Guía docente
390438 - FMBE - Introducción a la Modelización de Fluidos en la Bioingeniería

Última modificación: 19/01/2026

Unidad responsable: Escuela de Ingeniería Agroalimentaria y de Biosistemas de Barcelona
Unidad que imparte: 745 - DEAB - Departamento de Ingeniería Agroalimentaria y Biotecnología.

Titulación: GRADO EN INGENIERÍA DE SISTEMAS BIOLÓGICOS (Plan 2009). (Asignatura optativa).

Curso: 2025 **Créditos ECTS:** 3.0 **Idiomas:** Inglés

PROFESORADO

Profesorado responsable: Ramón Salcedo Cidoncha

Otros:

METODOLOGÍAS DOCENTES

OBJETIVOS DE APRENDIZAJE DE LA ASIGNATURA

The main objective of this course is for students to learn, understand, and apply the principles of fluid modeling in bioengineering contexts through the practical use of Computational Fluid Dynamics (CFD) tools. By the end of the course, students will be able to formulate a basic two-dimensional CFD problem relevant to the field of biosystems engineering. Students will learn the concepts of computational meshes, boundary conditions, and numerical simulations, as well as how to critically analyze and interpret results. Furthermore, the course aims to develop students' ability to validate models, evaluate their limitations, and clearly communicate technical findings in writing.

HORAS TOTALES DE DEDICACIÓN DEL ESTUDIANTADO

Tipo	Horas	Porcentaje
Grupo pequeño/Laboratorio	10,0	8.33
Grupo mediano/Prácticas	20,0	16.67
Aprendizaje autónomo	90,0	75.00

Dedicación total: 120 h

CONTENIDOS

1. Problem definition and physical flow interpretation

Descripción:
Definition of the bioengineering problem to be modeled throughout the course. Analysis of the physical system, identification of the computational domain, main modeling assumptions, and relevant flow variables. Introduction to the case study that will be progressively developed using CFD.

Objetivos específicos:
To understand the bioengineering problem to be modeled, correctly interpret the physical flow behavior, identify the required assumptions, and select the relevant variables for CFD model formulation.

Actividades vinculadas:
- Theoretical introduction to the bioengineering problem and physical system.
- Definition of the computational domain and boundary conditions.
- Guided analysis of the case study and flow variables.
- Deliverable

Dedicación: 10h
Grupo mediano/Prácticas: 2h
Grupo pequeño/Laboratorio: 1h
Aprendizaje autónomo: 7h

2. Geometry creation and mesh generation

Descripción:
Creation of the two-dimensional geometry of the case study and generation of the computational mesh. Mesh refinement strategies, quality criteria, and assessment of mesh suitability for CFD simulations are addressed

Objetivos específicos:
To define the two-dimensional geometry of the problem, generate an appropriate computational mesh, and assess its quality to ensure the consistency of the CFD model.

Actividades vinculadas:
- Creation of the two-dimensional geometry of the case study.
- Generation of the computational mesh.
- Application of mesh refinement strategies.
- Analysis of mesh quality using technical criteria.
- Deliverable

Dedicación: 13h
Grupo mediano/Prácticas: 3h
Grupo pequeño/Laboratorio: 2h
Aprendizaje autónomo: 8h

3. Boundary conditions and numerical setup

Descripción:
Definition of boundary conditions, selection of physical models, solver configuration, and specification of numerical parameters required for CFD simulations.

Objetivos específicos:
To learn how to define boundary conditions and numerical parameters for ensuring a stable and physically consistent CFD simulation.

Actividades vinculadas:
- Definition of boundary conditions for the case study.
- Selection of physical models (turbulence, flow, etc.).
- Configuration of the solver and numerical parameters.
- Deliverable

Dedicación: 20h
Grupo mediano/Prácticas: 3h
Grupo pequeño/Laboratorio: 2h 30m
Aprendizaje autónomo: 14h 30m

4. Simulation and post-processing

Descripción:
Execution of CFD simulations and post-processing of results, including visualization and extraction of relevant flow variables.

Objetivos específicos:
To run CFD simulations correctly and interpret flow results through appropriate post-processing techniques

Actividades vinculadas:
- Execution of CFD simulations for the case study.
- Analysis of numerical solution convergence.
- Visualization of results using post-processing tools.
- Extraction of velocity fields, profiles and other variables.
- Physical interpretation of the obtained results.
- Deliverable

Dedicación: 31h 45m
Grupo mediano/Prácticas: 7h
Grupo pequeño/Laboratorio: 2h 30m
Aprendizaje autónomo: 22h 15m

5. Analysis, validation and reporting

Descripción:
Critical analysis and validation of CFD results, discussion of model limitations, and preparation of the final technical report.

Objetivos específicos:
To evaluate CFD results, assess model validity, and communicate findings in a structured technical report.

Actividades vinculadas:
- Critical analysis of simulation results.
- Model validation using physical criteria or reference comparisons.
- Discussion of model limitations.
- Writing of the final technical report.
- Deliverable
- Presentation and discussion of results.

Dedicación: 45h 15m
Grupo mediano/Prácticas: 5h
Grupo pequeño/Laboratorio: 1h
Aprendizaje autónomo: 39h 15m

ACTIVIDADES

Applied theory

Descripción:
Applied theory sessions will focus on introducing the fundamental concepts of the course through practical examples and real-case applications. Theoretical explanations will be combined with activities linked to the development of the course project.

Objetivos específicos:
To understand the bioengineering problem to be modeled, correctly interpret the physical flow behavior, identify the required assumptions, and select the relevant variables for CFD model formulation.

Material:
Computer classroom and computers with Autodesk CFD software available through the UPC software platform.

Entregable:
1) Short document describing the bioengineering problem to be modeled, the physical interpretation of the flow, the assumptions made, and the key variables selected for the CFD model.
2) CFD model file including the defined two-dimensional geometry and the generated computational mesh, together with a brief description of mesh refinement criteria and mesh quality assessment
3) CFD model setup file including defined boundary conditions, physical models, and solver settings.
4) Post-processed results including figures and extracted flow variables.
5) Final technical report summarizing methodology, results, validation, discussion, and conclusions.

Dedicación: 50h
Grupo mediano/Prácticas: 20h
Aprendizaje autónomo: 30h

Practical exercises

Descripción:
Practical exercise sessions will focus on applying theoretical concepts through problem-solving activities and the use of Autodesk CFD software. Students will carry out simulations, analyze results, and work on the course case study under the guidance of the instructor.

Objetivos específicos:
- Apply CFD theoretical concepts to practical problems.
- Develop simulations using Autodesk CFD software.
- Correctly configure geometry, mesh, and boundary conditions.
- Analyze and interpret the obtained results.
- Solve technical problems related to the course case study.

Material:
Computer classroom with computers and Autodesk CFD software available through the UPC software platform.

Entregable:
1) Short document describing the bioengineering problem to be modeled, the physical interpretation of the flow, the assumptions made, and the key variables selected for the CFD model.
2) CFD model file including the defined two-dimensional geometry and the generated computational mesh, together with a brief description of mesh refinement criteria and mesh quality assessment
3) CFD model setup file including defined boundary conditions, physical models, and solver settings.
4) Post-processed results including figures and extracted flow variables.
5) Final technical report summarizing methodology, results, validation, discussion, and conclusions.

Dedicación: 70h
Grupo pequeño/Laboratorio: 10h
Aprendizaje autónomo: 60h

SISTEMA DE CALIFICACIÓN

BIBLIOGRAFÍA

Complementaria:

- Versteeg, H. K; Malalasekera, W. An Introduction to computational fluid dynamics : the finite volume method . 2nd ed. London : Pearson Education, cop. 2007. ISBN 9780131274983.
- Anderson, John David. Computational fluid dynamics . New York [etc.] : McGraw-Hill, cop. 1995. ISBN 9780070016859.
- Patankar, Suhas V. Numerical heat transfer and fluid flow . Boca Ratón : Washington : Taylor & Francis ; Hemisphere Pub. Co, 1980. ISBN 9780891165224.

RECURSOS

Enlace web:
- Autodesk CFD – Software de simulació CFD per a estudiants UPC. Recurso