Introduction to OpenFOAM

Basic course

Content

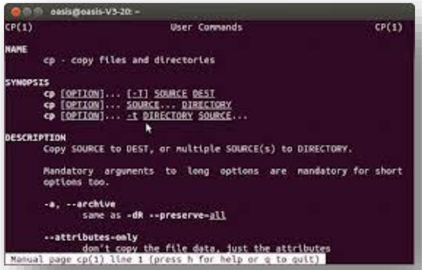
- What is OpenFOAM
- Challenges & gains
- Capabilities
- Workflow
- File structure

What is OpenFOAM

- OpenFOAM = Open Field Operation And Manipulation
- Open: open source → free to use, open to customize, …
- Field: can be many things, not just CFD:
  - Mechanical stress
  - Discrete particles
  - Electromagnetics
  - …

OpenFOAM

Challenges & gains

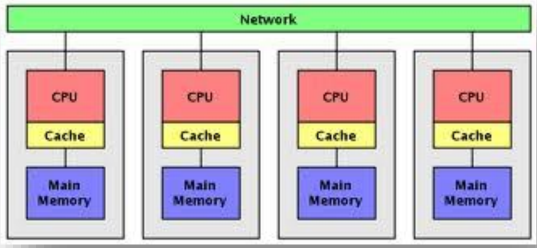


- Biggest challenges:
  - Steep learning curve
  - No GUI (graphical user interface)
  - Command line input (“terminal” window)
  - No link to CAD packages. STL files for meshing.
  - No standard workflow from CAD to result
  - Not all combinations of solvers are readily available (e.g. multiphase flow + chemical reaction) → customization required

www.airshaper.com

Challenges & gains

- Biggest gains:
  - Open source: free, customization, …
  - Massive user base (e.g. www.cfd-online.be)
  - Compatible with powerful post-processing tools (e.g. Paraview)
  - Very advanced solver capabilities (2-phase flows, combustion, multiple RANS models, LES, …)
  - Parallel computing



Capabilities

Solver Capabilities

- Incompressible flows
- Multiphase flows
- Combustion
- Buoyancy-driven flows
- Conjugate heat transfer
- Compressible flows
- Particle methods (DEM, DSMC, MD)
- Other (Solid dynamics, electromagnetics)

Code Customisation

- Creating solvers in OpenFOAM
- Extending library functionality

Meshing Tools

- Mesh generation in OpenFOAM
- Converting meshes into OpenFOAM format
- Tools to manipulate meshes

Library Functionality

- Turbulence models
- Transport/rheology models
- Thermophysical models
- Lagrangian particle tracking
- Reaction kinetics / chemistry

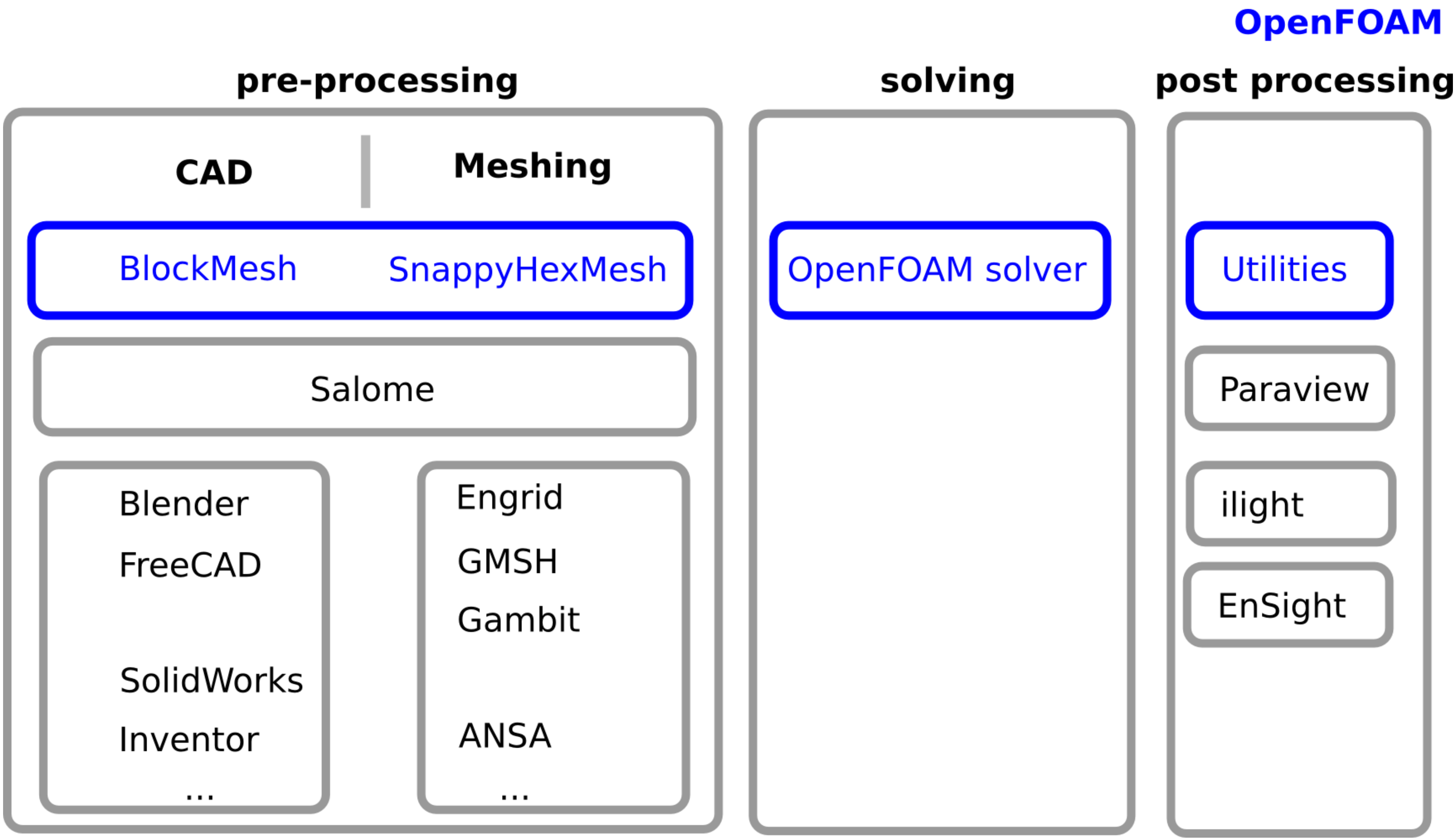
Post-processing

- ParaView and VTK post-processing
- Run-time post-processing
- Third-party post-processing

Core Technology

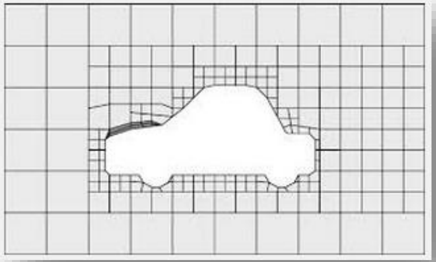
- Numerical method
- Linear system solvers
- ODE system solvers
- Parallel computing
- Dynamic mesh

Workflow


OpenFOAM
pre-processing
CAD
Meshing
BlockMesh
SnappyHexMesh
Salome
Blender
FreeCAD
SolidWorks
Inventor
...
Engrid
GMSH
Gambit
ANSA
...
solving
OpenFOAM solver
post processing
Utilities
Paraview
ilight
EnSight

Workflow

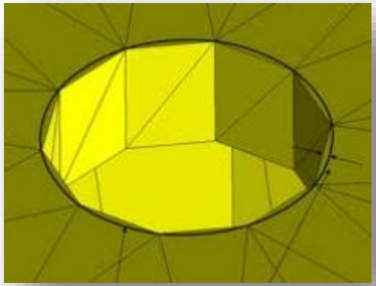
- Pre-processing:
  - CAD geometry:
    - OpenFOAM:
      - only blockMesh for very basic shapes
    - CAD packages:
      - Without internal mesher: export geometry as STL, IGES, STEP, …
      - With internal mesher: export mesh directly
  - Meshing:
    - OpenFOAM – snappyHexMesh
      - Very good mesh quality
      - Excellent compatibility
      - Based on STL (sadly no IGES/STEP support)



    - Meshers
      - Import STL, IGES, STEP, … depending on the package
      - Generate & export the mesh
      - Use openFOAM translation tools to convert the mesh

Workflow

- Pre-processing:
  – Challenges:
    - No decent open source CAD package available
    - Meshing in OpenFOAM - snappyHexMesh:
      – Input through STL looses parametrisation
      – Logical surfaces are split into smaller ones
      – Patch definition is diffcult
    - Meshing in 3rd party package:
      – Not always free
      – Different element types (tet, hex, …)
      – Sometimes good coupling with CAD models
      – Patch definition requires attention

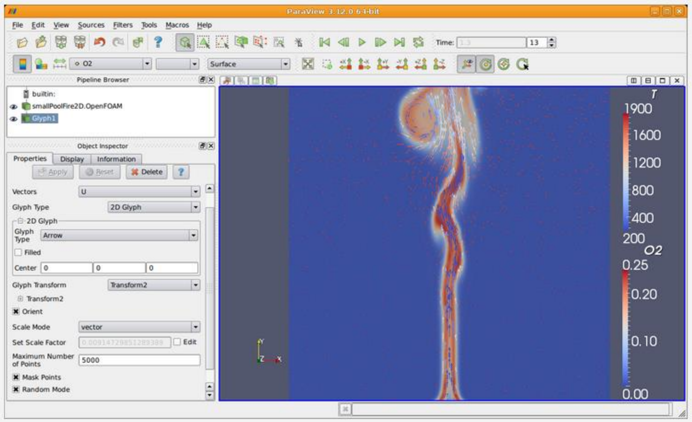


Workflow

- Solving:
  - This is the core function of openFOAM
  - Different solvers for different problems
  - Parallel computating: on multiple cores, CPU's and even PC's
  - Real-time monitoring of convergens, forces, …
  - Results: folders per time step containing field data (pressure, velocity, …) for each cell

Workflow

- Post-processing:
  - Visualisation of the field data
    - Cut-plots
    - Streamlines
    - Isosurfaces
    - …
  - Calculation of extra parameters
    - Relative velocities
    - Forces (not always easy)
    - …
  - Performed in ParaView: excellent open source tool



www.airshaper.com

File structure

- "system" directory:
  - controlDict: contains the run control parameters (start time, time step, data output, …)
  - fvSchemes: discretisation parameters
  - fvSolution: parameters for equation solver, tolerances, …