NUMERICAL ANALYSIS OF QUASI-ONE-DIMENSIONAL NOZZLE BY UPWIND SCHEMES

Venkata Sowjanya Malikireddy[1], Harinath Reddy Nakkala[2]

[1,2]Assistant Professor,
Y.S.R. Engineering College of YVU
[1]Department of Civil Engineering
[2]Department of Mechanical Engineering
Proddatur – 516 360, Andhra Pradesh, India
[1]mv.sowjanya@gmail.com, [2]reddynhn@gmail.com

Abstract - The governing equations of fluid flow are hyperbolic and flows are characterized by the preferred direction of information propagation. As the analytical solution of these non-linear partial differential equations is not possible in most cases, a numerical solution is the only alternative. Computation of flows governed by compressible Euler equations around supersonic aircraft and launch vehicles are some situations where upwind methods are effective. All these schemes, however, are not uniformly good for all situations. Different aspects of these methods such as CPU time-wise efficiency, accuracy, and robustness are expected to be brought out by the present study to assist a CFD investigator to decide upon the scheme to be used for a particular physical situation. Keeping in this view quasi-one-dimensional nozzle problem is studied.

Key Words: Computational Fluid Dynamics (CFD), Quasi-one-dimensional Nozzle, CFL Condition, Numerical Scheme, Upwind schemes, Subsonic Flow, Supersonic Flow.

1. INTRODUCTION

For numerical analysis of flow at high Mach numbers around objects like missiles, launch vehicles etc., Euler equations are frequently used. The family of upwind schemes, whose origin may be taken back to Courant, Isaacson and Reeves (1952), is directed towards the introduction of the physical properties of the flow equations into discretized formulation and has led to the family of techniques known as Upwinding.

As analytical solution of these non-linear partial differential equations is not possible in most cases, a numerical solution is the only alternative. The numerical method of one-sided differencing is upwind scheme. As upwind schemes are well known for their ability to capture shocks and compute flows over a wide range of speed and geometry, their popularity is on the rise and considerable research is going on to refine technique and extend their range of applicability. They are extensively used in aerodynamic design of different aerospace configurations.

There are different upwind schemes Van Leer's scheme (1988), Zha-Bilgen scheme (1993), Advection Upstream Splitting method (AUSM), Steger Warming (1981), and MacCormack schemes (1969) were used to solve the quasi-one-dimensional nozzle problem. Upwind schemes were tested for different problems by different authors. Harinath Reddy N, Venkata Sowjanya M (2019) were tested upwind schemes for shock tube problem.

1.1 Quasi-one-dimensional Nozzle:

The nozzle geometry for which computations have been carried out is a series of two-dimensional converging-diverging nozzles designed and tested at NASA Langley Research Center, namely, nozzle A2 and the geometry as shown in Figure 2. The geometry is formed by a plane upstream and downstream of the throat region with slope angles of and, respectively. In the throat region, it has a circular-arc surface for transition. The geometry is symmetric about the central axis plane, and only upper half is shown here.

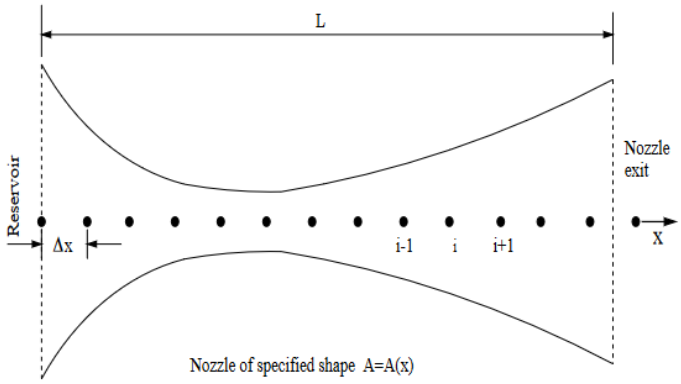

Fig.1. Nozzle – Grid point Distribution

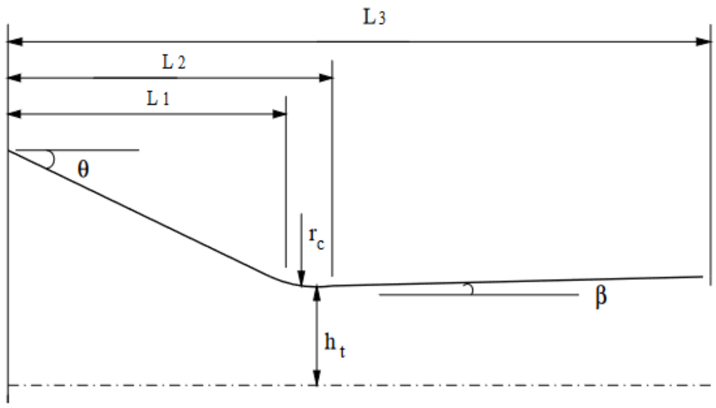

Fig.2. Nozzle geometry for problem

The formulations describing the geometry are

$y = \tan(\theta)x + h_i$	for $0 \leq x \leq L_1$
$y = y_c$ -sqrt(r_c-(x-x_c)^2	for $L_1 \leq x \leq L_2$
$y = \tan(\beta)(x-x_t) + y_t$	for $L_1 \leq x \leq L_2$

where θ = -22:33^o , β= 1:21^o , L_1 = 4:74 cm, L_2 = 5:84 cm,
L_3 = 11:56 cm, x_t = 5:84 cm, y_t = 1:37 cm, x_c = 5:78 cm,
y_c = 4:11 cm, r_c = 2:74 cm, h_t = 1:37 cm, and h_i = 3:52 cm.

2. NUMERICAL PROCEDURE

The governing equations are one-dimensional Euler equations and the conservation form of these equations is given below.

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} = 0$$

where U, F are vectors containing conservative variables and conservative fluxes respectively and S is the cross-sectional area and are given by,

$$U = S\begin{bmatrix}\rho \\ \rho u \\ e\end{bmatrix}; \quad F = \begin{bmatrix}\rho u \\ p + \rho u^2 \\ (e+p)u\end{bmatrix}$$

where ρ, ρu, e = mass, momentum, total energy per unit volume. And e is defined as,

$$e = \frac{p}{\gamma - 1} + \frac{1}{2}\rho u^2$$

The above equation is solved by using different schemes for a given standard problem, quasi, one-dimensional nozzle problem

To solve a nozzle problem, it is divided into a number of grid points in the x-direction as shown in the Figure 1. The spacing between the adjacent grid points is x. Now assume the flow field variables at all grid points as initial conditions at time t = 0. For faster time marching procedure, one has to choose the initial conditions very carefully. Generally initial conditions should be closer to final steady-state results for faster convergence. The first step in solving the nozzle problem is to feed the nozzle shape and initial conditions into the program. Calculate all the flow properties for the next time step and compare them with the previous time. Repeat this procedure until steady-state is reached.

For the subsonic boundary conditions at the entrance, the velocity is extrapolated from the inner domain, and the other variables are determined by the total temperature and total pressure. For supersonic exit boundary conditions, all of the variables are extrapolated from inside of the nozzle. The analytical solution was used as the initial ow eld. The computation is proceeded using global time step in time accurate fashion. The nozzle contour can be input as a function y(x) measured from the centerline. For an axisymmetric nozzle, the function A(x) is just, $A(x) = y^2(x)$. On the other hand, if a 2-D nozzle is assumed, A(x)=2 y(x). To solve, a computer code in C has been used. The solution has been obtained for the geometries shown above by running the code written in C until the steady state is obtained.

2.1 Stability (CFL) Condition:

The stability condition for the convergence of any numerical solution is known as CFL (Courant, Friedrich, and Lewy) condition. Every scheme requires the specification of a time increment, Δt. For explicit methods, the value of Δt cannot be arbitrary, rather it must be less than some maximum value allowable for stability.

$\Delta t = \lambda(\frac{\Delta x}{u+c})$ where λ is the Courant number.

CFL criterion is that Δt must be less than or at most equal to the time required for a sound wave to propagate between two adjacent grid points. i.e.,

$$\Delta t \leq \lambda(\frac{\Delta x}{u + c})$$

3. RESULTS AND DISCUSSION

Figures 3 and 5 shows the variation of Mach number and flow rate as a function of distance for the steady subsonic-supersonic isentropic flow through a nozzle with different schemes for problem 1. From Figure 5, it is clear that the flow rate computed with Van Leer's scheme, Steger and Warming scheme, and MacCormack's scheme is closer to the analytical result but AUSM and Zha-Bilgen are farther from it. Important conclusions can be drawn by studying the variation of Mach number at the throat region. It is clearly demonstrated in Figure 4.

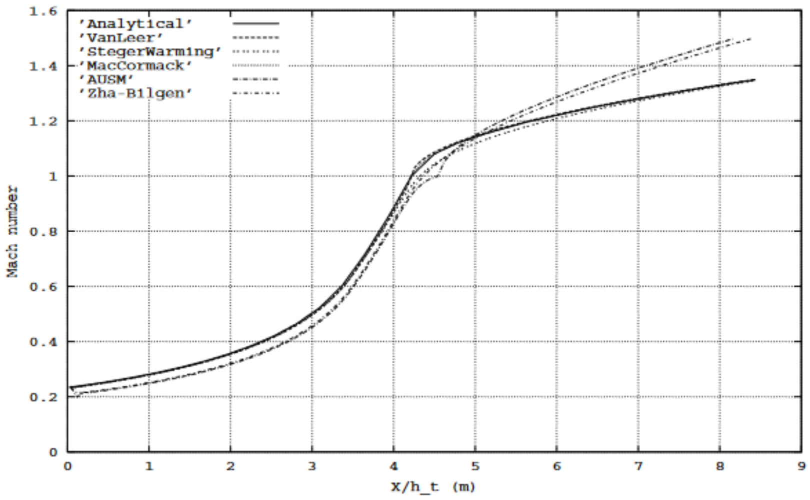

Fig.3. Comparison of Mach number variation for different schemes

MacCormack's scheme agrees the best with analytical result at the throat region, but with a little jump. This scheme is performing extremely well when there is no shock and contact discontinuity. Van Leer's scheme also agrees well with analytical result throughout, but at the transition region there is a large jump. Steger and Warming scheme perform well without any jump at the throat region. But AUSM scheme is giving a small jump at the throat region and is deviating at the outlet, similarly Zha-Bilgen scheme is also deviating more from the analytical results at outlet.

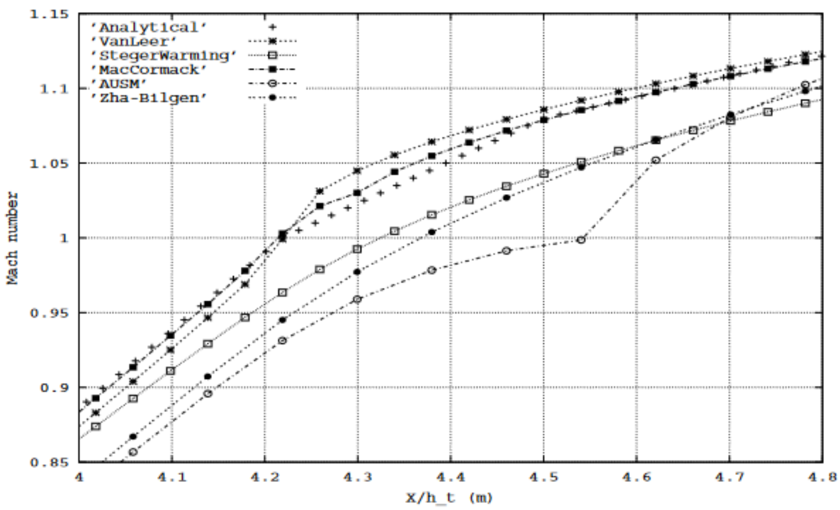

Fig.4. Comparison of Mach number variation at throat region of nozzle

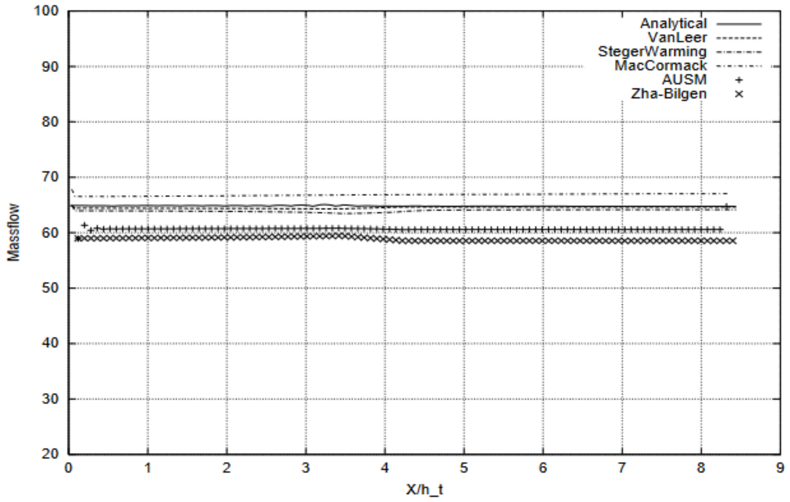

Fig.5. Comparison of Mass flow region variation for different schemes

4. CONCLUSIONS

For mixed subsonic-supersonic flow without shocks (as in the nozzle), the results obtained with Steger and Warming scheme is closer to the analytical result as compared to other upwind schemes. It tells us that Steger and Warming scheme works well, when there is no shock and contact discontinuity. It is clear from the Figure 3. Van Leer's scheme gives a large jump at the transition region (Fig. 4). The computation of the problems involving shock and contact discontinuity with different schemes shows that shock and contact discontinuity is greatly smeared with almost all the schemes but VanLeer's scheme performs better. For mixed subsonic-supersonic flow without shocks. Steger and Warming scheme perform well and AUSM gives a small jump at the transition region. Thus, no scheme is seen to be uniformly good in all situations and care has to be exercised before choosing the scheme to be used.

REFERENCES

[1] Courant, R., Isaacson, E., and Reeves, M. (1952). `On the Solution of Non-linear Hyperbolic Differential Equations by Finite Differences.' Comm. Pure and Applied Mathematics, 5, 243-55.

[2] Van Leer, B. (1982). `Flux Vector Splitting for Euler Equations.' In Proc. 8 the International Conference on Numerical Methods in Fluid Dynamics, Springer.

[3] Zha, G. C., and Bilgen, E. (1993). `Numerical solutions of Euler Equations by Using a New Flux Vector Splitting Scheme.' International Journal for Numerical Methods in Fluids, 17, 115-144

[4] MacCormack, R. W. (1969). `The Effect of Viscosity in
Hypervelocity Impact Cratering.' AIAA Paper, 69-354

[5] Steger, J. L., and Warming, R. F. (1981). `Flux Vector Splitting of the Inviscid Gas dynamic Equations with Application to Finite Difference Methods.' Journal of Computational Physics, 40, 263-93

[6] Harinath Reddy, N, Venkata Sowjanya, M (2019), 'Computation of high-speed inviscid compressible flow by upwind schemes', IJREAM, Vol-4, Issue-12.