Simulation of Supersonic flow over wedge 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 "Supersonic flow over simple wedge" using pisoCentralFoam and compare the results with analytical solution.**

Keywords*—*Wedge, 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:

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

Momentum Conservation:

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

Energy Conservation:

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

IV. Wedge Case

Geometry

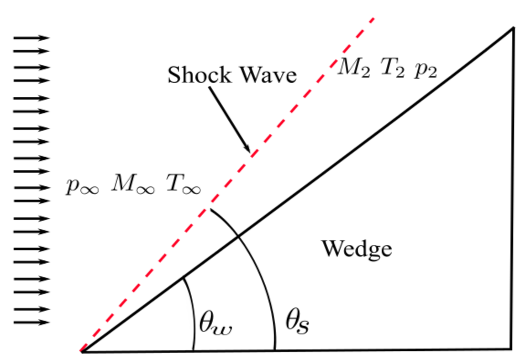

Fig. 1: Wedge Configuration

Conditions

Mach Numbers	1.65,1.75,2,2.25,2.5,3.0,3.5
Semi wedge angle	15 deg
Freestream Pressure	100000 Pa
Freestream Temperature	270 K
Fluid used	Air

A. Steps followed to do Simulation

1) Create the mesh of wedge geometry by running blockMesh command.
2) Setup initial and boundary conditions in the 0 folder. (p,U,T).
3) Editing the thermophysicalProperties file in the constant folder.

   Molecular weight = 28.96
   Specific heat at constant pressure = 1004 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	48000

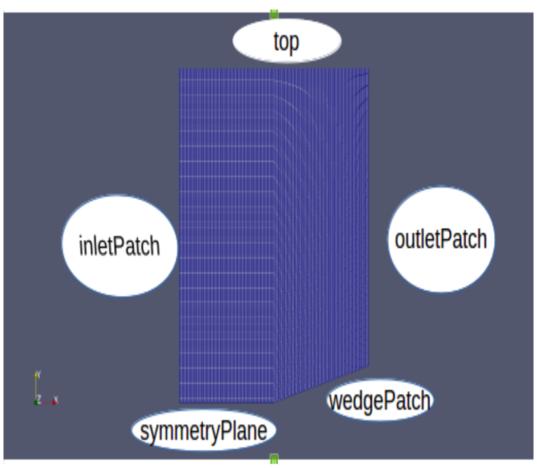

Figure 3: Wedge Mesh

Boundary Conditions Used

	p	U	T
inlet	freeStream	freeStream	freeStream
outlet	waveTransmissive	zeroGradient	zeroGradient
top	freeStream	freeStream	freeStream
wedge	zeroGradient	noSlip	zeroGradient
symmetryPlane	symmetryPlane	symmetryPlane	symmetryPlane

Results

The Mach number after the shock is taken from the paraFoam and plotted with respect to the freestream Mach numbers. The same is compared with the analytical solution.

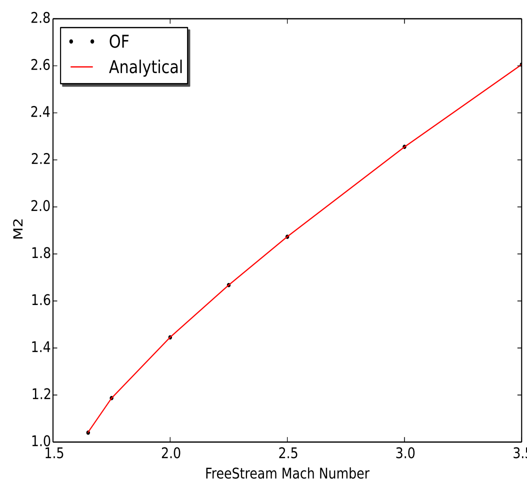

Fig. 2: Simulation Result

The plot of after shock Mach number plotted with respect to the freestream Mach numbers from literature is shown.

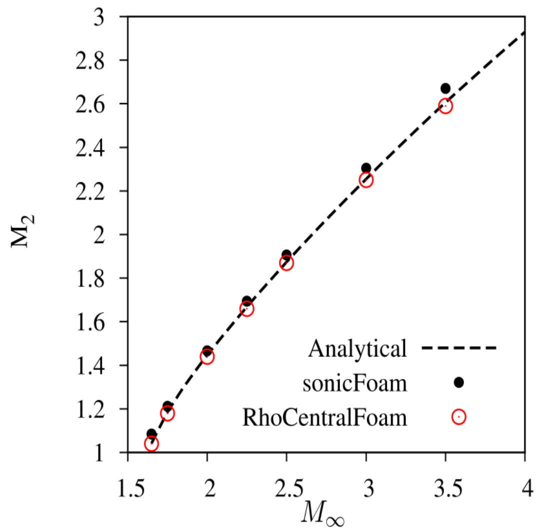

Fig. 3: From Literature [1]

The pressure ratio across shock is taken from the paraFoam and plotted with respect to the freestream Mach numbers. The same is compared with the analytical solution.

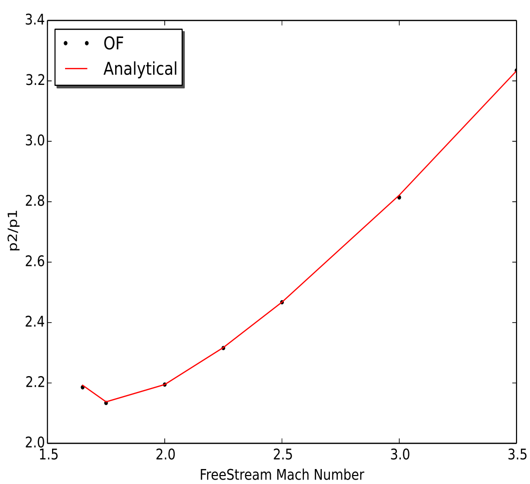

Fig. 4: Simulation Result

The plot of pressure ratio plotted with respect to the freestream Mach numbers from literature is shown.

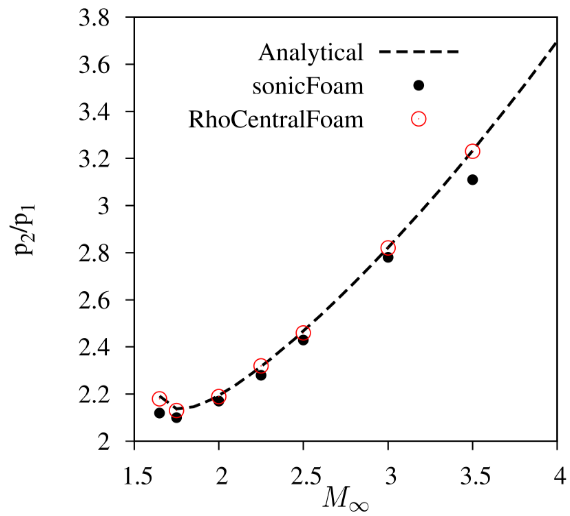

Fig. 5: From Literature [1]

The temperature ratio across shock is taken from the paraFoam and plotted with respect to the freestream Mach numbers. The same is compared with the analytical solution.

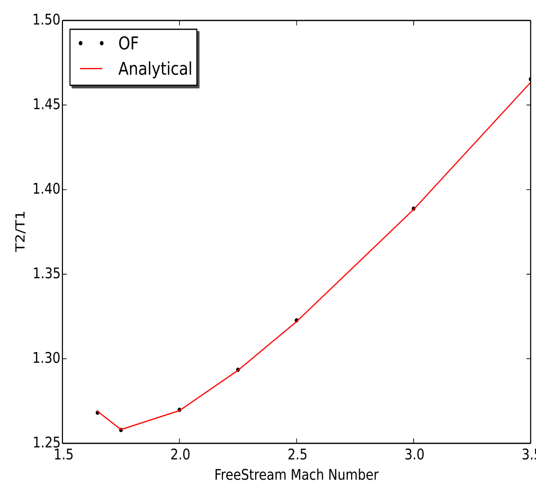

Fig. 6: Simulation Result

The plot of temperature ratio plotted with respect to the freestream Mach numbers from literature is shown.

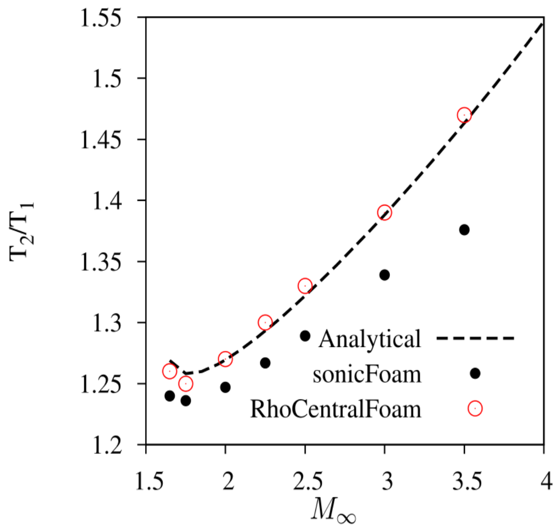

Fig. 7: From Literature [1]

The shock angle is measured from the paraFoam and plotted with respect to the freestream Mach numbers. The same is compared with the analytical solution.

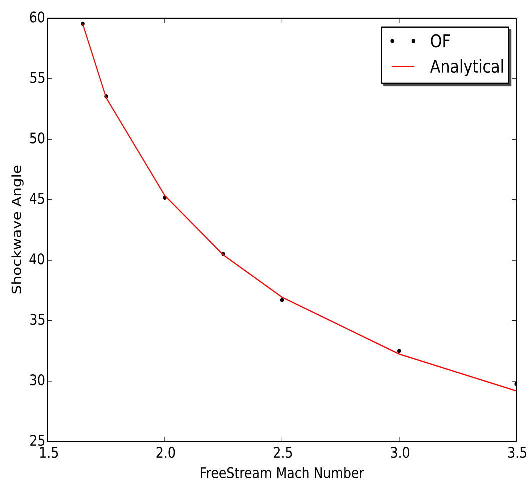

Fig. 8: Simulation Result

The plot of variation of shock angle with respect to the freestream Mach numbers from literature is shown.

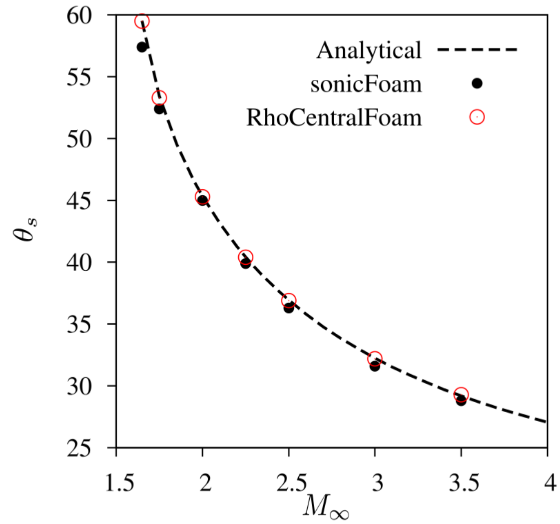

Fig. 9: From Literature [1]

V. Conclusion

The Supersonic flow of wide range of Mach numbers is simulated over a simple wedge case and compared with the analytical results. It matches with the anlaytical results and also with the literature [1] which shows the results obtained from the sonicFoam and rhoCentralFoam. The maximum Mach number achieved is compared and pisoCentralFoam works fine with wide range of Mach number from incompressible range to Mach number upto 5.0, which allows to apply this for various cases.

VI. Reference

1) "High Speed Flow Simulation using OpenFOAM", Luis F.Gutierrez Marcantoni,Jose P.Tamagno and Sergio A.Elaskar, Mecanica Computacional Vol XXXI, pags. 2939-2959
2) Used Compressible Aerodynamics Calculator 3.0 from Virginia Tech for Analytical Calculations