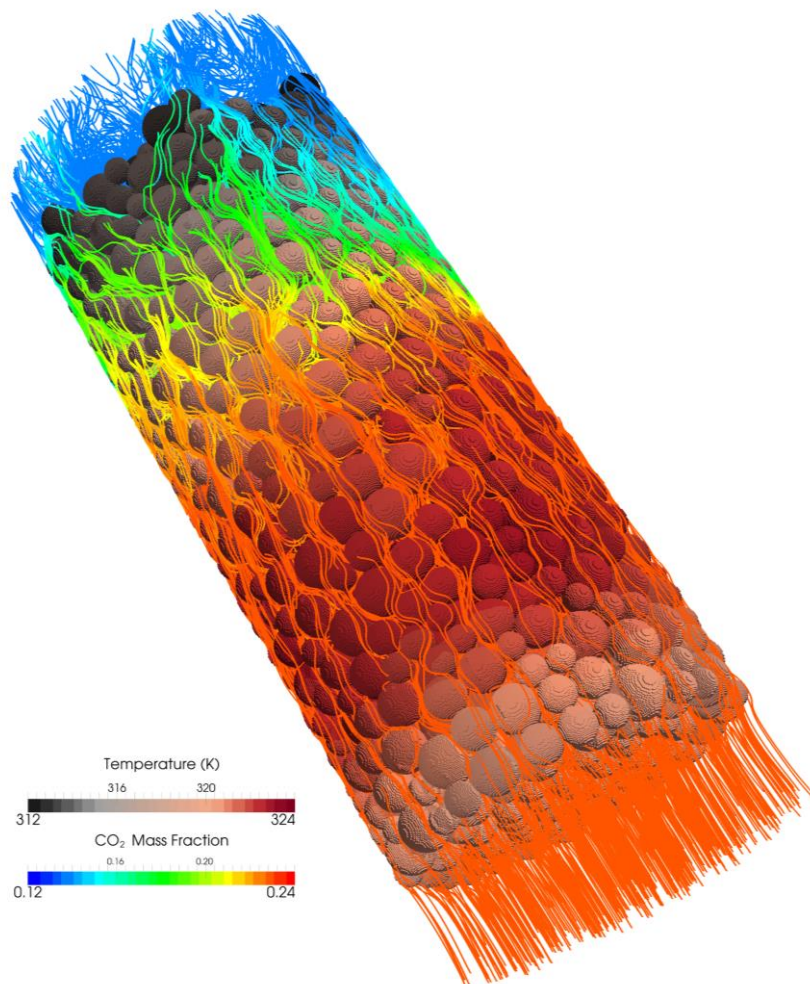


## OpenFOAM<sup>®</sup> Basic Training

### Tutorial Four



3<sup>rd</sup> edition, Feb. 2015



This offering is not approved or endorsed by ESI<sup>®</sup> Group, ESI-OpenCFD<sup>®</sup> or the OpenFOAM<sup>®</sup> Foundation, the producer of the OpenFOAM<sup>®</sup> software and owner of the OpenFOAM<sup>®</sup> 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 – shockTube (discretization)

### Simulation

Use the scalarTransportFoam solver, simulate 5 s of flow inside a shock tube, with 1D mesh of 1000 cells (10 m long geometry from -5 m to 5 m). Patch with a scalar of 1 from -0.5 to 0.5. Simulate following cases:

- Set U to uniform (0 0 0). Vary diffusion coefficient (low, medium and high value).
- Set the diffusion coefficient to zero and also U to (1 0 0) and run the simulation in the case of pure advection using following discretization schemes:
  - upwind
  - linear
  - linearUpwind
  - Gauss linear
  - QUICK
  - cubic

### Objectives

- Understanding different discretization schemes.

### Post processing

Import your simulation into ParaView, and plot temperature along tube length.



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

As it was mentioned before, the discretization scheme for each operator of the governing equations can be set in fvSchemes.

```
// ***** //
ddtSchemes
{
    default          Euler;
}

gradSchemes
{
    default          Gauss linear;
}

divSchemes
{
    default          none;
    div(phi,T)      Gauss linearUpwind grad(T);
}

laplacianSchemes
{
    default          none;
    laplacian(DT,T) Gauss linear corrected;
}

interpolationSchemes
{
    default          linear;
}

snGradSchemes
{
    default          corrected;
}

fluxRequired
{
    default          no;
    T                ;
}
// ***** //
```

For each type of operation a default scheme can be set (e.g. for `divSchemes` is set to `no default`, it means no default scheme is set). Also a special type of discretization for each element can be assigned (e.g. `div(phi,T)` it is set to `linearUpwind`). For each element, which a discretization method has not been set, the default method will be applied.

*Note:* The general transport equation for property  $\varphi$  looks like the following:

$$\frac{\partial(\rho\varphi)}{\partial t} + \nabla \cdot (\rho\varphi\mathbf{u}) - \nabla \cdot (\Gamma\nabla\varphi) = S_\varphi$$

*In this equation the first term shows the rate of change of property  $\varphi$  with time. The second term is responsible for the advection of property  $\varphi$  by the fluid flow and the third term shows the diffusion of property  $\varphi$ .*

*The right hand side of the equation also refers to the source terms. By setting the diffusion coefficient ( $\Gamma$ , in this simulation it is `DT`) to zero, the case will be switched to a pure advection simulation with no diffusion.*

*Note: In fvSchemes, the schemes for the time term of the general transport equation are set in ddtSchemes sub-dictionary. divSchemes are responsible for the advection term schemes and laplacianSchemes set the diffusion term schemes.*

*Note: divSchemes should be applied like this: Gauss + scheme. The Gauss keyword specifies the standard finite volume discretization of Gaussian integration which requires the interpolation of values from cell centers to face centers. Therefore, the Gauss entry must be followed by the choice of interpolation scheme (www.openfoam.org).*

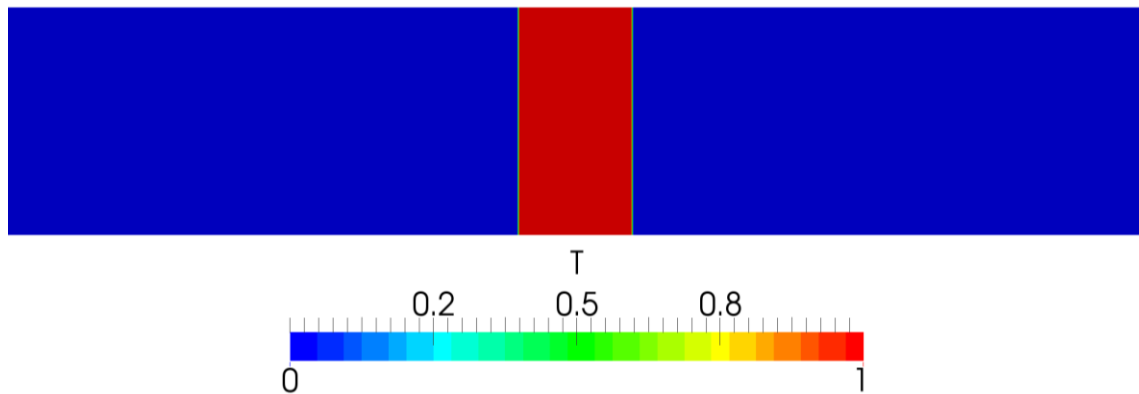
### Running simulation

```
>blockMesh
>setFields
>scalarTransportFoam
```

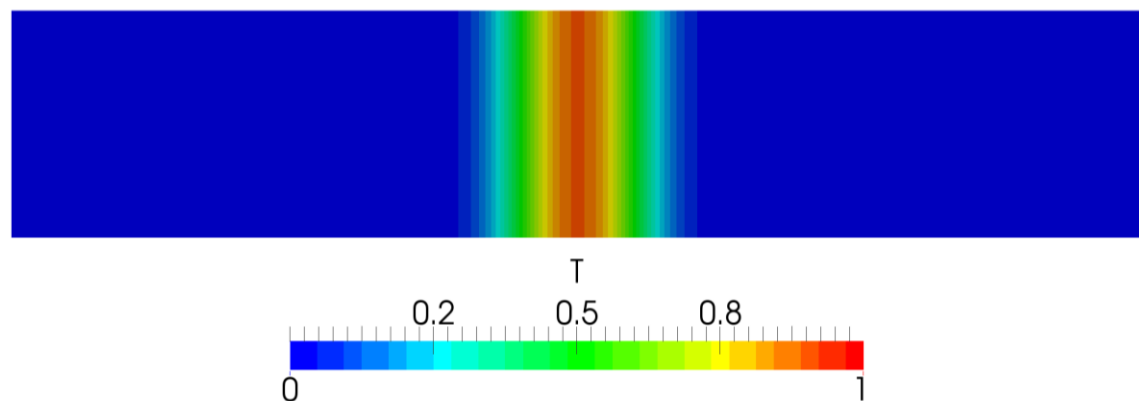
### Exporting simulation

The simulation results are as follows.

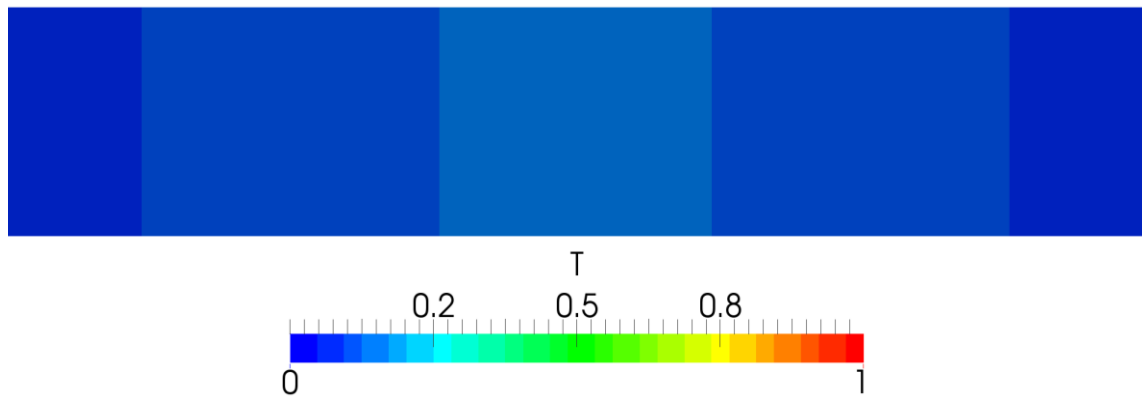
A) Case with zero velocity (pure diffusion):



**Figure 4.1** Pure diffusion with low diffusivity (0.00001) at  $t = 5$  s

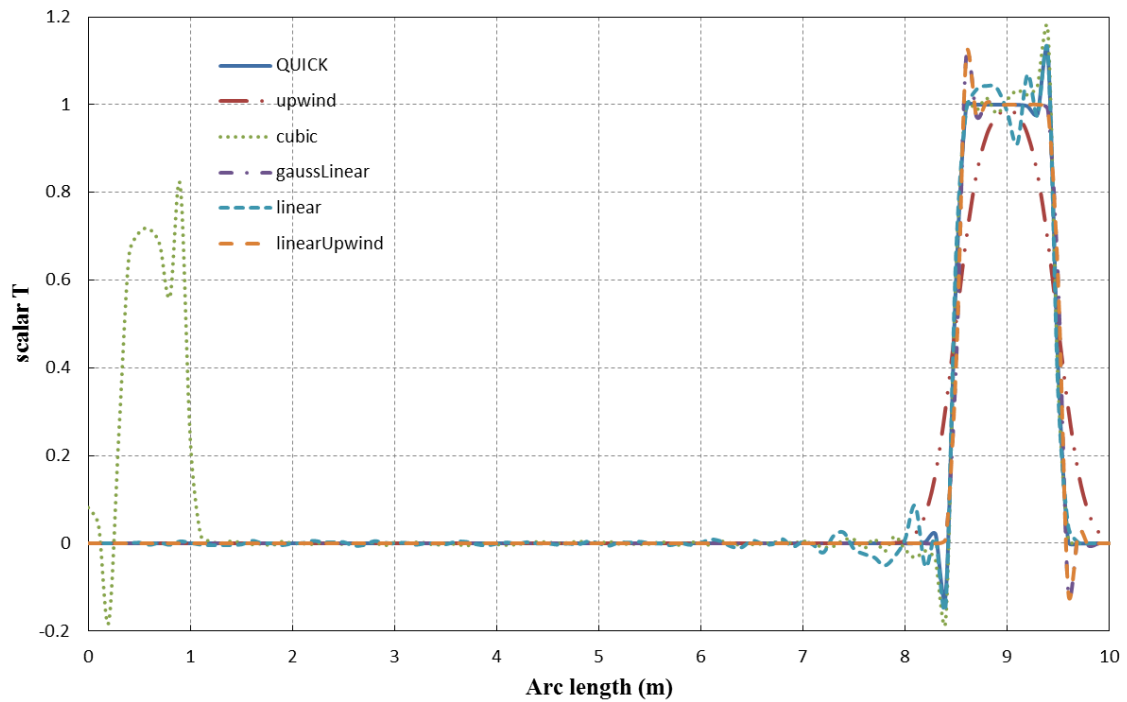


**Figure 4.2** Pure diffusion with medium diffusivity (0.01) at  $t = 5$  s



**Figure 4.3** Pure diffusion with high diffusivity (1) at  $t = 5$  s

B) Case with pure advection (diffusion coefficient = 0):



**Figure 4.4** Scalar T along tube at  $t = 4$  s