CFD with OpenFOAM online course

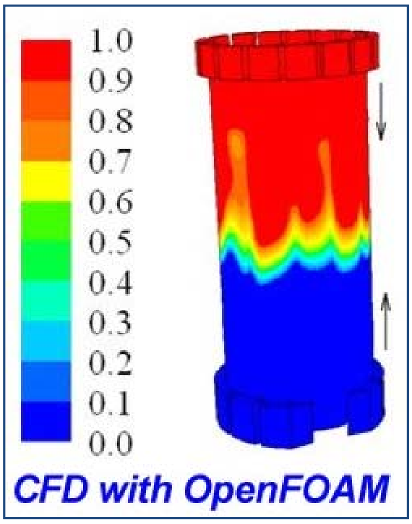
Online course. The content of the course is 60 hours, and the maximum time to complete it is 3 months. At the end of the course an aptitude certificate will be issued to the students.

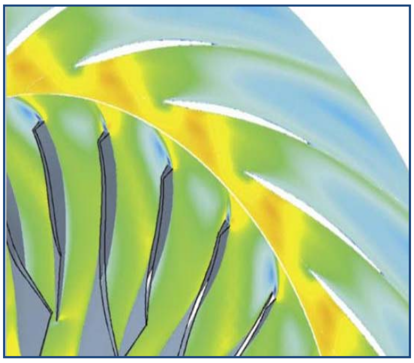
CFD with OpenFOAM course includes manuals, videolessons and exercises. Our website has chat, forums, remote desktop connection, video conferencing, internal mail, etc. The teachers (M.I. Lamas and C.G. Rodriguez) have an extensive experience in CFD and OpenFOAM and papers in important international journals.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

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

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

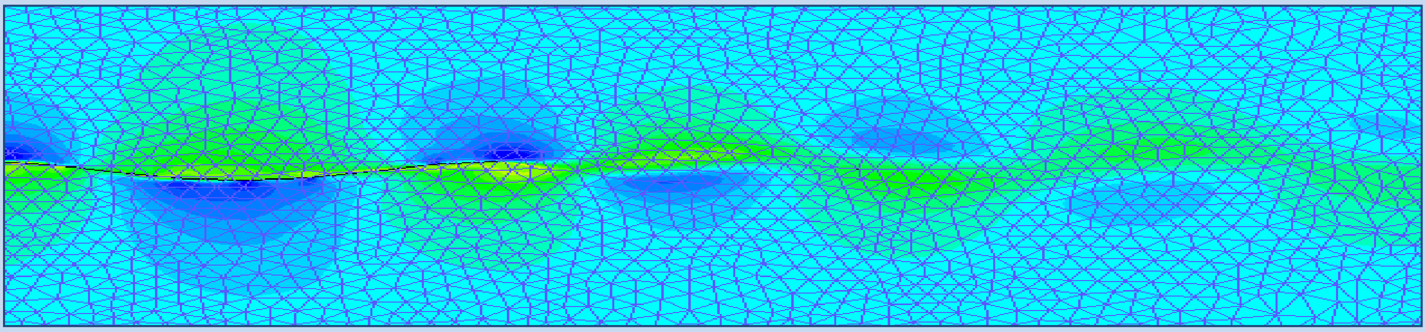




As OpenFOAM is an open software, it allows users to edit the source code. The code is written in C++.

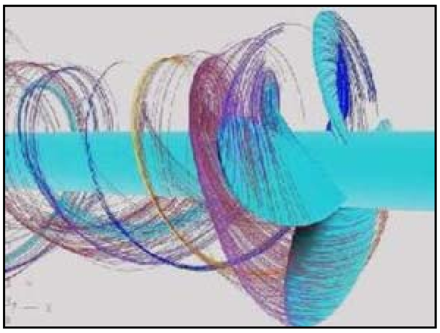
Another advantage is that it can be run in parallel mode using multiple processors on a multiprocessor computer or on many computers across a network.

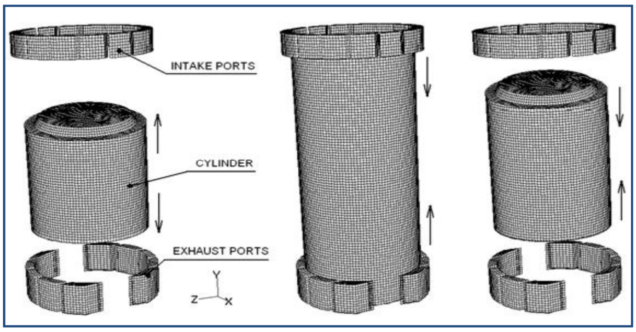
OpenFOAM also includes free meshing tools.

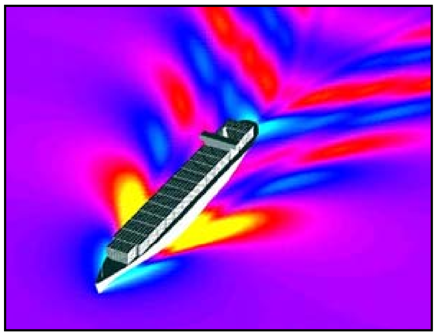


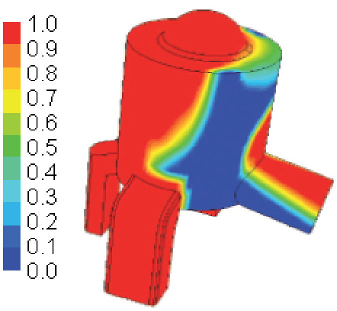
CFD with OpenFOAM online course

CONTENTS









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