Project Report

ON

Numerical Simulation of shockwave propagation through a Y-Split

Submitted to

Prof. Dr Ankit Bansal

As a project report for practical assignment given in course

MIN-345 Compressible Flow

By

Putluru Deshik Reddy – 18119026

Shreya Agarwal – 19119037

Rathod Snehith Raj - 19117095

DEPARTMENT OF MECHANICAL AND INDUSTRIAL ENGINEERING

Date of submission: 14-11-2021

1 Introduction

In the search for protection from explosions, a variety of underground shelters were studied to minimize the risks related to the propagation of shock waves in closed areas as shown by [4] and [5]. An important feature in designing such protection is the knowledge of shock or blast wave propagation in ducts leading to the shelter. This knowledge is also of importance for safety pre- cautions in mines, tunnels, or corridors after an explosion for protecting humans and materials in case of sudden explosions. The propagation of a planar shock wave in a complex ducts system can create serious personal injury and property damage due to numerous reflections that generate local zones of dangerous high-pressure.

2 Theory

Reflection of shockwaves is the most common phenomenon seen in this type of modeling. With multiple reflections from the bottom and top wall when passing through angled channel, the turbulence created dissipates the shock energy, thus weakens the shock. Further at the end wall reflection, there is a very significant loss, (at least 50% loss) which will be proved from the simulaiton below.
Mach reflection is also a common phenemenon observed in duct flow. When a shock travels in a passage, the reflection from the top wall, reflection from the bottom wall and the mach stem together coincides/makes up at a point which is called triple point, and termed as Mach reflection. Mach reflections are the biggest contributors of turbulence in the flow behind shocks.

3 Problem Statement

An experimental work was conducted by Marty et.al on the passage of shockwaves through a Y-Split channel. The same work is to be replicated using the numerical simulation tools and finally validate the framework. In the experiments, the angle of bifurcation is constant, but it can be easily parametrized in the simulations, so we also aimed to study the variation of behavior with respect to the angle of split.

4 Numerical Modeling

The flow field can be modelled by the Euler Equations that express the conservation of mass, momentum and energy for an inviscid compressible fluid obeying perfect gas equation of state. For a two-dimensional flow, the governing equations, expressed in cartesian coordinate system, are:

The continuity equation,

$$\frac{\partial \rho}{\partial t} + \frac{\partial (u_i \rho)}{\partial x_i} = 0$$

Inviscid momentum equation

$$(\frac{\partial \hat{u}_i}{\partial t})_I + \frac{\partial (u_i \hat{u}_j)}{\partial x_j} + \frac{\partial p}{\partial x_i} = 0$$

The energy equation,

$$(\frac{\partial \hat{E}}{\partial t})_I + \frac{\partial [u_k(\hat{E} + p)]}{\partial x_k} - \frac{\partial}{\partial x_i}\mu.u_j(\frac{\partial u_i}{\partial x_j} - \frac{2}{3}\frac{\partial u_k}{x_k}\partial x_i \delta_{ij}) = 0$$

From $\hat{E}$ the temperature T is calculated through,

$$T = \frac{1}{C_v}(\frac{\hat{E}}{\rho} - \frac{u_k u_k}{2}).$$

All the equations are initially guessed, then corrected using a corrector term added later after first iteration. All these equations are enclosed in the rhoCentralFoam solver with customized finite volume schemes. We have chosen the Euler Implicit scheme and upwind schemes for the calculations.

5 Computational Domain

The geometry was made using the blockMesh utility which is a simple tool comprises of only hex elements. We defined few lines of code to specify the number of blocks, grid size, cell density. The same dictionary also contains the face type specification such as symmetry Plane, Axis Symmetry, patch, wall etc. As shown in the figure, the mid line which bisects the entire geometry is defined as symmetryPlane and all the

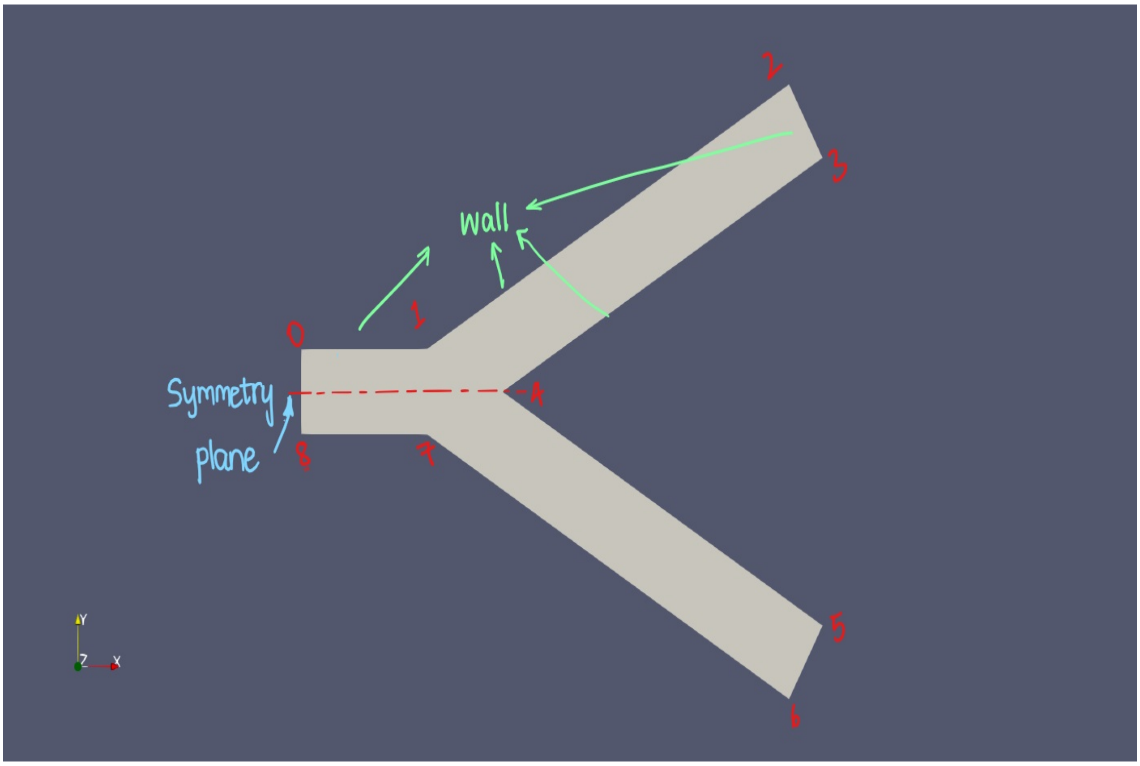

Fig. 1. Solid geometry showing all the boundary types and symmetry plane. (Only one half is considered for computation and later reflected the same results about the symmetryPlane)

faces 0-1, 1-2, 2-3, 3-4 and the S-0 are defined as patches and later specified the wall boundary conditions in the '0' folder.

The remaining faces which face towards the viewer and away (front and back walls) are considered as empty which makes the simulation 2D.

For the discretization, the same blockMesh itself also does the meshing. The grid size is specified just after defining the block, the grid sizes for the main channel block is 40x150, the bifurcation region is 40x80 and the end spitted channels has grid of 80x500.

6 Boundary Conditions

Both the initial and domain related boundary conditions were specified as follows, *

Driver gas properties -

Type	hePsiThermo
Equation of State	perfectGas

Contnd.

---

*All the code files with test cases can be found here - Link

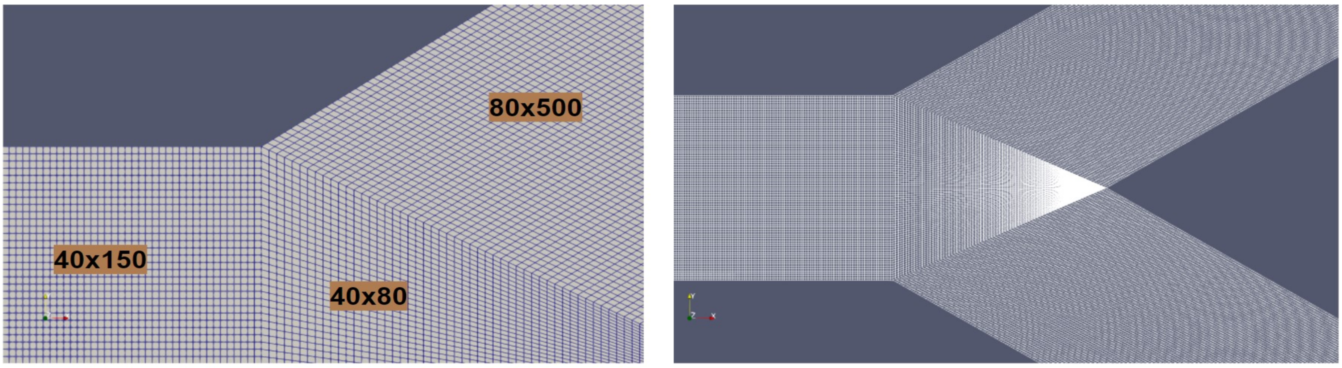

Fig. 2. Snapshot of grid showing all the three blocks. Left-Main channel, Center-Bifurcation region, Right-split channels

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      p;
}
// * * * * * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * * * * //

dimensions      [1 -1 -2 0 0 0 0];

internalField   uniform 0;

boundaryField
{
    sides
    {
        type            zeroGradient;
    }
    interfaces
    {
        type            symmetry;
    }
}

//
************************************************
************************* //
```

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      T;
}
// * * * * * * * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * //

dimensions      [0 0 0 1 0 0 0];

internalField   uniform 1;

boundaryField
{
    sides
    {
        type            zeroGradient;
    }
    interfaces
    {
        type            symmetry;
    }
}

//
****************************************************
********************* //
```

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * * * * *
* * * * * * * * * * * * //

dimensions      [0 1 -1 0 0 0 0];

internalField   uniform (0 0 0);

boundaryField
{
    sides
    {
        type            zeroGradient;
    }
    interfaces
    {
        type            symmetry;
    }
}

//
**************************************************
*********************** //
```

Fig. 3. Code showing all the boundary conditions of Pressure, Temperature and velocity.

Energy	sensibleInternalEnergy
Molecular Weight	28.96
C_p (Sp.Heat)	1004.5
H_f (Sp.Enthalpy)	2.544e+06
mu	0
Pr (Prandtl Number)	1

7 Setup

The entire modeling was done using the OpenFoam v6 software and the post-processing with Paraview v5.9.1 tool. The solver for our case is required to be transient, density-based and able to capture shocks, all the jobs can be best performed by the "rhoCentralFoam". Following are the properties of the solver,

1. Compressible
2. Density-based
3. Transient
4. Shock capturing
5. Heat Transfer
6. Equations are solved using upwind schemes of Kurganov and Tadmor

No specific turbulence was selected of RANS and LES, the default laminar model works well for the case.

Test Cases:- On overall seven test cases are sufficient to study the bifurcation phenomenon and parametrization.

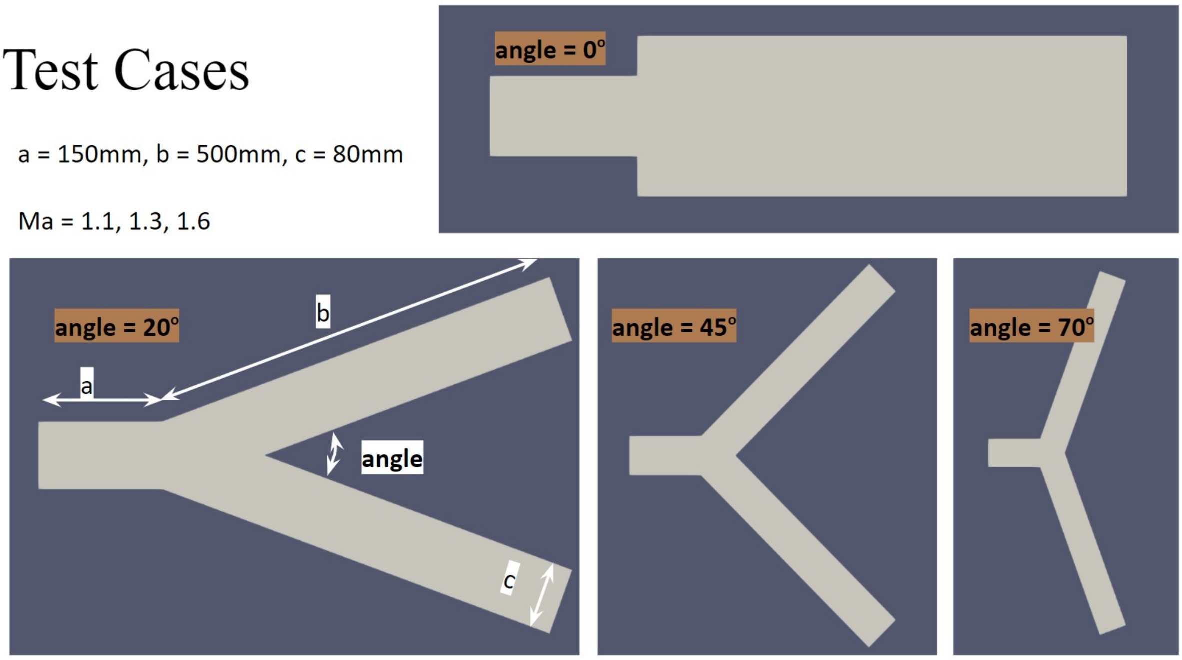

Fig. 4. All the different geometrical test cases, (Note: 90^0 case not shown)

8 Results

The above are the schlieren snaps captured from the experiments for Ma $= 1.12$ and Ma $= 1.36$.

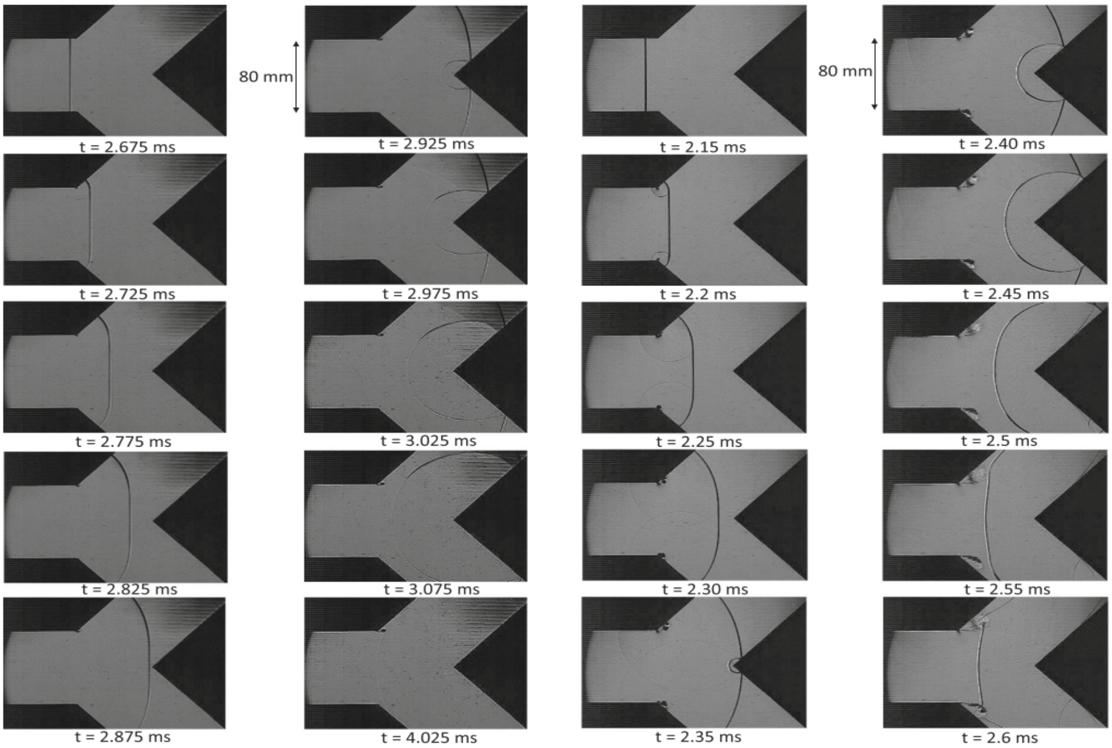

Fig. 5. Results of experiments conducted by Marty research group at incident Mach 1.12

The numerical schlieren is calculated as,

$$S_n = log_{10}[1 + |\vec{\nabla}\rho|]$$

The numerical results of schlieren S_n is shown below for Mach 1.1, Some of the key observations in the experimental results are, 1. Low pressure region formation 2. flow separation near the corner 3. Shock reflection at bifurcation 4. Mach reflection in the split channel 5. Turbulence generated behind shock after bifurcation. All the above except (2) are clearly captured in the numerical simulations. As seen in Fig. 6 the 4th picture from left in first row shows the low pressure region. The 3rd picture in the second row depicts the shock reflection and the next snap has multiple reflections, the triple point linking the reflected shock wave from the peak of bifurcation, the transmitted shock wave which propagates in the branch and the Mach stem can be

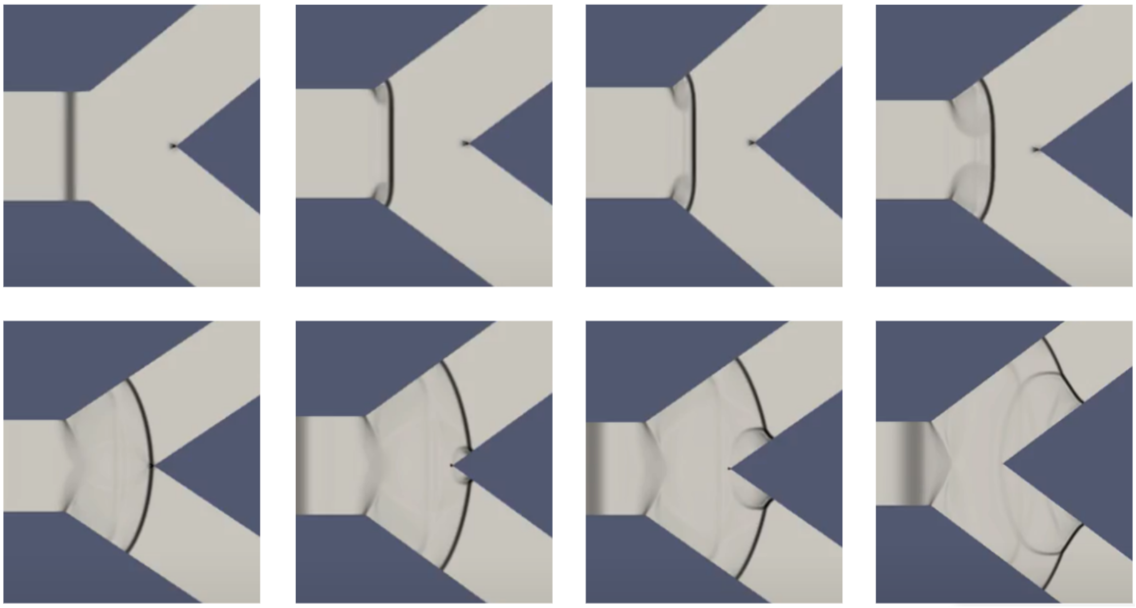

Fig. 6. Numerical schlieren results obtained from OpenFOAM simulation

seen, which is also called Mach reflection.

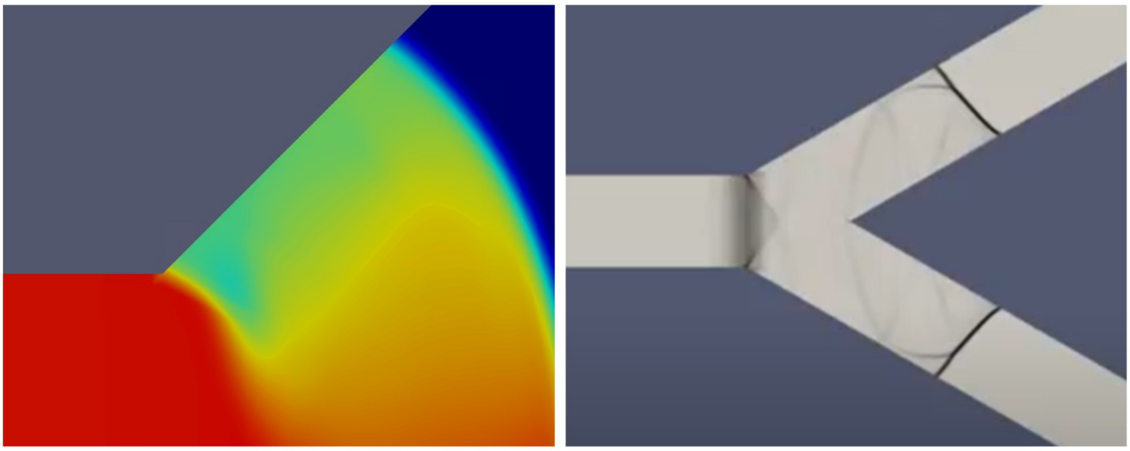

Fig. 7. L-R: a. Pressure contours showing low pressure region near the corner b. Mach reflection and Turbulent flow behind the shock wave

Pressure signals: The pressure signals are determined at the end wall in the experiments using pressure gauges. In the Fig.9 the right sided plot shows the experimental and numerical results, where the dark lines are of experiments and the pale red line depicts the numerical simulation results. The red and black lines are pressure signals of both the end walls in experiments, whereas the blue line represents the pressure signal as if there is no bifurcation. It is clear that the sudden rise in pale red plot is very close to the experimental results, thus validated. The left plot is the pressure vs time chart at a point just near the bifurcation. The small dip in pressure is associated with the shock reflection at the bifurcation edge and the sudden rise in pressure initially is due to the first shock incidence. The same data is plotted for the Mach 1.3 and Mach 1.6 as shown in Fig.10 and 11. [†]

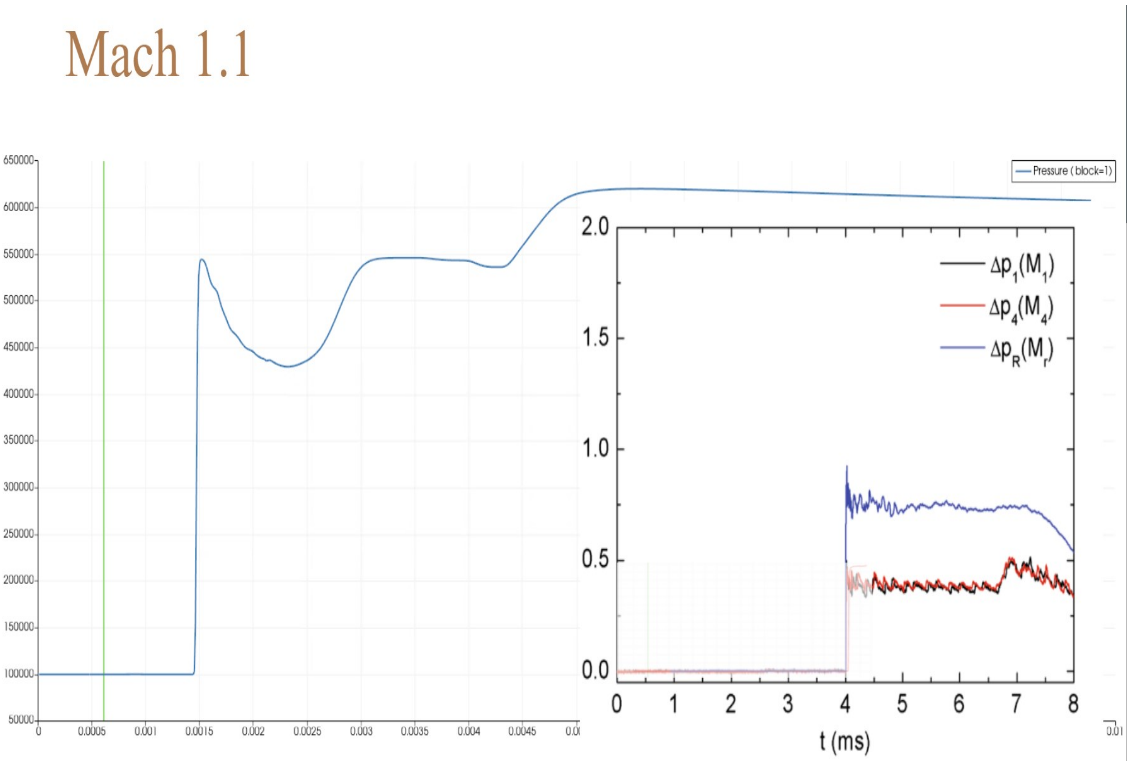

Fig. 8

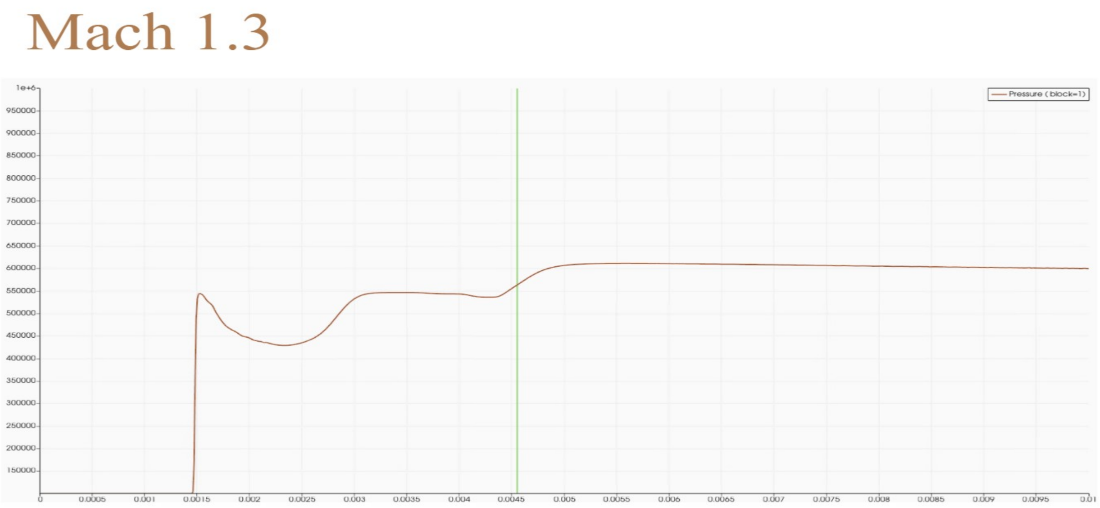

Fig. 9

[†] Here is the link for numerical schlieren animation movie - Link

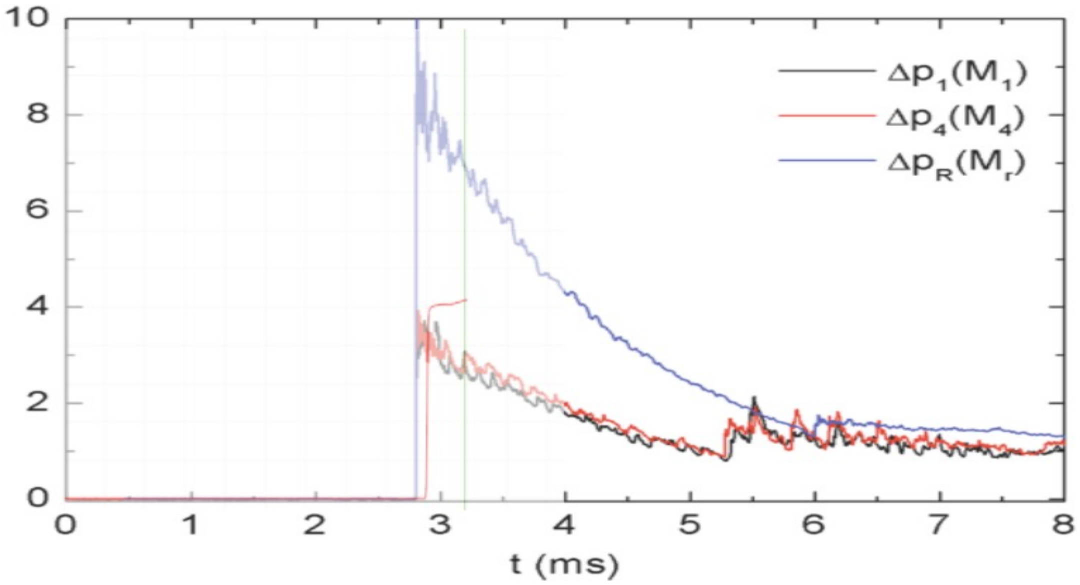

Fig. 10

Parameterization Study: The simulation is conducted by varying the angle of split at Mach 1.6. The angle of split is considered to be 0, 20, 45, 70, 90 degrees which makes 5 test cases. Following are the end wall overpressures recorded for each case,

Y Split Angle	Over Pressure
0	3.73
20	3.64
45	3.55
70	3.27
90	3.18

9 Conclusion

Pressure at the end wall and junction is studied with varying Mach Numbers and angle of ‘Y’. The numerical results are close to th experimental results with some error. Very less change in overpressure is observed with change in bifurcation angle. The overpressure is dependent only on the incident Mach Number and the area of cross section profiles. The pressure recorded at the end wall is less than half of the pressure as if there is no bifurcation. Hence, it can be considered as a better approach to attenuate shocks in case of duct systems.

10 References

1. Detailed simulations of shock-bifurcation and ignition of an argon-diluted hydrogen/oxygen mixture in a shock tube (Ihme, Matthias, Sun, Yong, Deiterding, Ralf)

2. Experimental and numerical investigations of shock wave propagation through a bifurcation (Marty, A.,Daniel, E. Massoni, J. Biamino, L. Houas, L. Leriche, D. Jourdan, G.)

3. The numerical simulation of shock bifurcation near the end wall of a shock tube (Weber, Y. S., Oran, E. S., Boris, J. P., Anderson, J. D.)

4. Shock wave propagation in a branched duct (Igra, O., Wang, L., Falcovitz, J., Heilig, W.)

5. Ben-Dor, G., Igra, O., Elperin, T.: Handbook of Shock Waves. Academic Press, NewYork (2000)