



051176 - Computational Techniques for Thermochemical Propulsion
Master of Science in Aeronautical Engineering

Course Content and Practical Info

Prof. Federico Piscaglia

Dept. of Aerospace Science and Technology (DAER)

POLITECNICO DI MILANO, Italy

federico.piscaglia@polimi.it

This course will provide students with an introduction to numerical methods and analysis techniques used in **computational solutions of fluid mechanics involving turbulent reacting flows** and to provide an introduction to the basic approaches and models commonly used in the literature. The specific objectives of the course are:

- to introduce students with the major approaches and methodologies used in **Computational Fluid Dynamics (CFD) of turbulent reacting flows**, the interplay of physics and numeric, the methods and results of numerical analysis;
- to cover a range of **modern approaches for numerical and computational fluid dynamics**, aiming to provide students with a general knowledge and understanding of the subject, including recommendations for further studies;
- **to familiarize students with the numerical implementation of these techniques**, so as to provide them with the means to implement and use basic CFD methods, computer use and programming, debugging, and so acquire the knowledge necessary for the skillful utilization of CFD software.

1. **Introduction.** The Finite Volume (FV) method in CFD. Overview of the conservation principles governing fluid flows and related transport phenomena. Solution of the Navier-Stokes equations; elements of linear algebra.
2. **Numerical solvers:** algorithms for steady (SIMPLE-based) and transient (PISO-based) flow calculations.
3. **Turbulence Modeling**
4. **Combustion modeling**
5. **Wall film modeling**
6. **Spray modeling**
7. **Mesh motion strategies** for moving boundary problems.

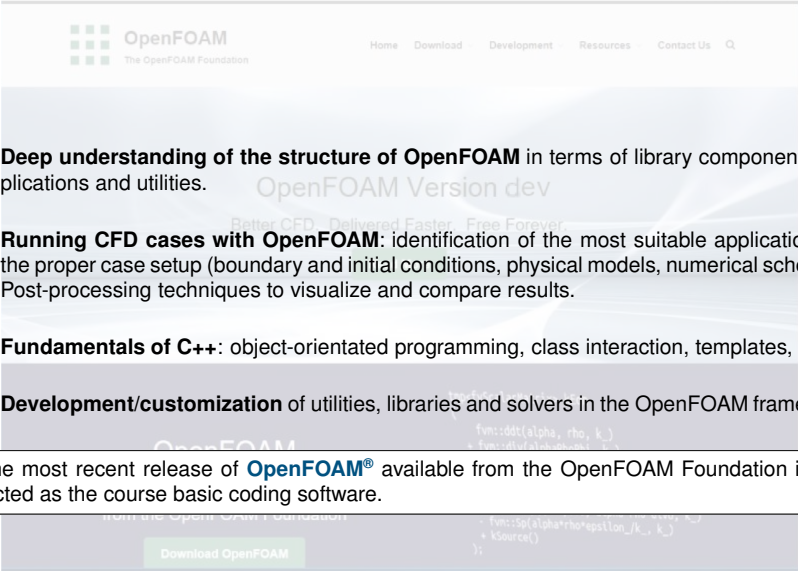
OpenFOAM is selected as the course basic coding software. OpenFOAM® is a free, open-source software, licensed under the GNU General Public License. **A significant part of this class will be devoted to understand the complex structure of the code, how to use and extend it.**

1. **Numerical methods for reacting and non-reacting flows.** General forms of conservation equations for reacting flows; choice of primitive variables, pressure-velocity coupling algorithm for segregated solvers with static and dynamic grids. Discretization of the operators, solution algorithms, momentum interpolation methods. Unsteady convection-diffusion-reaction equation. Numerical solvers for stiff differential equations in chemical systems.
2. **Some comments on turbulence and mixing.** Statistical description of turbulence, turbulent scales, temporal and spatial correlations, Reynolds average equations. Phenomenological aspects of turbulent mixing. The meaning of scalar dissipation rate.
3. **Flows with non-premixed reactants.** Phenomenological description, flame structure, specific features of turbulent non-premixed flames. Modeling techniques: Moment methods, well-stirred reactor, conserved scalar methods, transported pdfs.
4. **Flows with premixed reactants.** Laminar premixed flames, turbulent premixed flames. Turbulent premixed combustion modeling.
5. **Spray Modeling.** Definition of spray regimes. Thin and dilute spray assumption. Spray modeling: continuum droplet model (Volume of Fluid), Lagrangian particle tracking.
6. **Wall film modeling.** Physics of the wall film formation and evolution. Liquid film transport modeling on coupled surface regions; coupling of the film model to the bulk flow, both for continuum and discrete (particle) phases.
7. **Mesh motion strategies for moving boundary problems.** Study of the main techniques for dynamic mesh handling.

This is a course about

CFD modeling of Turbulent Reacting Flows

OpenFOAM is selected as the course basic coding software. OpenFOAM® is a free, open-source software, licensed under the GNU General Public License and widely used in Industry and Academia worldwide; it is first and foremost a C++ library, that allows users to create new solvers and utilities with some pre-requisite knowledge of the underlying method, physics and programming techniques involved; hence, it often helps concentrate on the algorithms themselves rather than on the “coding” of basic and elementary steps.

- 
- The background of the slide is a screenshot of the OpenFOAM website. At the top left is the OpenFOAM logo (a 3x3 grid of squares) and the text 'OpenFOAM The OpenFOAM Foundation'. To the right are navigation links: 'Home', 'Download', 'Development', 'Resources', 'Contact Us', and a search icon. Below the navigation bar is a large banner with the text 'OpenFOAM Version dev' and 'Better CFD. Delivered Faster. Free Forever.' in a stylized font. At the bottom of the banner is a 'Download OpenFOAM' button. On the right side of the banner, there is a code snippet showing C++ code for calculating the divergence of a vector field.
- **Deep understanding of the structure of OpenFOAM** in terms of library components, applications and utilities.
 - **Running CFD cases with OpenFOAM:** identification of the most suitable applications, of the proper case setup (boundary and initial conditions, physical models, numerical schemes). Post-processing techniques to visualize and compare results.
 - **Fundamentals of C++:** object-orientated programming, class interaction, templates, ...
 - **Development/customization** of utilities, libraries and solvers in the OpenFOAM framework.

The most recent release of **OpenFOAM®** available from the OpenFOAM Foundation is selected as the course basic coding software.



🕒 10th July 2018

OpenFOAM 6 Released

The OpenFOAM Foundation is pleased to announce the release of version 6 of the OpenFOAM open source CFD toolbox. Version 6 is a snapshot of the OpenFOAM development version which, through [sustainable development](#), is always-releasable, provides new functionality and major improvements to existing code, with strict development usability, robustness and extensibility.

OpenFOAM 6 includes the following key developments.

- **Conjugate heat transfer (CHT):** improved usability, with simplified set up and run.
- **Rotating/sliding geometries:** more robust AML, and support for periodic cases.
- **Particle tracking:** optimised computation and improved robustness.
- **Reacting multiphase models:** phase change, reactions, drag, breakup, coalescence.
- **Reactions/combustion:** faster, full algebraic Jacobian, significant code redesign.
- **Other models:** water waves and films, turbulence, thermophysics, atmospheric flow.
- **General:** new boundary conditions, function objects, improved code compilation.
- **Further tools for more productive CFD with OpenFOAM.**
- Approximately 750 code commits, 350+ resolved issues
- **ISO/IEC 14882:2011 (C++11):** tested for GCC v4.8+, Clang v3.7+, Intel ICC v17.0.4+.

Credits

OpenFOAM 6 was produced by:

- **Core Team (CFD Direct):** [Henry Weller](#) (co-founder & lead developer); [Chris Greenshields](#) (co-founder), [Will Bainbridge](#)
- **Developers/Maintainers:** Bruno Santos, Francesco Contino, Mattijs Janssens (co-founder), Juho Peltola, Fabian Schlegel, Ronald Oertel, Timo Niemi
- **Patch Contributors:** Tobias Holzmann, Kevin Nordin-Bates, Lorenzo Trevisan, [Federico Piscaglia](#), Björn Pfeiffelmann, Jakub Benda, Nicolas Bourgeois, SeongMo Yeon, Stefan Hildenbrand

Licence

OpenFOAM 6 is distributed under the [General Public Licence v3](#) by the [OpenFOAM Foundation](#).

🕒 10th July 2018 👤 [Chris Greenshields](#) 📁 [Release Announcements](#), [Source](#)

<https://openfoam.org/release/6/>

Our Commitment to OpenFOAM®



🕒 8th July 2019

OpenFOAM 7 Released

The OpenFOAM Foundation is pleased to announce the release of version 7 of the OpenFOAM open source CFD toolbox. Version 7 is a snapshot of the OpenFOAM development version which, through sustainable development, is always-releasable. It provides new functionality and major improvements to existing code, with strict demands on usability, robustness and extensibility.

OpenFOAM 7 includes the following key developments.

- **Heat transfer:** consolidated solvers and improved convergence and robustness.
- **Particle tracking:** improved robustness and optimized computation.
- **Multiphase:** wave damping, configurable inlet phase properties, better settling numerics.
- **Reacting multiphase models:** heat transfer, population balance, breakup, coalescence.
- **Reactions/combustion:** simplified case setup.
- **Turbulence:** improved consistency and stability of wall functions, added sources.
- **Thermophysical:** thermodynamic functions, temperature-strain-dependent viscosity.
- **Other models:** atmospheric, rigid body dynamics, boundary conditions, sources.
- **Mesh:** standardized dynamic mesh capability, improved motion solvers.
- **Case Configuration:** improved data visualization, setup tools, function objects.
- **Computation:** improvements to containers, fields, parallel running, etc.
- Approximately 550 code commits, 250+ resolved issues
- **ISO/IEC 14882:2011 (C++11):** tested for GCC v4.8+, Clang v7.0+, Intel ICC v18.0+.

Credits

OpenFOAM 7 was produced by:

- **Core Team (CFD Direct):** **Henry Weller** (co-founder & lead developer); **Chris Greenshields** (co-founder), **Will Bainbridge**
- **Developers/Maintainers:** Mattijs Janssens (co-founder), Juho Peltola, Timo Niemi, Fabian Schlegel, Ronald Oertel, Bruno Santos
- **Patch Contributors:** Francesco Contino, Lorenzo Trevisan, Federico Piscaglia, Robert Lee, SeongMo Yeon, Alberto Passalacqua

Licence

OpenFOAM 7 is distributed under the **General Public Licence v3** by the **OpenFOAM Foundation**.

🕒 8th July 2019 👤 Chris Greenshields ➡ Release Announcements, Source

Getting the Best OpenFOAM Training

Productive CFD in OpenFOAM

OpenFOAM User Guide

OpenFOAM Open Day 2018

<https://openfoam.org/release/7/>

The class consists of:

1. [theory](#) (4 hours/week) about **CFD modeling of turbulent compressible reacting flows**. The different topics of the program will be discussed by commenting the slides provided with the class;
2. [exercises/laboratory](#) (4 hours/week), about the **advanced use of OpenFOAM**. A significant part of this class will be devoted to learn how to use the code, to understand its complex object-oriented structure and, finally, to extend its capabilities.

THESES AVAILABLE:

1. [We have large availability of theses in OpenFOAM](#). If you are interested please contact me. Provided that you want to work in OpenFOAM, we will find for sure a topic matching your interests!
2. We sometime have [thesis offers by Companies](#). This offers are loaded on the [Beep](#) portal. Please check, in case you are interested!

Schedule & Contact Info



Course Title		CFU	Semester	Campus	Language	No. of students enrolled	Course details						
051176 - COMPUTATIONAL TECHNIQUES FOR THERMOCHEMICAL PROPULSION		8.00	1	Milano Bovisà		—							
Alphabetical grouping		Class schedule planning											
Date	Where	09:00	10:00	11:00	12:00	13:00	14:00	15:00	16:00	17:00	18:00	19:00	20:00
Monday													
Tuesday	BL 27.15									Resonance COMPUTATIONAL TECHNIQUES FOR THERMOCHEMICAL PROPULSION (from 17/09/2019 to 17/11/2019)			
Wednesday													
Thursday	BL 2.9			Resonance COMPUTATIONAL TECHNIQUES FOR THERMOCHEMICAL PROPULSION (from 16/09/2019 to 16/11/2019)									
	LM.3												
										Interstazionale COMPUTATIONAL TECHNIQUES FOR THERMOCHEMICAL PROPULSION (from 16/09/2019 to 16/11/2019)			
Friday	BL 28.1.2			Resonance COMPUTATIONAL TECHNIQUES FOR THERMOCHEMICAL PROPULSION (from 20/09/2019 to 20/11/2019)									
Saturday													

Office hours:
upon appointment

Prof. Federico Piscaglia

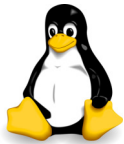
address Dept. of Aerospace Science and Technology (DAER)
POLITECNICO DI MILANO
via La Masa 34, 20158 Milano - Italy

office: building B12, 2nd floor

phone: (+39) 02 2399 8620

e-mail: federico.piscaglia@polimi.it

web: piscaglia.aero.polimi.it



OpenFOAM is written for the UNIX and GNU/Linux operating systems, therefore installation on a native *nix OS remains the best choice.

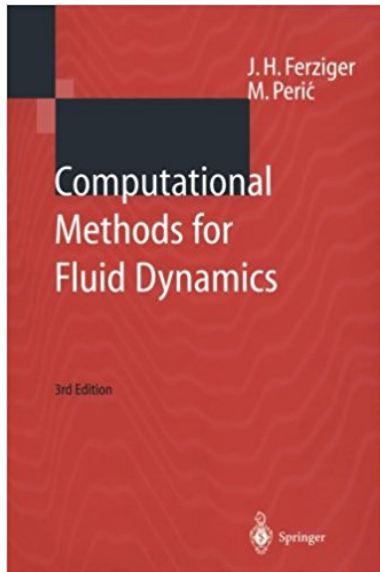
Several options are available to **install OpenFOAM on your own machine**. You can install OpenFOAM on:

- a native `Linux/Unix` distribution on your Laptop
→ you can install and start `Linux` from a bootable USB key!
- a “containerized” version of OpenFOAM (`Docker`, any OS¹)
- a `Linux` distribution running on a Virtual Machine (`VirtualBox`, any OS¹)
- Win10 only: use an embedded `Linux` emulator (`Ubuntu bash shell`)

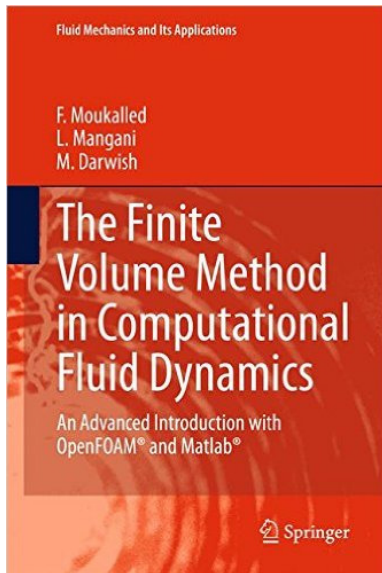
¹ `Linux/Unix`, `macOS`, `Windows`

1. **J.H. Ferziger and M. Perić.** *Computational Methods for Fluid Dynamics.* Springer-Verlag, Berlin, 1999.
2. **F. Moukalled, L. Mangani, M. Darwish.** *The Finite Volume Method in Computational Fluid Dynamics. An Advanced Introduction with OpenFOAM and Matlab.* Fluid Mechanics and Its Applications, vol. 113, Springer International Publishing, 2016.
3. **T. Poinso and D. Veynante.** *Theoretical and Numerical Combustion.* Fourth Edition, 2015.
4. **The OpenFOAM Foundation.** *OpenFOAM User Guide.*
<https://cfd.direct/openfoam/user-guide>

OpenFOAM is selected as the course basic coding software. OpenFOAM® is a free, open-source software, licensed under the GNU General Public License and widely used in Industry and Academia worldwide; it is first and foremost a C++ library, that allows users to create new solvers and utilities with some pre-requisite knowledge of the underlying method, physics and programming techniques involved; hence, it often helps concentrate on the algorithms themselves rather than on the “coding” of basic and elementary steps.



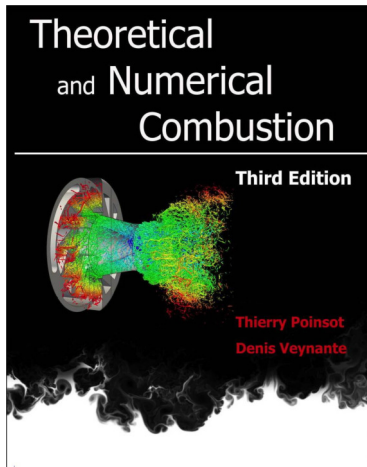
Ferziger, Joel H., Peric, Milovan. “Computational Methods for Fluid Dynamics”, Third Edition, Springer 2002.



Some figures in this slides are taken from the reference text book:

F. Moukalled, L. Mangani, M. Darwish. "The Finite Volume Method in Computational Fluid Dynamics", Springer International Publishing Switzerland 2016.

Prof. Marwan Darwish is greatly acknowledged for sharing the images from his book and for allowing to include them in this course's material.



T. Poinso, D. Veynante. **"Theoretical and Numerical Combustion"** (Third Edition), Cerfacs, 2015.

Evaluation and grading



Hours of lecture: 48 (6 CFU)

Other activities: 32 (2 CFU)

The final course grade will be weighted as follows:

ACTIVITIES	PERCENTAGE OF FINAL GRADE
Homework (5 in total, 4% each)	20%
Final project (1)	10%
Written Exam	30%
Oral Exam	40%

Homework

Five problem sets (+1 final..) will be given. They will be due during the class. We encourage students to work with each other on the homework assignments, but we do not condone copying. Make your own honest collaborative efforts to contribute to the solution and, based on your own understanding, write up the answers in your own words and style. If you worked closely with other students on a given homework assignment and feel that your understanding was substantially influenced by the mutual learning process, you should cite the names of those students with whom you worked.

Evaluation and grading



Hours of lecture: 48 (6 CFU)

Other activities: 32 (2 CFU)

The final course grade will be weighted as follows:

ACTIVITIES	PERCENTAGE OF FINAL GRADE
Homework (5 in total, 4% each)	20%
Final project (1)	10%
Written Exam	30%
Oral Exam	40%

Final Project

Each student (or group of students) will work to a project (final assignment) using the OpenFOAM® code. We plan to have a final session where all students will make a presentation of their projects to the whole class and staff. Such presentations provide an excellent means for additional learning and sharing. Evaluation will be based on presentation and related material quality.

mesh generation (1)



steady flow tests (2)



turbulence and heat transfer (3)



lagrangian spray modeling (4)



finite-volume wall-film modeling (5)



reactive flow modeling (6)

SOME NOTES:

- The final project consists of the application and/or development of OpenFOAM to the [simulation of a SWIRLED COMBUSTOR](#).
- The short CFD project carried out by [groups of 2 students \(or individually\)](#). Each group has to provide:
 - short report ($\sim 10/20$ pages);
 - working cases and applications in OpenFOAM®.
- **Final projects will be evaluated and discussed during the oral exam sessions.**

Thank you for your attention!

contact: federico.piscaglia@polimi.it

Prof. **Federico Piscaglia**, Ph.D.

CONTACT INFORMATION

address Dept. of Aerospace Science and Technology (DAER)
 POLITECNICO DI MILANO
 via La Masa 34, 20156 Milano - Italy

e-mail: federico.piscaglia@polimi.it

phone: (+39) 02 2399 8620

web: piscaglia.aero.polimi.it