Simulation of Shock Tube problem using pisoCentralFoam

Sathish Kumar K
, OpenFOAM team, FOSSEE Project, IIT Bombay

***Abstract*—OpenFOAM has became a powerful Opensource CFD tool for the research work in different fields of fluid dynamics. But it lacks detailed documentation for the solvers made and to use different tutorials which comes with the installation. For this purpose the current work tends to illustrate to setup and run the case of "Simulation of Shock Tube problem using pisoCentralFoam" and compare the results with analytical solution.**

***Keywords*—Shock tube, pisoCentralFoam, OpenFOAM.**

I. Introduction

Numerical solution of compressible flows demands a solution in a wide range of Mach numbers. The schemes such as Kurganov-Tadmor's scheme (KT), AUSM+ scheme etc are used to ensure the same by considereing the monotonocity in disscontinuities. There is also a range of semi-implicit methods such as PISO, SIMPLE, PIMPLE, for solving subsonic problems are developed to simulate high Mach number flows. The inconvenience of these methods consists in occurrence of numerical oscillations in the regions of flow properties discontinuities, that take place in high-speed flows. Hence hybrid solvers were developed. pisoCentralFoam is one of hybrid solver which uses PISO algorithm and AUSM+ scheme are employed. In this report it is explained how to use pisoCentralFoam to simulate supersonic flow over wedge at range of Mach numbers.

II. Mach number range

The pisoCentralFoam is tested with the rhocentralFoam solver and the maximum Mach number achieved is compared.

- Maximum Mach number for rhoCentralFoam : 5.0
- Maximum Mach number for pisoCentralFoam : 8.0

III. Governing Equations

The governing equations solved are mass, momentum and energy.

Mass Conservation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\mathbf{U}) = 0 \quad (1)$$

Momentum Conservation:

$$\frac{\partial \rho\mathbf{U}}{\partial t} + \nabla.(\rho\mathbf{U}\mathbf{U}) = \nabla.\Pi + \mathbf{F}_b \quad (2)$$

Energy Conservation:

$$\frac{\partial \rho\mathbf{e}}{\partial t} + \nabla.(\rho\mathbf{U}\mathbf{e}) = \nabla.(\Pi.\mathbf{U}) - \nabla.\mathbf{q} \quad (3)$$

IV. Shock Tube Theory

The shock tube is a long tube of uniform cross section and with uniform internal dimensions. The diaphragm separates the high pressure driver section from the low pressure driven or test section. The material of the diaphragm and its thickness are dictated by the pressure ratio between the sections. On abrupt rupturing of the diaphragm, pressure waves emanating from the diaphragm station coalesce to form the shock front which propagates in to the low pressure section. As the shock front moves into the low pressure section, a contact surface which is an imaginary line of separation between the driven and driver gases follows the shock front.
Simultaneously an expansion fan travels in to the driver section. The shock tube is a versatile experimental facility for the study of gaseous phenomena at elevated temperature and pressure.

GEOMETRY

Length of the Shock Tube = 0.3 m
Location of Diaphragm = 0.15 m
Breadth of the Shock Tube = 0.06 m

CONDITIONS

Pressure at Driver section = 300000 Pa
Pressure at Driven section = 30000 Pa
Temperature at Driver and Driven section = 288 K
Fluid at Driver and Driven section = Air

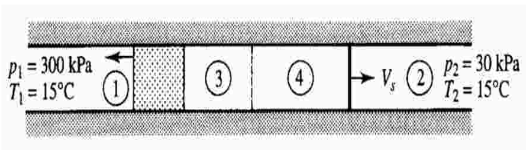

Fig. 1: Schematic of ShockTube

A. *Steps followed to do Simulation*

1) Create the mesh of shock tube geometry by running blockMesh command.
2) Setup initial conditions of pressure, velocity and temperature in the driver and driven section by running "setFields" command.
3) Editing the thermophysicalProperties file in the constant folder.

   Molecular weight = 28.96
   Specific heat at constant pressure = 1005 J/kg-K
   Dynamic viscosity = 0 (inviscid case).
   Prandtl number = 1
4) Modify the controlDict file.
5) Run the pisoCentralFoam solver.
6) Postprocess the results.

B. *Meshing*

The meshing for this simulation was done using the OpenFOAM Mesh utility blockMesh.

Total Number of cells	1000

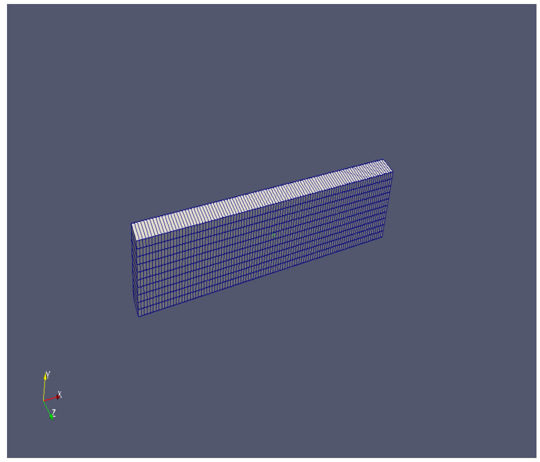

Figure 3: Wedge Mesh

RESULTS

The variation of pressure, velocity, temperature and density are compared with the analytical solution at t=0.00025 sec.

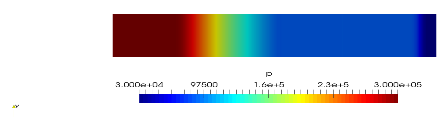

Fig. 2: Pressure contour across the shock tube

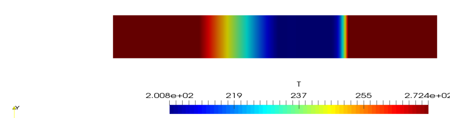

Fig. 3: Temperature contour across the shock tube

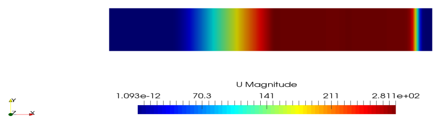

Fig. 4: Velocity contour across the shock tube

The pressure variation across the shock tube is compared with the analytical result and shown below

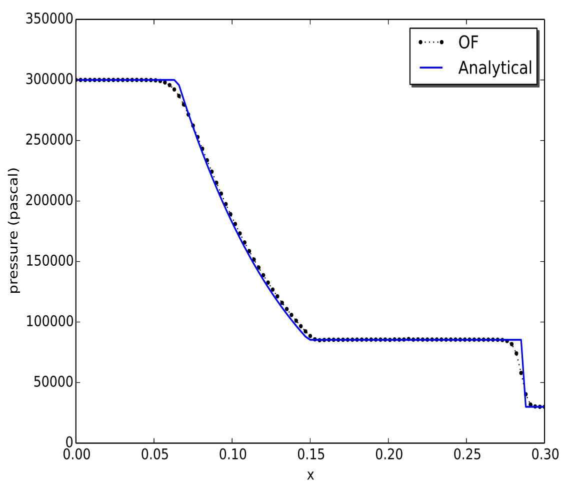

Fig. 5: Pressure variation across the shock tube

The density variation across the shock tube is compared with the analytical result and shown below

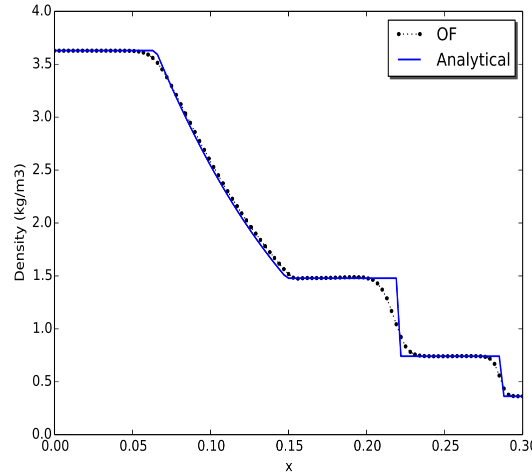

Fig. 6: From Literature [1]

The temperature variation across the shock tube is compared with the analytical result and shown below

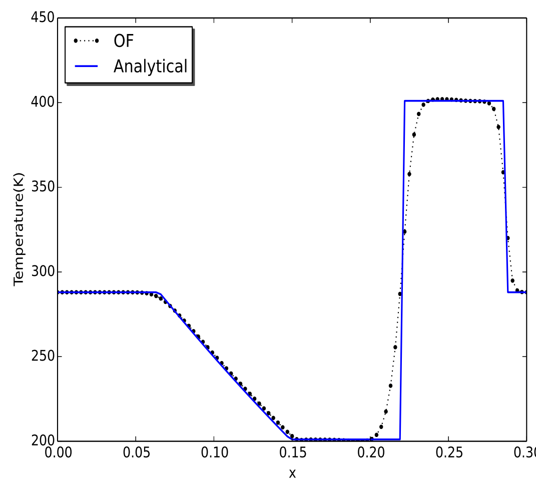

Fig. 7: Simulation Result

The velocity variation across the shock tube is compared with the analytical result and shown below

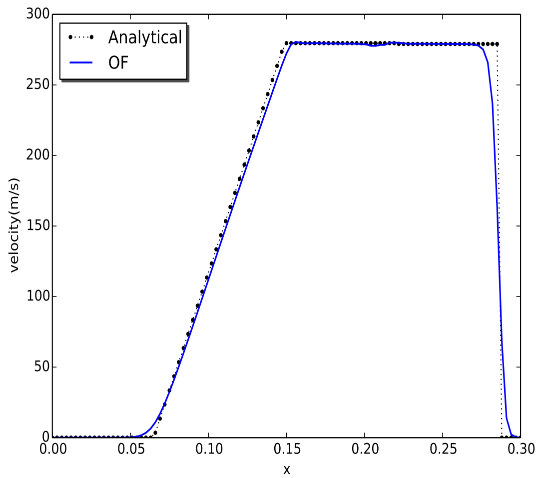

Fig. 8: From Literature [1]

V. Conclusion

The shock tube case which is the basic test case for any compressible flow solvers is simulated using pisoCentralFoam solver using OpenFOAM. The python script is written to calculate and plot the analytical results of shock the problem and compare with the simulation results, which shows good agreement. The various numerical schemes can be modified in fvSchemes and tested using this case to apply the solver for complex problems.

VI. Reference

1) "Compressible Fluid Flow", Patrick H.Oosthuizen, William E.Carscallen,The McGraw-Hill Companies, Inc.