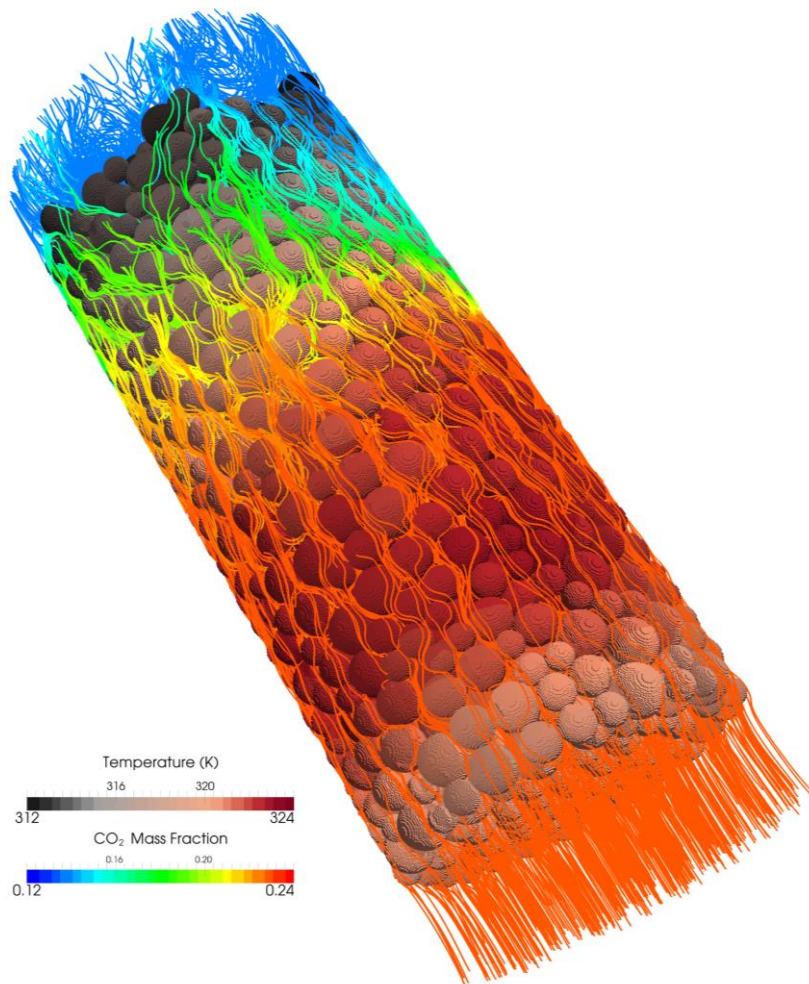


OpenFOAM[®] Basic Training

Tutorial Three



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.

sonicFoam – shockTube

Simulation

Use the sonicFoam solver, simulate 0.007 s of flow inside a shock tube, with a mesh with 100, 1000 and 10000 cells in one dimension, for initial values 1 bar/0.1 bar and 10 bar/0.1 bar.

Objectives

- Understanding setFields
- Investigate effect of grid resolution

Post processing

Import your simulation into ParaView, and compare results.

Step by step simulation

Open tutorial

Copy the tutorial from the following directory to your working directory

```
~/OpenFOAM/OpenFOAM-2.3.0/tutorials/compressible/sonicFoam/  
laminar/shockTube
```

constant directory

By checking the geometry and the mesh, it is obvious that it is a 1D mesh, because of the number of mesh cells in y and z directions is one, and also in the patches, plates vertical to these directions are defined as empty boundary condition. The mesh density can be set in the `blocks` part by changing x direction mesh size (e.g. change it from 1000 to 100 or 10000).

system directory

Checking system directory, there is a file “setFieldDict” which is used by the tool `setFields` for patching (assign an amount to a region) in the simulation. For example, here the pressure of 0.1 bar should be patched to half of the region (the geometry is from -5 to 5, so from 0 to 5 will be patched) and 10 bar to the other half.

```
// * * * * * //  
  
defaultFieldValues ( volVectorFieldValue U ( 0 0 0 ) volScalarFieldValue T  
348.432 volScalarFieldValue p 1000000 );  
  
regions ( boxToCell { box ( 0 -1 -1 ) ( 5 1 1 ) ; fieldValues (  
volScalarFieldValue T 278.746 volScalarFieldValue p 10000 ) ; } );  
  
// * * * * * //
```

In the `defaultFieldValues`, a value is assigned to the whole domain, for example here, the velocity has been set everywhere to zero, the temperature to 348.432 K, and the pressure to 1000000 Pa. In the `regions`, `boxToCell` defines the region to which a special amount must be patched. With `boxToCell` the region is chosen by a cube, and the cube is defined by giving the coordinates of one of its diagonals.

After choosing the region, the new values are assigned to the parameters (e.g. here temperature 278.746 K and pressure 10000 Pa).

Running simulation

In order to assign the values which were set in the `setFieldDict`:

```
>setFields
```

Then run:

```
>sonicFoam
```

Note: Checking `deltaT` in `controlDict` in the system directory, it is $1e-5$ s. The question is: What is the criteria for setting `deltaT`? If `deltaT` is bigger, the simulation will run faster, but a too big `deltaT` makes the simulation unstable and sometimes physically meaningless. Therefore `deltaT` should be selected in a way to

have a fast and at the same time stable and also physically reasonable simulation. The Courant (Co) number is a dimensionless number, which is usually used as a necessary condition for having a convergent solution, for one dimension:

$$Co = \frac{u\Delta t}{\Delta x}$$

Where u is velocity magnitude in that direction, Δt is `deltaT` and Δx is the mesh size in this direction. For having a convergent solution in most of the cases Co should be less than one in all the cells in the domain.

As it is obvious from the equation by decreasing the Δx or the mesh size `deltaT` should be also adjusted (decreased) for having a stable and convergent solution.

In the OpenFOAM® simulations usually Co is calculated in a way to make sure maximum Courant number in the whole domain is less than 1. It is assumed $Co = 1$, and for Δx smallest cell size and for u maximum velocity magnitude in domain is selected. Then using these data, `deltaT` is calculated. It is a rough estimation, but always helps to keep $Co < 1$!

Note: In the 10000 cell case with 10 bar and 0.1 bar, the simulation will crash with the default `deltaT` ($1e-5$); After checking the same case with 1000 cells, you will find that the maximum Co is around 0.6:

```
Time = 0.001
```

```
Courant Number mean: 0.0508555 max: 0.589018
diagonal: Solving for rho, Initial residual = 0, Final residual = 0, No
Iterations 0
```

In the case with 10000 cells, the number of cells is increased by factor 10, so the cell size is reduced by factor 10. For keeping the Courant number in the same range (around 0.6), according to the above equation, `deltaT` should be decreased by factor 10. After reducing it to $1e-6$ the simulation will run smoothly.

Note: After running `setFields` for the first time, the files in the 0 directory are overwritten. If the mesh will be changed these files are not compatible with the new mesh and the simulation will fail. To solve this problem replace the files in the 0 directory with the files in the 0.org. In the OpenFOAM® files or directories with suffix “.org” (“original”) usually contain the backup files. If a command changes the original files these files can be replaced.

Exporting simulation

The simulation results are as follows:

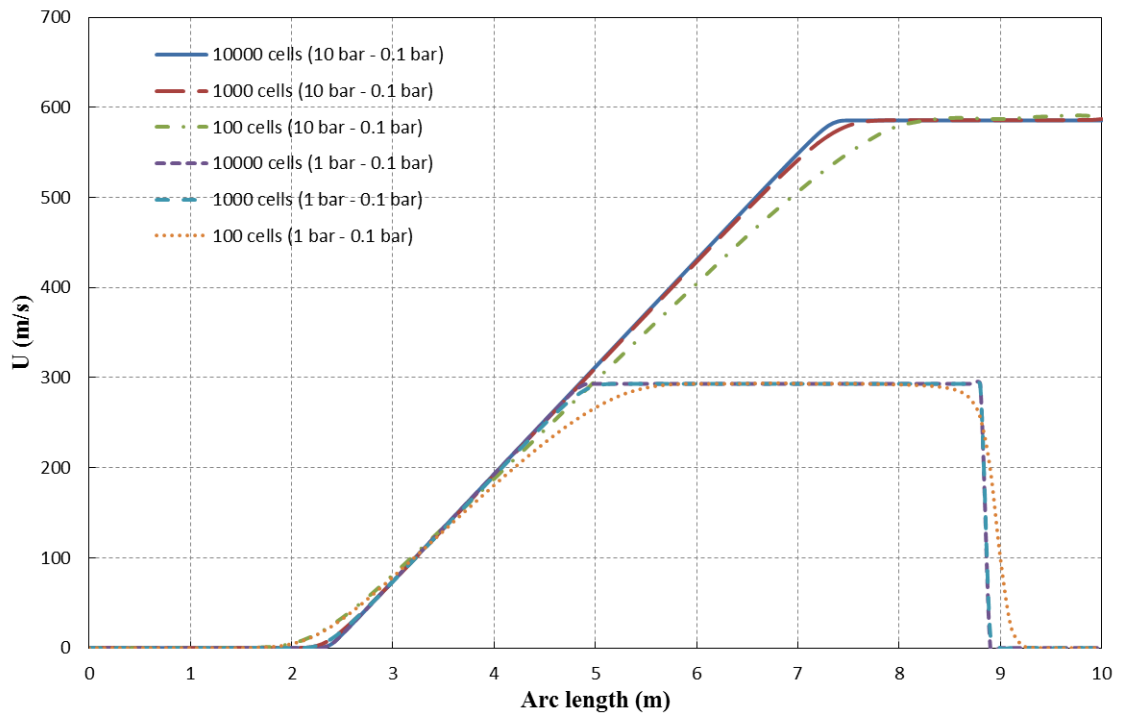


Figure 3.1 Velocities for different configurations along tube at $t = 0.007$ s

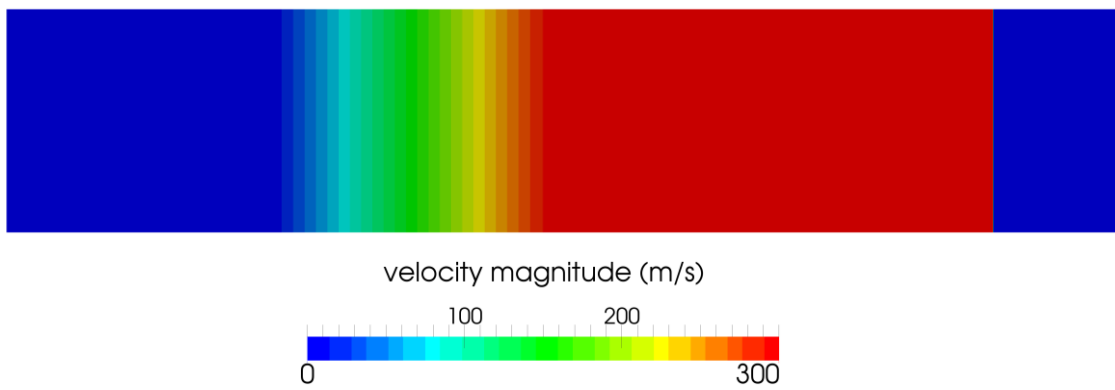


Figure 3.2 Velocity along tube axis for 10 bar/0.1 bar and 10000 cells case at $t = 0.007$ s

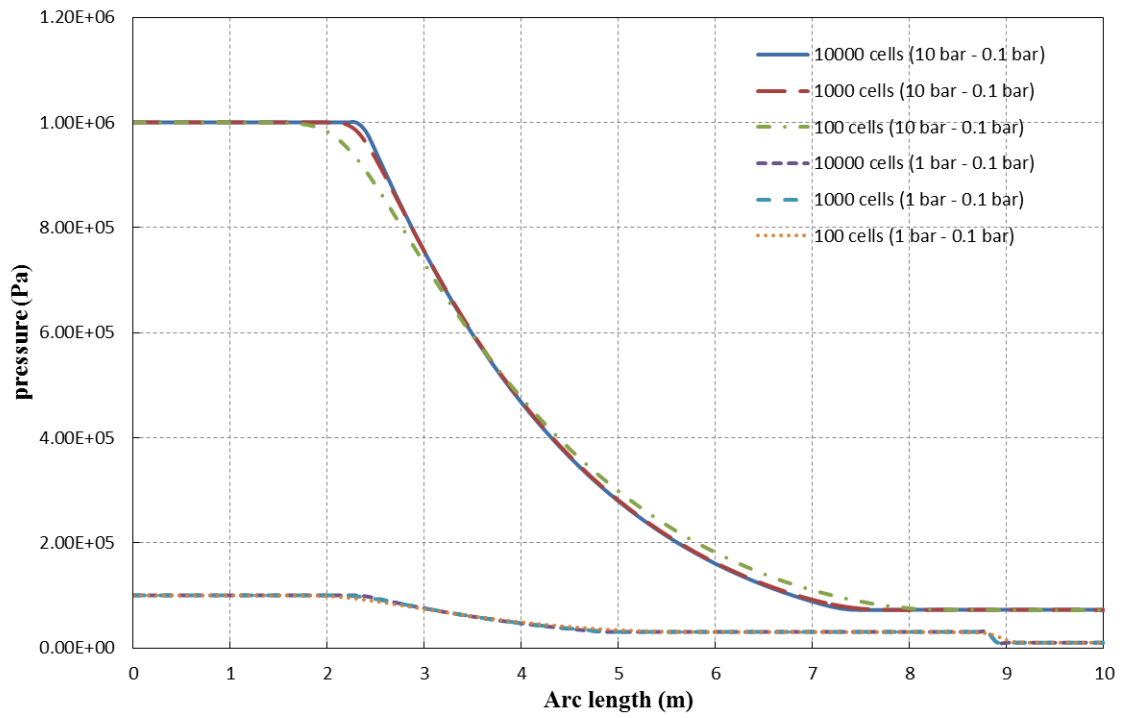


Figure 3.3 Pressures for different configurations along tube at $t = 0.007$ s

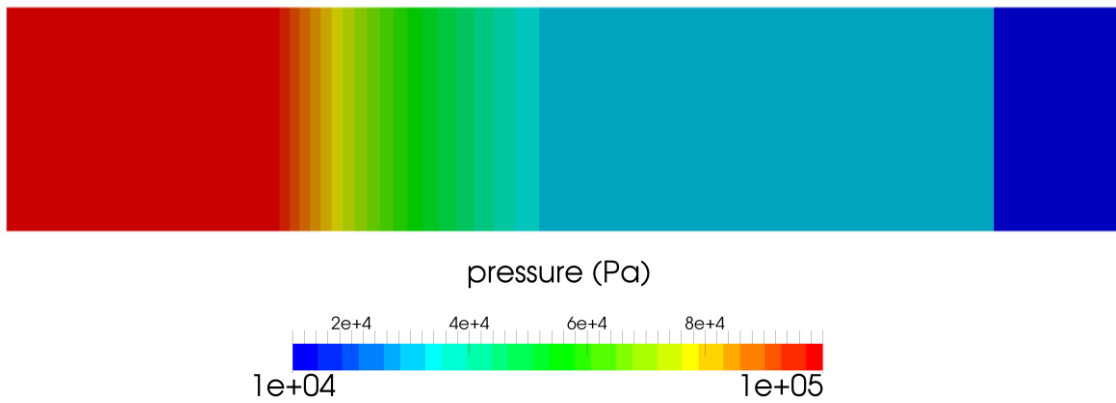


Figure 3.4 Pressure along tube axis for 10 bar/0.1bar and 10000 cells case at $t = 0.007$ s

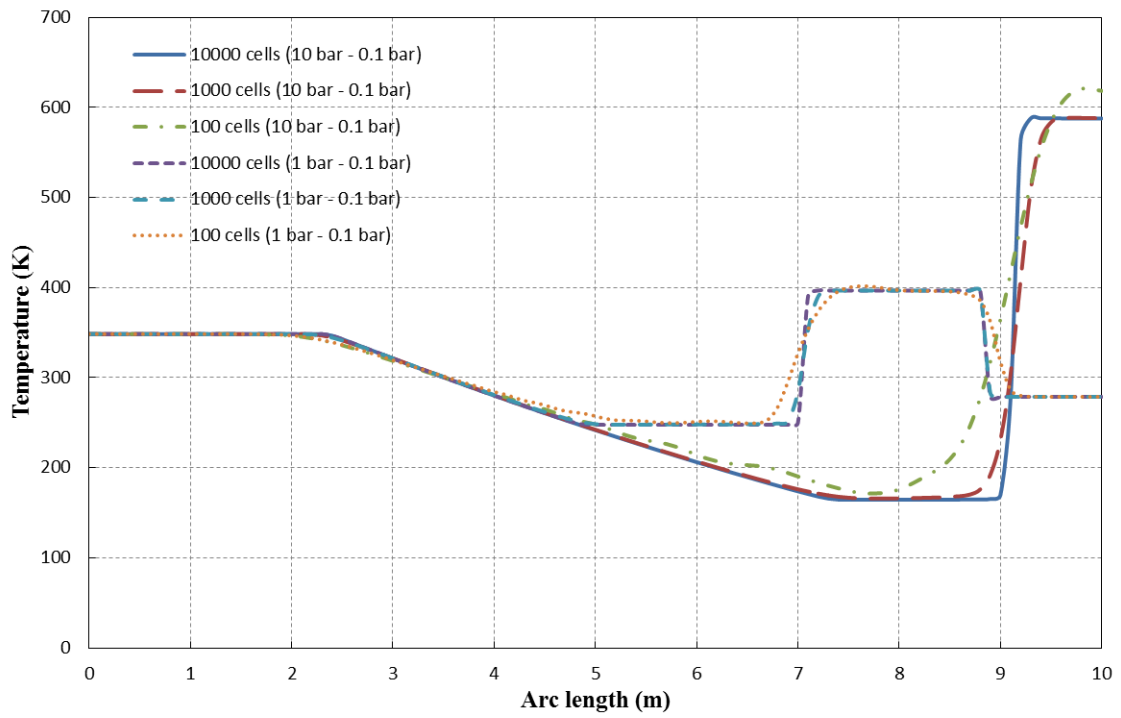


Figure 3.5 Temperature for different configurations along tube at $t = 0.007$ s

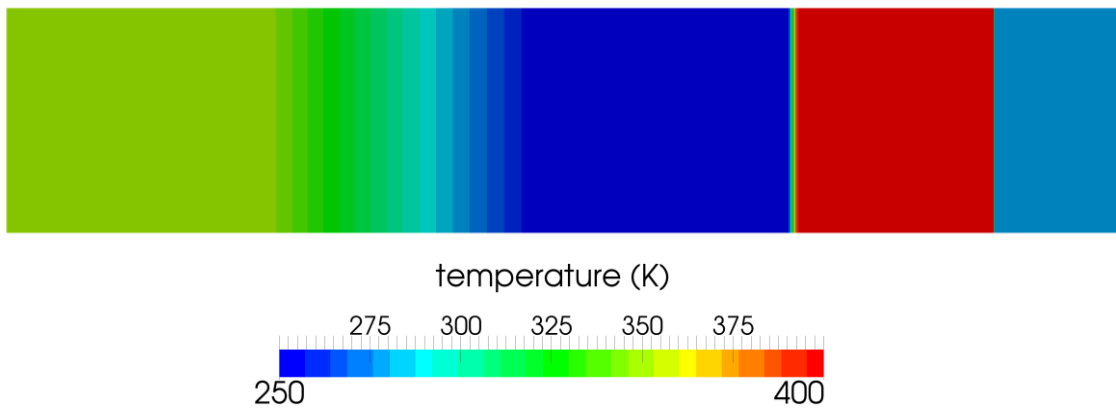


Figure 3.6 Temperature along tube axis for 10 bar/0.1bar and 10000 cells case at $t = 0.007$ s