Introduction to OpenFOAM

Roberto Pieri - *SCS Italy*

17 June 2014

Overview on *OpenFOAM*

OpenFOAM structure

Official links

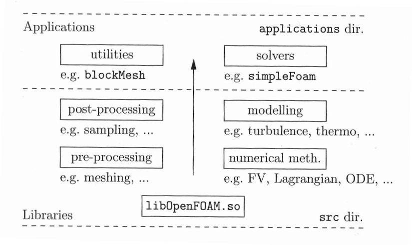
Hands on a real case study

Tutorial session

What is OpenFOAM ?

- Free and open-source toolbox of C++ libraries, licensed under the GNU General Public Licence.
- Produced by *OpenCFD Ltd*.
- Mostly used for computational fluid dynamics.
- Top level code represents the equations being solved.

A toolbox, not a black box

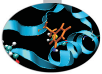


- *OpenFOAM* consists of (80+) libraries.
- Libraries are used to create more than 200 applications.
- *OpenFOAM* consists of about 1.3 millions of code lines.

Case structure

- *<case>*
  - *system*
    - *controlDict*
    - *fvSchemes*
    - *fvSolution*
  - *constant*
    - *. . . Properties*
    - *polyMesh*
      - *points*
      - *faces*
      - *owner*
      - *neighbour*
      - *boundary*
  - time directories

- constant directory contains the directory *polyMesh* and dictionaries specifing all physical properties of the case;
- polyMesh subdirectory contains a full description of the mesh case;
- system directory contains the properties of the solver.
- Time directories contain files of data for every field of the simulation.



Links

- *www.openfoam.org*
  - Download guide for different O.S.
  - Official user guide (html or PDF format).
  - Official programmers guide.
  - C++ source guide.
- *OpenFOAM* wiki : *http://openfoamwiki.net/index.php/Main_Page*
- Online forum : *www.cfd-online.com/Forums/*

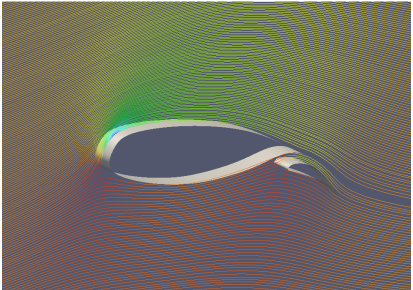
Tutorial

Physical properties

- NLR-7301 airfoil with 20^o flap
- $\mathrm{M} \sim 0.2 \rightarrow$ incompressible
- $\mathrm{Re} \sim 2.6 \cdot 10^6 \rightarrow$ turbulent
- $\alpha = 6^o$
- $|\mathbf{U}| = 40 m/s$

Modellation

- rectangular domain
- $k - \omega$ turbulence model
- steady-state simulation
- *simpleFoam* solver



SuperComputing Applications and Innovation

Tutorial

- Copy extruded mesh in the right directory of the case.
- Set appropriate boundary conditions.
- Set the dictionary for the decomposition.
- Set the directions of lift and drag in the appropriate dictionary.
- Run the solver.