simulation.



> *TECH's team of experts has designed this CFD program for you to reach your most ambitious professional goals in a short period of time"*

Management

Dr. José Pedro García Galache

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

Professors

Dr. Daniel Espinoza Vásquez

- Consultant Aeronautical Engineer at Alten SAU
- Freelance CFD and Programming Consultant
- CFD Specialist at Particle Analytics Limitations
- 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 the Polytechnic University of Madrid

Dr. Enrique Mata Bueso

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

Ms. Maider Pérez Tainta

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

> *Take the opportunity to learn about the latest advances in this field in order to apply it to your daily practice"*

05

Structure and Content

The structure and content of this program have been designed by the professionals that make up TECH's team of experts in Computational Fluid Dynamics. These teachers have poured their experience and their most specialized knowledge in materials to guarantee a curriculum that meets most demanding expectations. In addition, based on the requirements of the most efficient teaching methodology, the *Relearning*, in which TECH is a pioneer

The most updated and complete content, with which you will be able to adapt your professional profile to the latest trends in CFD"

Module 1. Fluid Mechanics and High-Performance Computing

- 1.1. Dynamics of Computational Fluid Dynamics
    - 1.1.1. The Origin of 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. Future Supercomputing
    - 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, Iterative 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 RANS
  - 3.1.3. Artificial Intelligence
- 3.2. Finite Differences
  - 3.2.1. Presentation and Application in 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. The Runge-Kutta Method
- 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 Von the Karmanos Finite Volumes Constant
    - 3.10.2. Aerodynamics: Boundary Layers
    - 3.10.3. Noise in CFD Problems

Module 4. CFD in Application Environments: Finite Volume 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 (Evaporation, Cavitation and Chemical) Source Term
    - 4.2.3. Scalar Source Term
        - 4.2.3.1. Temperature and Species
- 4.3. Applications of Boundary Conditions
    - 4.3.1. Inputs and Outputs
    - 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. Vectorial
    - 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. 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 and 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 Studies 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. From Boltzmann to 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 Fluids

- 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. Non-Linear 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. Dimensional *Splitting*
    - 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 Phases
  - 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 Liquid-Gas 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 Populations
    8.8.2. Population Boundary Conditions
    8.8.3. Population Interactions
    8.8.4. Extending the Discrete Phase to Populations
8.9. Water Film
    8.9.1. Water Film 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 and Wobbe Index
    - 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. Post-Processing in CFD I
    - 10.1.1. Post-Processing on Plane and Surfaces
        - 10.1.1.1. Post-Processing in the Plane
        - 10.1.1.2. Post-Processing on Surfaces
- 10.2. Post-Processing in CFD II
    - 10.2.1. Volumetric Post-Processing
        - 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 Post-Processing 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. Experimentation
    - 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

> *Access a wide variety of additional material on the Virtual Campus and expand your knowledge in the aspects of CFD simulation that interest you most"*

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 has been the most widely used learning system among the world's leading Information Technology schools for as long as they have existed. 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 course, students 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

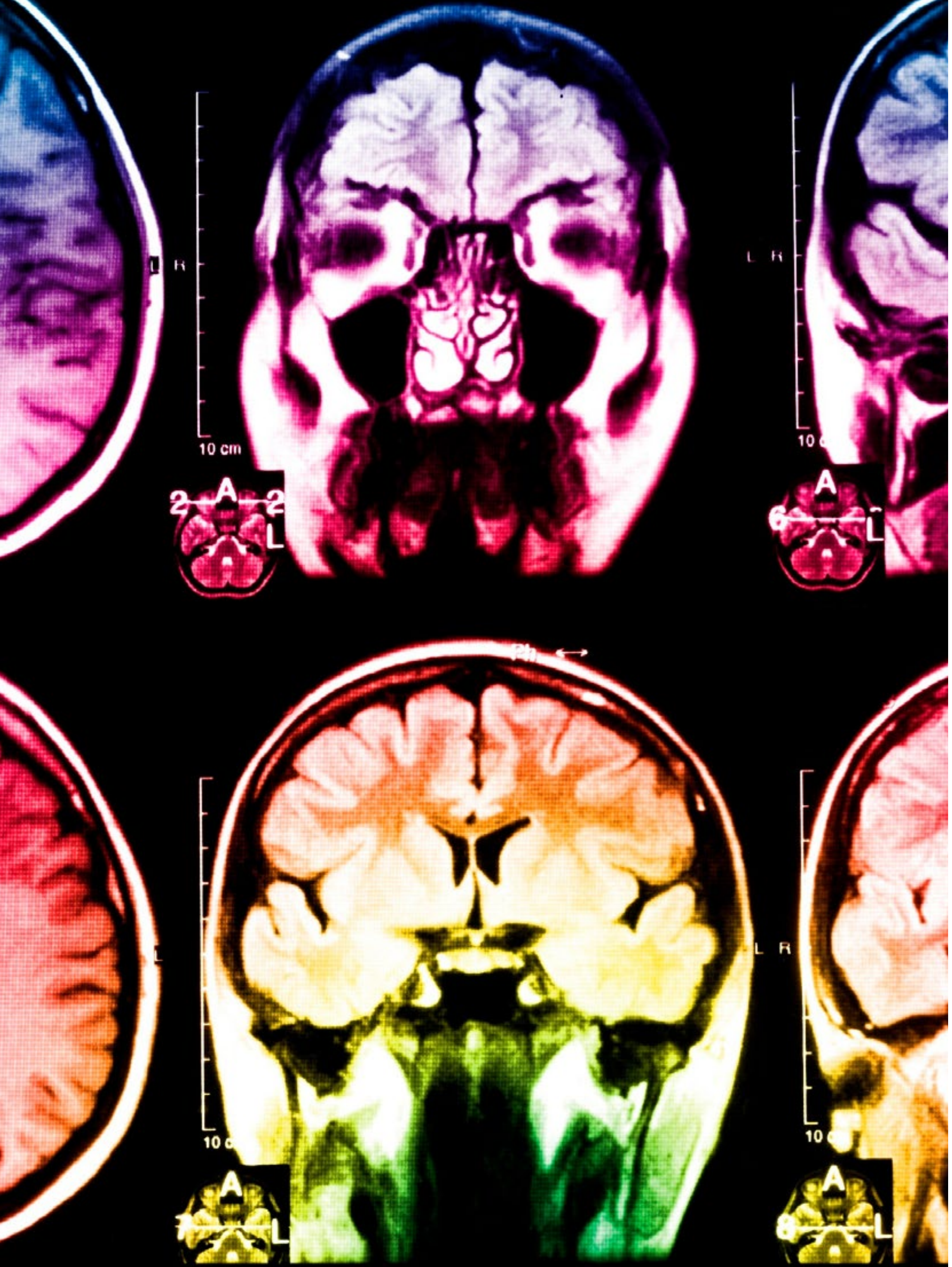
TECH effectively combines the Case Study methodology with a 100% online learning system based on repetition, which combines different teaching elements in each lesson.

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.

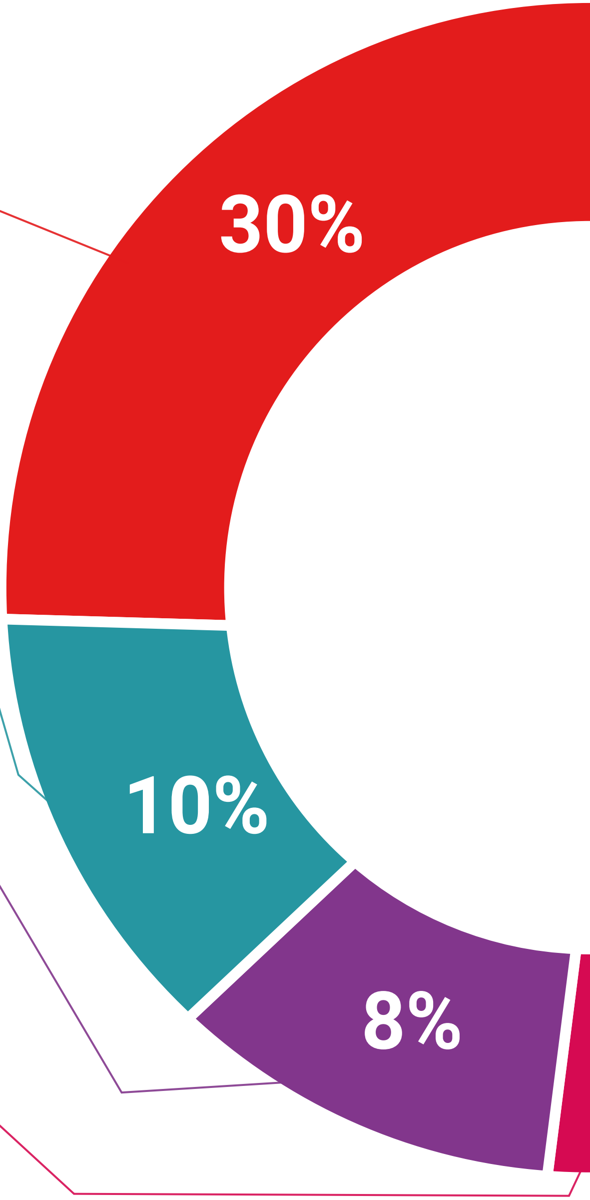
Learning from an Expert strengthens knowledge and memory, and generates confidence in future difficult decisions.

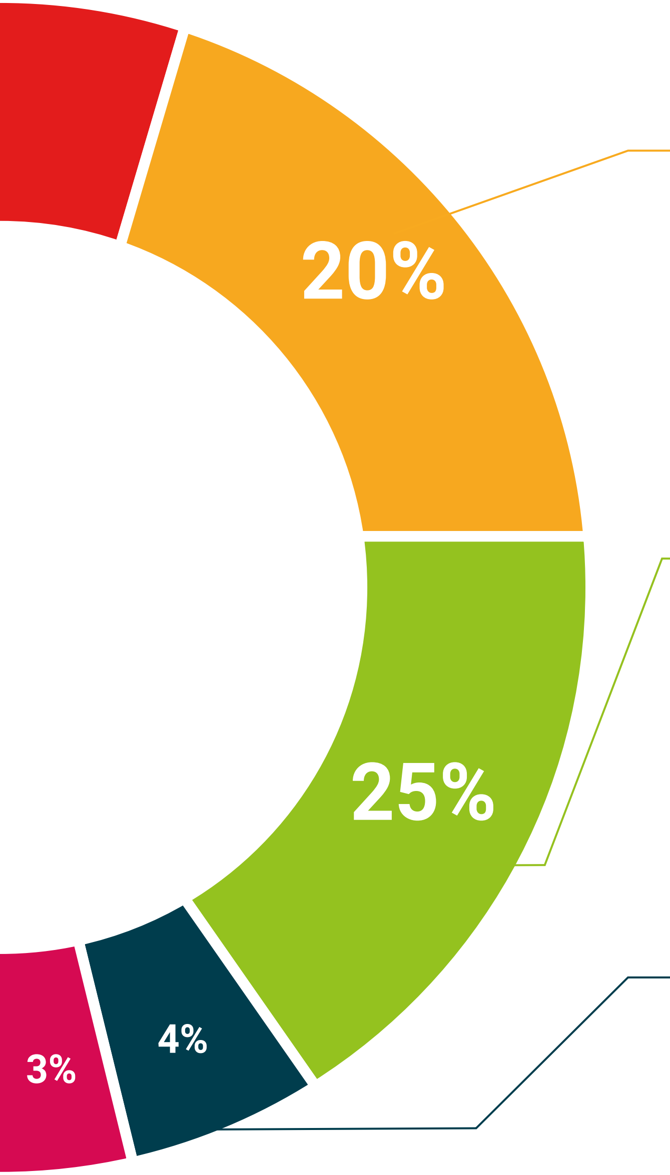
Practising Skills and Abilities

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 Advanced Master's Degree in Computational Fluid Dynamics guarantees students, in addition to the most rigorous and up-to-date education, access to a Postgraduate Certificate issued by TECH Global University.

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

This **Professional Master's Degree in Computational Fluid Dynamics** contains the most complete and up-to-date scientific on the market.

After the student has passed the assessments, they will receive their corresponding **Professional Master's Degree** issued by TECH Technological University via tracked delivery*.

The diploma issued by **TECH Technological University** will reflect the qualification obtained in the Professional Master's Degree,and meets the requirements commonly demanded by labor exchanges, competitive examinations, and professional career evaluation committees.

Title: **Professional Master's Degree in Computational Fluid Dynamics**

Official Nº of Hours: **1,500 h.**

Awards the following

DIPLOMA

to

Mr./Ms. _________________, with identification number ____________.

For having successfully passed and accredited the following program

PROFESSIONAL MASTER'S DEGREE

in

Computational Fluid Dynamics

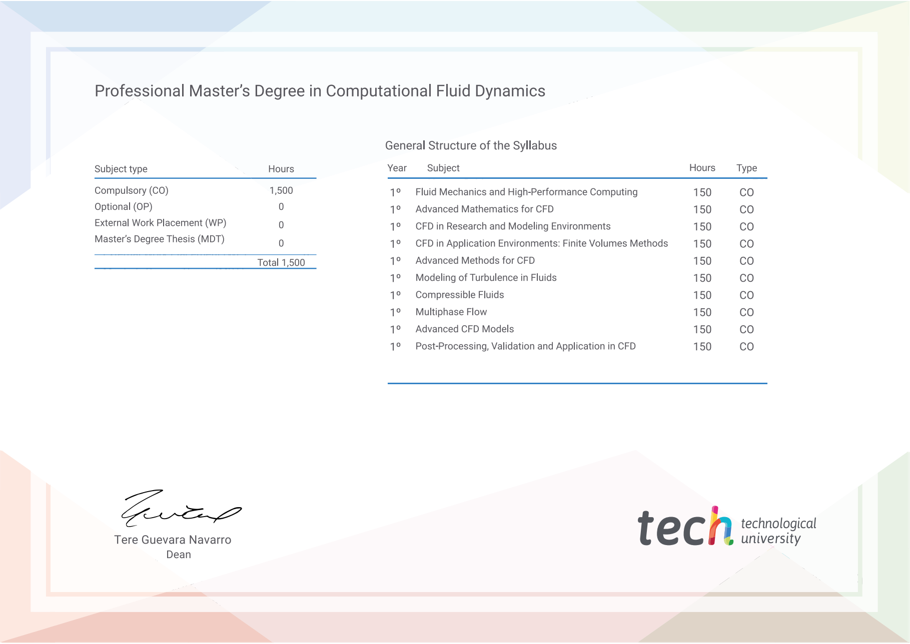
This is a qualification awarded by this University, equivalent to 1,500 hours, with a start date of dd/mm/yyyy and an end date of dd/mm/yyyy.

TECH is a Private Institution of Higher Education recognized by the Ministry of Public Education as of June 28, 2018.

June 17, 2020

Tere Guevara Navarro
Dean

Unique TECH Code: AFWORD23S techtitute.com/certificates



Professional Master's Degree in Computational Fluid Dynamics

Subject type	Hours
Compulsory (CO)	1,500
Optional (OP)	0
External Work Placement (WP)	0
Master's Degree Thesis (MDT)	0
	Total 1,500

General Structure of the Syllabus

Year	Subject	Hours	Type
1º	Fluid Mechanics and High-Performance Computing	150	CO
1º	Advanced Mathematics for CFD	150	CO
1º	CFD in Research and Modeling Environments	150	CO
1º	CFD in Application Environments: Finite Volumes Methods	150	CO
1º	Advanced Methods for CFD	150	CO
1º	Modeling of Turbulence in Fluids	150	CO
1º	Compressible Fluids	150	CO
1º	Multiphase Flow	150	CO
1º	Advanced CFD Models	150	CO
1º	Post-Processing, Validation and Application in CFD	150	CO

Tere Guevara Navarro
Dean

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

tech technological university

Professional Master's Degree
Computational Fluid Dynamics

- Modality: **online**
- Duration: **12 months**
- Certificate: **TECH Technological University**
- Schedule: **at your own pace**
- Exams: **online**

Professional Master's Degree
Computational Fluid Dynamics