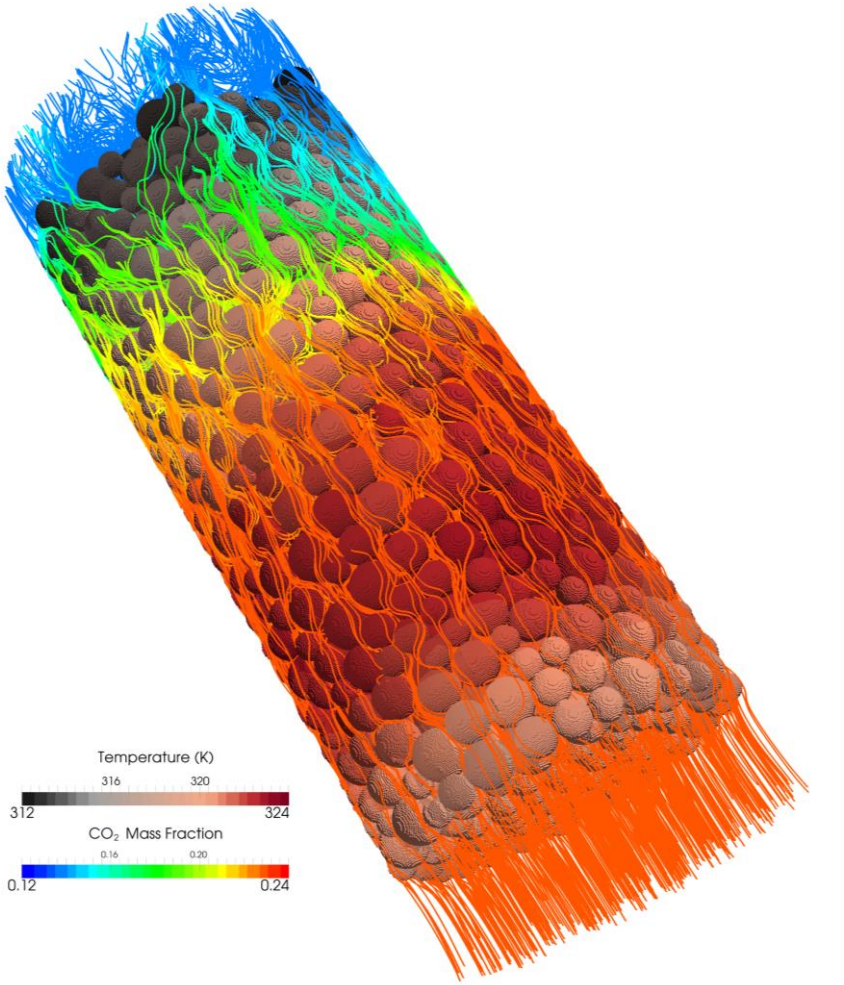
OpenFOAM® Basic Training

Tutorial Five



3rd edition, Feb. 2015

www.thvt.at

Editors and Contributors:

- Bahram Haddadi (TU Wien)
- Christian Jordan (TU Wien)
- Jozsef Nagy (JKU Linz)
- Clemens Gößnitzer (TU Wien)
- Vikram Natarajan (TU Wien)
- Sylvia Zibuschka (TU Wien)
- Michael Harasek (TU Wien)

Cover picture from:

- Bahram Haddadi, The image presented on the cover page has been prepared using the Vienna Scientific Cluster (VSC).

scalarTransportFoam – circle (discretization)

Simulation

Use the scalarTransportFoam solver, do simulate the movement of a circular scalar spot region (radius = 1 m) at the middle of a 100×100 cell mesh (10 m×10 m), then move it to the right, to the top and diagonally.

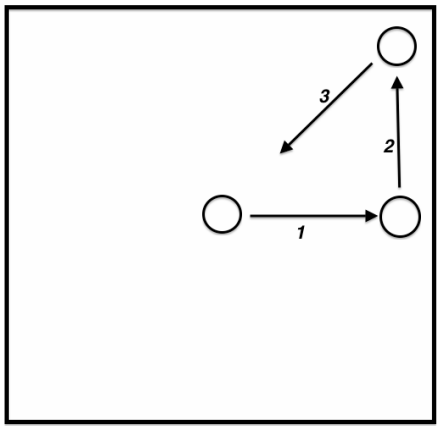


Figure 5.1 Schematic sketch of the problem

Objectives

- Choosing the best scheme of discretization

Post processing

Examine your simulation in ParaView.

Step by step simulation

Compile tutorial

Create the new case in your working directory like in example four.

0 directory

To move the circle to right change the `internalField` to `(1 0 0)` in the U file for setting the velocity field towards right for moving the circle to the right. Modify U at suitable times, to obtain a velocity field which will move the circle up and also diagonally.

constant directory

In the polyMesh directory, modify the blockMeshDict for creating a 2D geometry with 100x100 cells mesh.

```
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
convertToMeters 1;

vertices
(
    (-5 -5 -0.01)
    (5 -5 -0.01)
    (5 5 -0.01)
    (-5 5 -0.01)
    (-5 -5 0.01)
    (5 -5 0.01)
    (5 5 0.01)
    (-5 5 0.01)
);
blocks
(
    hex (0 1 2 3 4 5 6 7) (100 100 1) simpleGrading (1 1 1)
);
edges
(
);
boundary
(
    sides
    {
        type patch;
        faces
        (
            (1 2 6 5)
            (0 4 7 3)
            (3 7 6 2)
            (0 1 5 4)
        );
    }
    empty
    {
        type empty;
        faces
        (
            (5 6 7 4)
            (0 3 2 1)
        );
    }
);
// ************************************************************************* //
```

In the transportProperties set `DT` to zero (No diffusion!).

system directory

Choose a discretization scheme based on the results from the previous example and set the fvSchemes.

In the setFieldDict patch a circle to the middle of the geometry using the following lines.

```
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

defaultFieldValues (volScalarFieldValue T 0 );

regions
(
        cylinderToCell
        {
                p1 ( 0 0  -1 );
                p2 ( 0 0  1 );
                radius 0.5;
                fieldValues
                (
                        volScalarFieldValue T 1
                ) ;
        }
);


// ************************************************************************* //
```

`cylinderToCell` command is used to patch a cylinder to the region, `p1` and `p2` show the two ends of cylinder center line, in the `radius` the radius is set.

Check controlDict, in the first part of simulation, where the circle should move to the right set the `startFrom` to `startTime` and `startTime` to `0`. By a simple calculation it can be seen that the `endTime` should be `3` s. Similar calculations need to be done for the two other parts, except the `startTime` is set to the `endTime` of previous part, and new `endTime` should be that part "simulation time" plus `endTime` of the previous part.

Running simulation

```
>blockMesh

>setFields

>scalarTransportFoam
```

For running the further parts (moving the circle to top, and then diagonally) change the velocity field in the last time step directory.

After moving the circle to the right and changing the velocity field, the simulation is resumed. It was seen that the circle does not go up and moves to the right. This occurs due to the fact that OpenFOAM® used the previous time step fluxes (phi) to do the calculations. We can solve this problem by deleting phi file from the latest time step (of the previous part of simulation, e.g. 3). In this way, OpenFOAM® creates new fluxes based on the new velocity field that we just updated. So, easily delete phi and enjoy!

Exporting simulation

The simulation results are as follows:

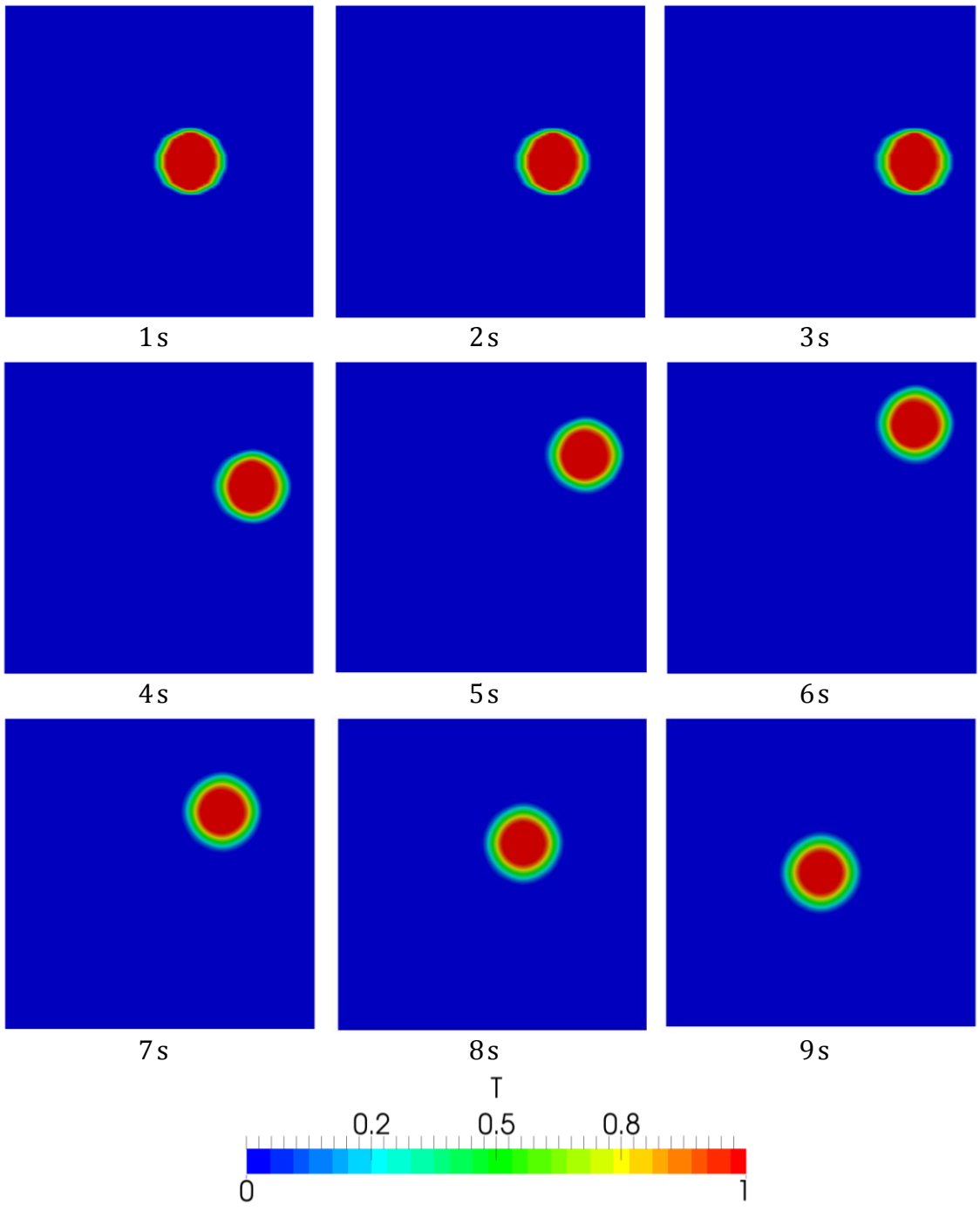


Figure 5.2 Position of the circle at different time steps