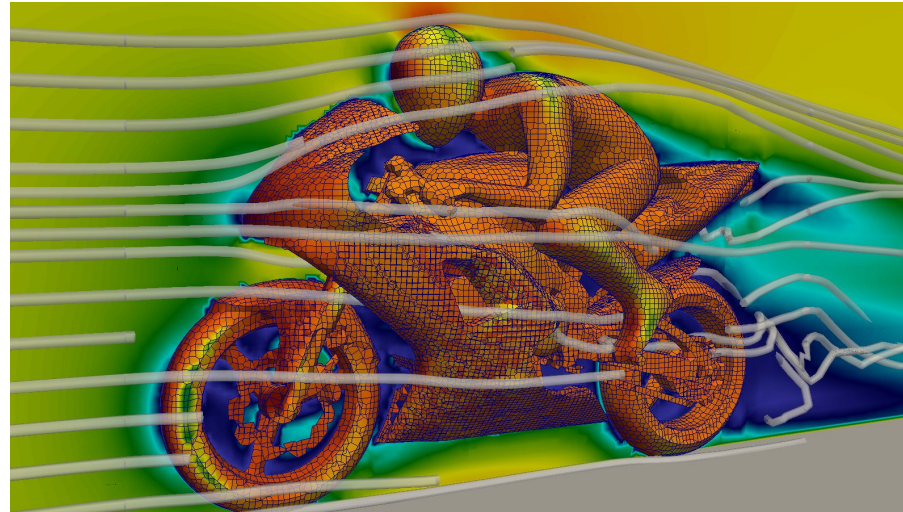# Open Source Computational Fluid Dynamics

An MSc course to gain extended knowledge in Computational Fluid Dynamics (CFD) using open source software.
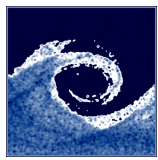
Zoltán Hernádi

Department of Fluid Mechanics

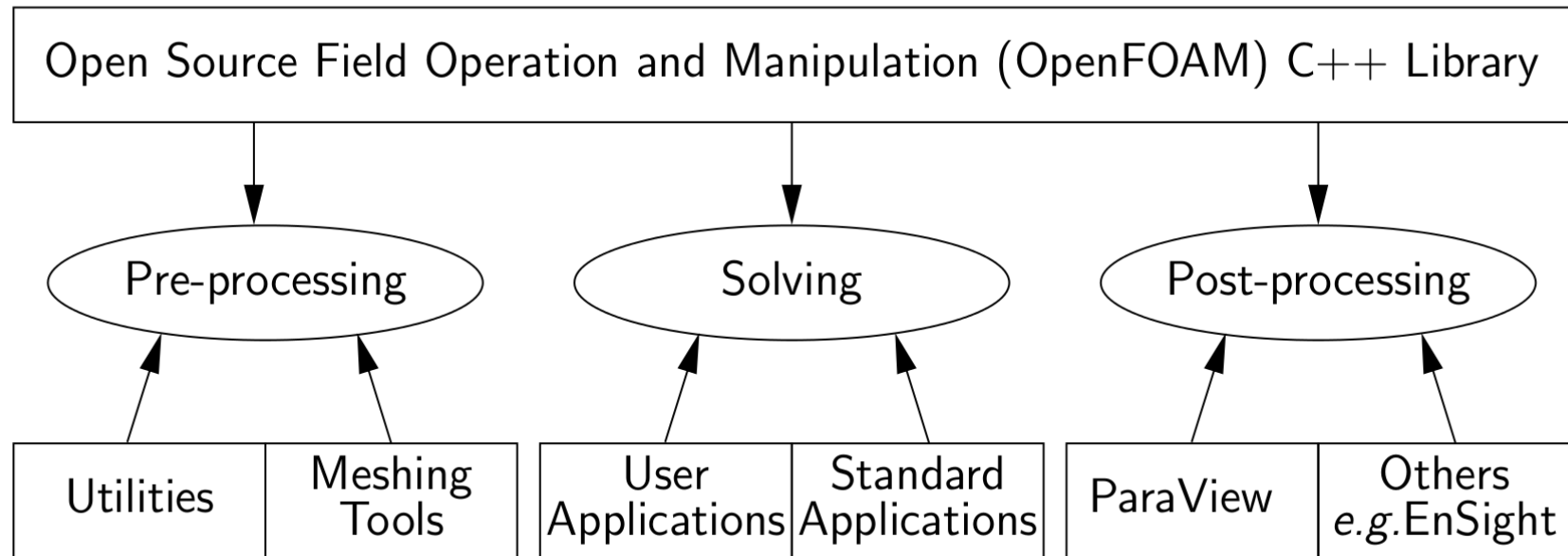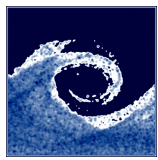Budapest University of Technology and Economics

MŰEGYETEM 1782

Open Source CFD @ Budapest University of Technology and Economics

OpenFOAM software components

Open Source Field Operation and Manipulation (OpenFOAM) C++ Library

Pre-processing

Solving

Post-processing

Utilities	Meshing Tools

User Applications	Standard Applications

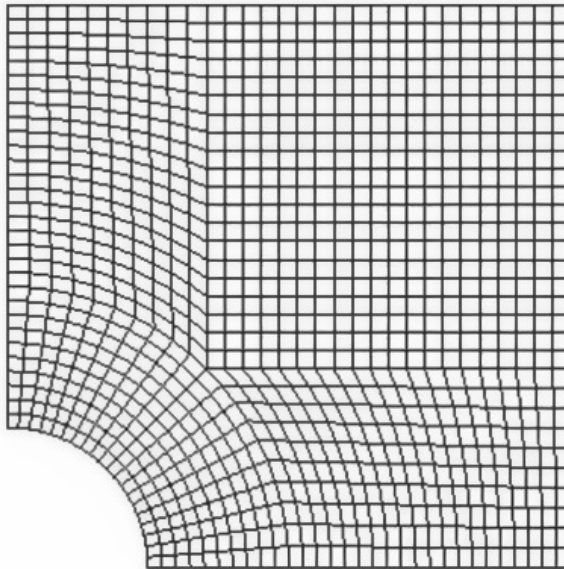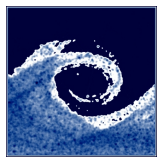| ParaView	Others *e.g.*EnSight

Pre-processing

- Utilities

  - mapFields: interpolate fields to new mesh
  - setFields: initialize fields through dictionary
  - changeDictionary: change dictionary entries, e.g. patch type

- Mesh generation

  - blockMesh: multi-block mesh generator
  - snappyHexMesh: automatic hex mesher, snapping to surface
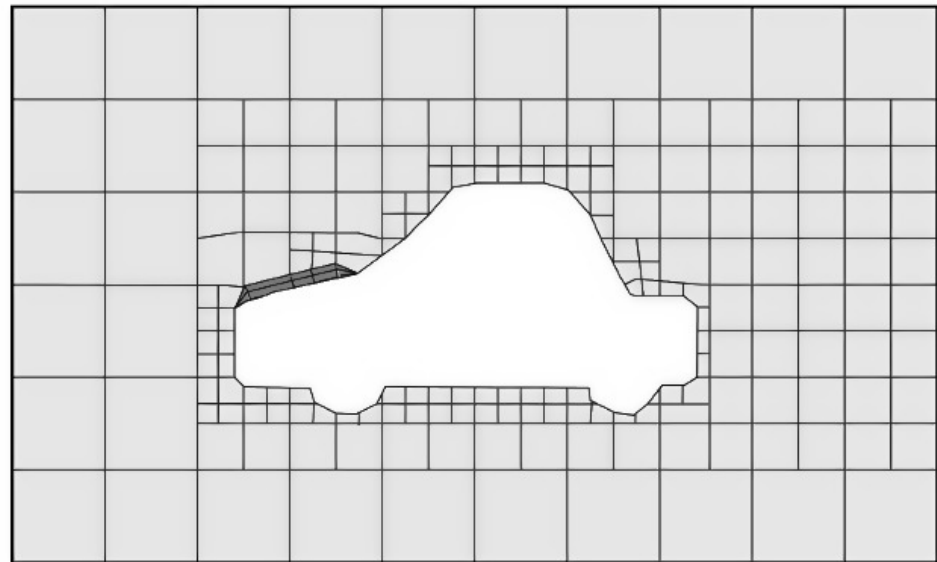
Pre-processing

- Mesh conversion

    - fluentMeshToFoam: convert mesh from Fluent to OpenFOAM
    - gambitToFoam: convert mesh from Gambit to OpenFOAM
    - gmshToFoam: convert mesh from GMSH to OpenFOAM

- Mesh manipulation

    - checkMesh: checks validity of a mesh
    - refineMesh: mesh refinement in multiple directions
    - autoRefineMesh: refine cells near to a surface
    - transportPoints: translate, rotate, scale mesh points
   - renumberMesh: renumbers the cell list to reduce bandwith
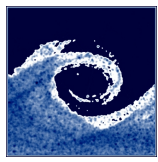
Solving

- Basic solvers

  - potentialFoam: starting fields for Navier-Stokes
  - laplacianFoam: Laplace equation, e.g. thermal diffusion
  - scalarTransportFoam: passive scalar transport

- Incompressible flow

  - icoFoam: transient laminar Newtonian fluid flow
  - simpleFoam: steady-state incompressible flow
  - pisoFoam: transient incompressible flow
  - pimpleFoam: transient incompressible flow - large time-step

- Compressible flow

  - rhoCentralFoam: density-based compressible flow solver
  - sonicFoam: trans-sonic/supersonic flow

- Multiphase flow

  - interFoam: 2 incompressible fluids VOF (volume of fluid)
  - compressibleInterFoam: 2 compressible fluids VOF
 - twoPhaseEulerFoam: 2 incompressible fluids Euler-Euler
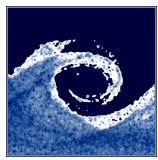
Solving

- Combustion

    - chemFoam: single cell chemistry solver
    - reactingFoam: combustion with chemical reactions
    - engineFoam: internal combustion engines
    - XiFoam: premixed combustion with turbulence modelling

- Heat transfer

    - buoyantBoussinesqSimpleFoam: steady-state buoyant incompressible
    - buoyantSimpleFoam: steady-state buoyant compressible
    - chtMultiRegionSimpleFoam: steady-state conjugate heat transfer

- Others

    - dnsFoam: direct numerical simulation of turbulence
    - mdFoam: molecular dynamics
    - dsmcFoam: direct simulation Monte-Carlo
    - mhdFoam: magnetohydrodynamics
    - solidDispalcementFoam: solid body linear-elastic deformation
    - financialFoam: Black-Scholes equation to price commodities
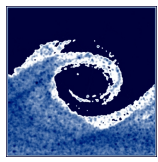
Post-processing

- Paraview
- Utilities

    - Co: Courant number field
    - vorticity: vorticity field
    - patchSummary: display fields and boundary conditions
    - patchAverage: average of a field over a patch
    - patchIntegrate: integral of a field over a patch
    - wallHeatFlux: heat flux over patches
    - wallShearStress: wall shear stress
    - sample: sample a field
    - foamLog: extract data of residuals, iterations, etc.
    - foamCalc: simple calculations of fields

- Libraries for run-time post-processing

    - fieldFunctionObjects: averaging, min/max, etc.
    - forces: lift/drag forces
    - probeLocations: probes at chosen points

- gnuplot
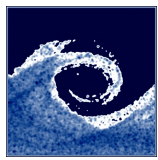
Dimensions

- Dimensions are stored as a list of 7 scalars

  1. Mass [kg]
  2. Length [m]
  3. Time [s]
  4. Temperature [K]
  5. Quantity [kgmol]
  6. Current [A]
  7. Luminous intensity [cd]

- To put it together, remember: [kg m s K kgmol A cd]
- Examples:

  - [0 1 -1 0 0 0 0] : m/s
  - [0 2 -1 0 0 0 0] : m^2/s
  - [1 -1 -2 0 0 0 0] : kg/m/s^2 = Pa