OpenFOAM – Opensource and CFD

Andrew King

Department of Mechanical Engineering
Curtin University

Outline

- What is Opensource Software
- OpenFOAM – Overview
  - Utilities, Libraries and Solvers
  - Data Formats
- The CFD ‘toolchain’
  - Solid Modelling (Geometry)
  - Meshing
  - Solution
  - Postprocessing

Outline

- Examples
  - Lid driven Cavity
  - Buoyancy driven flow
  - Free Surface Flows
  - Dynamic Meshing
  - Fluid Structure Interaction

Opensource – what is it?

Opensource Software

- has source code freely available.
- permits users to study, change, and improve the code.
- allows redistribution in modified or unmodified form.
- promotes development in a public, collaborative manner.

Examples

- Apache (web server) – runs ~60% of the internet
- Linux – Operating System based on UNIX.
- Firefox – Standards Compliant Web Browser
- ...many more ....

And of course

- Open∇FOAM – Open source CFD

OpenFOAM - Overview

- OpenFOAM stands for:
  '**Open** **F**ield **O**perations **A**nd **M**anipulation'
- Consists of a library of efficient CFD related C++ modules.
- These can be combined together to create
  - ***solvers***
  - ***utilities*** (for example pre/post-processing, mesh checking, manipulation, conversion, etc.)
- Or, additional functionality can be introduced through new libraries or modules

OpenFOAM - Overview

- So far modules, libraries and applications are available for the following tasks

SOLVERS

'Basic' CFD
Incompressible flows
Compressible flows
Multiphase flows
DNS and LES
Combustion
Heat transfer
Electromagnetics
Solid dynamics
Finance

UTILITIES

Pre-processing
The FoamX case manager
Other pre-processing utilities

Post-processing
The paraFoam post-processor
Third-party post-processing
Other post-processing utilities

Mesh processing
Mesh generation
Mesh converters
Mesh manipulation

LIBRARIES

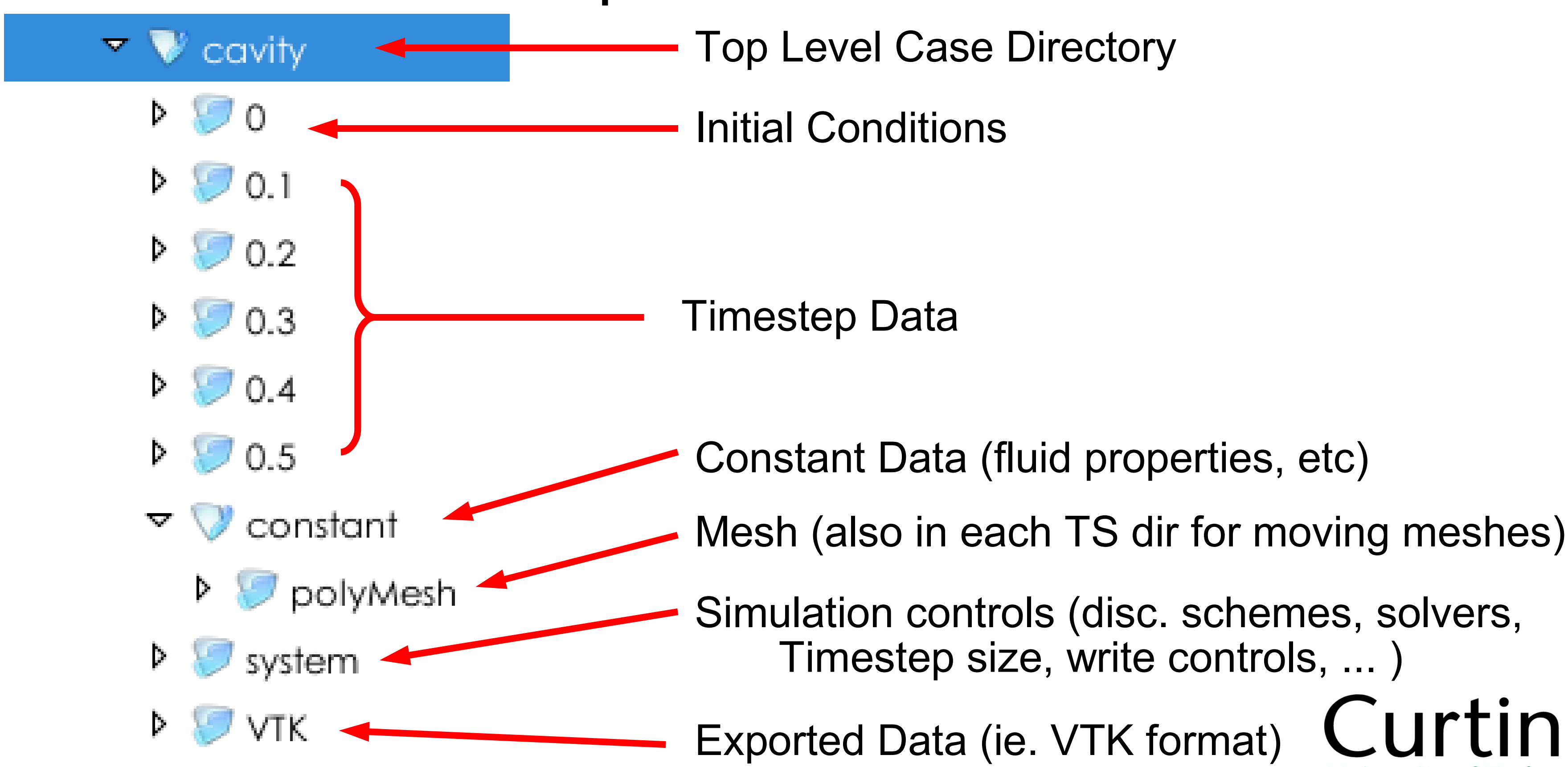
Model libraries
Turbulence
Large-eddy simulation (LES)
Transport models
Thermophysical models
Lagrangian particle tracking
Chemical kinetics

Other features
Linear system solvers
ODE system solvers
Parallel computing
Mesh motion
Numerical method

- More details of solvers later....

OpenFOAM - Overview

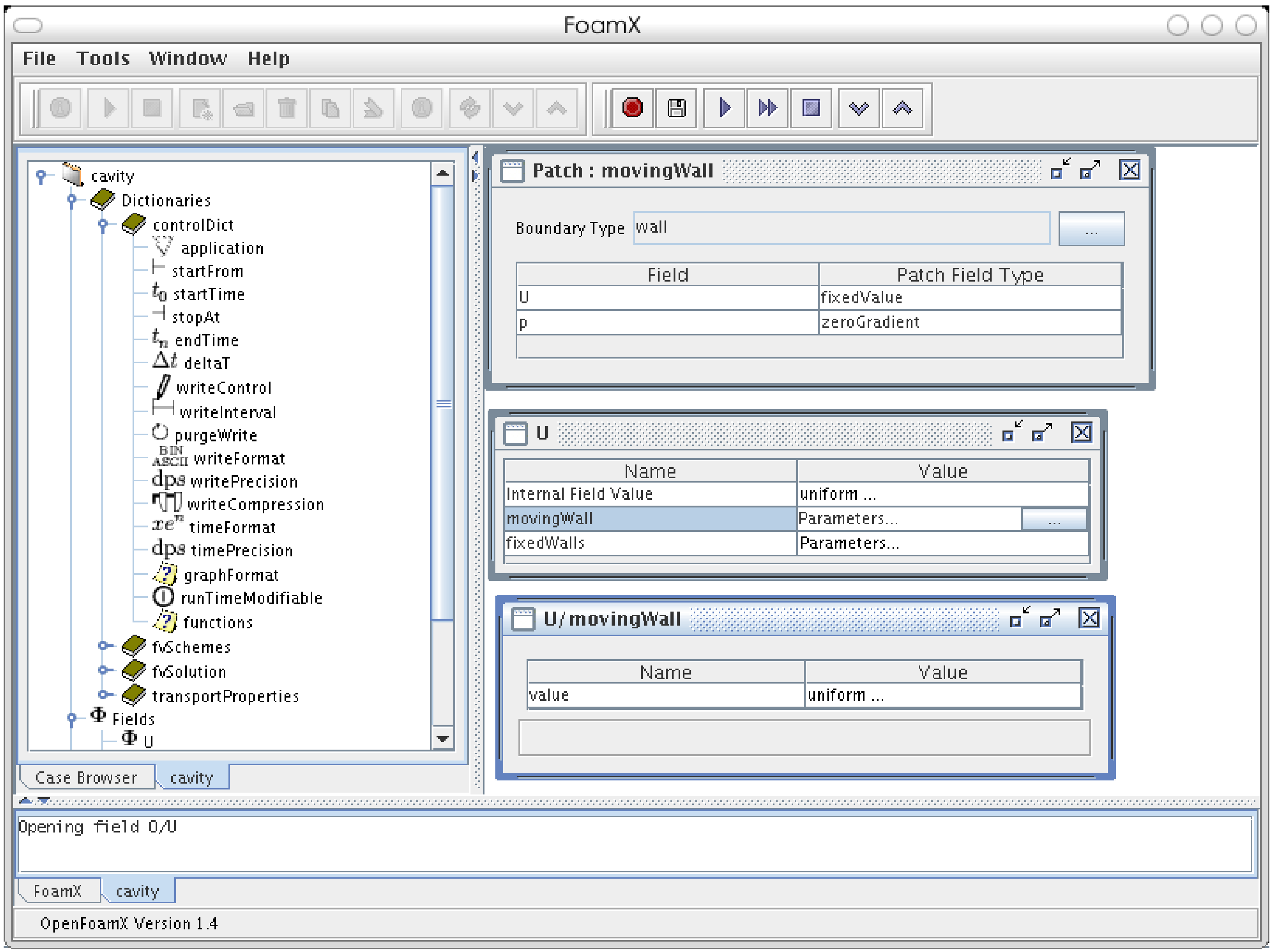
- Data Formats
  - Open foam cases consist of a directory, and subdirectories with specific tasks.



Curtin University of Technology

OpenFOAM - Overview

- A GUI (FoamX) allows easy access to the files, to control all facets, and run simulations.



FoamX GUI

OpenFOAM - Overview

- Alternatively direct access to files is possible
  - Boundary and initial conditions are set in '**0**' directory.

```
    class           volVectorField;
    object          U;

// * * * * * * * * * * * * * * * * * * * * * * * * *

dimensions      [0 1 -1 0 0 0 0];

internalField   uniform (0 0 0);

boundaryField
{
    movingWall
    {
        type            fixedValue;
        value           uniform (1 0 0);
    }

    fixedWalls
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }
```

file **`'.../cavity/0/U'`** – Velocities

Note that OpenFOAM keeps track of dimensions and type (ie scalar, vector, tensor) and checks these when compiling a solver.

Initial internal field values

boundary type and value (where required)

OpenFOAM - Overview

```
    class           volScalarField;
    object          p;
}

// * * * * * * * * * * * * * * * * * * * * * * * * *


dimensions      [0 2 -2 0 0 0 0];

internalField   uniform 0;

boundaryField
{
    movingWall
    {
        type            zeroGradient;
    }

    fixedWalls
    {
        type            zeroGradient;
    }
```

file **'.../cavity/0/p'** – Pressures

pressure is a scalar

dimensions (ie $m^2/s^2 = N/m^2$)

Initial internal field values

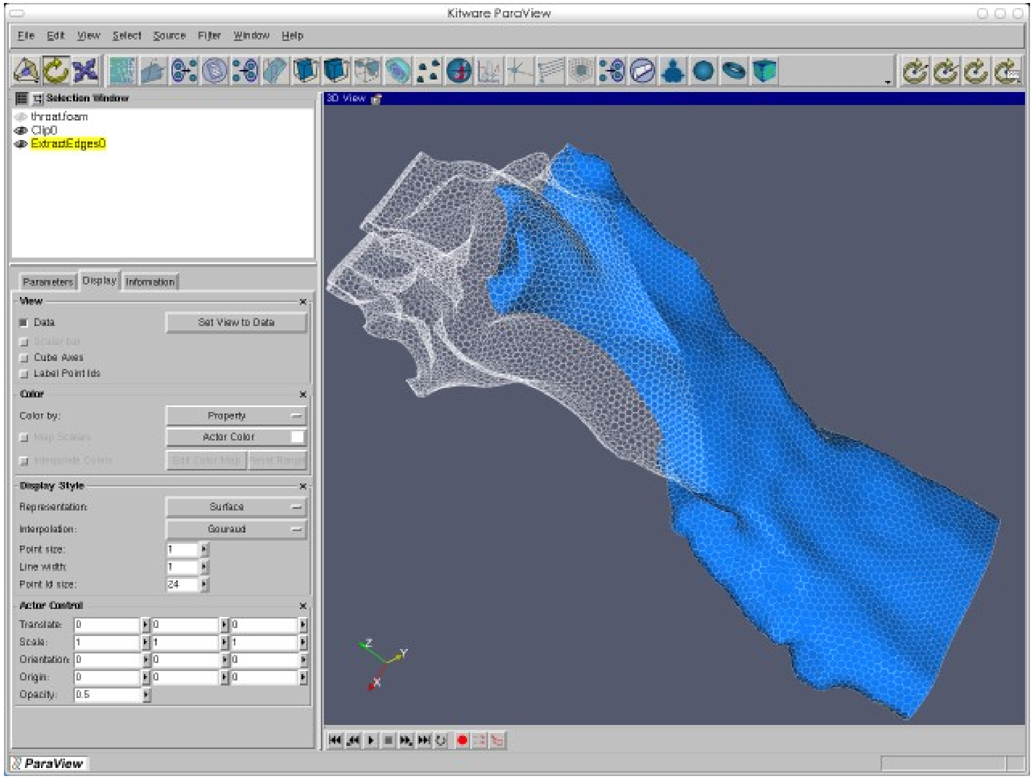
boundary type (zero normal gradient, no value req'd)

OpenFOAM - Overview

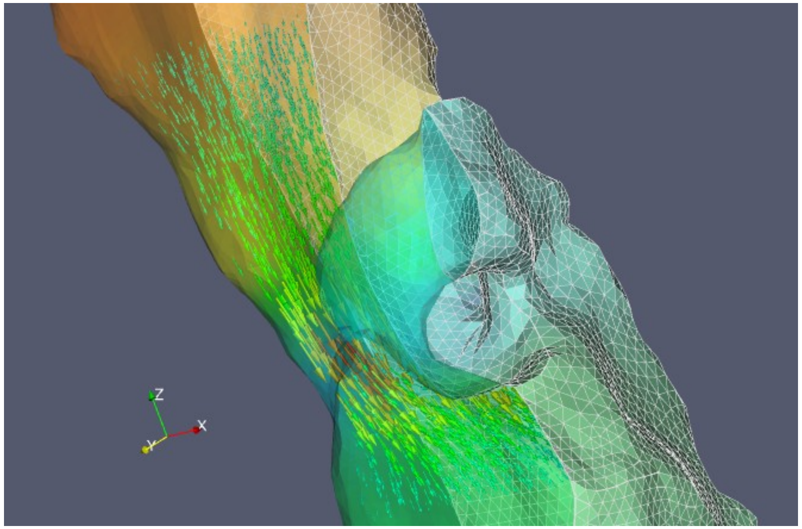
- Postprocessing
  - Quantitative data available using utility applications, ie.
    - *`sampleSurface`*
    - *`patchAverage`*
    - *`patchIntegrate`*
    - *`wallHeatFlux`*
    - *`checkYPlus`*
    - *`wallGradU`*
    - etc...
  - Or from `paraFoam` graphical utility

OpenFOAM - Overview

- ParaFoam



ParaFoam imports fields in general formats and filters are applied to construct contours, graphs, etc.



Grid visualisation (above) and overlay of pressure contours, velocity vectors and grid (right)

Opensource CFD ‘Toolchain’

- OpenFOAM provides the solver (and a bit more), however the CFD tool chain is more than this.
- It is possible to use an entirely OSS toolchain, or a combination of commercial and OSS tools.

Geometry

↓

Mesh

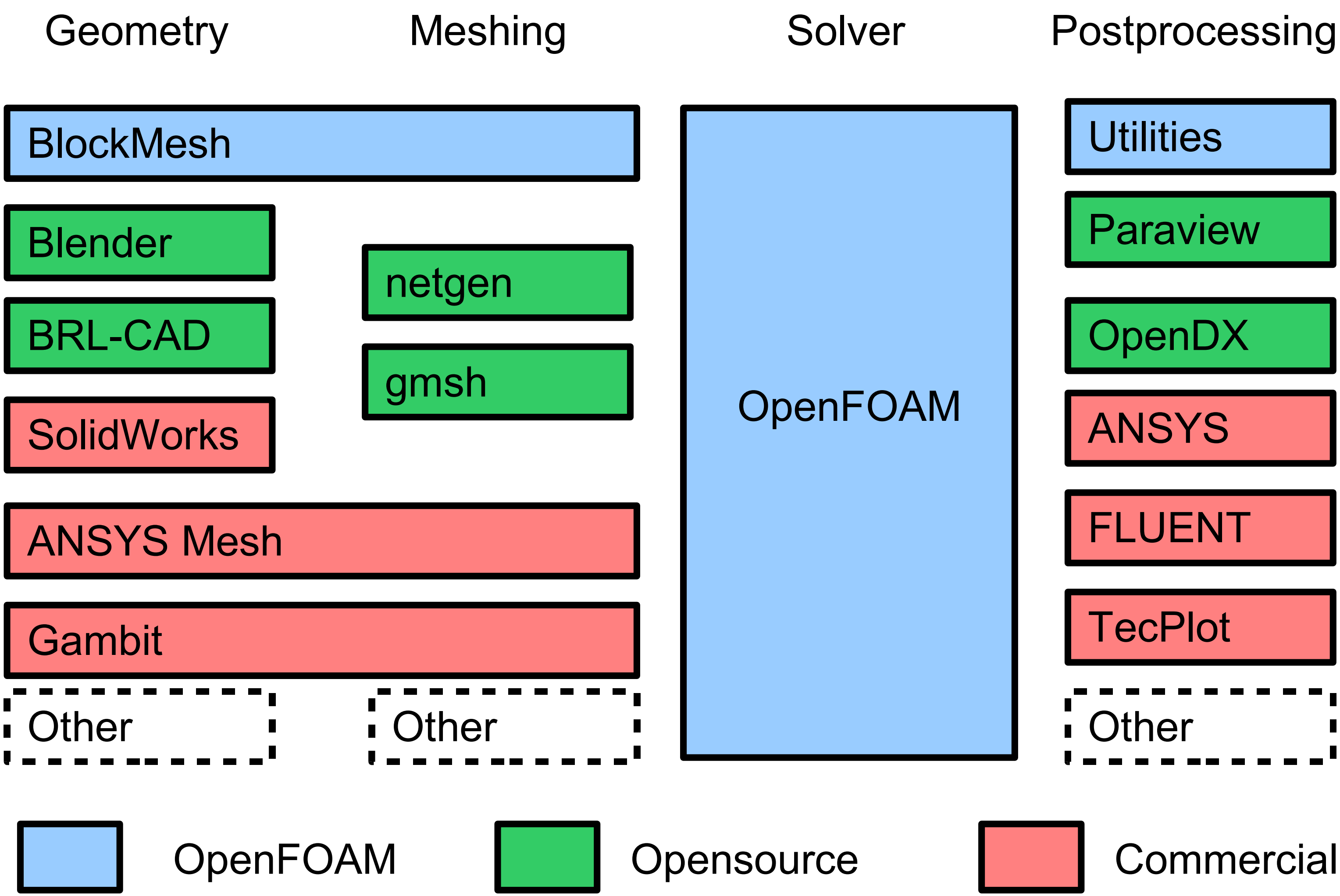
↓

Solver

↓

Postprocessing

Opensource CFD ‘Toolchain’



Customisation

- Due to opensource code, customisation is easy.
- for solvers, a typical flow equation can be descibed as follows. (ie momentum)

```
fvVectorMatrix UEqn
(
        fvm::ddt(U)              Unsteady term
      + fvm::div(phi, U)         Momentum term
      - fvm::laplacian(nu, U)    Diffusion term
);

solve(UEqn == -fvc::grad(p));    Pressure gradient
```

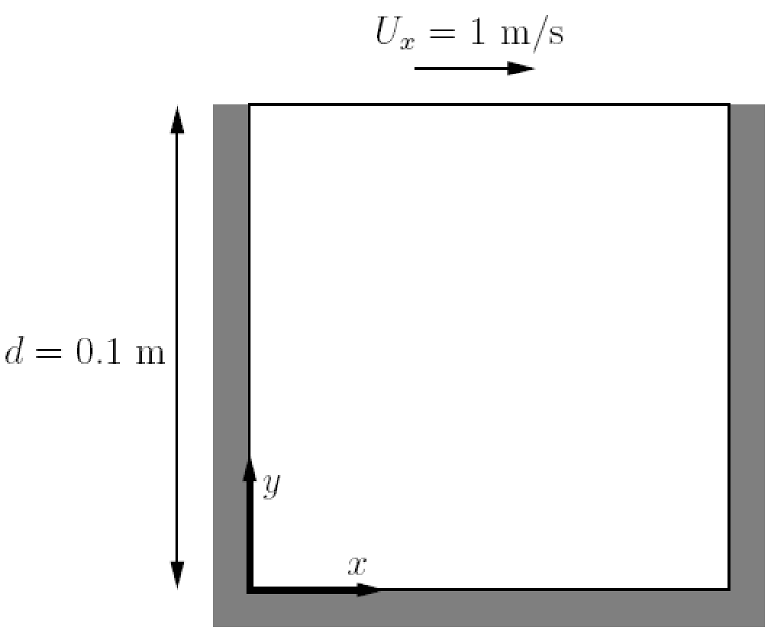
New solvers can be created by modifying existing solvers, for example to add scalar transport, equation is similar to above.

OpenFOAM - Overview

- In combination, the above utilities and solvers are at least as powerful as the commercial offerings. (Fluent, ANSYS CFX, StarCD, etc) – though some initial investment in learning is required.

OpenFOAM - Examples

- Lid driven cavity



Standard test case, bottom and side walls are no-slip and stationary, top wall is driven at 1 m/s.

Flow is assumed incompressible, isothermal and laminar.

Solution obtained using the standard `icoFoam` solver

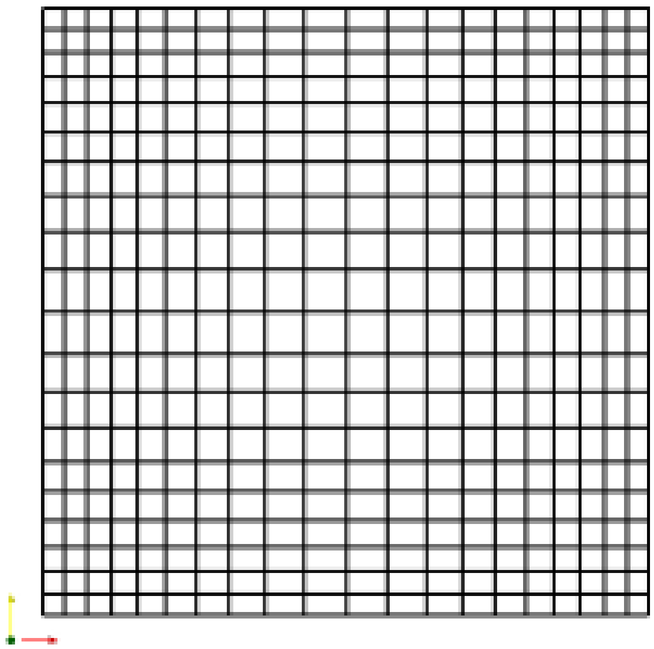
OpenFOAM - Examples

- Mesh generation

Mesh for this problem is created using the *blockMesh* utility

Geometry, grid spacing and labels for boundaries (patches) are all specified in a *blockMeshDict*.

Running the utility gives a mesh as shown.



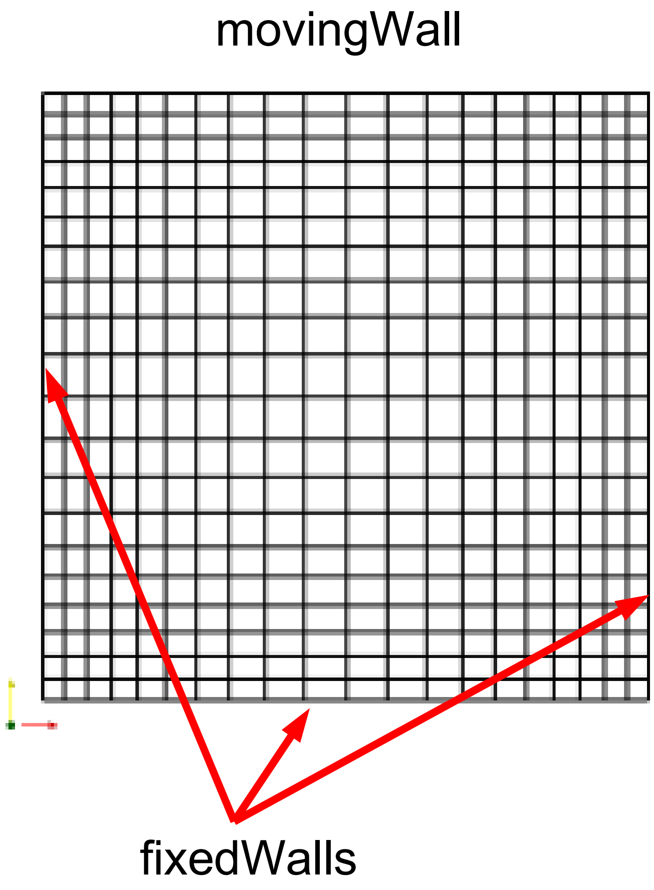
OpenFOAM - Examples

- Boundary Conditions

Set in 0/U

fixedWalls
    type fixedValue
    value uniform (0 0 0)
movingWall
    type fixedValue
    value uniform (1 0 0)

Set in 0/p
fixedWalls
    type zeroGradient
movingWall
    type zeroGradient



OpenFOAM - Examples

- Solution run using icoFoam
- Typical Output (at each timestep)

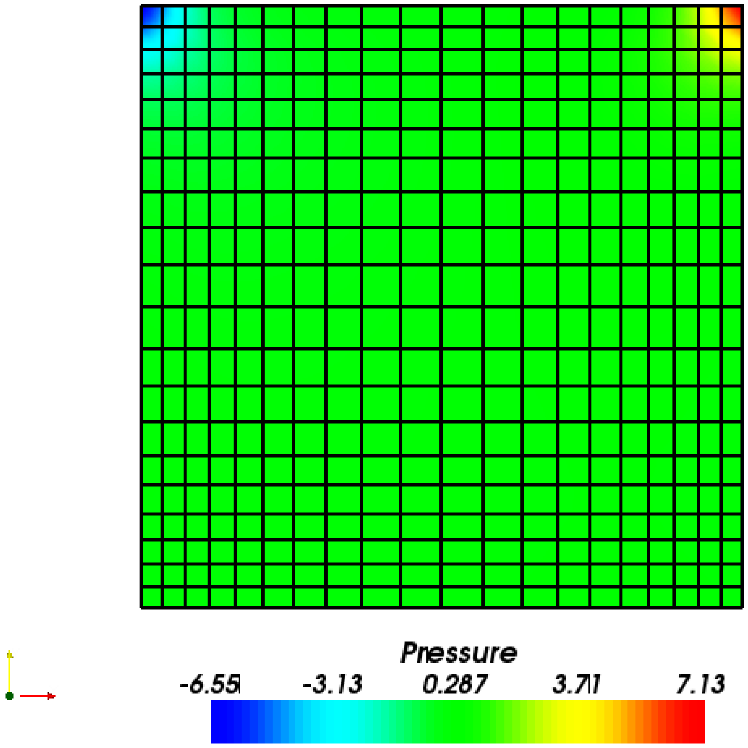
```
Time = 0.8

Mean and max Courant Numbers = 0.0500943 0.42312
BICCG:  Solving for Ux, Initial residual = 8.9862e-09,
    Final residual = 8.9862e-09, No Iterations 0
BICCG:  Solving for Uy, Initial residual = 2.28622e-08,
    Final residual = 2.28622e-08, No Iterations 0
ICCG:  Solving for p, Initial residual = 1.02719e-06,
    Final residual = 1.61421e-07, No Iterations 1
time step continuity errors : sum local = 1.01957e-09,
    global = -1.23457e-19, cumulative = -1.32236e-18
ICCG:  Solving for p, Initial residual = 3.48287e-07,
    Final residual = 3.48287e-07, No Iterations 0
time step continuity errors : sum local = 1.62239e-09,
    global = -5.22778e-20, cumulative = -1.37464e-18
ExecutionTime = 1.73 s  ClockTime = 3 s

End...
```

OpenFOAM - Examples

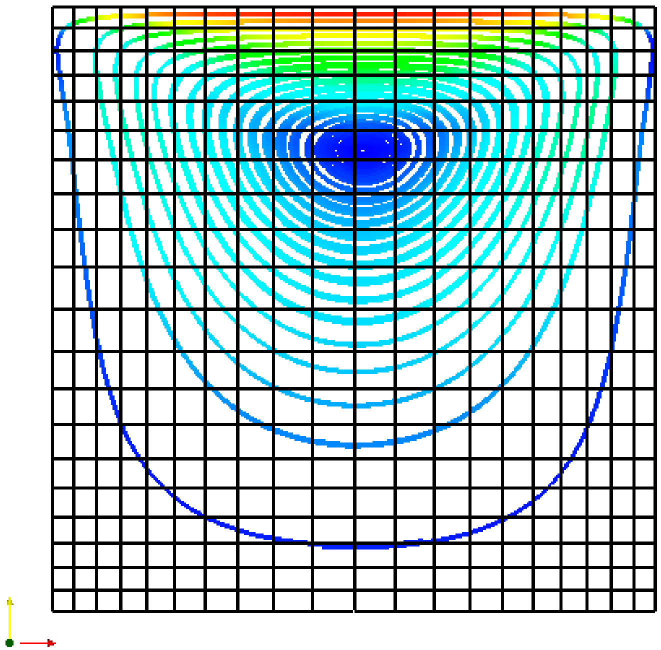
- Results

From *paraview* graphical results can be obtained.

Pressure

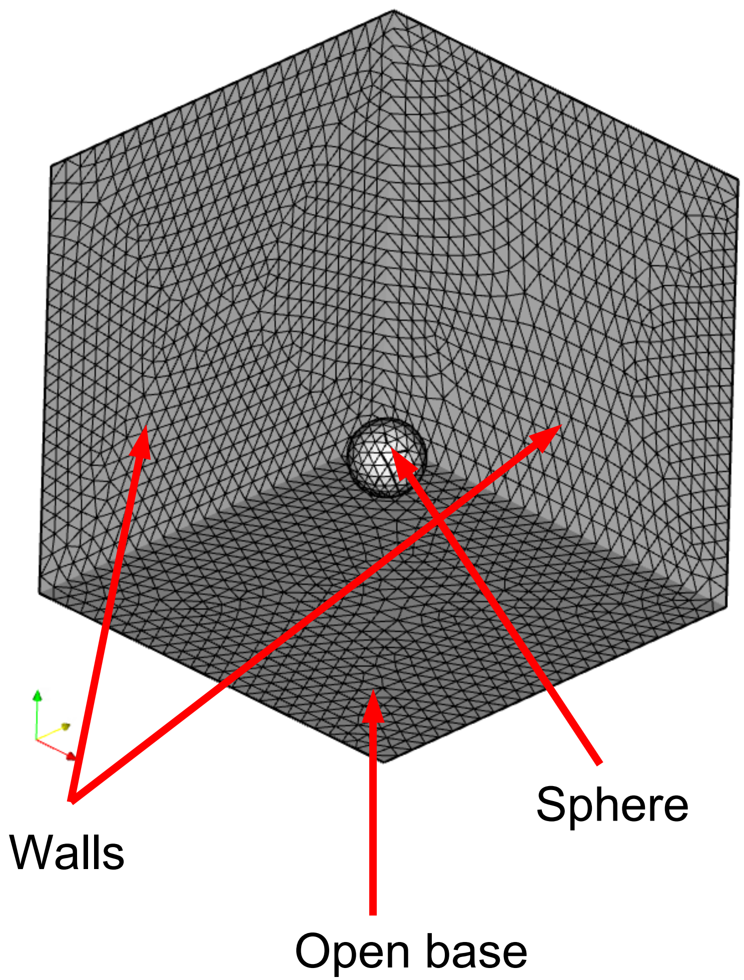


Stream Lines



OpenFOAM - Examples

- Buoyancy driven flow



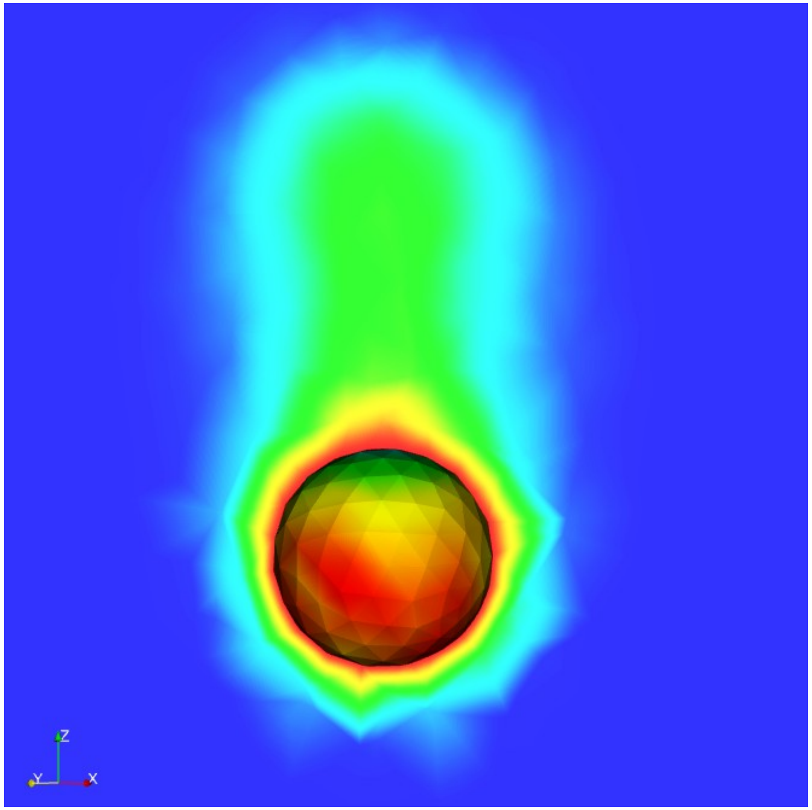
Isothermal sphere, in walled enclose open at top and bottom

Flow is assumed incompressible and laminar with boussinesq assumption for buoyant forces.

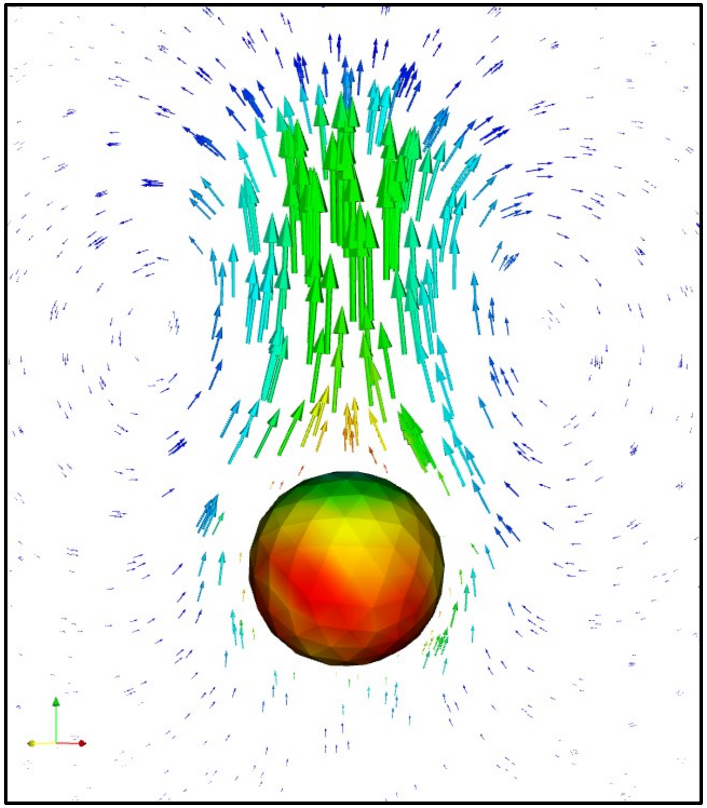
Solution obtained using contributed `boussinesqBuoyantFoam` solver

OpenFOAM - Examples

- Buoyancy driven flow



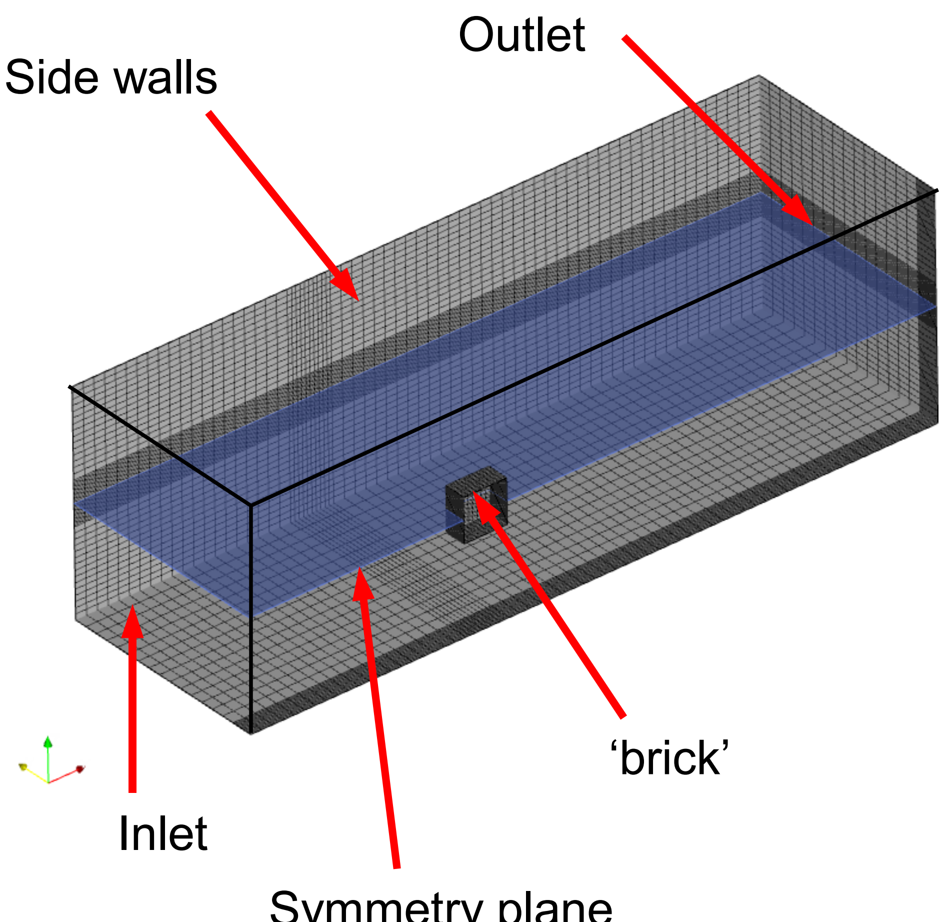
Temperature Field after 0.3s, sphere coloured by wall heat flux



Velocity Vector Field

OpenFOAM - Examples

- Free-surface flow



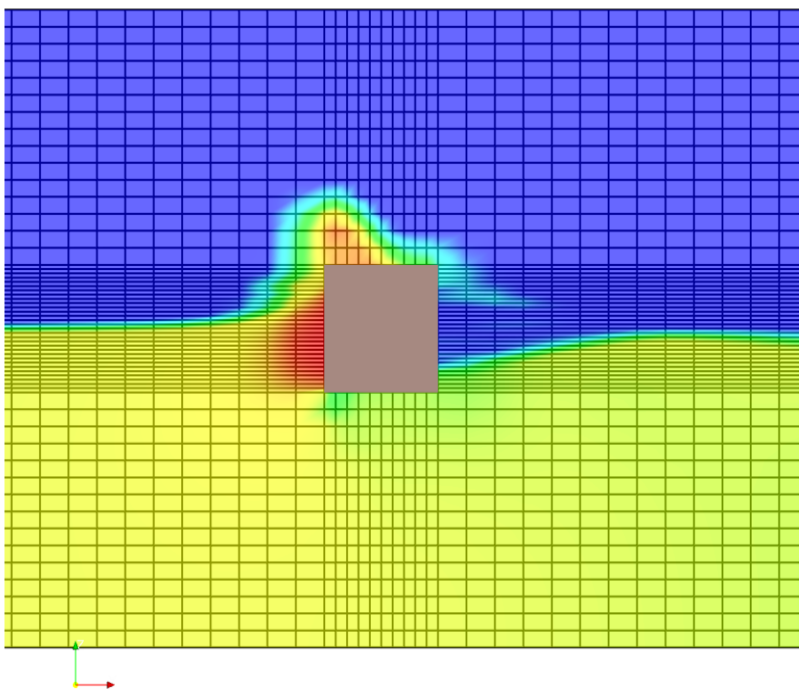
solid cube, 50% suspended in free-stream channel flow.

Flow is assumed incompressible and laminar with VOF tracking of interface.

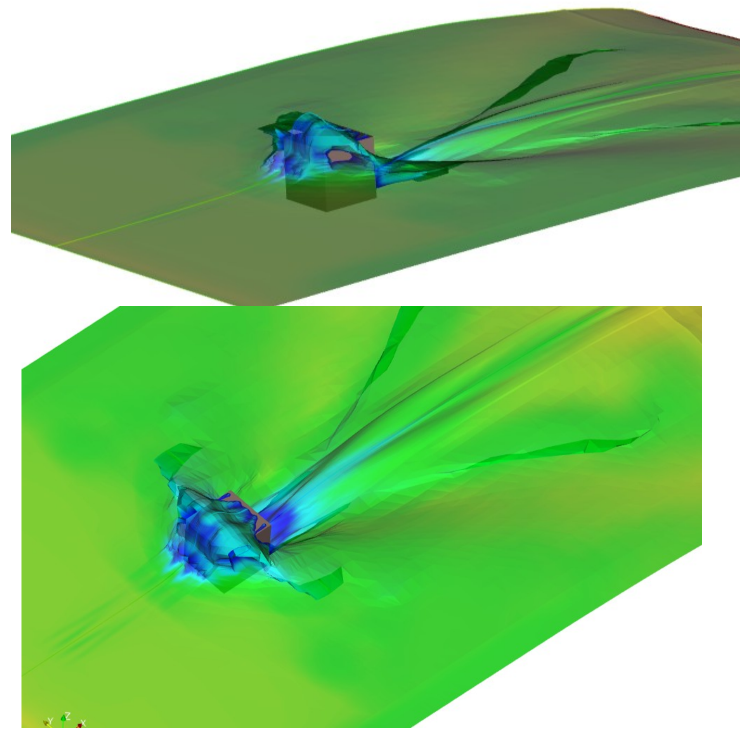
Solution obtained using the standard `interFoam` solver

OpenFOAM - Examples

- Free-surface flow



Pressure contours at brick centreline

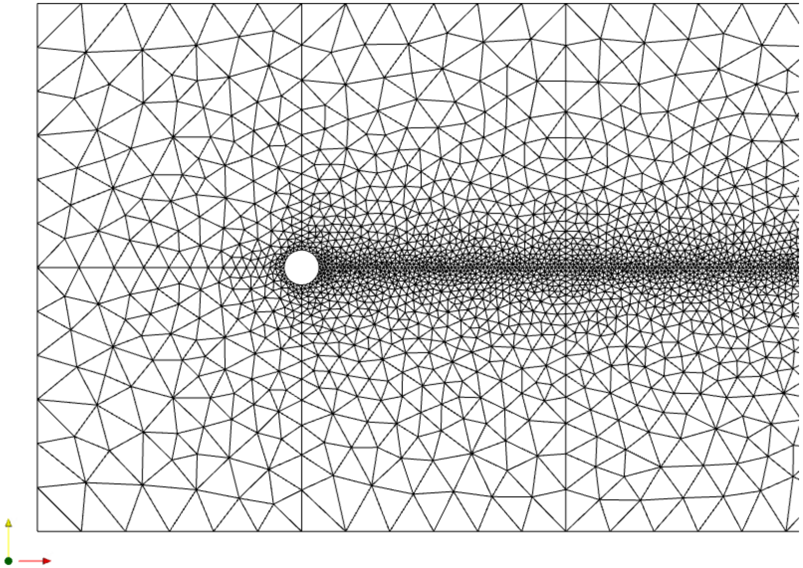


Free surface, coloured by velocity magnitude

Curtin University of Technology

OpenFOAM - Examples

- Dynamic Moving Mesh



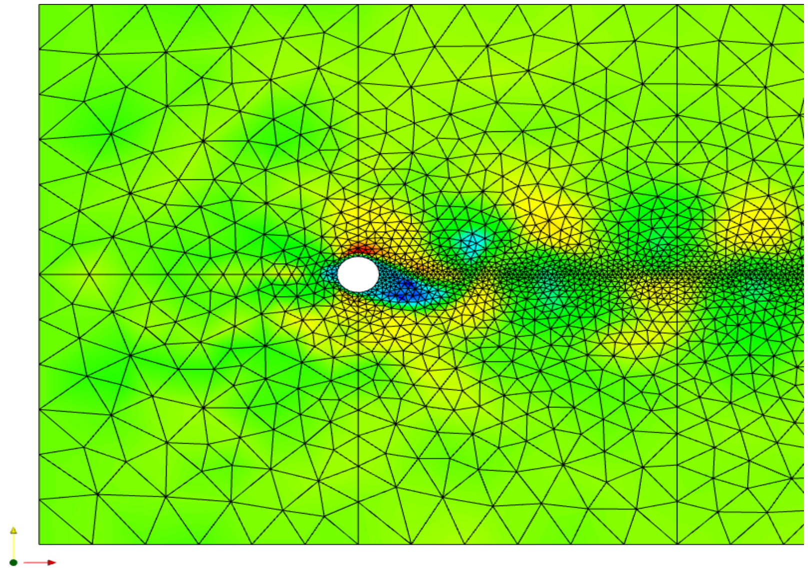
Cylinder in free stream flow

Flow is assumed incompressible and laminar
Re ~ 100

Solution obtained using the standard *icoDyMFoam* solver

OpenFOAM - Examples

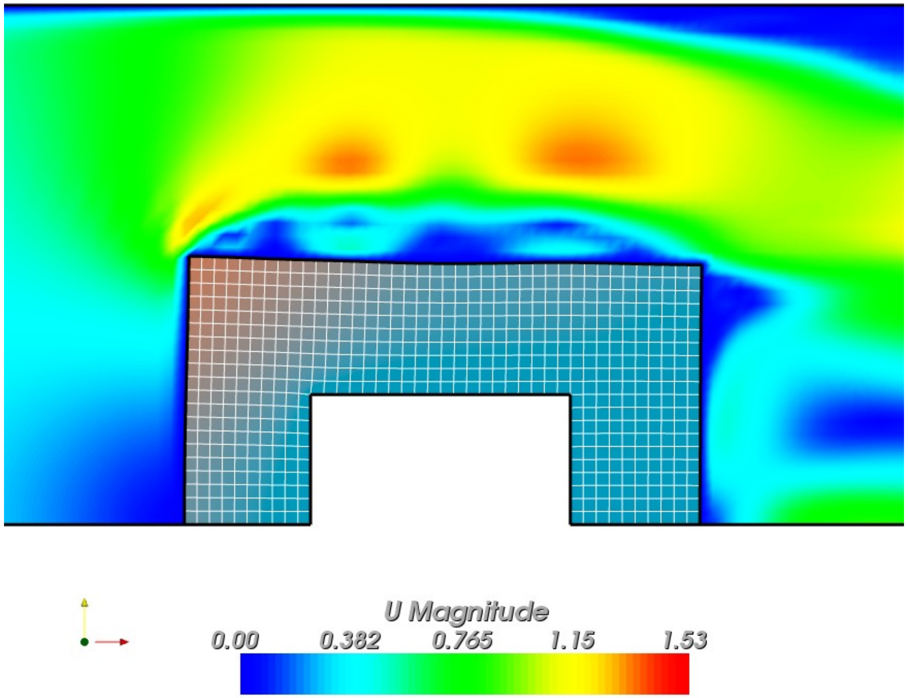
- Dynamic Moving Mesh



In the solution, periodic shedding of vortices in the wake can be observed.

OpenFOAM - Examples

- Fluid Structure Interaction



Channel flow, obstructed by solid block, covered with flexible casing.

Fluid stucture interaction solution obtained using contributed `icoStructFoam` solver