CFD with OpenFOAM online course

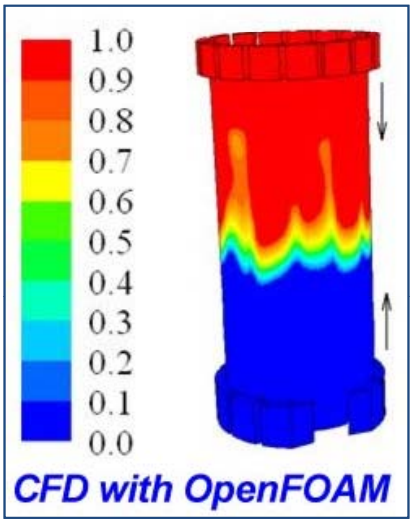
Online course. The content of the course is 50 hours, and the maximum time to complete it is 2 months. At the end of the course an aptitude certificate will be issued to the students.

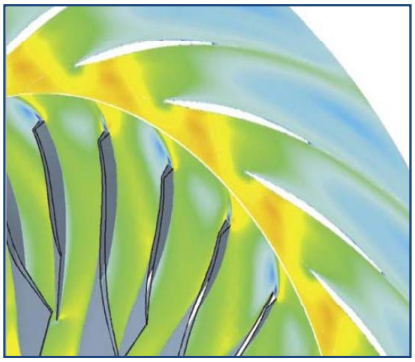
CFD with OpenFOAM course includes manuals, videolessons and exercises. Our website has chat, forums, remote desktop connection, video conferencing, internal mail, etc. The teachers (M.I. Lamas and C.G. Rodriguez) have an extensive experience in CFD and OpenFOAM and papers in important international journals.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

OpenFOAM is an open CFD (Computational Fluid Dynamics) software available for free at www.openfoam.org.

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

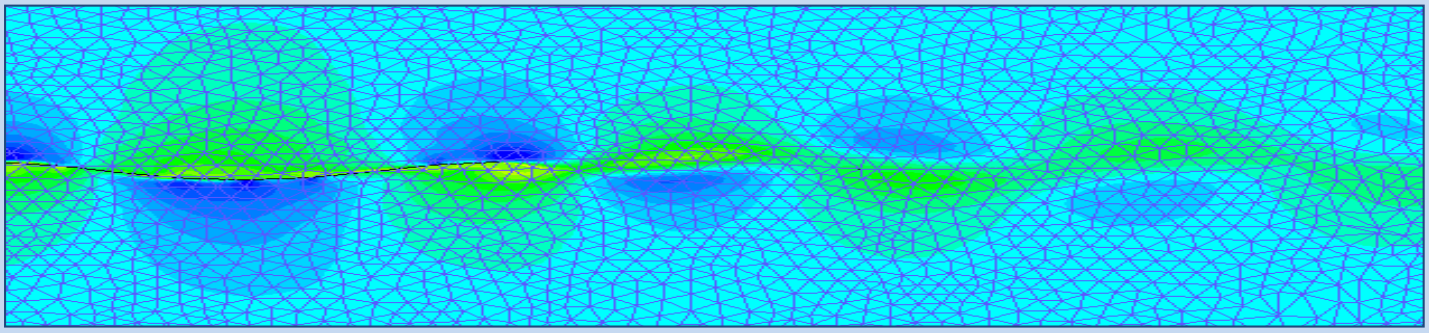




As OpenFOAM is an open software, it allows users to edit the source code. The code is written in C++.

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.



CFD with OpenFOAM online course

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 repesentation 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

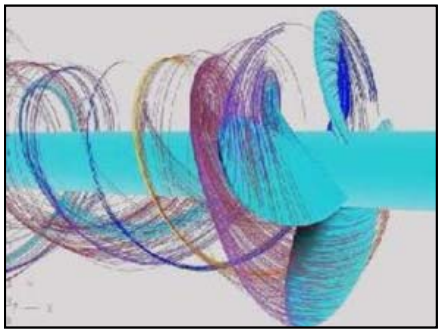
Chapter 9: Structure of an OpenFOAM solver
(This chapter includes a 45 pages text and a videolesson)
9.1 Introduction
9.2 Discretization of the transport equations
9.3 Structure of an OpenFOAM solver
9.4 Examples of OpenFOAM solvers

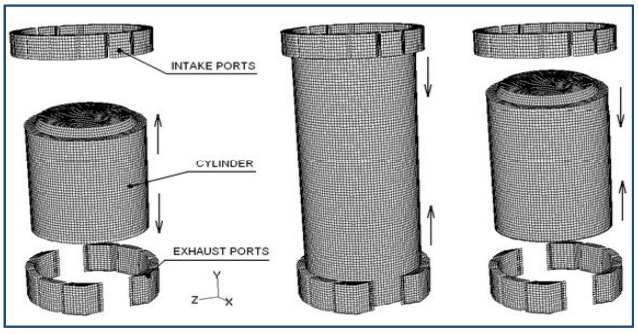
Chapter 10: Development of an own solver in OpenFOAM
(This chapter includes a 22 pages text and a videolesson)
10.1 Introduction
10.2 Modification of a solver
10.3 Development of a new solver

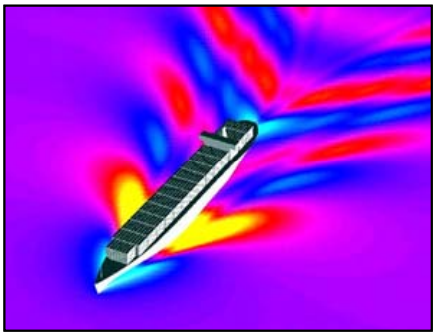
Chapter 11: Additional resources
(This chapter includes a 6 pages text and a videolesson)
11.1 Introduction
11.2 CFD online forum
11.3 OpenFOAM manuals and OpenFOAM user guides
11.4 ParaView manuals and ParaView user guides
11.5 OpenFOAM exercises and tutorials

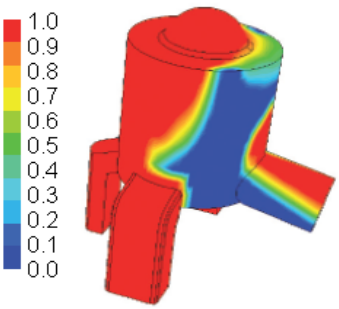
OpenFOAM EXERCISES (BASIC LEVEL):
OpenFOAM exercise 1: Mesh creation (4 pages and a videolesson)
OpenFOAM exercise 2: Meshing a plate (9 pages and a videolesson)
OpenFOAM exercise 3: Conversion of Fluent format to OpenFOAM format (4 pages and a videolesson)
OpenFOAM exercise 4: Transient laminar flow 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

Web: www.technicalcourses.net