Supersonic Flow over Multiple Wedge

Sahil Deepak Kukian

B.Tech in Aeronautical Engineering

Manipal Institute of Technology, Manipal

Abstract

The objective of the present project is simulate and study the shock on shock interaction formed due to the multi-angle wedge in supersonic flow conditions using the OpenFOAM. The geometry and mesh are created in ANSYS. Also the patch conditions and other necessary changes that should be done to the case setup are going to be shown in this project. The compressible solver 'rhoCentralFoam' is going to be used to run this simulation. The Results will then be analyzed and future experiments will be proposed.

1. Introduction

Supersonic flow is characterized as flow that is above 1.2 mach. For this project we are going to study the shock interaction at supersonic flows with a double wedge. For this we use the rhoCentralFoam Solver to compute and we use ParaView for viewing the results. One of the major issues with solving supersonic flows is the adverse temperature and pressure gradients that are caused due to shock formation making the solution extremely difficult to converge, all the issues and short coming of the project are going to be discussed in detail in the following chapters. We have used a double wedge of similar design as show in the fig 1 below. This figure also shows the formation of the oblique shock and the bow shock due to the second wedge.

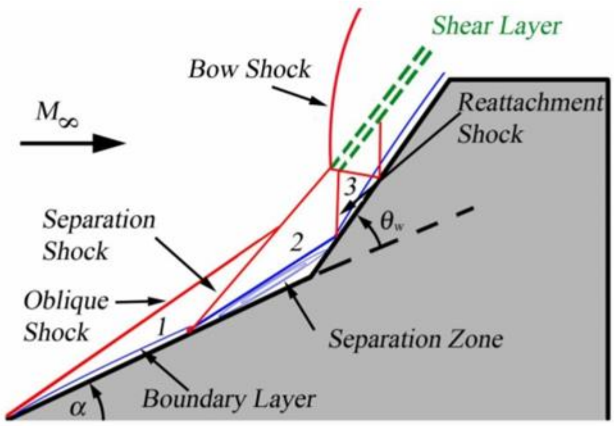


Fig 1 Double Wedge Design and shock interaction Diagram [1]

2. Problem Statement

The problem considers a supersonic flow of air at Mach 7.4 over a double wedge having different angles. The Free Stream properties for temperature are 696K and pressure is 8.13kPa. For this we are going to be using the rhoCentralFoam solver

3. Governing Equations

Conservation of Mass equation is followed directly from the control volume equation, by applying Gauss Divergence theorem, we can transform the surface integral into a volume integral finally becoming the Equation shown below

$$\frac{\partial \rho}{\partial t} + DIV(\rho v) = 0 \tag{1}$$

The Navies-Stokes equation in integral form is given by the equation below

$$\int_S \rho v \,. ndS = 0 \tag{2}$$

where ρ is pressure

v is velocity

n is the unit outward normal to the control volume

The angle of the oblique shock that is formed when the flow comes into contact with the wedge is given by the θ-β-M relation and it specifies θ as a unique function of M and β. This relation is given below

$$\tan\theta = 2\cot\beta\left[\frac{M^2\, sin^2\beta - 1}{M^2(\gamma + cos2\beta) + 2}\right] \tag{3}$$

where θ is the angle of the wedge

β is the angle of the oblique shock

M is the Mach number

From this relation we can see that for any Mach number, there exists a θ_{max} (maximum deflection angle), such that if $\theta > \theta_{max}$, then there would be no solution for oblique shock.

4. Case Setup

4.1 Geometry and Mesh

The geometry for this case study consists of inlet, outlet, topwall, bottomwall and finally the obstacle (double wedge). The total length of the model is 250cm and 120 cm in breadth. The angle of the first wedge is 30^0 and second wedge is 25^0. The total length of the wedge is given as 101.6cm. Since we want to simulate only the 2D simulation for this case but OpenFOAM operates only in 3D, we assign a thickness of 1cm. The mesh can be seen in Fig 2. It consists of 31000 hexahedral Mesh elements made using Ansys meshing tool. The mesh was exported into .msh format and then converted into OpenFOAM readable mesh by using the built-in function "fluentMeshToFoam ".

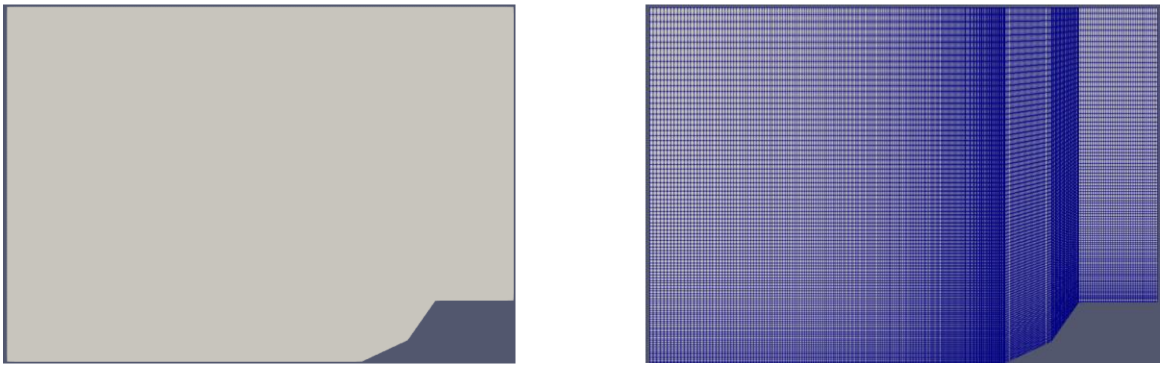


Fig 2 Geometry (left) and Mesh (right)

4.2 Boundary Conditions

The boundary conditions used for the patches are as shown below in Table 1.Selecting boundary conditions was one of the most difficult part of the simulation as incorrect selection will lead to the model diverging and not give a good result. Before being able to make the changes in the boundary conditions the necessary changes to the topwall, bottomwall and the defaultFaces must be done in the polyMesh folder after importing the mesh into OpenFOAM format. All the others should be changed to patch. Inlet Velocity is kept at 2664 m/s i.e. Mach 7.4.

The Temperature at inlet is 696K and the Pressure is 8.13kPa.

Boundary Name	U	T	P
Inlet	fixedValue	fixedValue	fixedValue
outlet	inletOutlet	inletOutlet	waveTransmissive
topWall	supersonicFreeStream	inletOutlet	zeroGradient

bottomWall	supersonicFreeStream	inletOutlet	zeroGradient
obstacle	slip	zeroGradient	zeroGradient
defaultFaces	empty	empty	empty

Table 1 Boundary conditions for rhoCentralFoam

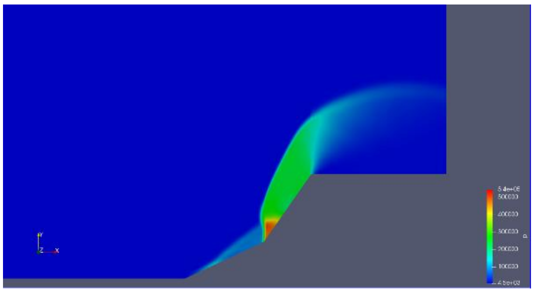
4.3 Solver and Simulation Controls

“rhoCentralFoam” is used for this simulation with the turbulence model set to Laminar. The time step is set to 1e-5 seconds and the simulation is run for 0.02 seconds. Other limiters that have been applied into the control dict file are maxCo set to 1 and also maxDeltaT set to 1e-6.

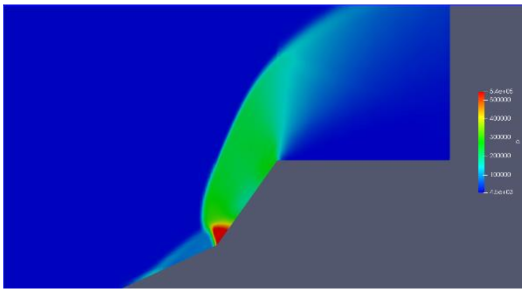
5. Result and Analysis

5.1 Pressure Contours

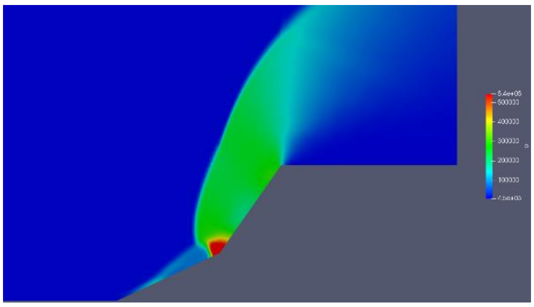
In Fig 3 we can see the pressure contour at various different time steps. The change in pressure is clearly visible as the supersonic flow interacts with the wedge. After 0.0004s we can see the oblique shock being formed at the first wedge as it satisfies the $\theta > \theta_{max}$ case in the Equation 3. However the second wedge does not satisfy this equation and conditions therefore a detached shock is formed.



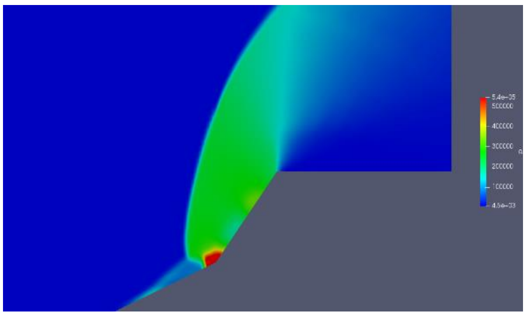
0.0004s



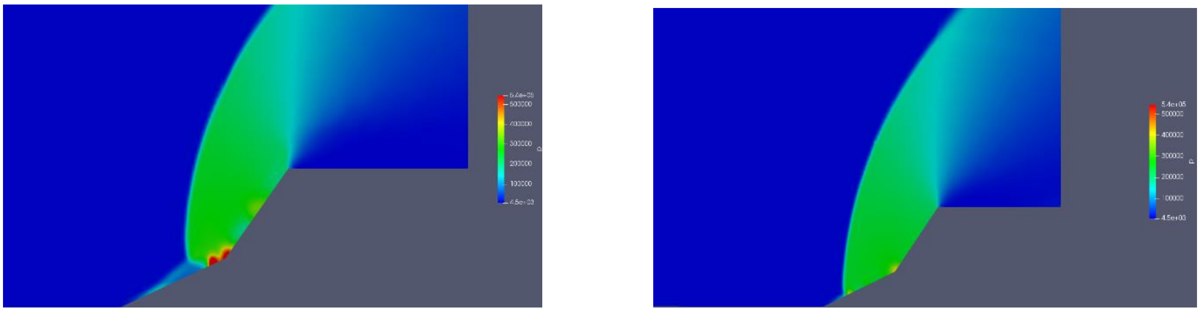
0.0007s



0.001s



0.0015s



0.002s 0.02s

Fig 3 Pressure Contour at various time steps

By 0.02 seconds the Case achieves steady state Condition.

As seen from the pressure contour a high pressure region (shown in green color) behind the bow shock can be seen, this is due to compression effects of the shock wave. This very principle is being used in the hypersonic propulsion systems to compress the air before it enters the engine.

Comparing the simulations results with the Schlieren images taken from A.B Swantek et al (2012)[1] . We can see the similarities between the two cases, we can clearly see the formation of the oblique shock, separation shock and the separation zone in the density contour (Refer to Fig 1 for detailed diagram).

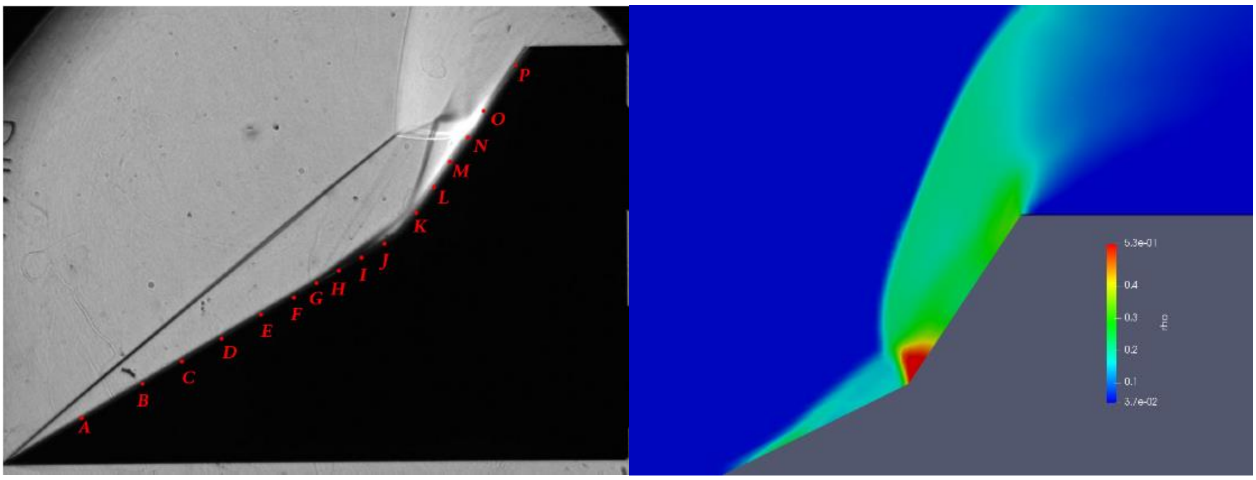


Fig 4 Schlieren image (left) and Density contour (right) at 0.0007s

6. Conclusion

This case study has explored the supersonic flow interaction between a multi angle wedges. This can also be taken forward into developing SCRAM jet inlets where majority of the compression happens outside the engine using a similar principle of the multiple wedge. Further research can be done by testing the design at different Mach No. and other parameters.

References

1. A.B Swantek, J.M Austin (2012), "Heat Transfer on a double Wedge geometry in Hypervelocity Air and Nitrogen Flows", 50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition.

2. Anderson, J.D., Modern Compressible Flow, McGraw Hill Inc., New York, 1984.

3. J. H Perziger , M .Peric , Computational Methods Of Fluid Dynamics ,Springer, ISBN 3-540-42074-6 Springer-Verlag Berlin Heidelberg NewYork