

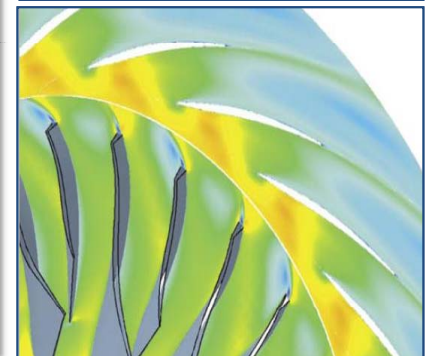
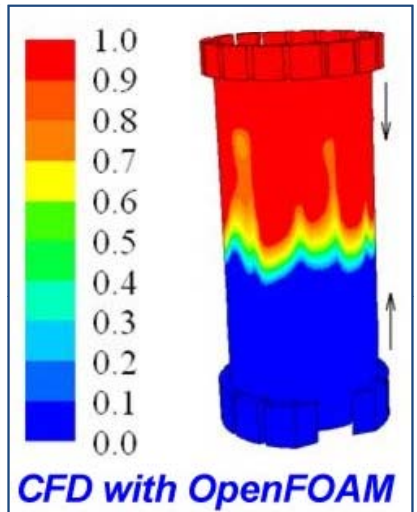
Online course. The content of the course is 60 hours, and the maximum time to complete it is 3 months. At the end of the course an aptitude certificate will be issued to the students.

CFD with OpenFOAM course includes manuals, videolessons and exercises. Our website has chat, forums, remote desktop connection, video conferencing, internal mail, etc. The teachers (M.I. Lamas and C.G. Rodriguez) have an extensive experience in CFD and OpenFOAM and papers in important international journals. Price: **400 €**

OpenFOAM is an open CFD (Computational Fluid Dynamics) software available for free at [www.openfoam.org](http://www.openfoam.org).

OpenFOAM community is growing fast and thus this software is becoming an important tool for both commercial companies and academics. OpenFOAM includes 80 solvers and more than 170 tutorials. They are useful to solve an extensive range of fluid flows, for instance the following ones:

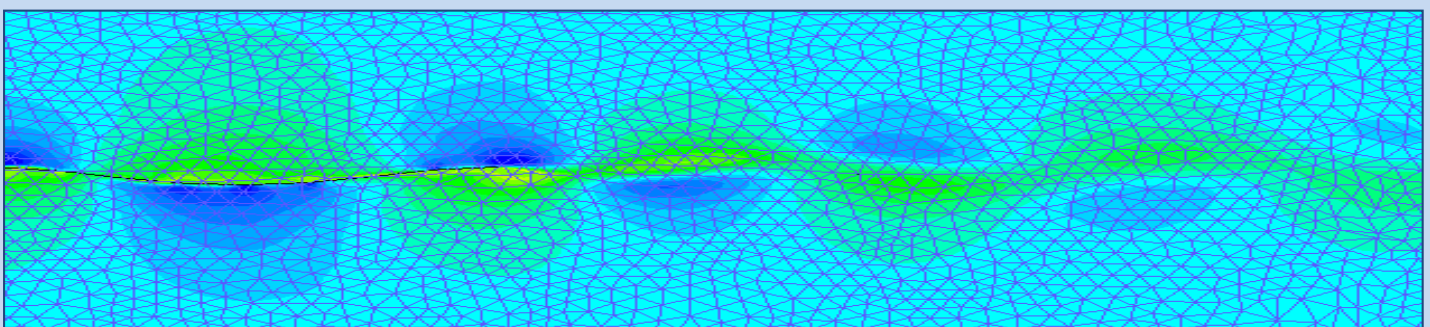
- Elemental computational fluid dynamics problems
- Compressible flows
- Chemical reactions
- Combustion
- Turbulence
- Heat transfer
- Engines and turbomachinery
- Solid dynamics
- Supersonic flows
- Electromagnetics
- Multiphase flows
- Etc



As OpenFOAM is an open software, it allows users to edit the source code. The code is written in C++.

Another advantage is that it can be run in parallel mode using multiple processors on a multiprocessor computer or on many computers across a network.

OpenFOAM also includes free meshing tools.



## CONTENTS

### Chapter 1: Computational Fluid Dynamics (CFD)

(This chapter includes a 33 pages text and a videolesson)

- 1.1 Introduction
- 1.2 Discretization process
  - 1.2.1 Spatial discretization of the domain
  - 1.2.2 Standard transport equation and equation discretization
- 1.3 Solution of discretized equations

### Chapter 2: Introduction to OpenFOAM

(This chapter includes a 41 pages text and a videolesson)

- 2.1 Introduction
- 2.2 Installation of OpenFOAM under Linux, Windows and Mac OS
  - 2.2.1 Installation under Linux
  - 2.2.2 Installation under Windows and Mac OS
- 2.3 States of a simulation in OpenFOAM
  - 2.3.1 Preprocessing
  - 2.3.2 Solving
  - 2.3.3 Postprocessing
- 2.4 Files and folders needed to execute a simulation
  - 2.4.1 Tutorials
  - 2.4.2 Solvers
- 2.5 Execution of a simulation

### Chapter 3: Postprocessing an OpenFOAM simulation

(This chapter includes a 20 pages text and a videolesson)

- 3.1 Introduction
- 3.2 Postprocessing with ParaView
- 3.3 Postprocessing with other software packages

### Chapter 4: Mesh generation for OpenFOAM

(This chapter includes a 16 pages text and a videolesson)

- 4.1 Introduction
- 4.2 Generation of a mesh
  - 4.2.1 Generation of a mesh using OpenFOAM
  - 4.2.2 Generation of a mesh using other software packages

### Chapter 5: Physical models included in OpenFOAM

(This chapter includes a 10 pages text and a videolesson)

- 5.1 Introduction
- 5.2 Solvers included in OpenFOAM
- 5.3 Tutorials included in OpenFOAM
- 5.4 Utilities included in OpenFOAM
- 5.5 Libraries included in OpenFOAM

### Chapter 6: Discretization schemes and solution controls in OpenFOAM

(This chapter includes a 11 pages text and a videolesson)

- 6.1 Introduction
- 6.2 Discretization schemes
  - 6.2.1 Temporal schemes
  - 6.2.2 Gradient schemes
  - 6.2.3 Divergence schemes
  - 6.2.4 Laplacian schemes
  - 6.2.5 Interpolation schemes
  - 6.2.6 Surface normal gradient schemes
  - 6.2.7 Flux calculation
- 6.3 Solution controls
  - 6.3.1 Solvers
  - 6.3.2 PISO and SIMPLE controls
  - 6.3.3 Relaxation factors

### Chapter 7: Parallelization in OpenFOAM

(This chapter includes a 10 pages text and a videolesson)

- 7.1 Introduction
- 7.2 Decomposition of a mesh
- 7.3 Running a solver in parallel
- 7.4 Postprocessing

### Chapter 8: Convergence in OpenFOAM

(This chapter includes a 14 pages text and a videolesson)

- 8.1 Introduction
- 8.2 Graphical representation of residuals
- 8.3 Mesh
- 8.4 Time step
- 8.5 Initial conditions
- 8.6 Discretization schemes
- 8.7 Under relaxation factors
- 8.8 Solvers

### Chapter 9: Structure of an OpenFOAM solver

(This chapter includes a 45 pages text and a videolesson)

- 9.1 Introduction
- 9.2 Discretization of the transport equations
- 9.3 Structure of an OpenFOAM solver
- 9.4 Examples of OpenFOAM solvers

### Chapter 10: Development of an own solver in OpenFOAM

(This chapter includes a 22 pages text and a videolesson)

- 10.1 Introduction
- 10.2 Modification of a solver
- 10.3 Development of a new solver

### Chapter 11: Additional resources

(This chapter includes a 6 pages text and a videolesson)

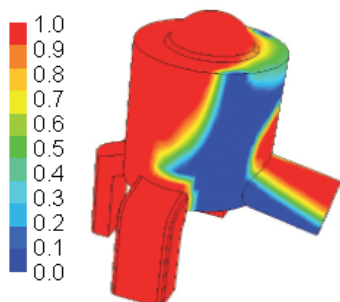
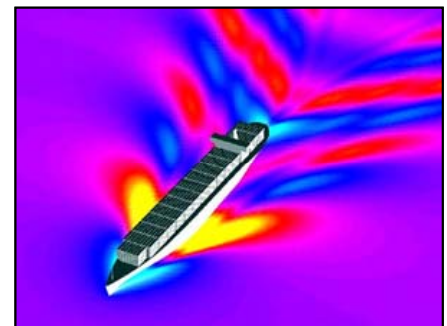
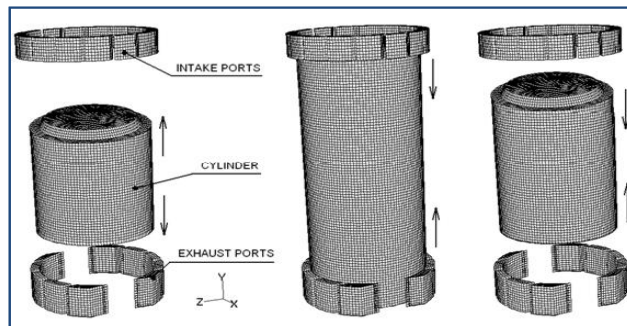
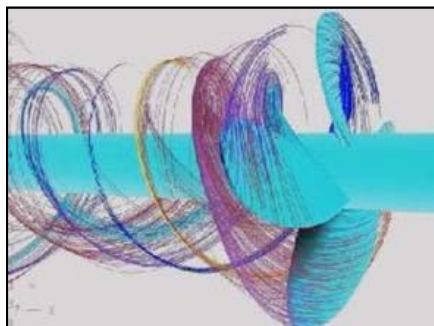
- 11.1 Introduction
- 11.2 CFD online forum
- 11.3 OpenFOAM manuals and OpenFOAM user guides
- 11.4 ParaView manuals and ParaView user guides
- 11.5 OpenFOAM exercises and tutorials

### OpenFOAM EXERCISES (BASIC LEVEL):

- OpenFOAM exercise 1: Mesh creation (4 pages and a videolesson)
- OpenFOAM exercise 2: Meshing a plate (9 pages and a videolesson)
- OpenFOAM exercise 3: Conversion of Fluent format to OpenFOAM format (4 pages and a videolesson)
- OpenFOAM exercise 4: Transient laminar flow in a duct (8 pages and a videolesson)
- OpenFOAM exercise 5: Steady laminar flow in a duct (5 pages and a videolesson)
- OpenFOAM exercise 6: Steady heating of a solid wall (5 pages and a videolesson)
- OpenFOAM exercise 7: Development of an own solver. Steady heating of a solid wall with internal energy generation (8 pages and a videolesson)
- OpenFOAM exercise 8: Development of an own solver. Transient level-set reinitialization (13 pages and a videolesson)

### OpenFOAM EXERCISES (INTERMEDIATE LEVEL - OPTIONAL):

- OpenFOAM exercise 9: Development of an own solver. Evaporation (15 pages and a videolesson)
- OpenFOAM exercise 10: Gas leak (15 pages and a videolesson)
- OpenFOAM exercise 11: Cavitation (18 pages and a videolesson)
- OpenFOAM exercise 12: Chemical reactions (19 pages and a videolesson)
- OpenFOAM exercise 13: Combustion (12 pages and a videolesson)
- OpenFOAM exercise 14: Fan (9 pages and a videolesson)
- OpenFOAM exercise 15: Moving mesh (15 pages and a videolesson)



If you are interest in this course, please contact us at:

Telephone: +34 686-691-703

E-mail: [info@technicalcourses.net](mailto:info@technicalcourses.net)

Web: [www.technicalcourses.net](http://www.technicalcourses.net)