

Cloud Computing 2020/2021

# Air-driven 3D shock tube simulation using OpenFOAM

## 1. Introduction

Several air-driven 3D shock tube simulations took place using OpenFOAM open-source CFD software and AWS EC2 instance to guarantee efficient computing.

## 2. Geometry, mesh, boundary/initial conditions and setup

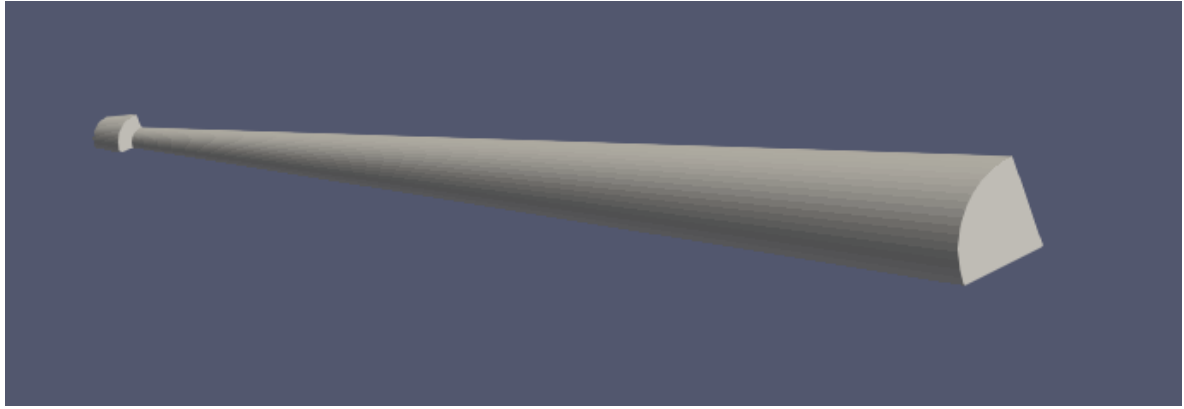
The aim of this project was to model 3D compressible, supersonic flow in the shock tube consisting of 1 m long driver section (150 mm in diameter) and a 4 m long driven section ( $\phi 80$ ), both containing pure air. The future aim is to change gas compositions in both sections to model combustion in helium-driven shock tube with air-fuel mixture in the driven section, including RAS or even LES turbulence model.

Geometry representing internal volume of quarter of a pipe and a structured mesh were generated using ANSYS 2020 R1 software. Then, existing mesh was converted to OpenFOAM PolyMesh text file. The mesh contained of 464400 Hex-dominant cells and had a very good average orthogonal quality (0.99).

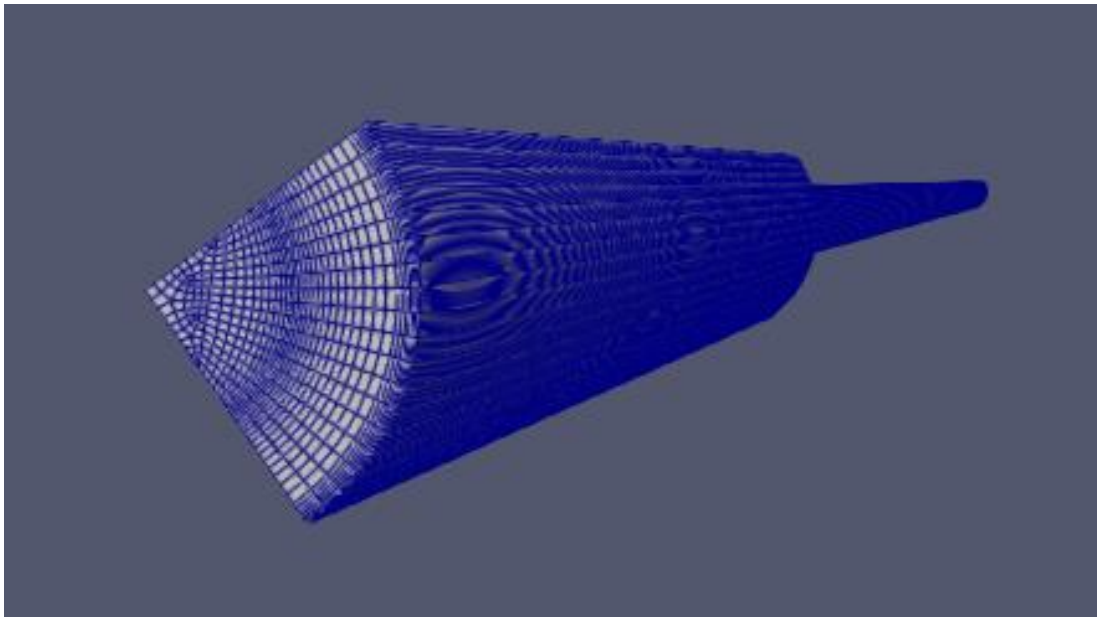
All meshing bodies were merged using CyclicAMI boundary condition on all contact regions. The walls were established as adiabatic. There were 2 symmetry planes. Initial pressure and temperature values in the driver and driven section were, respectively: 2 MPa, 403 K and 1000 hPa, 293 K. Simulation time step was set to  $1e-07$  s to achieve precise results. The end time was 0.1 s, so the total number of time steps was 1000000.



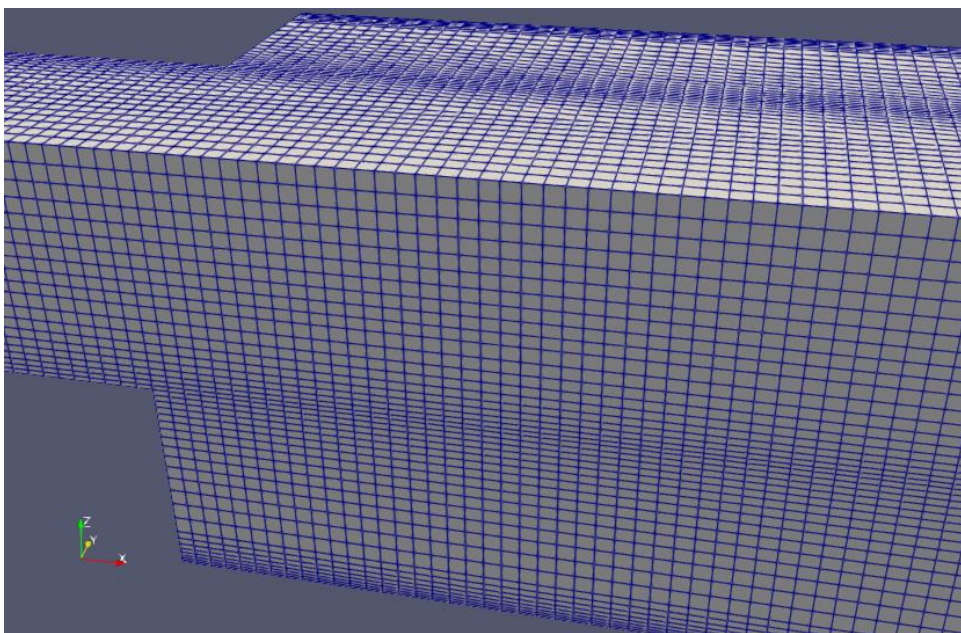
Figure 1. Geometry – symmetry plane view



*Figure 2. Geometry – driven section view*



*Figure 3. Mesh - driver section view*



*Figure 4. Mesh - symmetry planes detail view*

### 3. Solver

Calculations were made using sonicFoam and rhoPimpleFoam solvers in order to make a comparison between their performance, but in macro-scale the results for both of them occurred to be identical, so only sonicFoam results are presented.

### 4. Cloud

Calculations took place using Amazon Web Services Elastic Computing (EC2) offer, more precisely: c5.4xlarge instance (16 CPUs, 32 GiB Memory). The cluster was located in US East (N. Virginia) region. In order to solve some licensing problems with CFDDirect, the connection with the cloud took place using Docker Container so that CFD calculations could have been performed.

### 5. Results and summary

Pressure ( $p$ ), temperature ( $T$ ) and velocity magnitude ( $U$ ) contours are shown below for the few most important time steps ( $t$ ), including: start time and the time steps just before and just after shock wave being reflected.



Figure 5.  $p$  ( $t=0.00005$  s)



Figure 6.  $U$  ( $t=0.00005$  s)

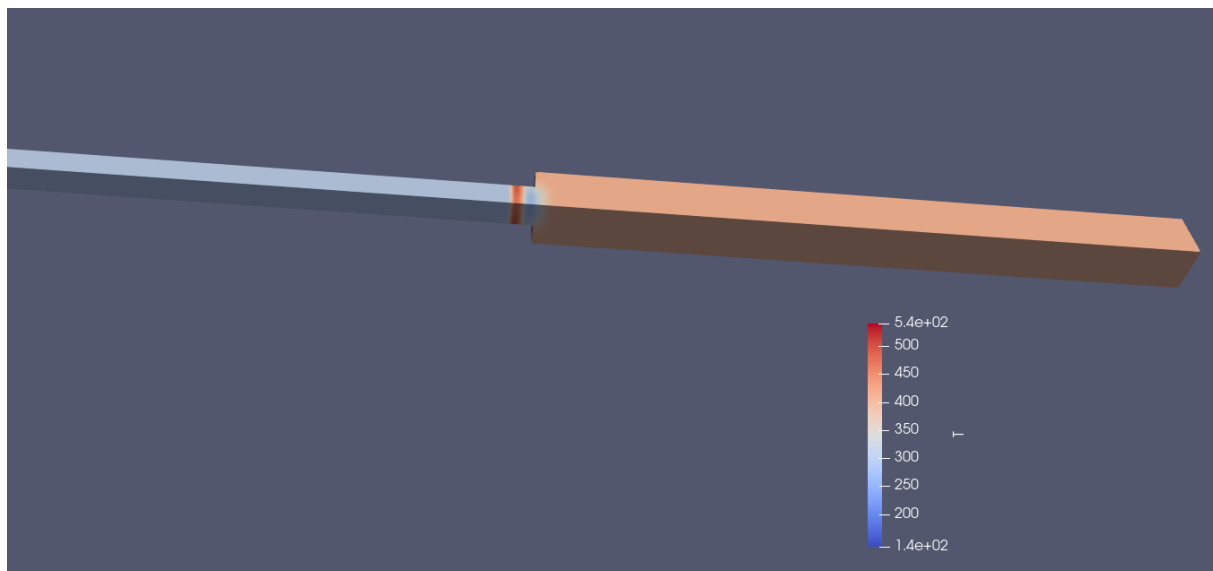


Figure 7.  $T$  ( $t=0.00005$  s)

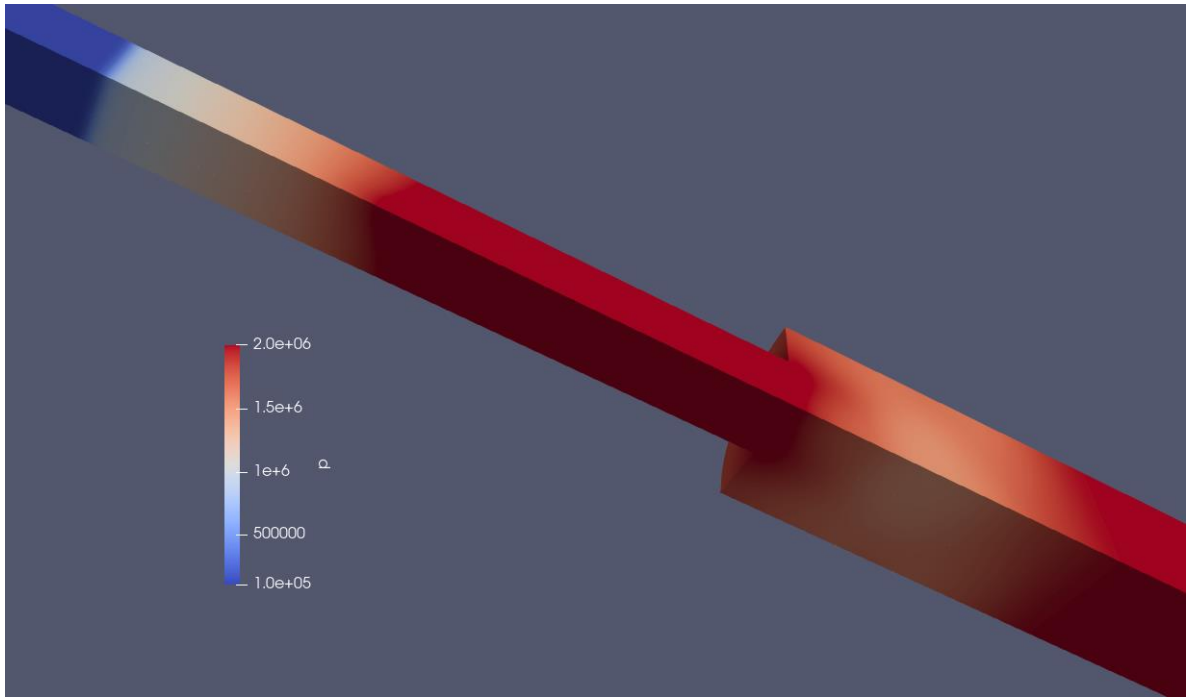


Figure 8.  $p$  ( $t=0.0005$  s) – detail view

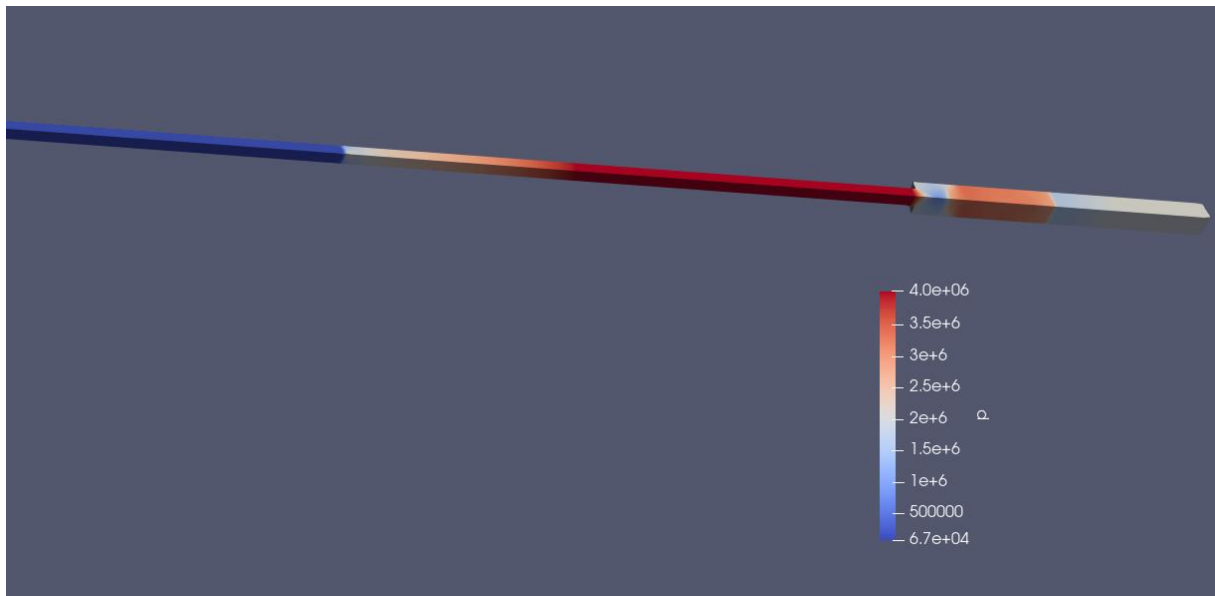


Figure 9.  $p$  ( $t=0.00165$  s) – shock and expansion wave fronts are clearly visible

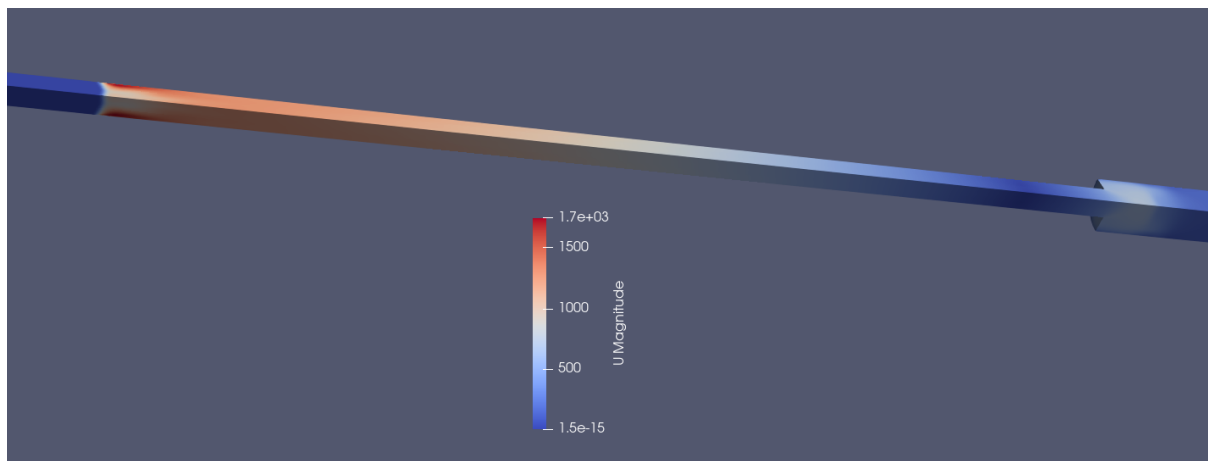


Figure 10.  $U$  ( $t=0.00165$  s)

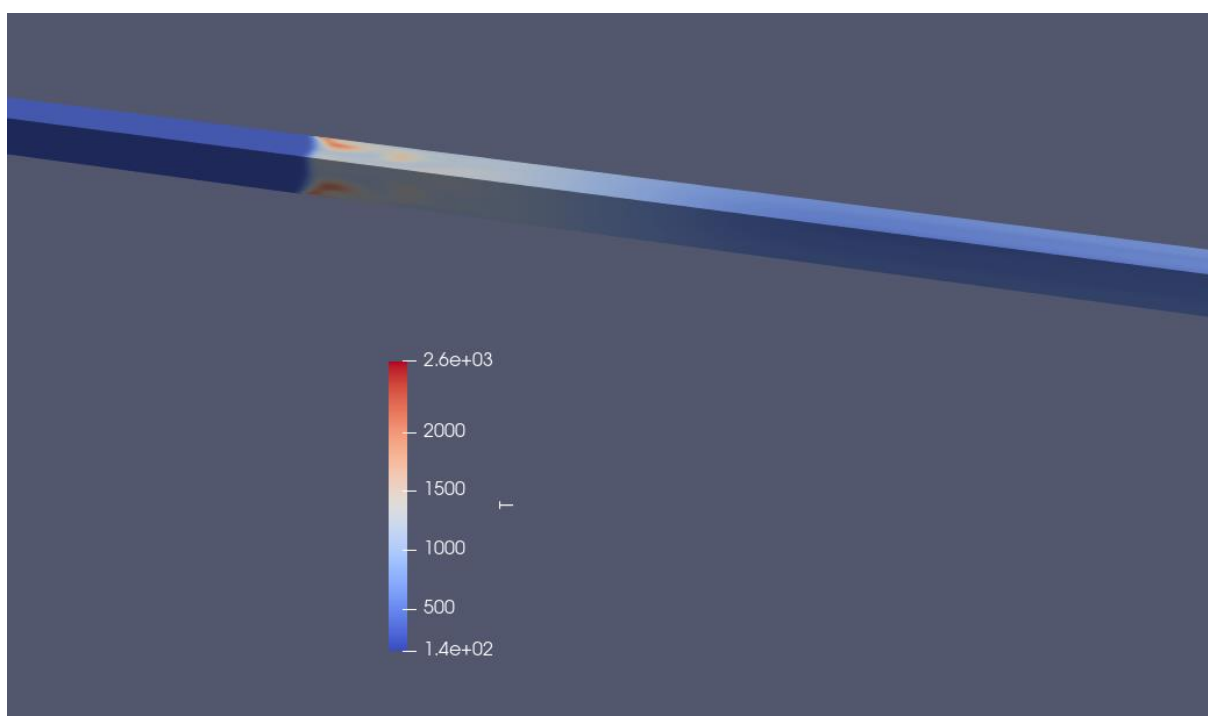


Figure 11.  $T$  ( $t=0.00165$  s) – temperature in the front of the shock wave

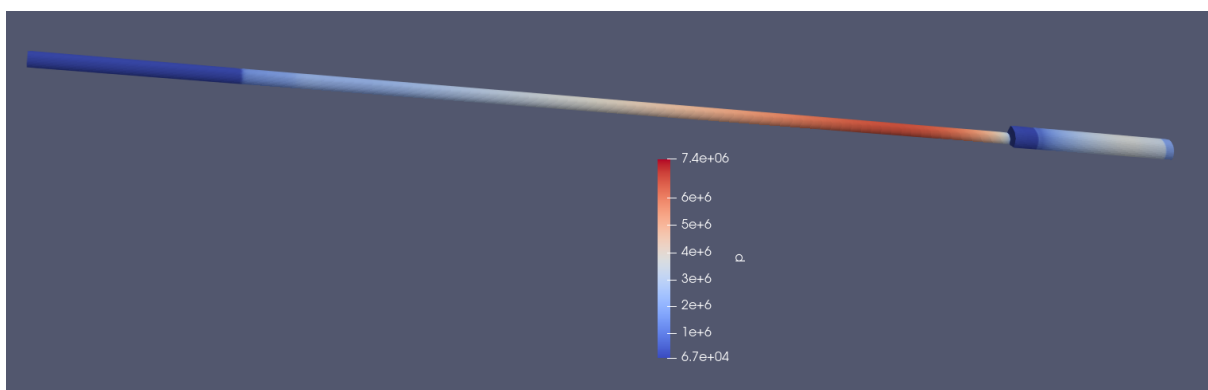


Figure 12.  $p$  ( $t=0.0027$  s)

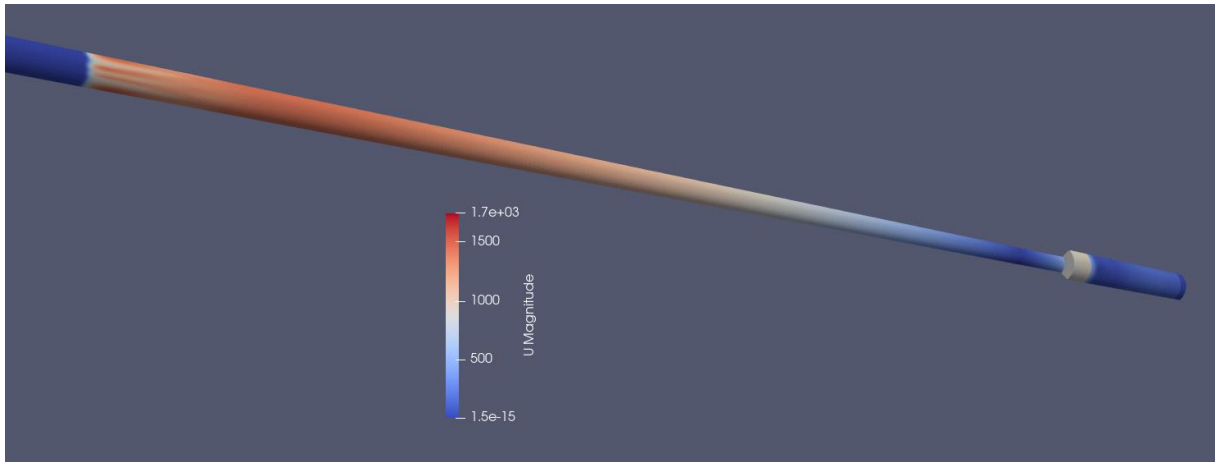


Figure 13.  $U$  ( $t=0.0027$  s)

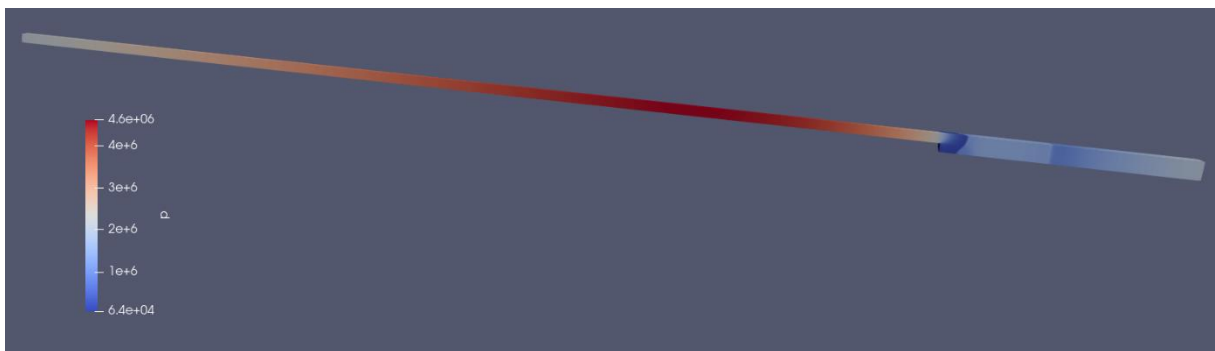


Figure 14.  $p$  ( $t=0.0042$  s) – the wave has just collided with the end of the tube

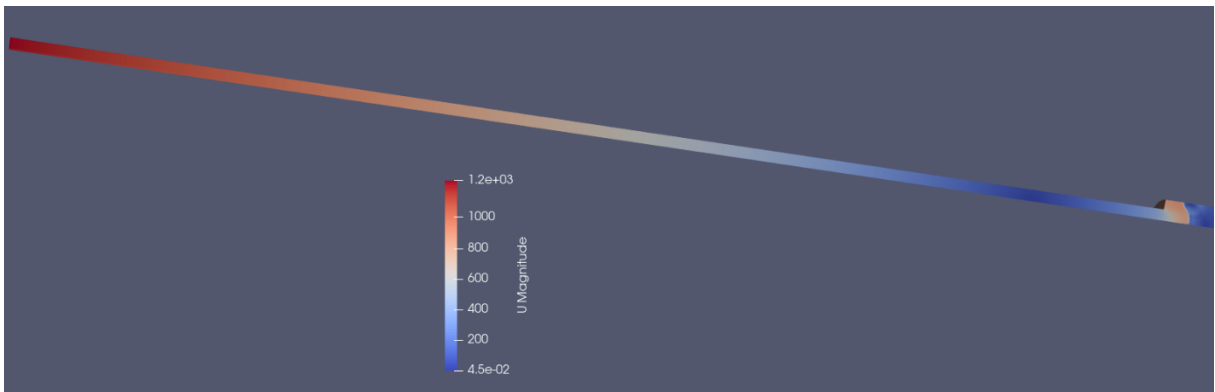


Figure 15.  $U$  ( $t=0.0042$  s)

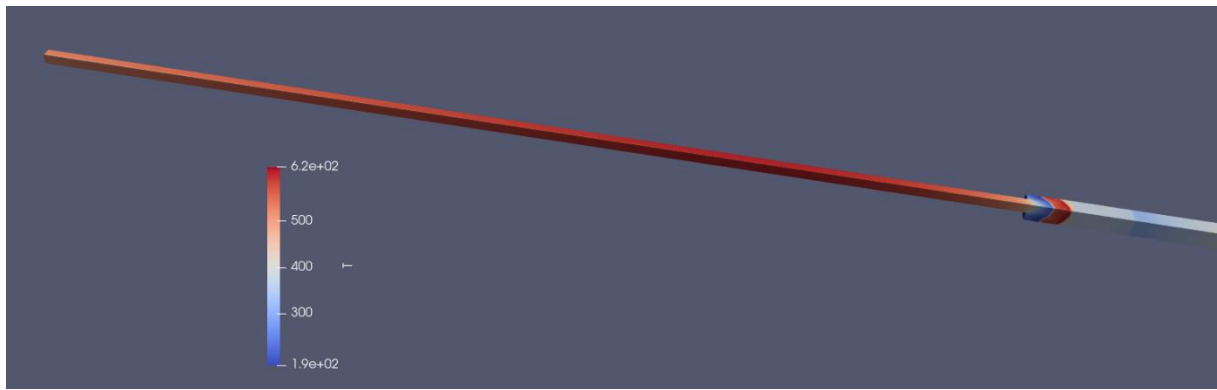


Figure 16.  $T$  ( $t=0.0042$  s)

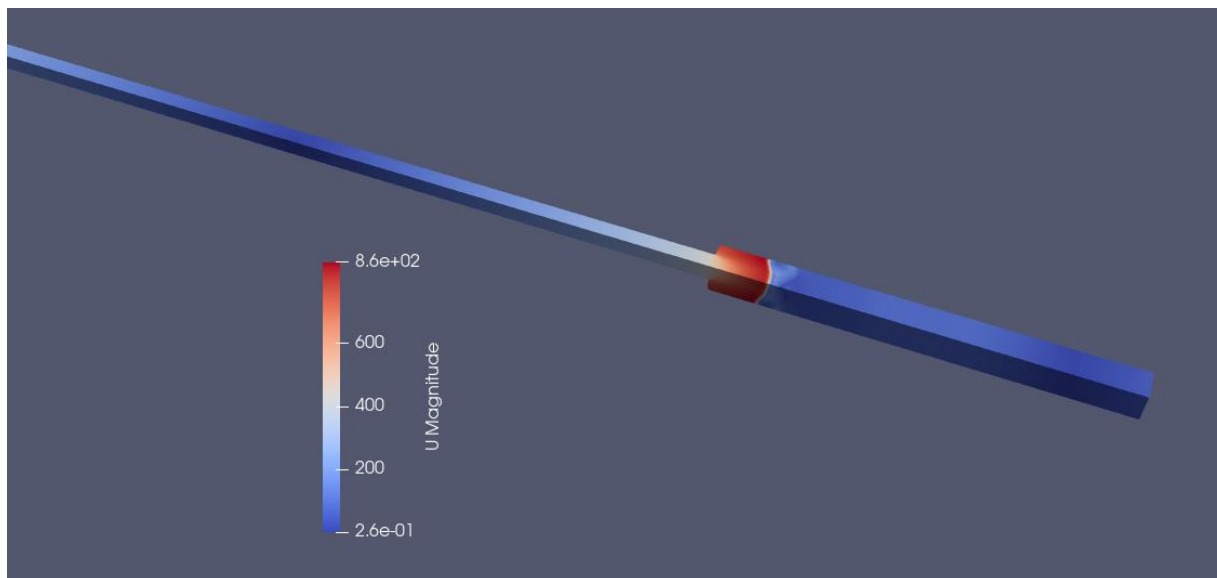


Figure 17.  $U$  ( $t=0.00975$  s) – reflected shock wave is entering the driver section again

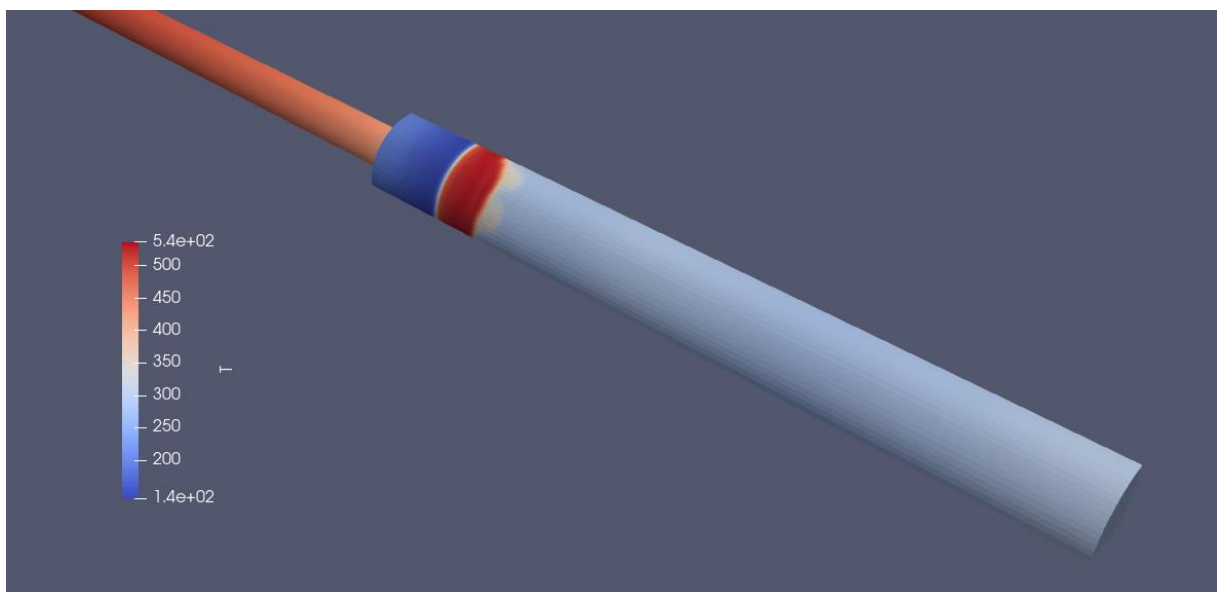


Figure 18.  $T$  ( $t=0.00975$  s)



In the beginning, an internal shock wave in the driver section (due to the diameter difference) can also be observed.

## **Resources**

- 1) Official OpenFOAM tutorials, e.g.:  
<https://develop.openfoam.com/Development/openfoam/-/tree/master/tutorials/compressible/sonicFoam/laminar/shockTube>
- 2) Jozsef Nagy YouTube tutorials, e.g.:  
<https://www.youtube.com/watch?v=KznljrgWSvo>