UNIVERSITAT POLITÈCNICA DE CATALUNYA BARCELONATECH

Course guide
390438 - FMBE - Introduction to Fluid Modelling in Bio-Engineering

Last modified: 19/01/2026

Unit in charge: Barcelona School of Agri-Food and Biosystems Engineering
Teaching unit: 745 - DEAB - Department of Agri-Food Engineering and Biotechnology.

Degree: BACHELOR'S DEGREE IN BIOSYSTEMS ENGINEERING (Syllabus 2009). (Optional subject).

Academic year: 2025 **ECTS Credits:** 3.0 **Languages:** English

LECTURER

Coordinating lecturer: Ramón Salcedo Cidoncha

Others:

PRIOR SKILLS

Students are expected to have a basic background in physics and mathematics. Prior knowledge of hydraulics and differential equations is recommended, including familiarity with fundamental concepts such as conservation laws, pressure, velocity fields, and flow regimes. Basic skills in scientific reasoning, data interpretation, and the use of engineering software tools are also advisable. No prior experience with Computational Fluid Dynamics (CFD) is required, as the course introduces these concepts progressively.

REQUIREMENTS

There are no mandatory prerequisites for enrollment in this course. It is recommended that students follow the guidance provided in the "Previous capacities" section to ensure an adequate background for the successful completion of the subject.

TEACHING METHODOLOGY

The course is eminently applied and structured around the progressive development of a single two-dimensional CFD case study throughout the course. Most class time is dedicated to practical work, during which students actively build, analyze, and refine a numerical model using Computational Fluid Dynamics (CFD) tools.

Preliminary theoretical explanations will provide the necessary physical and mathematical foundations for understanding modeling decisions, such as turbulence representation, meshing strategies, boundary condition definition, and results interpretation. These concepts are immediately applied to the ongoing practical case study, reinforcing learning through direct implementation.

Students continuously work on the same CFD problem: a 2D flow simulation explored in different fluid scenarios relevant to bioengineering applications. This approach allows students to thoroughly experience the entire modeling workflow, from problem definition to simulation validation. Learning is based on active participation, guided problem-solving, and iterative improvement of the model.

LEARNING OBJECTIVES OF THE SUBJECT

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.

STUDY LOAD

Type	Hours	Percentage
Self study	90,0	75.00
Laboratory classes	10,0	8.33
Practical classes	20,0	16.67

Total learning time: 120 h

CONTENTS

1. Problem definition and physical flow interpretation

Description:
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.

Specific objectives:
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.

Related activities:
- 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

Full-or-part-time: 10h
Practical classes: 2h
Laboratory classes: 1h
Self study : 7h

2. Geometry creation and mesh generation

Description:
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

Specific objectives:
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.

Related activities:
- 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

Full-or-part-time: 13h
Practical classes: 3h
Laboratory classes: 2h
Self study : 8h

UNIVERSITAT POLITÈCNICA DE CATALUNYA BARCELONATECH

3. Boundary conditions and numerical setup

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

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

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

Full-or-part-time: 20h
Practical classes: 3h
Laboratory classes: 2h 30m
Self study : 14h 30m

4. Simulation and post-processing

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

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

Related activities:
- 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

Full-or-part-time: 31h 45m
Practical classes: 7h
Laboratory classes: 2h 30m
Self study : 22h 15m

5. Analysis, validation and reporting

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

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

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

Full-or-part-time: 45h 15m
Practical classes: 5h
Laboratory classes: 1h
Self study : 39h 15m

ACTIVITIES

Applied theory

Description:
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.

Specific objectives:
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.

Delivery:
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.

Full-or-part-time: 50h
Practical classes: 20h
Self study: 30h

Practical exercises

Description:
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.

Specific objectives:
- 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.

Delivery:
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.

Full-or-part-time: 70h
Laboratory classes: 10h
Self study: 60h

GRADING SYSTEM

The course is assessed through a continuous assessment system.

N1: assignment corresponding to phase 1 of the CFD case study (problem definition).

N2: assignment corresponding to phase 2 of the CFD case study (numerical setup).

N3: assignment corresponding to phase 3 of the CFD case study (simulation).

N4: assignment corresponding to phase 4 of the CFD case study (analysis and interpretation of results).

N5: regular attendance and active participation in class, including participation in practical sessions and guided problem-solving activities.

NF = 0.20 N1 + 0.20 N2 + 0.20 N3 + 0.20 N4 + 0.20 N5

EXAMINATION RULES.

All assessment activities are part of the continuous assessment process. Practical assignments must be submitted within the deadlines established during the course and are directly related to the tasks carried out in class.

BIBLIOGRAPHY

Complementary:

- 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 0070016852.
- Patankar, Suhas V. Numerical heat transfer and fluid flow [on line]. First edition. Boca Ratón, FL: CRC Press, 1980 [Consultation: 06/02/2026]. Available on: https://www-taylorfrancis-com.recursos.biblioteca.upc.edu/books/mono/10.1201/9781482234213/numerical-heat-transfer-fluid-flow-suhas-patankar. ISBN 9781315275130.

RESOURCES

Hyperlink:

- Autodesk CFD – Software de simulació CFD per a estudiants UPC. Resource

Other resources:

https://www.autodesk.com/education/edu-software/overview />https://www.autodesk.com/es/products/cfd/overview