



## REPORT

# Mach 7 Hypersonic Ramp

Hareesh Chinthakuntla, Naveen Teja, Rajesh Ranjan \*

\*Corresponding author. Email: hareesh23@iitk.ac.in, bnteja20@iitk.ac.in

### Abstract

In this paper we will discuss about hypersonic flow over a ramp. A mach 7 flow over a ramp of  $15^\circ$  angle at a temperature  $15.6^\circ\text{C}$  and pressure  $101352.9\text{ Pa}$  was simulated using PRAVAH code generated in fortran language. The grid of the ramp was generated for different grid size and run the simulation. The code has run successfully giving an image of shock wave. The results of different grids were compared and the optimal grid is chosen considering computational time and the results for the further analysis. The pressure, density and velocity components, U velocity and V velocity contours are plotted.

**Keywords:** CFD, Hypersonic, Shock, Velocity, Pressure, Density

## 1. Introduction

With Supersonic and Hypersonic speeds playing very crucial rolls in space exploration, missile technology, military requirements, ramjet and scramjet engines etc. It is very important to study and understand the flow dynamics of hypersonic velocities. At hypersonic flows boundary layer plays a very major roll. It is very important to study the interaction of boundary layer and external flow. Because the external flow is in hypersonic regim and the flow on wall is having zero velocity there will be a very high velocity gradient in the boundary layer. Also there will be very low mass flux in the boundary layer and because of this the streamlines entering the boundary layer are almost parallel to the boundary layer. For a flow over a sharp nosed slender body, there exist a shock and this shock will have a very less angle of inclination with respect to the wall of the slender body. Because the shock is so close to the wall of the body, the boundary layer influences the shock[3]. As the fluid enter the viscous region the flow decelerate very rapidly which will raise in the temperature at the wall which leads to raise in the boundary layer thickness. For a constant Reynolds number, the laminar hypersonic boundary layer is 10 to 100 times larger than that of the laminar subsonic boundary layer. When it comes to the stability and transition of the boundary layer of the hypersonic flows, a distinct characteristic of a hypersonic boundary layer is the existence of the Mack mode family of instability modes. Tollmien–Schlichting (T–S) waves are the only unstable mode found in a low-speed boundary layer; in contrast, higher unstable modes are present in a hypersonic boundary layer in addition to the first-mode instability, which is the low-speed boundary layer's analogue to T–S waves. These linearly unstable modes make up the so-called Mack modes. Furthermore, they provide considerable amplitude variations to the transition. This causes the patterns at low speeds, like the impact of surface temperature, to differ from those in hypersonic zones[2].

we discuss about the hypersonic flow over a ramp of 15 degrees. The ramp consists of 0.5ft horizontal wall infront of the leading edge of the ramp and the ramp is of 1 ft length. Air flows at mach 7 at a temperature of 15.6°C, when calculated the velocity is 2383.6976 m/s. As the fluid approaches the ramp it should experience a sudden drop in velocity and increase in pressure and density because there should exist a shock. Because the inclination angle of the ramp is 15°, and the shock should be attached oblique shock. To address this problem we use the tool of CDF. PRAVAH is a program written in fortran, which solves the compressible navier stocks equation. We will also discuss about the grid convergence test, where we did simulations for multiple grid size and we took the optimal grid which give better results and computationally cheaper.

## 2. Numerical Methodology

Pravah code solves the compressible navier stocks equations which are continuity equation, momentum conservation equation and energy conservation equation as written bellow.

### Continuity Equation (Conservation of Mass):

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

This equation represents the conservation of mass, where  $\rho$  is the fluid density,  $\mathbf{u}$  is the velocity vector, and  $t$  is time.

### Momentum Equation:

$$\frac{\partial(\rho u_i)}{\partial t} + \nabla \cdot (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} \quad (2)$$

where,

$$\sigma_{ij} = \mu \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \right) \quad (3)$$

These equations represents the momentum conservathion of the flow. Here  $i$  and  $j$  are repeated index and dummy index. This is discussed more indepth in the book Fluid Mechanics by PK Kundu and IM Cohen[1].

### Energy Equation:

$$\frac{\partial(\rho E)}{\partial t} + \nabla \cdot (\rho \mathbf{u} H) = -\nabla \cdot \dot{\mathbf{q}} + \nabla \cdot (\sigma_{ij} \cdot \mathbf{u}) \quad (4)$$

Where,

$$H = E + \frac{p}{\rho} \quad (5)$$

$$\dot{\mathbf{q}} = -K \nabla T \quad (6)$$

This equation is energy conservation equation.

These equations are solved using second order metric scheme, third order Roe flux Recon scheme, second order viscous scheme and third order time scheme.

## 2.1 Boundary and initial conditions

The boundary conditions and initial conditions play a major roll in modeling a problem. Here we four boundaries, namely inlet, outlet, upper wall and lower wall. The inlet boundary conditions are specified. The inlet velocity is calculated by  $7\sqrt{\gamma RT}$  where  $\gamma$  is the specific heat ratio of air which is 1.4 assuming the gas is calorically perfect. R is specific gas constant of air which is  $287\text{Jkg}^{-1}\text{K}^{-1}$ . T is temperature which is  $15.6^\circ\text{C}$  for this case. The velocity is calculated as  $2383.6976\text{ m/s}$ . The inlet pressure is  $101352.9\text{Pa}$ . and the density is  $1.225\text{kgm}^{-3}$ .

The outlet boundary conditions are given as the Neumann Boundary conditions for all the quantities pressure, velocity and density. The upper and lower wall boundary conditions are no-slip conditions, which are, zero velocity at the walls and for pressure and density, Neumann conditions are given. The initial conditions play a major roll in fast convergence of your solutions. It depends on the problem statement. In this case, if the initial conditions are given as zeros, the scheme takes time to converge to the solution. But if the initial conditions are given as the free stream conditions, the problem take less time to converge to the solution. Here we specified the initial conditions to be free stream conditions.

## 2.2 Computational Domain and Grid

The domain was meshed as seen in Fig 1. The grid is meshed for different grid sizes like  $61 \times 41$ ,  $91 \times 61$ ,  $121 \times 81$ ,  $151 \times 101$  and  $181 \times 121$  to do the grid convergence test. These grid are simulated using PRAVAH code at a time step of  $10^{-7}\text{s}$ . The results have shown that the grid size  $91 \times 61$  is optimal. We have tried even finer grid but the code was blowing but as we reduce the time step futher, the code was working. This is because as the grid size reduce the CFL number increases. When the CFL number crosses 1 the system becomes unstable. But doing this was computationally highly expensive. The code would run for about three hours when four processors working on the problem.

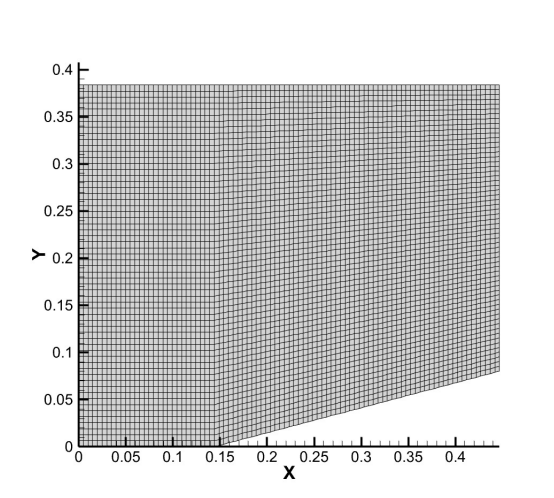
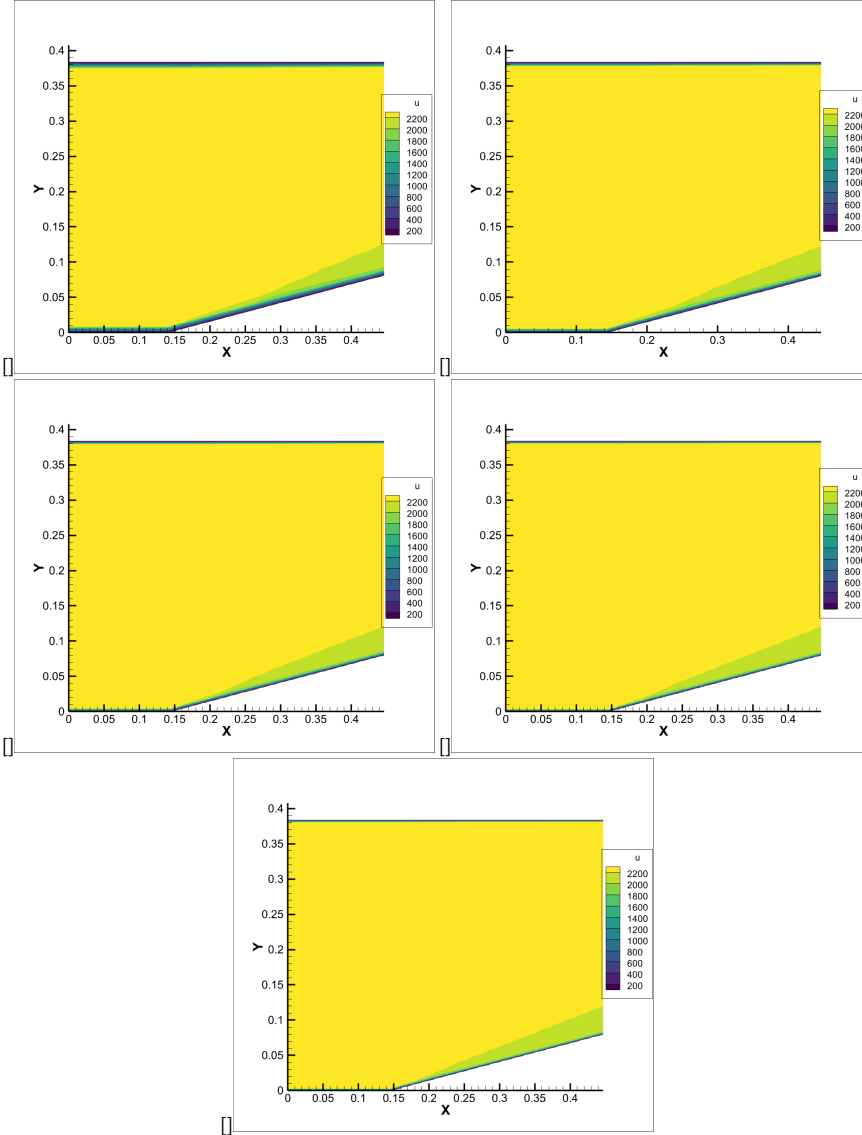


Figure 1. Grid

As we can see from the fig 1, the grid is uniform. A uniform mesh was formed as a rectangular block from the inlet side till the leading edge of the ramp. From the leading edge of the ramp, the y components of the grid points are added by a sine component in such a way that the upper wall stays horizontal.

### 3. Results and Discussion

The Fig 2 show the  $u$  velocity contours for every grid size simulated. We can clearly see that in Fig 2(a) the velocity contour for the most coarse grid size  $61 \times 41$ , the contour is dispersed. This is because the grid size is large and the number of grid points are less. We can also clearly see that as the grid becomes more fine, the contour is more clear as the number of grid points have increased. But as we increases the number of grid points, the simulation becomes more computationally expensive. Also when the number of grid points are increased, the CFL number increases, thus crossing the CFL number 1 will lead to the system instability and the code blows. To solve this problem we have to reduce the time step. This will again cause more expensive computation to simulate.

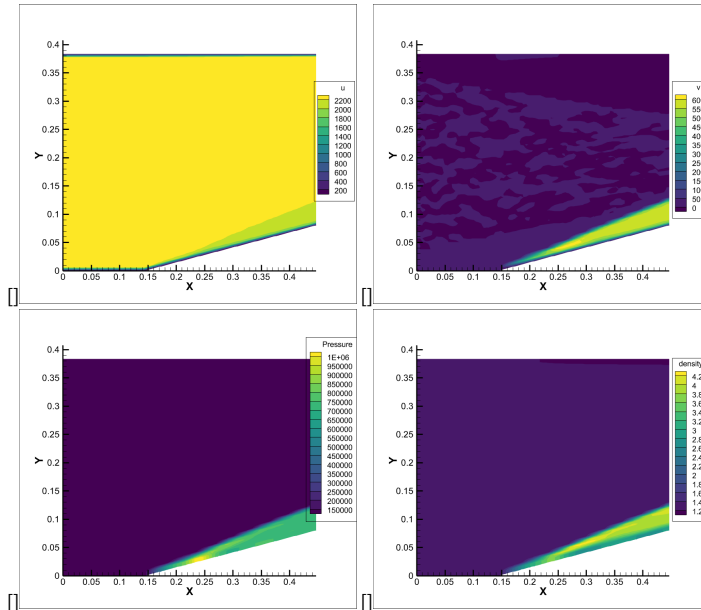


**Figure 2.** From top left to right and next row:  $u$  velocity contours for grid size (a)  $61 \times 41$  (b)  $91 \times 61$  (c)  $121 \times 81$  (d)  $151 \times 101$  and (e)  $181 \times 121$

In Fig 2(b), the contour is clear enough and the simulation of this grid structure less expensive.

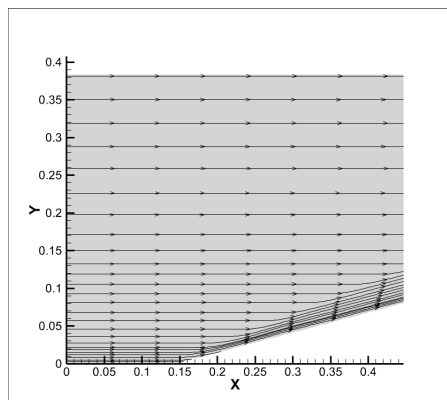
So as the quality of the grid 91x61 is good enough and less expensive, we select the grid 91x61 for the further analysis.

The Fig 3 shows the u velocity, v velocity, pressure and density contour. Here we can clearly see the attached oblique shock wave. These contours say that the velocity after the shock reduces hence the mach number. The pressure and density across the shock increases.



**Figure 3.** From top left to right and next row: (a)u velocity contour, (b)v velocity contour, (c)pressure contour and (d)density contour

The shock angle is calculated by measuring the position of the shock wave at the outlet and taking the tan inverse of the opposite/adjacent. The angle comes to be 21.68 which is approximately matching the analytical solution which is 21.59. The boundary layer is not completely developed or is not as expected, this is because the grid is uniform. To obtain a proper boundary layer it is supposed to give a very fine grid at the wall.



**Figure 4.** Grid

The figure 4 shows the stream lines in the flow. The stream lines are plot using Tecplot. Here we can clearly see that the streamlines are bending after a region which starts from the leading edge of the ramp and goes upstream at an angle. This is because there exist a shock there.

### 3.1 Comparison with literature

The hypersonic flows are very difficult to conduct experiments. Therefor the experimental data for this problem is not found. So we were not able to compare any experimental data. The Frg 5 compares the velocity contour we obtaine using PRAVAH code and the velocity profile that nasa obtained doing CFD. As we can see from both the images there exist a shock. and both of them seems to have same angle. The velocity in inviscid region is mach 7.0 and the flow after the shoch is approximatly of the mach number 5.0. This validated the simulation to be correct.

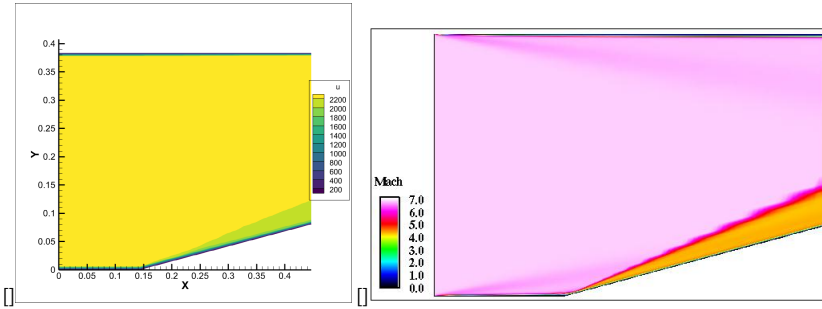


Figure 5. From top left to right:(a)Velocity contour we obtained and (b) velocity profile nasa obtained.

## 4. Conclusions and Perspectives

On concluding the Pravah code is run successfully for a flow over a ramp of 15 degree inclination with a mach number of 7.0. The simulation is run and the data is post processed in Tecplot. The velocity contours, pressure and density contours for different grids sizes are plotted for grid convergence test. The shock is detected in the contours. In Fig 3(c) we can see a high pressure region after the shock but near the boundary layer. This sees like a pressure bubble which may play a major roll in the hypersonic boundary layer and the shock interaction.

## References

- [1] Pijush K Kundu, Ira M Cohen, and David R Dowling, *Fluid mechanics*. Academic press, 2015.
- [2] Cunbiao Lee and Shiyi Chen. "Recent progress in the study of transition in the hypersonic boundary layer". In: *National Science Review* 6.1 (2019), pp. 155–170.
- [3] Lester Lees. "Hypersonic flow". In: *Journal of Spacecraft and Rockets* 40.5 (2003), pp. 700–735.