iJRASET
International Journal For Research in
Applied Science and Engineering Technology

# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

**Volume:** 4 **Issue:** I **Month of publication:** January 2016

**DOI:**

www.ijraset.com
IC Value: 13.98
Volume 4 Issue I, January 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

# Investigation Of Oblique Shock Using FVM Based CFD Solver

Shyam S. Kanwar[1], Aftab[2], Lukesh Kumar[3], Manoj K. Yadav[4], Nathu S. Singroul[5]
[1,2,3,4,5]*Guru Ghasidas university (a central university), Bilaspur, Chhattisgarh India*

***Abstract— The present research work focuses on the development of a 2-D explicit density based inviscid flow solver for oblique shock studies in compressible flows. Salient features of the in-house solver are separate discretisation in space and time, which provides the flexibility of having different accuracy for spatial and temporal discretisation. The spatial discretisation is carried out by unstructured cell-centered Finite Volume Method. Temporal discretisation was achieved by simple explicit time stepping scheme. Encouraging agreement has been noticed for the results obtained from solver and theoretical results.***
***Keywords— Oblique Shock, Euler equations, Finite Volume Method, Computational Fluid Dynamics (CFD)***

## I. INTRODUCTION

A Computational fluid dynamics is a mature and sophisticated technology. It provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of mathematical modeling (partial differential equations), numerical methods (discretization and solution techniques) and software tools (solvers, pre- and post processing utilities). The fundamental basis of any CFD problem is a governing equation which is Navier-Stokes equations in our case, which constitute a system of second-order nonlinear partial differential equations. These equations can be simplified by removing terms describing dissipation to yield the Euler equation. A shock wave is a special kind of pressure wave with steep pressure rise. It can be described as "a compression wave front in a supersonic flow field and flow across the wavefront results in abrupt change of fluid properties", i.e. across a shock there is always an extremely rapid rise in pressure, temperature and density of the flow. In supersonic flows, expansion is achieved through an expansion fan. A shock wave travels through most media at a higher speed than an ordinary wave. Shock wave is an irreversible process; the kinetic energy possessed by the incoming gas is utilized for compressing the gas across the wave. The two main types of shock waves produced by a supersonic body are Normal Shocks and Oblique Shocks. These two types are produced by the same phenomena, but the shape of the supersonic object dictates which type of shock wave is experienced. The shock wave may be classified as follows: a) Normal shock wave b) Oblique shock wave. In this paper only focuses on oblique shock wave. An Oblique Shock is a sharp edged shock wave that is formed when supersonic flow is turned on itself. These shocks are weaker than Normal Shocks, and although the temperature, pressure, density, and air stream velocity are reduced across the shock similar to the Normal Shock, the air stream behind the shock is not necessarily subsonic. The Mach number behind the Oblique shock is calculated from the upstream Mach number, defined by the angle at which the flow is tuned. The figure below shows a typical oblique shock formed by a sharp angle.

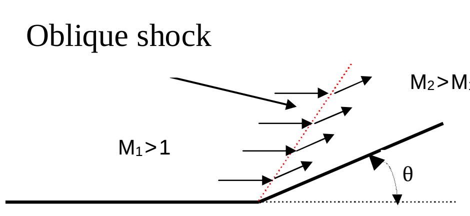



Fig. 1 Formation of oblique shock wave

Oblique shock waves are used predominantly in engineering applications when compared with normal shock waves. This can be attributed to the fact that using one or a combination of oblique shock waves results in more favourable post-shock conditions (lower post-shock temperature, etc.) when compared to utilizing a single normal shock. An example of this technique can be seen in the design of supersonic aircraft engine inlets, which are wedge-shaped to compress air flow into the combustion chamber while minimizing thermodynamic losses. Early supersonic aircraft jet engine inlets were designed using compression from a single normal shock, but this approach caps the maximum achievable Mach number to roughly 1.6. Concorde (which first flew in 1969) used variable geometry wedge-shaped inlets to achieve a maximum speed of Mach 2.2. A similar design was used on the F-14 Tomcat (the F14-D was first delivered in 1994) and achieved a maximum speed of Mach 2.34.

Many supersonic aircraft wings are designed around a thin diamond shape. Placing a diamond-shaped object at an angle of attack relative to the supersonic flow streamlines will result in two oblique shocks propagating from the front tip over the top and bottom of the wing, with Prandtl-Meyer expansion fans created at the two corners of the diamond closest to the front tip. When correctly designed, this generates lift. The calculation of oblique shock property on flow fields obviously requires thermodynamic and chemical relation which includes the real gas effects and the reactions and products. Anderson [1] carried out a simplified air model analysis, consisting of oblique shock property calculation. The use of simplified air model enables us to show the basic features of hypersonic flow.

## II. GOVERNING EQUATIONS

Considering a rectangular control volume passing through the shock perpendicular to the flow, we can derive the continuity, momentum and energy equation. The fundamental flow equations, namely, the equation of continuity, momentum equation and energy equation can apply across a shock wave. Since there is an increase in entropy across the shock wave, the isentropic flow assumptions are not applicable, but changes assumed to take place adiabatically across the shock wave [2].

$$\rho_1 u_1 = \rho_1 u_2 \tag{1}$$

$$p_1 + \rho_1 u_1^2 = p_2 + \rho_2 u_2^2 \tag{2}$$

$$h_1 + \frac{u_1^2}{2} + q = h_2 + \frac{u_2^2}{2} \tag{3}$$

Equations are represented as continuity, momentum and energy equation respectively. Where q is the heat added per unit mass and $h = e + pv$ is, by definition enthalpy. Now in order to find out the properties behind the shock wave we make use of above equations. In our research work has been confined to the solution procedures for the inviscid flow and the governing equations are known as the Euler equations. Euler equations are first order system of non-linear coupled equations, which can be expressed in various forms such as conservation form and primitive variable form. Conservation form of the equations is essential in order to compute correctly the propagation speed and the intensity of discontinuity, such as contact discontinuities or shocks that can occur in inviscid flows.

$$\frac{\partial \vec{U}_i}{\partial t} + \frac{\partial \vec{F}_i}{\partial x} + \frac{\partial \vec{G}_i}{\partial y} = 0 \tag{4}$$

$$\vec{U}_i = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \\ \rho m_i \end{bmatrix} \quad \vec{F}_i = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uH \\ \rho u m_i \end{bmatrix} \quad \vec{G}_i = \begin{bmatrix} \rho u \\ \rho uv \\ \rho v^2 + p \\ \rho uH \\ \rho v m_i \end{bmatrix}$$

The Euler equations governing the 2D flow [3] in the absence of body forces with species transport equation in the conservative and

Volume 4 Issue I, January 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

differential form are,

$$\frac{\partial(\rho)}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}=0 \tag{5}$$

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2+p)}{\partial x}+\frac{\partial(\rho v)}{\partial y}=0 \tag{6}$$

$$\frac{\partial(\rho E)}{\partial t}+\frac{\partial(\rho uH)}{\partial x}+\frac{\partial(\rho vH)}{\partial y}=0 \tag{7}$$

$$\frac{\partial(\rho m_i)}{\partial t}+\frac{\partial(\rho u m_i)}{\partial x}+\frac{\partial(\rho v m_i)}{\partial y}=0 \tag{8}$$

In the above versions of formulations, the total specific energy, E=e+0. 5 (u2+v2) the total specific enthalpy H=h+0. 5 (u2+v2), is the mass fraction of the species given by mi=ρi /ρ. This thesis considers a solution to unsteady state Euler

## III.NUMERICAL METHOD

Finite volume method formulation The basic idea of a FVM is to satisfy the integral form of the conservation laws to some degree of approximation for each of many adjacent control volumes which cover the domain of interest.

$$\frac{d}{dt}\int_{V(t)}\overline{U}dV+\oiint_{S(t)}\vec{n}.\vec{F}ds=0 \tag{9}$$

The average value of $U$ in a cell with volume V is

$$\overline{U}=\frac{1}{V}\iint UdV \tag{10}$$

Eq. 3.1 can be written as

$$V\frac{d}{dt}\overline{U}+\oiint_{S(t)}\vec{n}.\vec{F}ds=0 \tag{11}$$

$$\frac{d\overline{U}}{dt}+\frac{1}{V}\oiint_{S(t)}\vec{n}.\vec{F}ds=0 \tag{12}$$

$\overline{U}$ is the average value of U over the entire control volume, $\vec{F}$ is the flux vector and $\vec{n}$ is the unit normal to the surface. And $\vec{F}=F_I\vec{i}+G_I\vec{j}$ , is the total inviscid flux, upon integrating the inviscid flux over the faces of kth control volume the above equation becomes

$$\frac{\partial U_k}{\partial t}+\frac{1}{V_k}\left[\sum_{i=1}^{nf}\vec{F}.\vec{n}ds\right]_k=0 \tag{13}$$

Here, $\vec{n}=\frac{\Delta y_i}{\Delta s_i}\vec{i}-\frac{\Delta x_i}{\Delta s_i}\vec{j}$ and $\Delta s_i=\sqrt{(\Delta x)_i^{\,2}+(\Delta y)_i^{\,2}}$

For the 2-D axi-symmetric problems the finite volume formulation is given by

$$\frac{d\overline{U}}{dt}+\frac{1}{V}\oint_{S(t)}\vec{n}.\vec{F}ds=0 \tag{14}$$

*A. Discretisation Schemes*

In computational fluid dynamics discretisation of inviscid or convective fluxes is the critical part of Euler solver. One of the methods mentioned below is generally seen in the literature for computation of inviscid or convective fluxes-

Flux vector splitting scheme

Flux difference splitting scheme

Total Variation Diminishing (TVD) scheme

Fluctuation splitting scheme

In the present investigations Van-Leer scheme from the family of flux vector splitting schemes is preferred for present studies. The idea behind the flux vector splitting schemes is to divide the flux vector into positive and negative components.

*1) Flux Vector Splitting Scheme:* There are two class of flux vector splitting schemes, one class of flux vector scheme developed by Steger and Warming and Van-Leer which decomposes the vector of convective fluxes into two parts based on the sign of characteristics variables. The second class of flux vector splitting scheme developed by Liou and steffen [4] into convective and pressure part. Scheme like AUSM developed by Steger J L [5] belongs to the second class of the Flux vector splitting scheme. Advantages of the Flux vector scheme includes with a little or moderate increase in numerical effort gives better resolution of shocks. Also this is well suited for implicit methods where the computation of steady state solution is of great importance. Main importance is that the Flux vector schemes can be easily extended to real gas flow.

*a) Van-Leer Scheme:* Van Leer flux vector splitting scheme is based on the characteristics decomposition of convective fluxes. He splits the convective flux in to positive and negative part based on the normal Mach number to the face of the control volume.

$\vec{F}_c=\overrightarrow{F_c^+}+\overrightarrow{F_c^-}$, The Mach number normal to the face of the control volume is given by $M_n=\frac{u_\perp}{c}$ ,Where $u_\perp$ is the contra variant velocity, given by, $u_\perp=un_x+vn_y$ and c is the speed of the sound. The values of the flow variables $\rho,u,v$ and $p$ are respectively have to be interpolated to the faces of the control volume. Then the positive fluxes are computed with left state and negative fluxes are computed with right state. The advection Mach number is given by

$M_n=M_L^+ +M_R^-$ Where the split mach numbers are defined as

$$M_L^+ = \begin{cases} M_L & \text{if } M_L \geq 1 \\ \frac{1}{4}(M_L+1)^2 & \text{if } |M_L| < 1 \\ 0 & \text{if } M_L \leq -1 \end{cases}$$

$$M_R^- = \begin{cases} 0 & \text{if } M_R \geq +1 \\ \frac{1}{4}(M_R-1)^2, & \text{if } |M_R| < 1 \\ M_R, & \text{if } M_R \leq -1 \end{cases}$$

The mach numbers $M_L$ and $M_R$ are computed using the left and right state from the equation

$M_L = \frac{u_{\perp_L}}{c_L}$ $\quad M_R = \frac{u_{\perp_R}}{c_R}$, The normal flux vector is given by

$$F_\perp = \begin{bmatrix} \rho u_\perp \\ \rho u u_\perp + p n_x \\ \rho v u_\perp + p n_y \\ \rho H u_\perp \end{bmatrix}$$

In the case of subsonic flow ($|M_\perp|$<1) the positive and negative flux part are given by

$$\vec{F}_c^\pm = \begin{bmatrix} f_{mass}^\pm \\ f_{mass}^\pm \left[u + n_x(-u_{nor} \pm 2c)/\gamma\right] \\ f_{mass}^\pm \left[v + n_y\left(-u_{nor} \pm 2c\right)/\gamma\right] \\ f_{energy}^\pm \end{bmatrix}$$

The mass and energy flux components are defined as

$$f_{mass}^+ = +\rho_L c_L \frac{(M_L+1)^2}{4}$$

$$f_{mass}^- = -\rho_R c_R \frac{(M_R-1)^2}{4}$$

$$f_{energy}^\pm = f_{mass}^\pm \left\{ \frac{\left[(\gamma-1)u_{nor} \pm 2c\right]^2}{2(\gamma^2-1)} + \frac{u^2+v^2-u_{nor}^2}{2} \right\}$$

For supersonic flow $\left(|M_\perp| > 1\right)$ fluxes are computed as

$$F_\perp^+ = F_\perp,\ F_\perp^- = 0 \quad \text{if } M_\perp \geq 1$$

$$F_\perp^- = F_\perp,\ F_\perp^+ = 0 \quad \text{if } M_\perp \leq -1$$

$$F_\perp = F_\perp^+ + F_\perp^- \quad \text{if } -1 < M_\perp < 1$$

VanLeer flux vector splitting scheme performs very well in the case of Euler equations. But for viscous flow it smears out the boundary layer and also gives inaccurate stagnation and wall temperatures.

*B. Algorithm And Description About The Development Of In-House Solver*

A brief description about the algorithm of solver is planning to discuss. The flow chart of the algorithm is as shown in Fig 2. Once

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

the preprocessing is completed the primitive variables $\rho, u, v.T, e$ and conservative variables $\rho, \rho u, \rho v, \rho e$ are initialized in all cell centroids .Then the boundary conditions are initialized in the all boundary face centroids based upon the boundary type. Inlet boundary conditions are specified as same as that of free stream conditions. The outlet boundary conditions are extrapolated from the interior cell centroid. For inviscid wall boundary condition ghost cell approach is used for current solver. Intilization of the boundary condition is carried out by running loop over all the faces.

For a particular boundary face if its left cell is not existing flow variables are extrapolated to that face centroid from the right cell centroid and if its right cell is not existing flow variables are extrapolated to face centroid from its left cell centroid. For interior faces whose left cell and right cell exists, the average of the cell centroid values are extrapolated to face. Also for the inlet and outlet boundaries the mass, momentum and energy fluxes are directly specified on the boundary faces. The fluxes for the inviscid boundary faces and interior faces are computed from the upwind schemes. But upwind schemes required left and right state of the flow variables at faces. Residuals are calculated from the relation $\sqrt{\frac{\left(\rho^{n+1}-\rho^{n}\right)^{2}}{nc}}$ and it is being normalized by dividing the residual calculated from the first iteration.

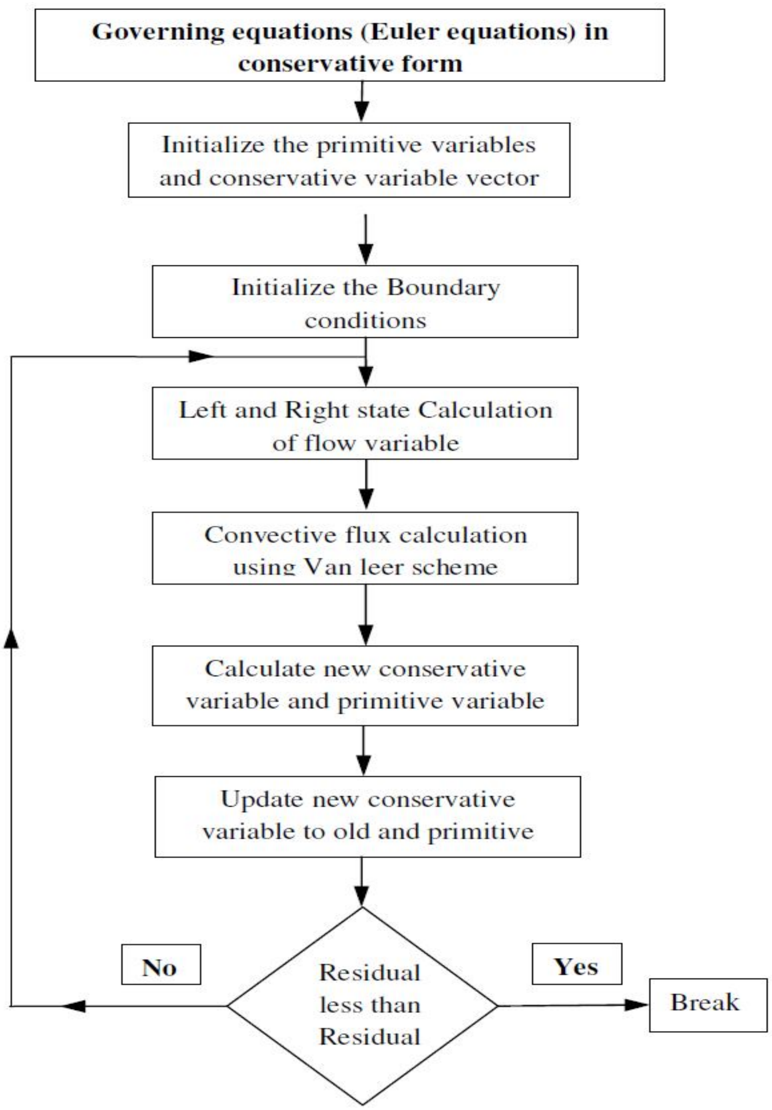



Fig. 2 Algorithm of the in-house solver

Volume 4 Issue I, January 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

## IV. RESULT AND DISCUSSION

The objective of this research paper is to discuss the results obtained from the present solver in comparison with the theories in hypersonic flow and with the standard oblique shock theory. Various test cases used to validate the code is planar flow past a cylinder. In this test case mentioned above the present solver results are validated across various theories in hypersonic flow theory.

### *A. Flow Through Wedge/Ramp In Channel*

The problem consists of a uniform flow being disturbed by a small disturbance, in this case the ramp. The supersonic flow through a wedge channel is a standard test case to study the oblique shock for validation of 2D inviscid flow solvers. To start with Euler computations, flow at inlet is supersonic, oblique shock will be generated at the wedge. Using the geometry the bottom wall is turned upward at the corner through a deflection angle $15°$ that is, the corner of concave. The length of the channel is 1.2 meter and height is 1 meter. Structured mesh used having grid size of 200×200.The grid consists of 80601 cells, 80000 points, 160600 faces .The flow geometry and corresponding computational mesh is shown in Fig 3. The lower wall, including the ramp, and the top wall are modeled as being impermeable boundaries i.e. slip- wall boundary condition is imposed. For computation, first order scheme, with Van Leer scheme is used for the calculation of the convective fluxes.

$$\left|\rho^{n+1}-\rho^{n}\right| \leq 10^{-6} \qquad (15)$$

where,

n = iteration index., ρ = density

The supersonic flow (Mach=8) at the wall must be tangent to the wall hence the stream line at the corner also deflected through the angle 15°. Due to this free stream flow is "turned into it-self" that is oblique shock which is clear in present simulation. From the property of oblique shock theory [6] the flow property must be increased where shock angle is determined by using θ-β-M chart. Fig 5.9 shows the Mach number discontinuously decreases. And pressure, density and temperature discontinuously increases which is demonstrate in Fig. 4, Fig. 5 and Fig.6 respectively. Eq no. 16 to Eq. no. 20 is used to calculate the property behind the oblique shock; the values are in the Table 2. Since oblique shock exists at the ramp in a channel, property relations obtained from the solver has been compared with the oblique shock relations [6] for as shown in the Table 2.

### *B. Formulation Of Parameters*

The following relations are to be used for calculation of property behind the oblique shock.

$$\frac{P_2}{P_1} = 1 + \frac{2\gamma}{\gamma+1}(M_{n_1} - 1) \qquad (16)$$

$$\frac{\rho_2}{\rho_1} = \frac{(\gamma+1)M_{n_1}{}^2}{(\gamma-1)M_{n1}{}^2 + 2} \qquad (17)$$

$$\frac{T_2}{T_1} = \frac{P_2}{P_1} \times \frac{\rho_1}{\rho 2} \qquad (18)$$

$$M_{n_2}{}^2 = \frac{M_{n_1}{}^2 + [\frac{2}{(\gamma-1)}]}{[\frac{2\gamma}{\gamma-1}]M_{n_1}{}^2 - 1} \qquad (19)$$

$$M_2 = \frac{M_{n_2}}{\sin(\beta-\theta)} \qquad (20)$$

Volume 4 Issue I, January 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

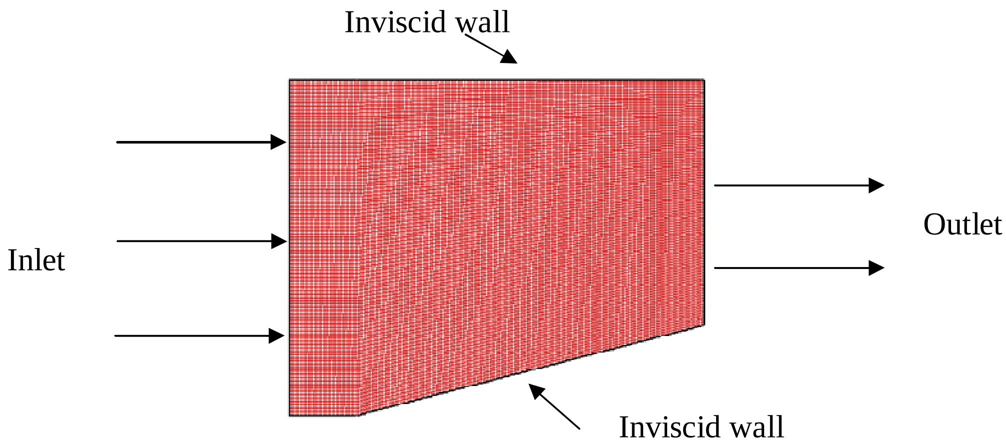



Table 1 Free stream condition for ramp in a channel

| Test case | | Free stream input conditions | | | |
|---|---|---|---|---|---|
| 1. | γ | $M_\infty$ | $U_\infty$ | $P_\infty$ | $T_\infty$ |
| 2. | 1.4 | 8 | 2777.50 m/s | 100000 Pa | 300 K |

Table 2 Comparison of various parameters across shock (wedge) for different species

| Parameter | Oblique shock theory | Solver Result with $\gamma$ =1.4 |
|---|---|---|
| $P_2/P_1$ | 9.301 | 9.31 |
| $T_2/T_1$ | 2.505 | 2.20 |
| $\rho_2/\rho_1$ | 3.712 | 3.00 |
| $M_2$ | 4.747 | 5.08 |
| β | 20.86 | 21 |

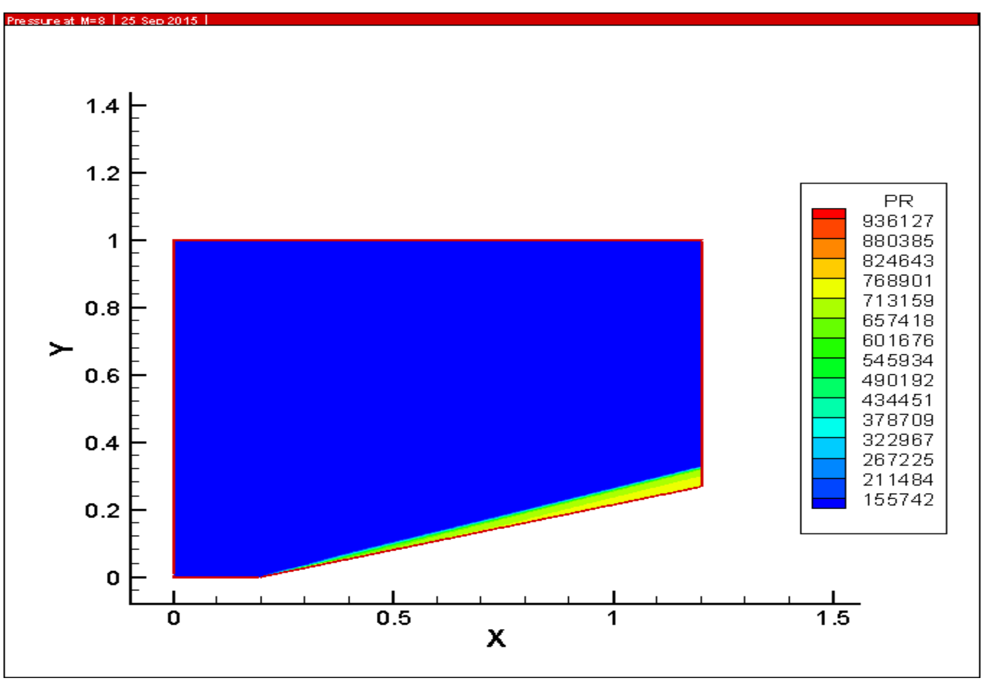



Fig. 4 Pressure contours (For M=8)

Volume 4 Issue I, January 2016
ISSN: 2321-9653

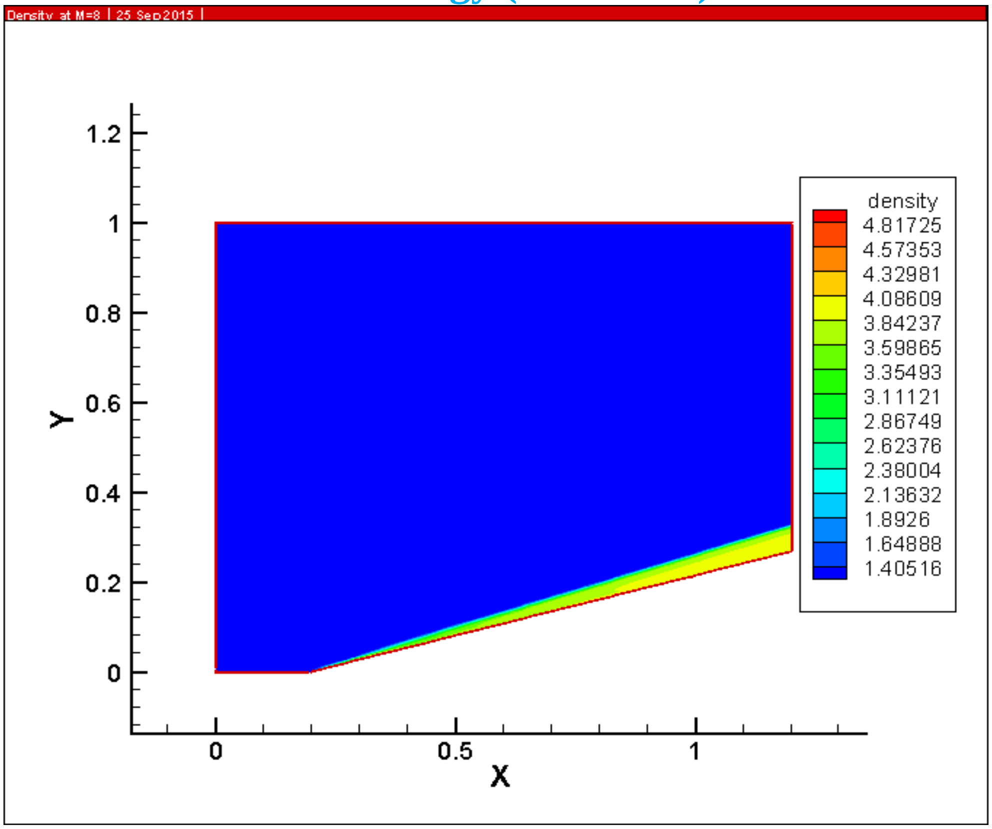




Fig. 5 Density contours (For M=8) in kg/m$^3$

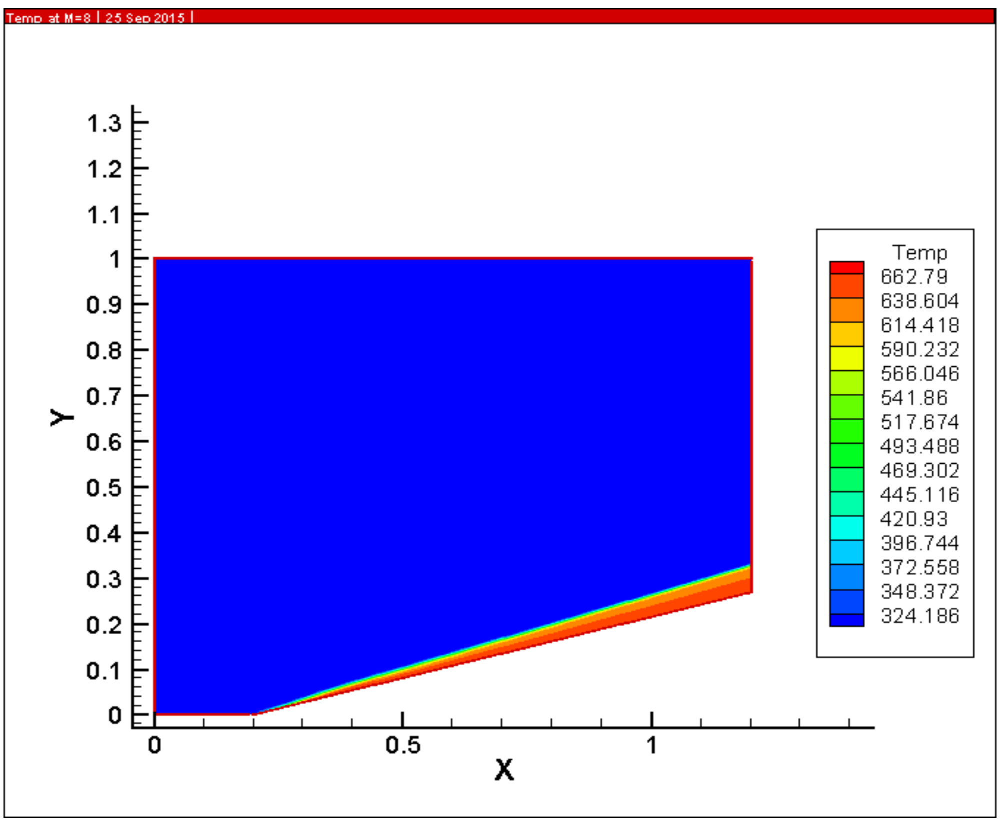



Fig. 6 Temperature contours (For M=8) in kg/m$^3$

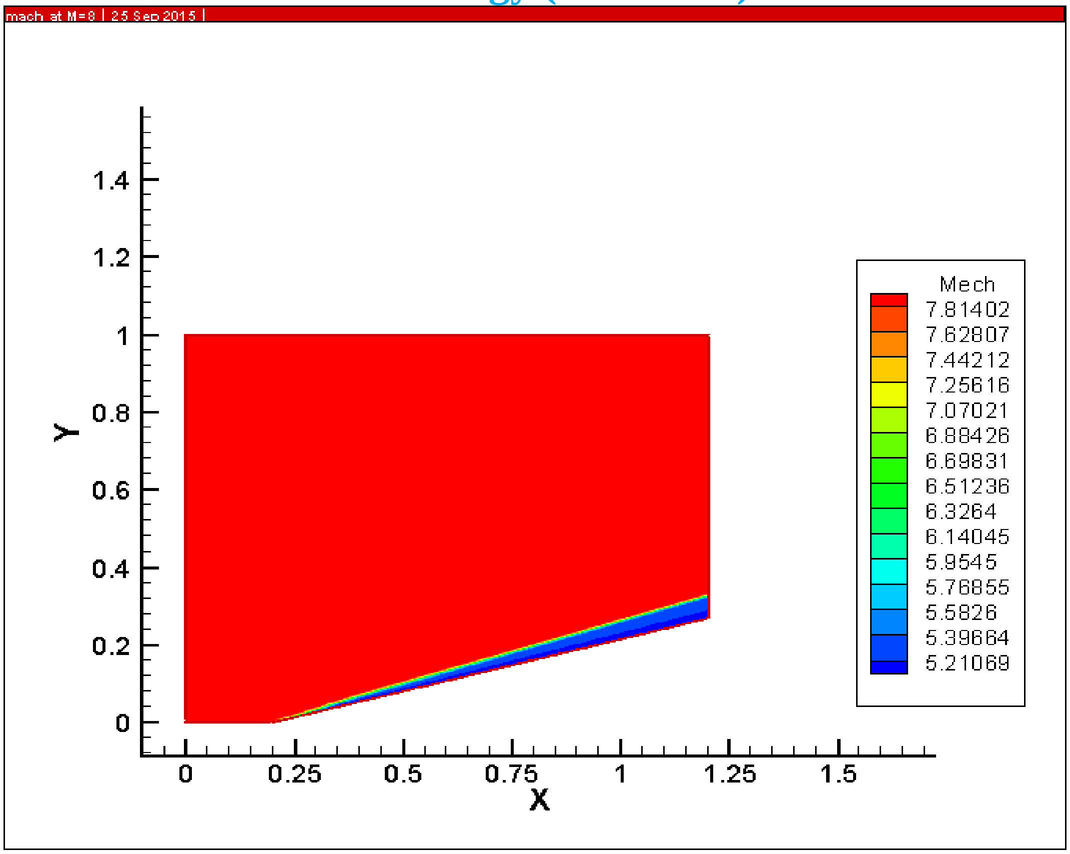


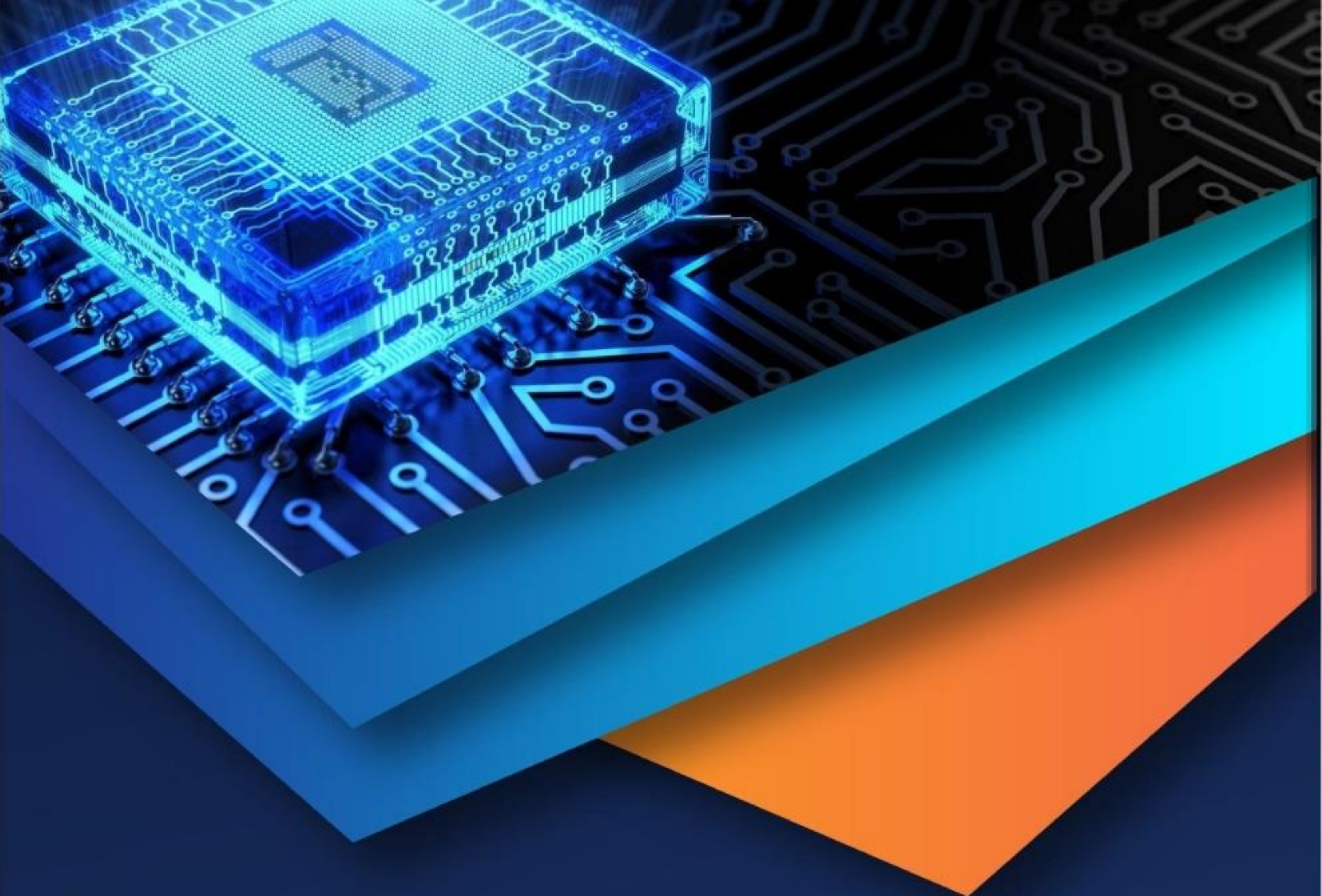

Fig. 7 Mach contours (For M=8) in kg/$m^3$

## V. CONCLUSION

An unstructured finite volume solver for high speed inviscid compressible flows for 2-D and 2-D axi-symmetric configurations has been successfully developed. Results obtained for all the test cases are in good agreement with the theoretical results (oblique shock theory).

## REFERENCES

[1] Anderson, J. D. (1989). Hypersonic and High Temperature Gas Dynamics. McGraw-Hill Inc.
[2] Blazek, J. (2001). Computational Fluid Dynamics Principal and Applications. Eselvier.
[3] Venkatakrishnan V (1995), Convergence to steady state solutions of the Euler equations on Unstructured grids with Limiters, Journal of computational physics 118,120-13.
[4] Liou, M. S. and C. J. Steffen (1999). A New Flux Splitting Scheme. Journal of Computational Physics 107, 23-39.
[5] Steger J L and Warming R F (1981), Flux vector splitting of inviscid gas dynamic equations with application to Finite difference methods, Journal of computational physics, 40, pp 263-293.
[6] Anderson, J. D. Jr. (2005), Fundamentals of Aerodynamics. McGraw-Hill Inc.




10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429


# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY