Hands-On Training with OpenFOAM

Flow Around a 2-D Airfoil

Hrvoje Jasak

`hrvoje.jasak@fsb.hr`

Faculty of Mechanical Engineering and Naval Architecture

University of Zagreb, Croatia

Outline

Summary of Objectives: Basic Code and Case Structure

- Basic structure of the OpenFOAM case and CFD runs

Tutorial Steps

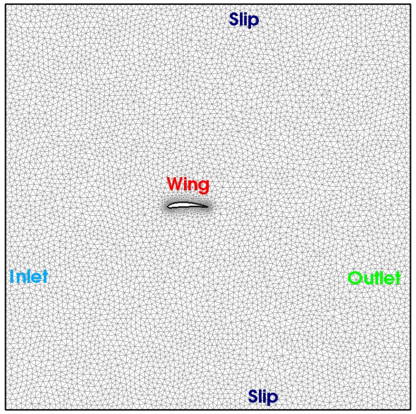
1. Basic review of case organisation
2. Convert the mesh generated in Pointwise
3. `constant/polyMesh/boundary`: set wall patch type
4. Set material properties: viscosity; turbulence model
5. 0 directory: set initial and boundary conditions for flow fields
6. `checkMesh`: analysis of mesh quality
7. Flow solver: `sonicFoam`
8. Basic post-processing with FieldView
9. Utilities and data manipulation: `Mach` number, `forceCoeffs` function object
10. Further post-processing
11. Basic review of solver and discretisation parameters

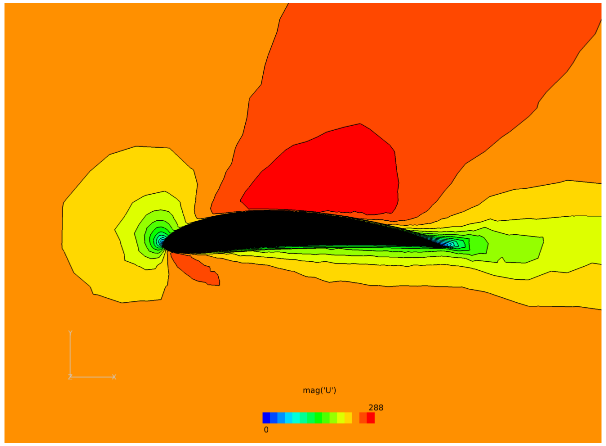
Case Setup

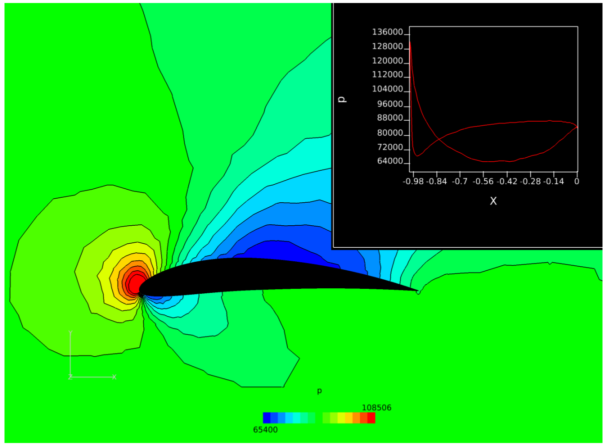
- Transient compressible turbulent flow simulation
- Material properties: air as ideal gas with constant properties
  `(name, nMoles, molWeight, Cv, Hf, mu, 1/Pr)`
- Inlet/far field conditions:

$$\mathbf{u} = (242.43 \quad 0 \quad 0)\,\mathrm{m/s} \quad p_{inf} = 85419\,\mathrm{Pa} \quad T = 260\,\mathrm{K}$$

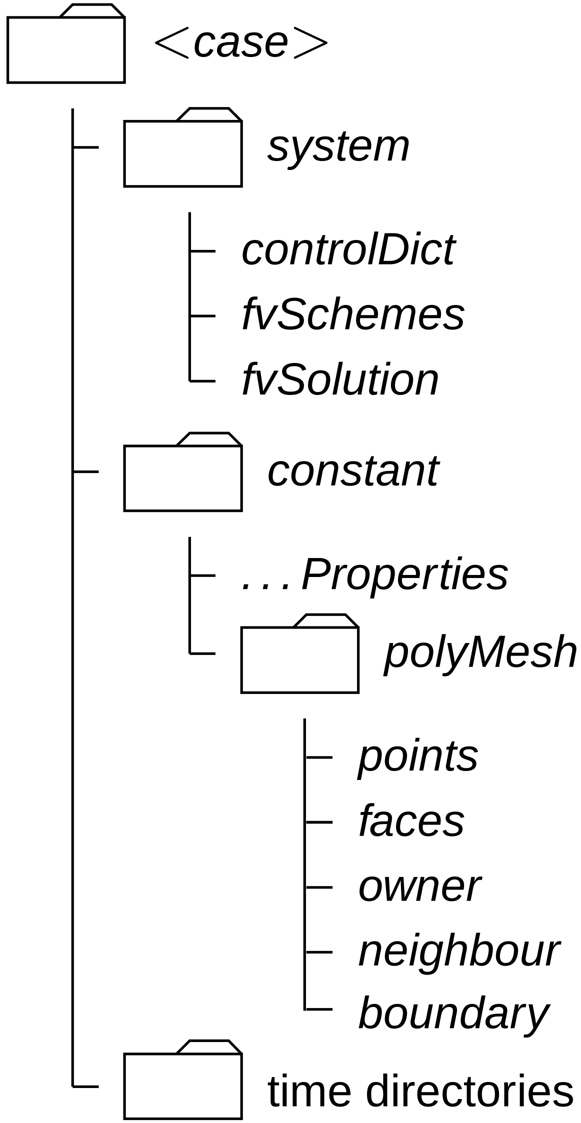
$$k = 220.4\,\mathrm{m}^2/\mathrm{s}^2 \quad \epsilon = 2688.26\,\mathrm{m}^2/\mathrm{s}^3$$







OpenFOAM Case Structure



Data Organisation and Management

- Unlike “standard CFD practice”, in OpenFOAM `case` is a **directory**: each self-contained piece of heavy-weight data stored in its own file
- Light-weight data is presented in **dictionary form**: keyword-value pairs in free format. It can be changed and re-read during the run: solution steering
- Mesh data split into components for efficient management of moving mesh cases
- Time directories contain solution and derived fields (one per file)
- Support for compressed I/O: more efficient I/O and less disk space

OpenFOAM Simulation: Data I/O

Data Input and Output

- Dictionary format: file header (IOobject) and keyword-value entry pairs

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      transportProperties;
}

// Diffusivity
DT           DT [0 2 -1 0 0 0 0] 0.01;
```

- Contents of dictionaries depends on their role
  - Material properties and physical model constants
  - Solution fields, initial and boundary conditions
  - Discretisation settings, solver controls I/O parameters etc.

OpenFOAM Simulation: Controls

Basic Controls: `controlDict`

- Basic controls of run start, end and write frequency

```
startFrom       startTime;  // latestTime // firstTime
startTime       0;

stopAt          endTime;    // writeNow // nextWrite
endTime         2500;

deltaT          1;

writeControl    timeStep;   // runTime // clockTime // cpuTime
writeInterval   50;

writeFormat     ascii;      // binary
writePrecision  6;
writeCompression uncompressed; // compressed

timeFormat      general;    // fixed // scientific
timePrecision   6;

runTimeModifiable yes;
```

OpenFOAM Simulation: Schemes

Basic Controls: `fvSchemes`

- Equation discretisation controls: per-term basis

```
ddtSchemes
{
    default steadyState;
}
gradSchemes
{
    default          cellLimited leastSquares 1.0;
//     grad(p)          Gauss linear;
}
divSchemes
{
    default          none;
    div(phi,U)       Gauss linearUpwindV Gauss linear;
    div(phi,k)       Gauss upwind;
    div(phi,omega)   Gauss upwind;
    div((nuEff*dev(grad(U).T()))) Gauss linear;
}
```

OpenFOAM Simulation: Schemes

Basic Controls: `fvSchemes`

- Equation discretisation controls, Cont'd

```
laplacianSchemes
{
    default          Gauss linear limited 0.5;
}
interpolationSchemes
{
    default          linear;
    interpolate(U)   linear;
}
snGradSchemes
{
    default          limited 0.5;
}
fluxRequired
{
    default          no;
    p;
}
```

OpenFOAM Simulation: Solution

Basic Controls: `fvSolution`

- Linear equation solver settings: per equation in top-level solver

```
solvers
{
    p
    {
        solver          PCG;
        preconditioner  DIC;

        tolerance       1e-8;
        relTol          0.01;
    }
    U
    {
        solver          PBiCG;
        preconditioner  DILU;

        tolerance       1e-07;
        relTol          0;
    }
}
```

OpenFOAM Simulation: Solution

Basic Controls: `fvSolution`

- Global algorithmic settings and under-relaxation factors

```
PISO
{
    momentumPredictor yes;

    nCorrectors     2;
    nNonOrthogonalCorrectors 0;

    nAlphaCorr      1;
    nAlphaSubCycles 2;
    cAlpha          1;
}
relaxationFactors
{
    p               0.3;
    U               0.7;
    k               0.7;
    omega           0.7;
}
```

OpenFOAM Simulation: Mesh

Basic Controls: Structure of Mesh Files

- Mesh files at start of simulation located in `constant/polyMesh` directory
- `points`, `faces`: basic lists of primitive entries
- `owner`, `neighbour`: lists of face-to-cell addressing
- Note: **OpenFOAM uses strongly ordered face lists** for efficiency

```
 11M 2011-07-22 11:19 points
 37M 2011-07-22 11:19 faces
4.0M 2011-07-22 11:19 owner
 14M 2011-07-22 11:19 neighbour
1.8K 2011-07-22 11:19 boundary
```

- Additional mesh files: sets and zones, mesh modifiers, parallel mapping etc.

```
 586 2011-03-14 09:43 pointZones
 24K 2011-03-14 09:43 faceZones
868K 2011-03-14 09:43 cellZones
 78K 2011-03-14 09:43 meshModifiers
4.0K 2011-03-14 09:43 sets/
```

OpenFOAM Simulation: Mesh

Basic Controls: Structure of Mesh Files

- Boundary definition: patch types and strong ordering

```
(
    Wing                            <-- patch name
    {
        type            wall;       <-- constrained patch type
        nFaces          154;        <-- number of faces
        startFace       23579;      <-- start face in face list
    }
    FrontAndBack
    {
        type            empty;       <-- constrained patch type
        nFaces          30712;
        startFace       23733;
    }
    Inlet
    {
        type            patch;       <-- (free) patch type
        nFaces          74;
        startFace       54445;
        ...
```

OpenFOAM Simulation: Mesh

Basic Controls: Structure of Mesh Files

- Boundary definition: patch types

```
...
  Slip
  {
      type                patch;
      nFaces              148;
      startFace           54519;
  }
  leftPlane
  {
      type                symmetryPlane;
      nFaces              74;
      startFace           54667;
  }
)
```

OpenFOAM Simulation: Fields

Field: Initial and Boundary Conditions

- Definition of initial and boundary conditions, per-field basis

```
dimensions      [0 1 -1 0 0 0 0]; // [kg m s K mol A Cd]
internalField   uniform (40 0 0);
boundaryField
{
    Body-4
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }
    Inlet-12
    {
        type            fixedValue;
        value           uniform (40 0 0);
    }
    Slip-10
    {
        type            slip;
    }
    ...
```

OpenFOAM Simulation: Fields

Field: Initial and Boundary Conditions

- Definition of initial and boundary conditions, Cont'd
- Fields located in time directories: `0/p`, `0/U`
- Boundary conditions defined on a **per-field basis**
- Note: consistency of boundary conditions related to the physics solver

OpenFOAM Simulation: Utilities

Basic Controls: Utility Controls

- Utility controls based under `system`. Example:

```
numberOfSubdomains 4;

method          metis;

globalFaceZones ( insideZone outsideZone );

simpleCoeffs
{
    n               (4 1 1);
    delta           0.0001;
}

metisCoeffs
{
    processorWeights 4( 1 1 1 1 );
}

roots ();
```