Familiarization with OpenFOAM

Open-Source CFD Course 2021 – Lab 2

Miklós BALOGH and Josh DAVIDSON

2021

Hydro-thermodynamical equation system

Conservation laws

- Momentum (Navier–Stokes equations):

$$\frac{\partial \boldsymbol{u}}{\partial t} + \boldsymbol{u} \cdot \nabla \boldsymbol{u} = -\frac{1}{\rho} \nabla p + \nu \left[\nabla^2 \boldsymbol{u} + \frac{1}{3} \nabla \left(\nabla \cdot \boldsymbol{u} \right) \right] + \boldsymbol{g}$$

- Mass (continuity):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{u}) = 0$$

- Energy:

$$\frac{\partial \left(\rho c_p T \right)}{\partial t} + \nabla \cdot (\rho c_p T \boldsymbol{u}) = \nabla \cdot (k \nabla T) + Q_\nu + Q_{ch.reaction}$$

Relationship between the material properties

- Ideal gas law:

$$p = \rho R T$$

Continous, general solution

A fundamental problem in analysis is to decide whether such smooth, physically reasonable solutions exist for the Navier–Stokes equations, thus the Clay mathematical institute posts 1 million dollar reward among the seven most important mathematical problems of the millennium. These are:

- Yang–Mills and Mass Gap
- Riemann Hypothesis
- P vs NP Problem
- Navier–Stokes Equation
- Hodge Conjecture
- Poincaré Conjecture (solved by Grigorij Perelman, 2003)
- Birch and Swinnerton-Dyer Conjecture

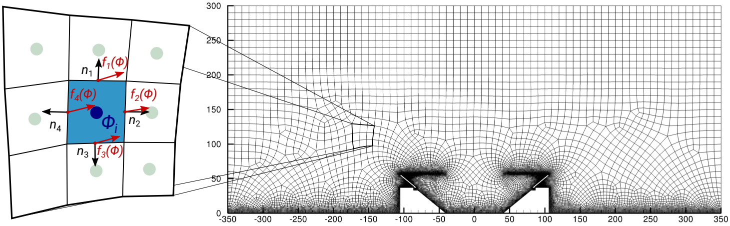
Numerical solution of the N–S equations

- General analytical solution of the N–S equation are not known, but numerical approximation is possible:
  - Spatial discretization (mesh: grid or cell network)
  - Boundary conditions (at the bounding surfaces)
  - Temporal discretization (suitable time step, Δt)
  - Initial conditions (at $t = 0$)
- Simplification of geometry: sub-grid features and details
- Simplifications of equations:
  - Suitable coordinate system (Cartesian, cylindrical, spherical)
  - Steady vs. unsteady
  - Compressible vs. incompressible
  - Laminar vs. turbulent
  - External forces (gravitational, Coriolis, centripetal)

Numerical solution of the N–S equations

- Spatial discretization
    - Finite Volume Method (FVM)
    - Finite Element Method (FEM)
    - Finite Difference Method (FDM)
    - Spectral methods (e.g. for DNS on periodic domains)
    - Particle methods (e.g. SPH)
    - Lattice gas model, lattice-Boltzmann method
- Temporal discretization (unsteady problems)
    - Explicit and implicit schemes, stability criteria (e.g. CFL)
    - Local time-step, adaptive time-step control
- Pressure-velocity coupling
    - Pressure correction (sequential, e.g. SIMPLE, PISO)
    - Coupled: simultaneous solution of the equations

Finite Volume Method (FVM)



General transport equation: $$\frac{\partial \Phi}{\partial t} + \nabla \cdot f(\Phi) = 0$$

Its volume integral form: $$\int_V \left[\frac{\partial \Phi}{\partial t} + \nabla \cdot f(\Phi) \right] dV = 0$$

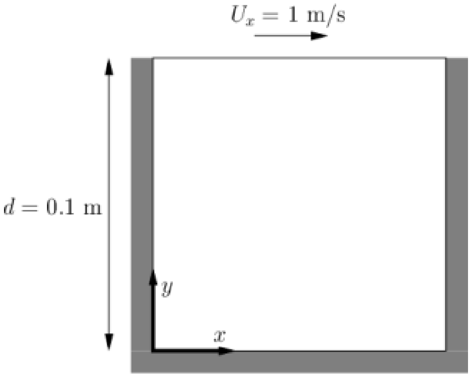
Using the Gauss-theorem: $$\int_V \frac{\partial \Phi}{\partial t} dV + \int_A f(\Phi) \cdot \boldsymbol{n} dA = 0$$

Discretized form for the i^{th} cell: $$V_i \frac{\partial \Phi_i}{\partial t} + \sum_j f_j(\Phi_i) \cdot \boldsymbol{n}_j A_j = 0$$

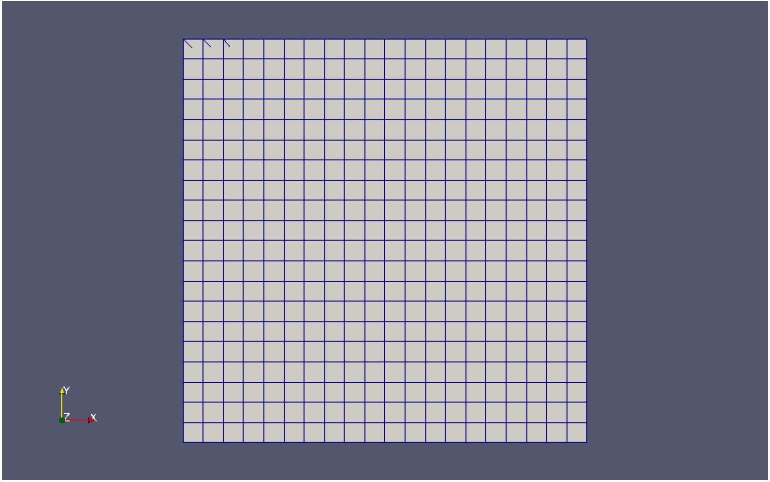
Steps of the numerical analysis

- Construction of the geometry (computational domain)
- Mesh generation
  - The basis of the spatial discretization
  - Decomposition of the domain to cells
- Definition of the boundary conditions
- Definition of the initial conditions
  - Constant - predefined values
  - Hybrid - potential flow solver
  - Patch - values given cell by cell (e.g. theoretical values)
  - Mapping - values from simulation (interpolation)
- Simulation (numerical integration of the equations)
- Post-processing

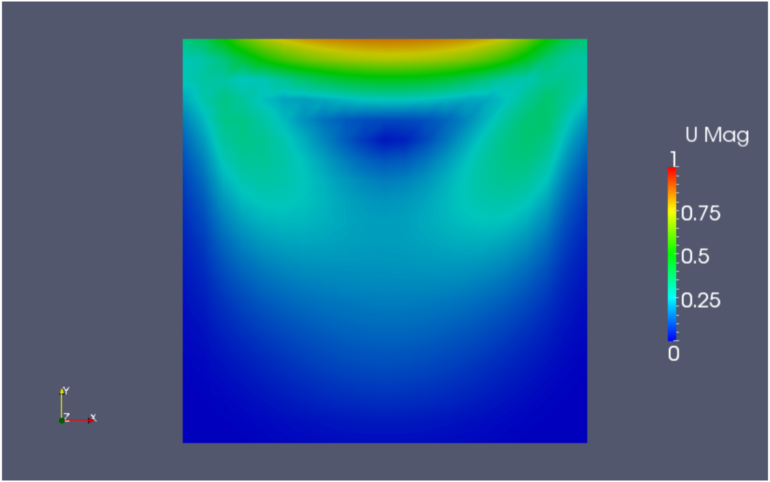
Lid-driven cavity – Geometry



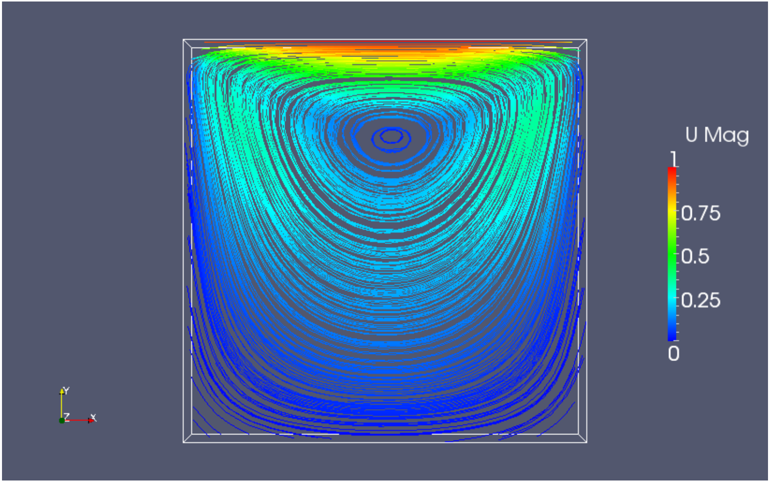
Lid-driven cavity – Mesh



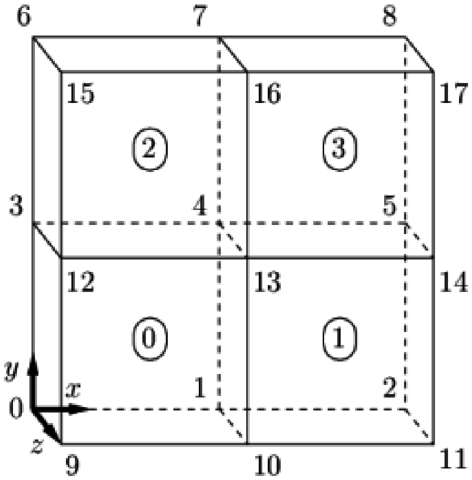
Lid-driven cavity – Velocity



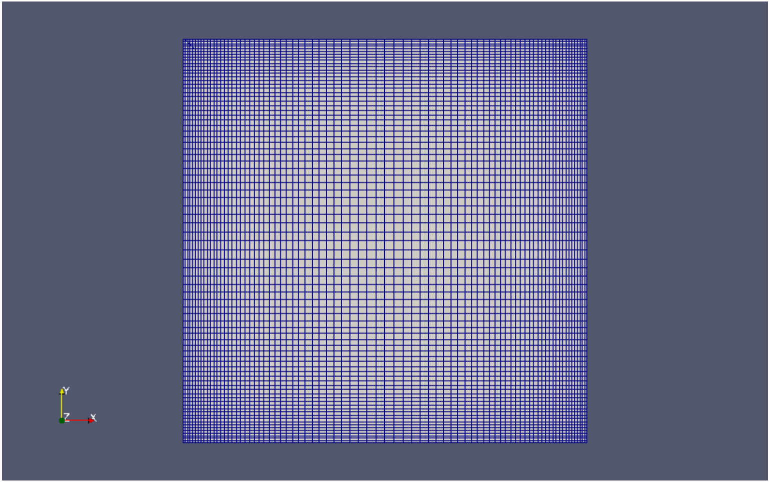
Lid-driven cavity – Streamlines



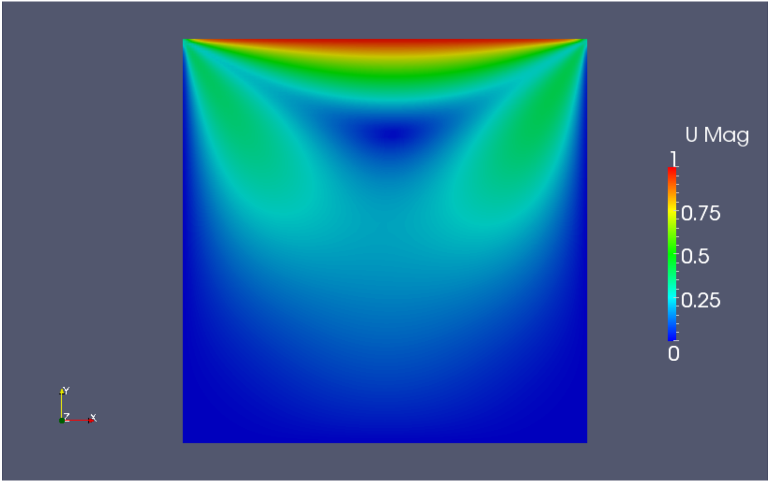
Refined lid-driven cavity – Geometry



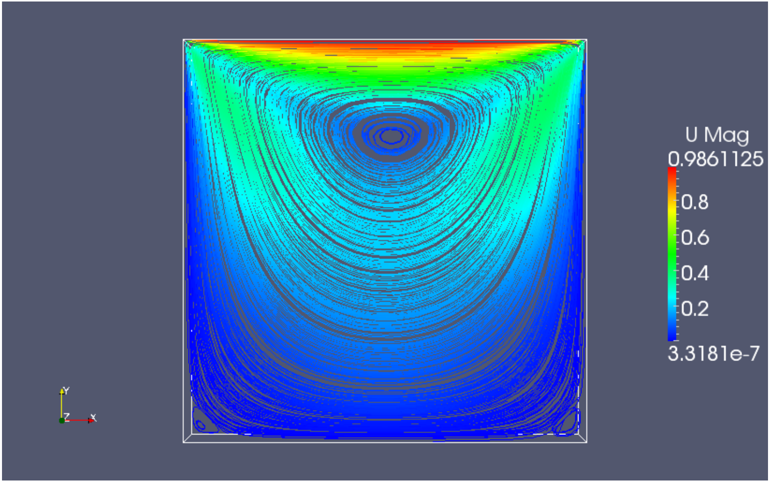
Refined lid-driven cavity – Mesh



Refined lid-driven cavity – Velocity



Refined lid-driven cavity – Streamlines



Mapping fields in OpenFOAM

- One can initialize a simulation with former results
  - obtained even on lower resolution,
  - via interpolating the fields to the new mesh

```
cd $FOAM_RUN/tutorials/incompressible
cd icoFoam/cavity/cavity
blockMesh > blockMesh.log
icoFoam > icoFoam.log
cd ../cavityGrade
blockMesh > blockMesh.log
mapFields ../cavity -consistent
icoFoam > icoFoam.log
```

Bash scripts – executes linux commands in a row

Listing 1: Hello World sample script

```
#!/bin/bash
STR="Hello World!"
echo $STR
```

Listing 2: OpenFOAM runner sample script

```
#!/bin/bash
blockMesh > blockMesh.log
icoFoam > icoFoam.log
```

Bash scripts – executes linux commands in a row

Listing 3: Clocking sample script

```
#!/bin/bash
START_T=$(date +%s.%N)
# Do something time consuming here...
END_T=$(date +%s.%N)
ELAPS_T=$(echo "$END_T - $START_T" | bc)
```

Listing 4: Running a script

```
# Save as name.bsh and run with sh command
sh name.bsh
# Or just change permissions and run it
chmod +x name.bsh
./name.bsh
```

Laboratory tasks I.

1. Write and run a script to perform the simulation of lid-driven cavity including
    - Mesh generation
    - Simulation (in controlDict set endTime to 1)
    - Redirecting the output to a logfile
    - Plotting the time consumption of every steps of the analysis
2. Visualize the results using paraFoam
    - Velocity map with vectors
    - Streamlines colored by the velocity
    - Mesh

Laboratory tasks II.

③ Modify the cavityGrade case and run (via bash script)
  - Modify constant/polyMesh/blockMeshDict (to have a fine, graded mesh)
  - Modify system/contolDict (according to the CFL)
  - Create the mesh
  - Map the fields from the simple cavity case
  - Run the simulation

```
blocks
(
    hex (0 1 4 3 9 10 13 12) (40 40 1) simpleGrading (4 4 1)
    hex (1 2 5 4 10 11 14 13) (40 40 1) simpleGrading (0.25 4 1)
    hex (3 4 7 6 12 13 16 15) (40 40 1) simpleGrading (4 0.25 1)
    hex (4 5 8 7 13 14 17 16) (40 40 1) simpleGrading (0.25 0.25 1)
);
```

```
deltaT          0.0005;

writeControl    timeStep;

writeInterval   200;
```

Assignments

1. How many finite volume cells are used in the performed simulation?
2. How many time-step is done for the cavityGraded case?
3. What is the mean and maximum Courant number for the cases in the last time-step?
4. How many iteration step was required when solving pEqn in the first and the last time-step?
5. How does the Courant number change, if the resolution is doubled and the time-step is halved?
6. What is the smallest cell size in case of the graded mesh?

Homework

1. Visualize the results of cavityGraded case
   - Velocity map with vectors
   - Streamlines colored by the velocity
   - Mesh
2. Compare the results to the basic cavity case

Listing 5: Open multiple cases with paraFoam

```
# Open a case (e.g. cavity)
cd $FOAM_RUN/tutorials/incompressible/icoFoam/cavity/cavity
paraFoam &

# Open another case (e.g. cavityGraded)
# Create a file in the case directory can be handled by paraFoam
touch ../cavityGrade/cavityGrade.OpenFOAM

# Open it with paraFoam (Open item of the File menu)
```

Questions?

Thanks for your attention!