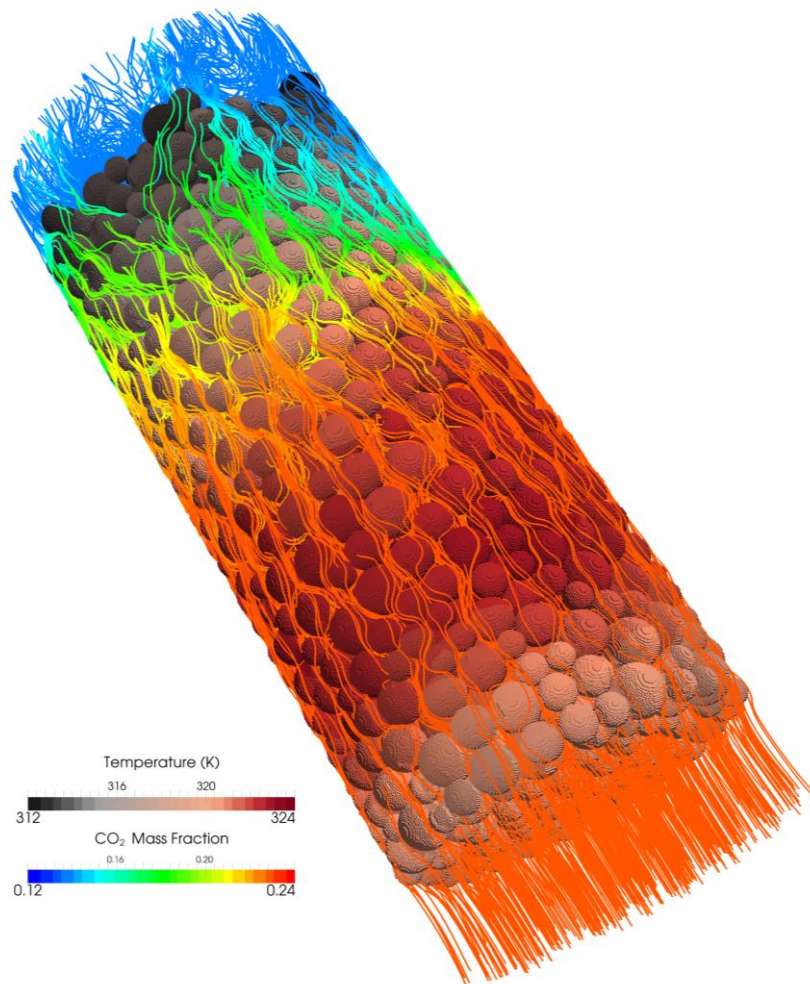




OpenFOAM® Basic Training

Tutorial Five



3rd edition, Feb. 2015



This offering is not approved or endorsed by ESI® Group, ESI-OpenCFD® or the OpenFOAM® Foundation, the producer of the OpenFOAM® software and owner of the OpenFOAM® trademark.

Except where otherwise noted, this work is licensed under <http://creativecommons.org/licenses/by-nc-sa/3.0/>

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).


 Except where otherwise noted, this work is licensed under <http://creativecommons.org/licenses/by-nc-sa/3.0/>

Attribution–NonCommercial–ShareAlike 3.0 Unported (CC BY–NC–SA 3.0)

This is a human-readable summary of the Legal Code (the full license).

Disclaimer

You are free:

- to Share — to copy, distribute and transmit the work
- to Remix — to adapt the work

Under the following conditions:

Attribution — You must attribute the work in the manner specified by the author or licensor (but not in any way that suggests that they endorse you or your use of the work).

Noncommercial — You may not use this work for commercial purposes.

Share Alike — If you alter, transform, or build upon this work, you may distribute the resulting work only under the same or similar license to this one.

With the understanding that:

Waiver — Any of the above conditions can be waived if you get permission from the copyright holder.

Public Domain — Where the work or any of its elements is in the public domain under applicable law, that status is in no way affected by the license.

Other Rights — In no way are any of the following rights affected by the license:

Your fair dealing or fair use rights, or other applicable copyright exceptions and limitations;

The author's moral rights;

Rights other persons may have either in the work itself or in how the work is used, such as publicity or privacy rights.

Notice — For any reuse or distribution, you must make clear to others the license terms of this work. The best way to do this is with a link to this web page.

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.

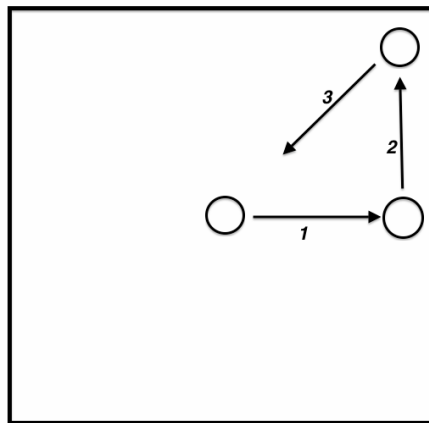


Figure 5.1 Schematic sketch of the problem

Objectives

- Choosing the best scheme of discretization

Post processing

Examine your simulation in ParaView.

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:

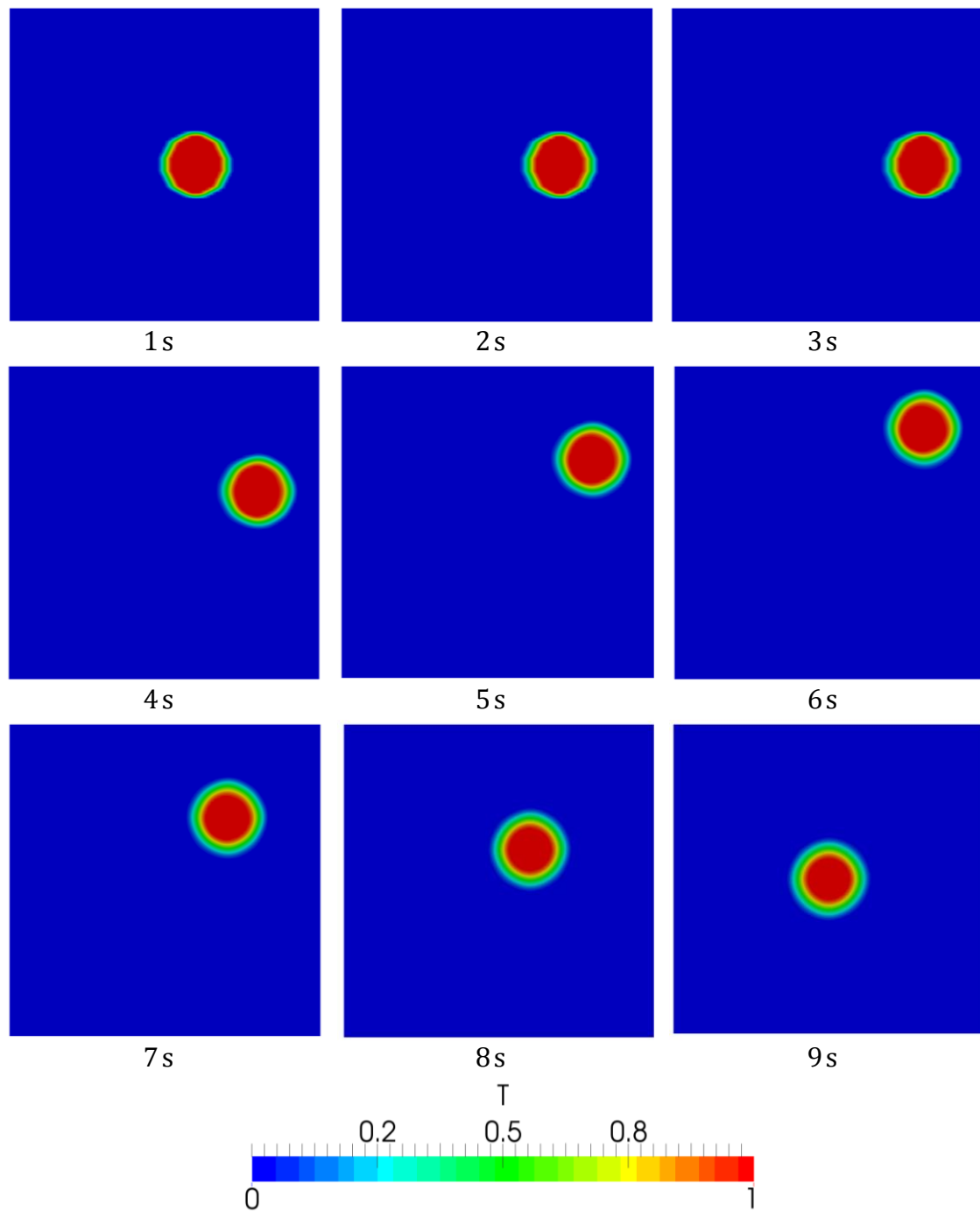


Figure 5.2 Position of the circle at different time steps