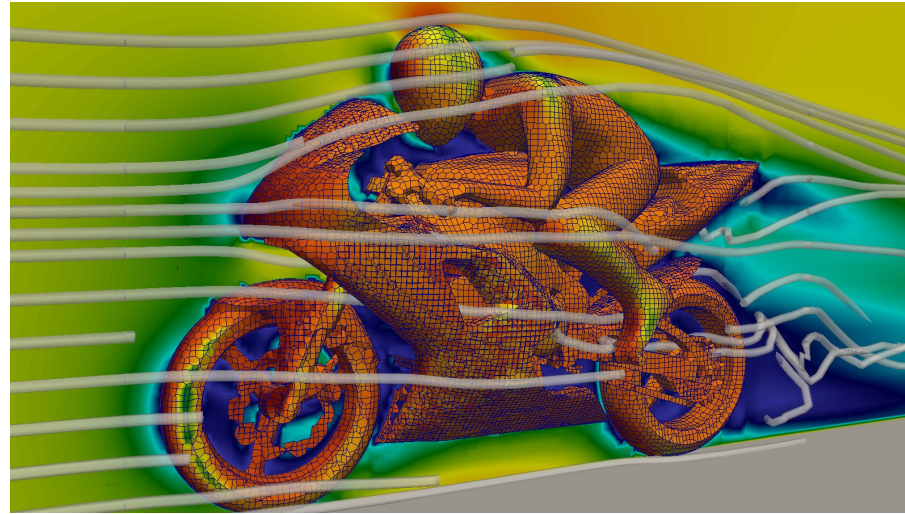


Open Source Computational Fluid Dynamics



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

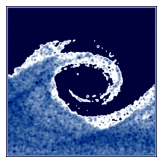
Zoltán Hernádi

Department of Fluid Mechanics

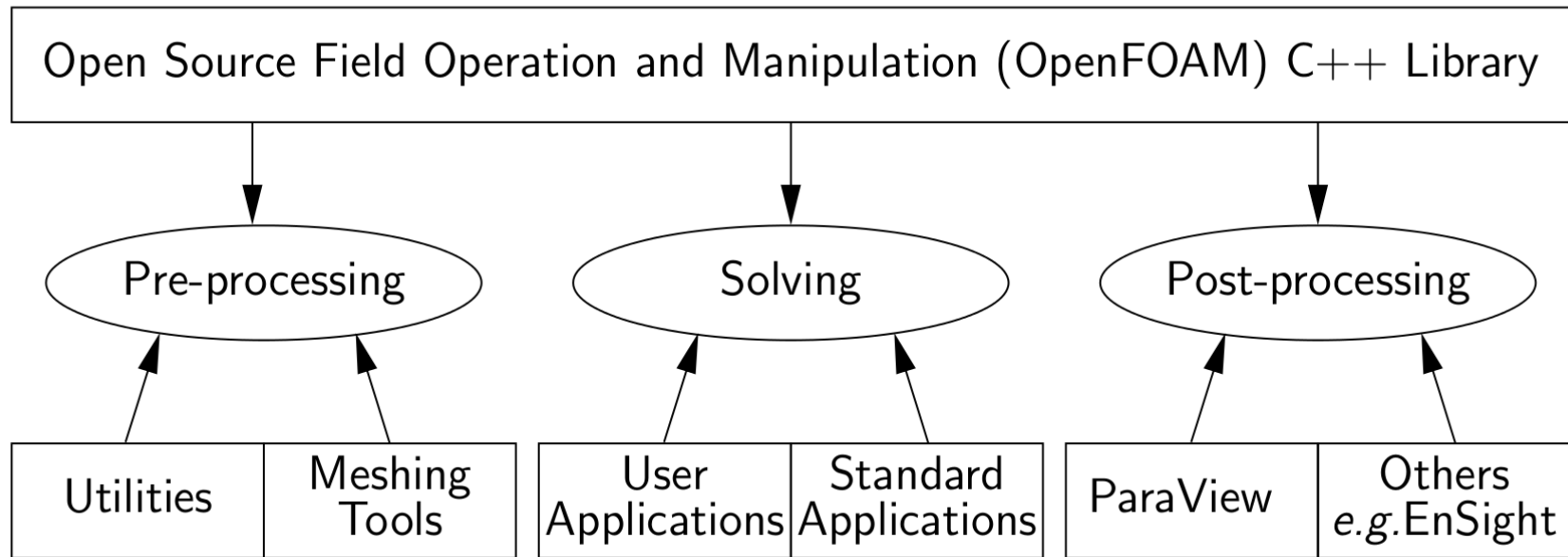
Budapest University of Technology and Economics

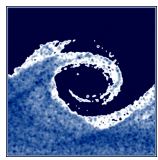


M Ű E G Y E T E M 1 7 8 2



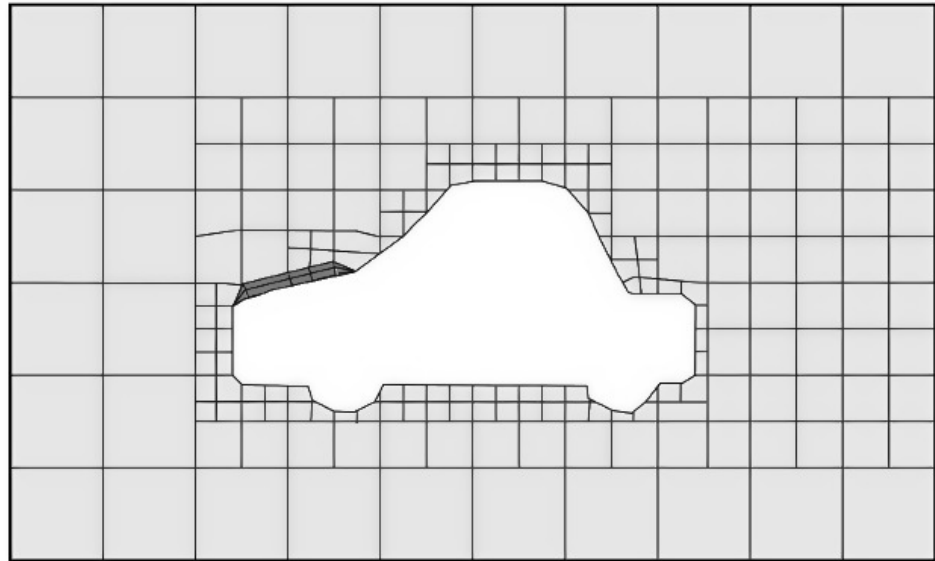
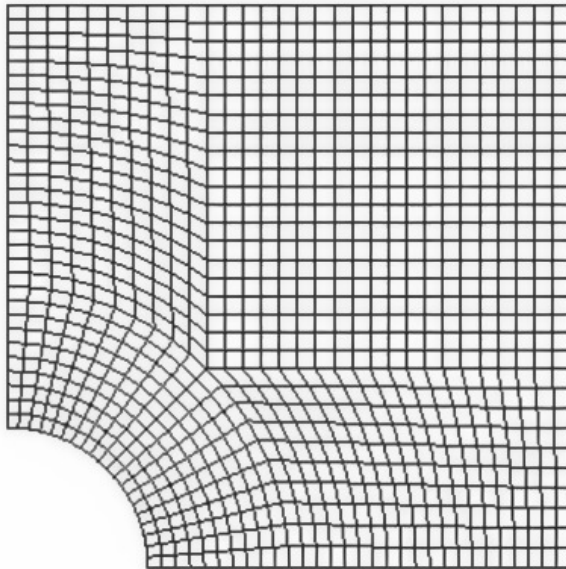
OpenFOAM software components

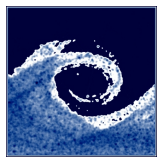




Pre-processing

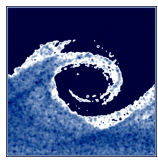
- Utilities
 - mapFields: interpolate fields to new mesh
 - setFields: initialize fields through dictionary
 - changeDictionary: change dictionary entries, e.g. patch type
- Mesh generation
 - blockMesh: multi-block mesh generator
 - snappyHexMesh: automatic hex mesher, snapping to surface





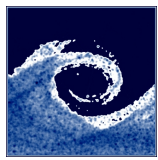
Pre-processing

- 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 bandwidth



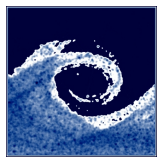
Solving

- 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



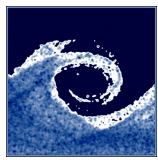
Solving

- 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
 - solidDisplacementFoam: 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] : m²/s
 - [1 -1 -2 0 0 0 0] : kg/m/s² = Pa