CFD Letters www.cfdl.issres.net Vol. 10 (1) – June 2018

Comparative study of upwind and averaging schemes with artificial dissipation for numerical solution for quasi-one-dimensional supersonic flow

Tohid Adibi[1c] and Aria Amrikachi[2]
[1] *Department of mechanical engineering*
University of Bonab, IRAN
[2] *Department of mechanical engineering*
Shahid Beheshti University, IRAN

Abstract

In this study, a quasi-one-dimensional supersonic flow in a divergent nozzle has been numerically simulated. For this purpose, codes have been written in FORTRAN software. In this study, the finite-volume method has been used for simulation. Time discretization has been progressive. Consequently, the equations obtained are explicit. Characteristic equations are extracted for one-dimensional compressible flows. The gradient of all characteristic lines is positive. To calculate convective terms, both upwind and averaging methods have been used. Forth order dissipation have been used for averaging methods' stability. The results of the two methods have been compared for the accuracy of the results and the convergence rate. The effect of artificial damping coefficient on the convergence rate and the accuracy of the results in the averaging method have been investigated. Independence of the network is also examined for both methods. The results of these two methods were compared with the results obtained from the exact method which was solved for one-dimensional isentropic flow. Results comparison illustrates that upstream method has a better accuracy and convergence speed.

Keywords: Upwind scheme; Averaging scheme; Supersonic; Artificial dissipation; convective flux.

1. Introduction

The equations governing fluid dynamics are very complex that does not have an exact solution in general. Apart from very specific cases that due to simple nature of the problem, the governing equations of the fluid flow are simplified and an exact solution is obtained, in other cases, alternative methods should be sought. One of the alternative methods is experimental methods. The major problem with these methods is the high cost of them. Dimensional analysis and similarity have, to a large extent, been able to reduce laboratory costs, but experimental methods are still costly. Numerical methods are another alternative technique which are very inexpensive compared to experimental methods. In recent years, these methods have been developed rapidly due to computers speed

[c] Corresponding Author: Tohid Adibi
Email: Tohidadibi@bonabu.ac.ir

development. In recent years, using numerical methods, many researchers have gained reliable results. Several of numerical methods have been produced and developed for solving fluid problems. To compare numerical methods, Researchers has used these numerical methods to solve previously solved problems to compare the effectiveness of different numerical methods in different conditions. In the following, some of the used and compared numerical methods have reviewed.

Comparison of pressure base method and artificial compressibility method shows a faster convergence of artificial compressibility method than base pressure method [1]. A characteristic base method is presented for laminar incompressible two-dimensional flows by Drikakis et al. [2]. The proposed method has extended to three-dimensional streams in unorganized networks by Tai et al. [3]. This method has also expanded to compressible and turbulent flows by Nithiarasu et al. [4]. The proposed equations in reference [2] were reviewed by Neofytou and improved equations were introduced which increased the speed of convergence [5]. These equations were re-evaluated more precisely by Su et al. [6]. In another study, the upwind method of the multi-dimensional character base was presented by Razavi et al. [7] to solve the incompressible Navier-Stokes equations. A finite-volume scheme was used in this work. Also, the upwind scheme of the multi-dimensional characteristic introduced in reference [7] was used in the flow with the inverse step and the steady and unsteady flow from the cylinder. In this work, the development of a multi-dimensional base design for calculating convective fluxes in non-Cartesian networks has been presented. A faster convergence and a higher accuracy of the proposed scheme were reported [8].

In another paper [9], a comparison was made between weighted averaging methods using pressure, Roe method, characteristic based method and corrected baseline characteristic method. A case study was conducted on NACA 0012. In the next paper [10], the crater flow around the cylinder is simulated with a characteristic base method and averaging. The higher convergence rate, stability, and greater accuracy were reported for the characteristic base method compared to the averaging. Fifth order Rang-Kutta was used to time discretization. In two separate works, Adibi and Razavi [11, 12] worked on the characteristic based methods and proposes a new characteristic scheme for fluid dynamic with heat transfer. For the calculation of convective fluxes in incompressible flows with artificial compression along with heat transfer, a new method is proposed. In this design, the basic characteristic features of compressible flows to incondensable flows along with heat transfer were expanded. Compatibility equations from the linear combination of governing equations (continuity equation, momentum equations in two directions and energy equation) were obtained after complex mathematical operations. Compatibility equations were discretized on characteristic lines and then convective fluxes were calculated at the cell boundaries using a new proposed method. To illustrate the ability of the new method, codes were written in the FORTRAN software and three benchmark flows were simulated. The flow between the two parallel plates, the flow into the cavity and the flow across the cylinder were simulated with various Reynolds, Prandtl and Grashof numbers. Also, other codes were written to calculate convective fluxes by averaging method. Friction coefficient, drag force, Nusselt number, shear stress, temperature and pressure contours have been reported in these simulations. The results show proposed characteristic based method has acceptable performance. In other work, Adibi et al. [13] simulated a convection heat transfer numerically. They used SIMPLE method. Sheu et al. [14] dealt with the non-stationary pure convection equation in two dimensions. The interfacial numerical fluxes are interpolated by virtue of the rational areas which depend on the corner velocity vectors. This leads to a discrete system containing dissipative artifacts in regions normal to the local streamline. Dick and Steelant [15] made a comparison of the accuracy of the central discretization scheme with artificial dissipation and the upwind flux-difference TVD scheme for the compressible Navier-Stokes equations for high Reynolds number flows. First, a comparison was made on two one-dimensional model problems. Then the schemes were compared on flat plate boundary layer flow. It is shown that a central scheme basically has poor accuracy due to the isotropic nature of the artificial dissipation. An upwind scheme decomposes the flow into different components and adapts the dissipation to the velocity of the components. Soltani et al. [16] presented an upwind unstructured grid cell-centered scheme for the solution of the compressible Euler and Navier-Stokes equations in two dimensions. The algorithm employs a finite volume formulation. Calculation of the inviscid fluxes is based on the approximate Riemann solver of Roe. Viscous fluxes are obtained from solution gradients computed by a variational recovery procedure. Higher order accuracy is achieved through performing a monotonic linear reconstruction of the solution over each cell.

Zadorin and Tikhovskaya [17] conveyed a Cauchy problem for a singular perturbation second-order ordinary differential equation. It is proved that the upwind difference scheme on a grid offered by Shishkin is uniformly convergent. The grid is well known only in application to a boundary value problem. They discussed on the outcomes of selected numerical results. Kabanikhin and Krivorot [18] offered A numerical multistep algorithm for computing tsunami wave front amplitudes. The leading step contains in solving a suitable eikonal equation. They solved the eikonal equation with Godunov's approach and a bicharacteristic schemes. Also they compared the results of two methods. Palymskii [19] numerically simulated two-dimensional and tree-dimensional flows. Their simulated flow was viscous incompressible turbulent flow with heat transfer. Munir et al [20] simulated the flow inside the two sided lid driven cavity using third order upwind compact finite difference method based on flux difference splitting in mixture with artificial compressibility approach. The results were compared with alternate direction implicit finite difference method.

Mostafa and Adel [21] studied a three- dimensional numerical for unsteady, turbulent flow inside the passage of the axial turbine. In this work a three- dimensional Navier-Stokes code was used (CFDRC, 2008) to model the two-phase flow field around a four blades axial turbine. The governing equations were discretized on a structured grid by an upwind difference method. The numerical simulation applied the standard K-ε turbulence model. Bagabir et al [22] investigated turbulent flow and heat transfer characteristics in a three-dimensional ribbed square channels numerically. The governing equations are discretized by the second order upwind differencing method, decoupling with the SIMPLE (semi-implicit method for pressure linked equations) algorithm and are solved by a finite volume method.

In this study, simulation of quasi-one-dimensional supersonic flow in a nozzle has been investigated. The finite-volume method has been used for this simulation for calculation of convective fluxes, upwind method and averaging method with artificial dissipation have been used. In this study two numerical methods and their results are compared. Comparing numerical techniques will allow researchers who do research in the numerical field to use a better method. The simplest method to calculate convective terms is simple averaging. But this method is unstable in most of the cases. For stability of this method, the dissipation terms are used, which makes the method somewhat stable but at the same time, the method becomes more complicated. The method which can be considered as an alternative to this method is the upwind method. This method is required in many cases and does not require dissipation terms. This paper discusses all the above issues in one-dimensional supersonic flows.

2. Equations governing the flow and their discretization

Governing equations are displayed in equation 1 for quasi-one-dimensional supersonic flows. These equations include the continuous equation, momentum and energy equations, and the three auxiliary relationships.

$$\frac{\partial}{\partial t}(\rho S)+\frac{\partial}{\partial x}(\rho u S)=0,$$

$$\frac{\partial}{\partial t}(\rho u S)+\frac{\partial}{\partial x}\left[\left(\rho u^2+p\right)S\right]-p\frac{ds}{dx}=0,$$

$$\frac{\partial}{\partial t}(\rho e_t S)+\frac{\partial}{\partial x}[(\rho e_t+p)uS]=0, \tag{1}$$

$$p=\rho RT,$$

$$e_t=e+\frac{1}{2}u^2=c_v T+\frac{1}{2}u^2=\frac{RT}{k-1}+\frac{1}{2}u^2$$

$$M=\frac{u}{a}=\frac{u}{\sqrt{kRT}}$$

Density, velocity, pressure, total energy, temperature and Mach number are unknown on these six equations. After solving the above equations, these six unknown parameters are obtained. It's very beneficial to work with dimensionless equations therefore first we try to make above equations dimensionless with a proper method. We used equation (2) to make the above equations dimensionless.

$$S^*=\frac{S}{S_{ref}}, u^*=\frac{u}{\sqrt{RT_{ref}}}, x^*=\frac{x}{L_{ref}},$$

$$t^*=\frac{\sqrt{RT_{ref}}\,t}{L_{ref}}, \rho^*=\frac{\rho}{\rho_{ref}}, p^*=\frac{p}{p_{ref}}, \tag{2}$$

$$e_t^{\,*}=\frac{e_t}{RT_{ref}}, T^*=\frac{T}{T_{ref}}$$

In this study, nozzle length, area, temperature, pressure and density at the entrance to the nozzle were selected as reference length, area, temperature, pressure and density. After making equations dimensionless and eliminating *. For simplicity, equations (3) are obtained notice that all parameters in equations (3) are dimensionless.

$$\frac{\partial}{\partial t}(\rho S)+\frac{\partial}{\partial x}(\rho u S)=0,$$

$$\frac{\partial}{\partial t}(\rho u S)+\frac{\partial}{\partial x}[(\rho u^2+p)S]-p\frac{ds}{dx}=0,$$

$$\frac{\partial}{\partial t}(\rho e_t S)+\frac{\partial}{\partial x}[(\rho e_t+p)uS]=0, \tag{3}$$

$$p=\rho T,$$

$$e_t=\frac{T}{k-1}+\frac{1}{2}u^2$$

$$M = \frac{u}{a} = \frac{u}{\sqrt{kT}}$$

In order to be able to use finite-volume methods, these equations must be written integrally.

$$\iiint \left[\frac{\partial}{\partial t}(\rho S) + \frac{\partial}{\partial x}(\rho u S) \right] dvol = 0,$$

$$\iiint \left[\frac{\partial}{\partial t}(\rho u S) + \frac{\partial}{\partial x}\left[\left(\rho u^2 + p\right)S\right] - p\frac{ds}{dx} \right] dvol = 0, \tag{4}$$

$$\iiint \left[\frac{\partial}{\partial t}(\rho e_t S) + \frac{\partial}{\partial x}\left[(\rho e_t + p)uS\right] \right] dvol = 0,$$

Equations (5) are the result of simplifying and using Greens theorem (transforming the triple integral into the integral on the surface) of equations (4).

$$\frac{\partial}{\partial t}(\rho S vol) + \oint \rho u S dA = 0,$$

$$\frac{\partial}{\partial t}(\rho u S vol) + \oint \rho\left(u^2 + p\right) S dA + p\frac{ds}{dx} vol = 0, \tag{5}$$

$$\frac{\partial}{\partial t}(\rho e_t S vol) + \oint (\rho e_t + p) S dA = 0,$$

After the discretization of the above equations on cell i, the relation (6) is obtained.

$$\frac{\rho_i^{n+1} - \rho_i^{n}}{\Delta t} = \frac{1}{vol_i}\left[(\rho u)_I - (\rho u)_{I-1}\right]$$

$$\frac{(\rho u)_i^{n+1} - (\rho u)_i^{n}}{\Delta t} = \frac{1}{vol_i}\left[\left(\rho u^2 + p\right)_I - \left(\rho u^2 + p\right)_{I-1} + \left(p\frac{ds}{dx}\right)_i\right] \tag{6}$$

$$\frac{(\rho e_t)_i^{n+1} - (\rho e_t)_i^{n}}{\Delta t} = \frac{1}{vol_i}\left[(\rho e_t + p)_I - (\rho e_t + p)_{I-1}\right]$$

After the simplification of (6), the relation (7) is obtained.

$$\rho_i^{n+1} = \frac{\Delta t}{vol_i}\left[(\rho u)_I - (\rho u)_{I-1}\right] + \rho_i^{n}$$

$$(\rho u)_i^{n+1} = \frac{\Delta t}{vol_i}\left[\left(\rho u^2 + p\right)_I - \left(\rho u^2 + p\right)_{I-1} + \left(p\frac{ds}{dx}\right)_i\right] + (\rho u)_i^{n} \tag{7}$$

$$(\rho e_t)_i^{n+1} = \frac{\Delta t}{vol_i}\left[(\rho e_t + p)_I - (\rho e_t + p)_{I-1}\right] + (\rho e_t)_i^n$$

Time discretization is forward to make obtained equations explicit. As a consequence, numerical solving of the above equations doesn't require to solve simultaneous equations. At each stage, new information is obtained in the first cell, and then the information in the subsequent cells is also arranged. The main problem in the above relationship is finding convective sentences at the boundary between the two cells. In this paper, two upwind and averaging methods have been used to calculate these terms. In the upwind method, the convective flux is obtained from equation (8).

$$\lambda_I = \lambda_{i-1}, \lambda = \rho, p, u, e_t \tag{8}$$

In the averaging method, the convective fluxes are obtained from equation (9).

$$\lambda_I = \frac{1}{2}(\lambda_{i-1} + \lambda_i), \lambda = \rho, p, u, e_t \tag{9}$$

In Sections 7 and 8, "I" is the cell boundary in which convective sentences are calculated, i-1, i are cell centers where the information is specified. (Figure 1)

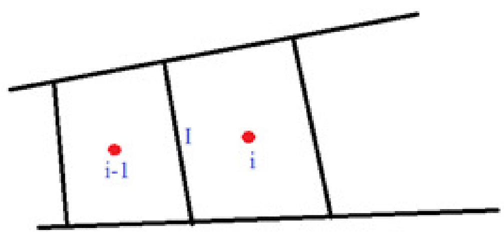

Figure 1. Specifying the naming of the centers and boundaries of the cells

To specify the stability of the two methods, the characteristic lines must be determined in supersonic flows. For this, the linear composition of the governing equations of flow (continuity, momentum, and energy) is considered as the relation (10).

$$\sigma_1 * continuety + \sigma_2 * momentum + \sigma_3 * energy = 0 \rightarrow$$

$$\sigma_1 * (\rho_t + u\rho_x + \rho u_x) + \sigma_2 * (\rho u_t + u\rho u_x + p_x) + \sigma_3 * (p_t + up_x + \rho a^2 u_x) = 0 \rightarrow \tag{10}$$

$$(\sigma_1 + \rho u \sigma_2)\left[u_x + \frac{\sigma_2}{\sigma_1 + \rho\sigma_2}\right] + (\sigma_2 + u\sigma_3)\left[p_x + \frac{\sigma_3}{\sigma_2 + u\sigma_3} p_t\right] +$$
$$(u\sigma_1 - a^2 u\sigma_3)\left[\rho_x + \frac{\sigma_1 - a^2\sigma_3}{u\sigma_1 - a^2 u\sigma_3}\rho_t\right] = 0$$

According to (10), the slope of the characteristic lines is defined as (11).

$$\lambda = \frac{dt}{dx} = \frac{\sigma_2}{\sigma_1 + \rho\sigma_2} = \frac{\sigma_3}{\sigma_2 + \rho\sigma_3} = \frac{\sigma_1 - a^2\sigma_3}{u\sigma_1 + a^2 u\sigma_3} \tag{11}$$

Equation (11) of the three equations with three unknowns will be in the form of (12).

$$\begin{aligned} &\sigma_1\lambda + \sigma_2(u\lambda - 1) = 0 \\ &\sigma_2\lambda + \sigma_3(u\lambda - 1) = 0 \\ &\sigma_1(u\lambda - 1) - \sigma_3 a^2(u\lambda - 1) \end{aligned} \tag{12}$$

$\sigma_1 = \sigma_2 = \sigma_3 = 0$ is an answer for (12). To have non-zero answers for the equations, the determinate of coefficients must be zero, as a result

$$(u\lambda - 1)\left[(u\lambda - 1)^2 - a^2\lambda^2\right] = 0 \tag{13}$$

The above third-degree equation has three answers, which are the slopes of the characteristic lines. The three answers are in the form of relation (14).

$$\lambda_1 = \frac{1}{u}, \lambda_2 = \frac{1}{u+a}, \lambda_3 = \frac{1}{u-a} \tag{14}$$

In the supersonic flows, all three λ are positive, on the other hand in subsonic flows λ_3 is negative. With regard to the positivity of the slope of each of the three characteristics in supersonic flows, the upwind method that obtains convective terms at the cell boundary from the center of the upstream cell is considered a stable method. Despite that, the averaging method that obtains constricted terms on the cell boundary from the mean of upstream and downstream cell information is an unsustainable method. For stability, dissipation terms must be added to this relationship. In this study, forth-order artificial dissipation has been used. For instance, relation 15 has been used to obtain density.

$$\rho_i^{n+1} = \frac{\Delta t}{vol_i}\left[(\rho u)_I - (\rho u)_{I-1}\right] + \rho_i^{\,n} - \frac{\varepsilon}{vol_i}\left(\rho_{i+2}^n - 4\rho_{i+1}^n + 6\rho_i^n + 4\rho_{i-1}^n + \rho_{i-2}^n\right) \tag{15}$$

In the above relation ε is coefficient of artificial dissipation. For inlet boundary conditions, three quantities, such as Mach number, temperature, and pressure, are known, and the remainder is derived from the thermodynamic relationships. For outlet boundary conditions, extrapolation is performed for three parameters and the rest of the quantities are determined by the thermodynamic relationship. The inlet and outlet flow is supersonic. For flow geometry, a divergent nozzle is considered, where the output area varies to input.

3. Results and discussion

In this study, codes were written in FORTRAN to simulate the quasi-one-dimensional supersonic flow. Due to the discrepancies in the previous section, these simulations have been done. Both upwind and averaging methods have been used to calculate convective fluxes. Firstly, the convective fluxes were calculated using the upwind method with different time steps. Boundary conditions have been given at the entrance of the divergent nozzle. The Mach number, the temperature, and the dimensionless pressure at the inlet were known. Other input parameters are calculated from the formulas mentioned in the previous section. For outlet boundary conditions, first-order extrapolation (equation 16) and the second order extrapolation (relationship 17) has been performed.

$$\lambda_{exit} = \lambda_N, \lambda = \rho, p, u, e_t \tag{16}$$

$$\lambda_{exit} = 1.5\lambda_N - 0.5\lambda_{N-1}, \lambda = \rho, p, u, e_t \tag{17}$$

In the above relation, N is the number of cells. After executing the code written for both methods, the values of the quantities are determined in the cell centres. For example, a diagram of changes in the Mach number and pressure is shown in Figures 2 and 3.

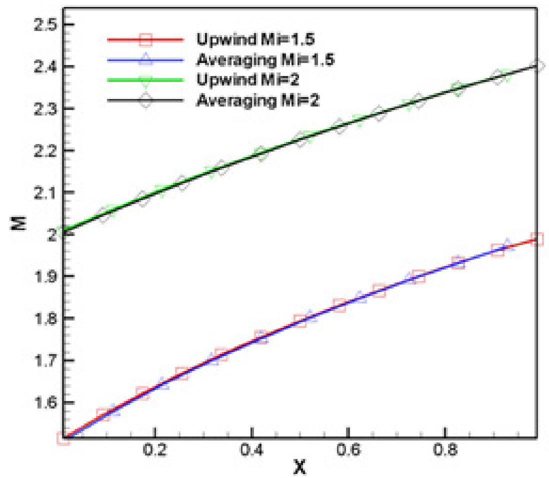

Figure 2. Mach number changes in the upwind and averaging method with different input Mach numbers.

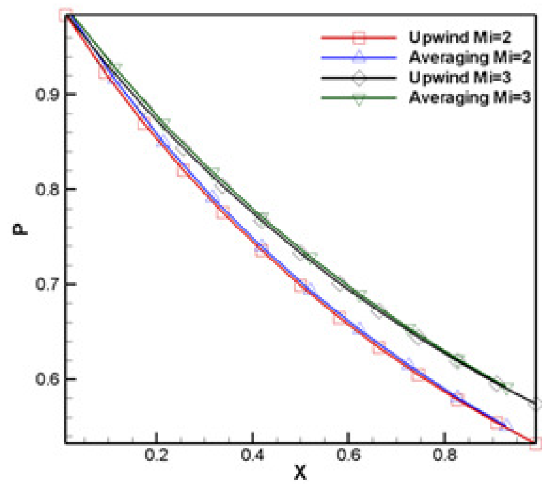

Figure 3. Pressure changes in upwind and averaging method with different input Mach number

As shown in Figures 2 and 3, along the divergent channel the Mach number increases and pressure decreases. Regarding to the relation $\frac{dA}{A} = \frac{dp}{\rho v^2}\left(1 - M^2\right)$. In convergent channels for supersonic flows, velocity and Mach number increases along the duct, while pressure decreases. In Figures 2 and 3, the relation (18) is used to calculate the error.

$$Error = \frac{\sqrt{\sum_{i=1}^{N}\left(p_i^{k+1} - p_1^{k}\right)^2}}{N} \tag{18}$$

The convergence diagram is shown in Figure 4 in the case where the convective fluxes are calculated by the upwind method.

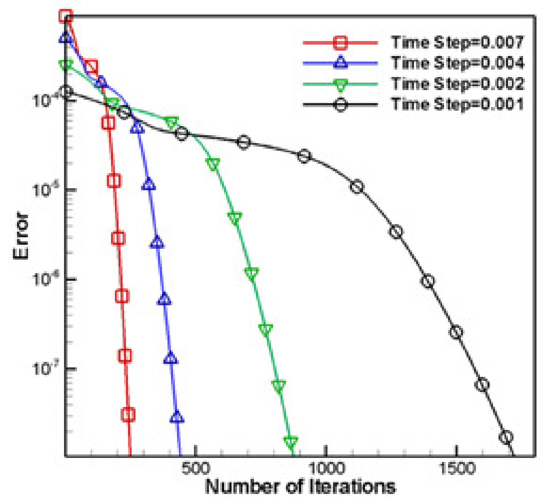

Figure 4. The convergence diagram is in different steps with the upwind method

As shown in Figure 4, convergence has taken place faster as the time steps has increased. But after time step 0.007 no answer obtains due to graph divergence. In the next step, the convective fluxes were calculated by averaging method the graph of convergence in different time steps is shown in Figure 5, in which convective fluxes are calculated using averaging without artificial dissipation.

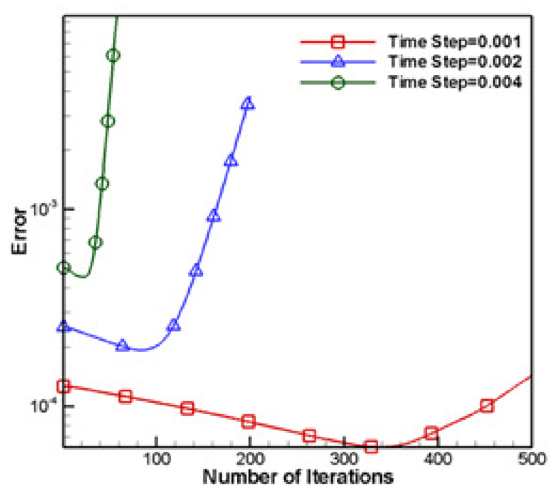

Figure 5. Convergence diagram in different time steps with averaging method without artificial dissipation

As shown in Figure 5, at all-time steps the averaging method is unstable and diverges. The cause of divergence is the nature of the averaging method. Due to the fact that in supersonic flows there are three characteristic lines, all of which with same sign of slope, the information is transmitted only from the upstream to downstream, so that the upwind method corresponds to the physics of the flow and so it converges. For the convergence of the averaging method, according to (9) we can use dissipation terms. So that, in this part of the research, new codes were written to calculate the convective terms using the averaging method along with the dissipation terms. The graph of convergence in different steps of time with the averaging method with artificial dissipation is shown in Figure 6.

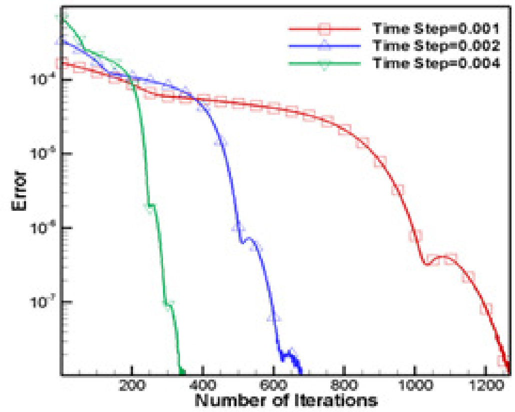

Figure 6. Convergence history in different time steps with averaging method with artificial dissipation

As shown in Figure 6, after the addition of the artificial dissipation terms, the method becomes stable and convergent. Also, as the time step increases, convergence occurs faster, but after a certain time step, the method leads to divergence due to the fact that the method is explicit. To compare the convergence rate of the two sustained methods, new simulations are performed and the results are shown in Figure 7.

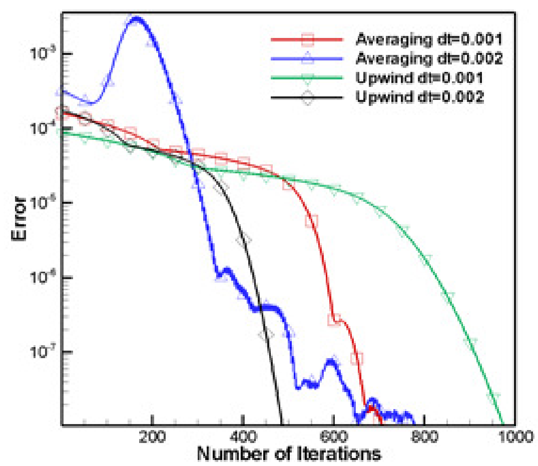

Figure 7. Comparison of the convergence history of upwind method with the averaging method with artificial dissipation

As shown in Figure 7, both methods have a relatively similar convergence history. But the averaging method, along with artificial dissipation terms, becomes unstable sooner. In the next step, the results obtained from numerical methods are compared with the exact results of the analytical method. Relation (19) is established for quasi-one-dimensional isentropic flow inside a nozzle.

$$\frac{A}{A^*} = \frac{1}{M}\left(\frac{1+0.5(k-1)M^2}{0.5(k+1)}\right)^{\frac{0.5(k+1)}{(k-1)}} \tag{19}$$

In the above relation, with the definite area and Mach number in the input, A* is obtained. This variable value is equal in inlet and outlet in isentropic flows. Due to the fact that outlet area is known, The Mach number is obtained by using the above relation again. In order to evaluate the accuracy of the results, the simulation was carried out using upstream method with 50 cells. The numerical results obtained in Table 1 were compared with the exact results and the error rate was determined.

TABLE 1: COMPARISON OF THE RESULTS OBTAINED FROM THE NUMERICAL METHOD AND THE EXACT RESULTS OF THE ANALYTICAL METHOD AND THE CALCULATION OF THE ERROR

Inlet Mach Number	Outlet to Inlet Area Ratio	Exact Outlet Mach Number	Mach Number Obtained from Upwind Method	Relative Error Rate (%)	Mach Number Obtained from Averaging Method	Relative Error Rate (%)
1.1	1.5	1.864	1.858	0.32	1.803	3.3
1.5	1.5	2.053	2.047	0.29	1.994	2.9
2	1.5	2.456	2.453	0.12	2.403	2.2
3	1.5	3.429	3.426	0.09	3.376	1.5

As the results of Table 1 show, the upwind method error is less than the averaging method, which is due to the good performance of upwind methods in supersonic flows that are consistent with problem physics.

The following factor which was in investigated in this study are the effect of artificial dissipation coefficient on the maximum time-step, convergence process and relative error. By re-simulating the flow with the input Mach number 2 and the number of cells 50 with different dissipation coefficients, this effect was determined and shown in Table 2. According to the results, with increasing damping coefficient, larger incremental steps are already selected, therefore convergence speed increases. According to the results, with increasing dissipation coefficient, larger time-steps could be selected, therefore convergence speed increases. But further increase of this number leads to divergence. In simulated conditions, the dissipation coefficient of 0.1 is very good. To verify the independence of the network, simulations were performed in cells number 50 to 200 for both methods and compared together in Figures 8 and 9.

TABLE 2: THE EFFECT OF ARTIFICIAL DISSIPATION COEFFICIENT ON THE MAXIMUM TIME STEP, THE NUMBER OF REPETITIONS FOR CONVERGENCE AND RELATIVE ERROR

Artificial dissipation coefficient	Maximum time-step	Number of iteration for convergence	Relative error
0.001	0.0005	3389	2.16
0.01	0.001	846	2.28
0.1	0.004	180	2.19
0.5	It will not converge at any time-stem		

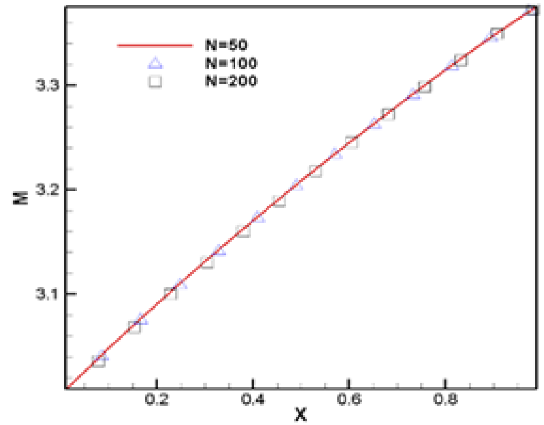

Figure 8. Graph of network independence for upwind method

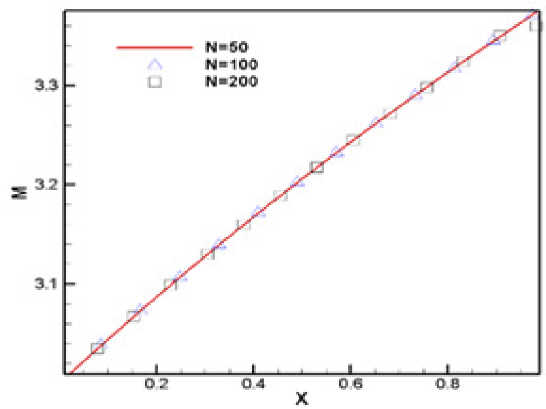

Figure 9. Graph of network independence for averaging method

4. Conclusion

In this paper, supersonic flow in a nozzle was simulated numerically. A finite-volume method was used for discretization. For the time discretization, the progressive method was used to make equations explicit to solve. Convective fluxes were calculated using two averaging methods with artificial dissipation and upwind method. According to the results, the method of averaging is unstable and the codes written couldn't converge to the correct answer. Artificial dissipations were used to stabilize the averaging method. This stabilized the averaging method and led it to convergence. Despite the averaging method, upwind method was a stable method, which had a faster convergence as time-step increased. Also, the results showed that the upwind method have better convergence and accuracy than the averaging method with artificial dissipation.

References

[1] Zhao, Y. and B. Zhang, *A high-order characteristics upwind FV method for incompressible flow and heat transfer simulation on unstructured grids.* Computer methods in applied mechanics and engineering, 2000. **190**(5-7): p. 733-756.

[2] Drikakis, D., P. Govatsos and D. Papantonis, *A characteristic‐based method for incompressible flows.* International Journal for Numerical Methods in Fluids, 1994. **19**(8): p. 667-685.

[3] Tai, C.H. and Y. Zhao, *Parallel unsteady incompressible viscous flow computations using an unstructured multigrid method.* Journal of Computational Physics, 2003. **192**(1): p. 277-311.

[4] Nithiarasu, P. and C.-B. Liu, *An artificial compressibility based characteristic based split (CBS) scheme for steady and unsteady turbulent incompressible flows.* Computer Methods in Applied Mechanics and Engineering, 2006. **195**(23-24): p. 2961-2982.

[5] Neofytou, P., *Short note: Revision of the characteristics-based scheme for incompressible flows.* Journal of Computational Physics, 2007. **222**(2): p. 475-484.

[6] Su, X., Y. Zhao and X. Huang, *On the characteristics-based ACM for incompressible flows.* Journal of Computational Physics, 2007. **227**(1): p. 1-11.

[7] Razavi, S., K. Zamzamian and A. Farzadi, *Genuinely multidimensional characteristic‐based scheme for incompressible flows.* International journal for numerical methods in fluids, 2008. **57**(8): p. 929-949.

[8] Zamzamian, K. and S.E. Razavi, *Multidimensional upwinding for incompressible flows based on characteristics.* Journal of Computational Physics, 2008. **227**(19): p. 8699-8713.

[9] Razavi, S.E. and A.A. Orang, *A comparative investigation of hydrofoil at angles of attack.* International Journal for Numerical Methods in Fluids, 2012. **68**(9): p. 1087-1101.

[10] Orang, A.A., S.E. Razavi and H. Pourmirzaagha, *Computational study of incompressible turbulent flows with method of characteristics.* Journal of Computational and Applied Mathematics, 2014. **259**: p. 741-751.

[11] Adibi, T. and S. Razavi, *A new characteristic approach for incompressible thermo‐flow in Cartesian and non‐Cartesian grids.* International Journal for Numerical Methods in Fluids, 2015. **79**(8): p. 371-393.

[12] Razavi, S. and T. Adibi, *A Novel Multidimensional Characteristic Modeling of Incompressible Convective Heat Transfer.* Journal of Applied Fluid Mechanics, 2016. **9**(3).

[13] Adibi, T., R.A. Kangarluei and V. Farhangmehr, *Numerical Study of Natural Convection Flow inside Squared and Trapezoidal Cavities in Various Conditions.*

[14] Sheu, W., S.-M. Lee and M. Wang, *Multi-dimensional monotone flux discretization scheme for convection dominated flows.* International Journal of Numerical Methods for Heat & Fluid Flow, 1997. **7**(8): p. 814-842.

[15] Dick, E. and J. Steelant, *Accuracy analysis of central schemes with artificial dissipation and upwind flux-difference tvd schemes for navier-stokes equations.* International Journal of Numerical Methods for Heat & Fluid Flow, 1995. **5**(10): p. 877-888.

[16] Soltani, S., K. Morgan and J. Peraire, *An upwind unstructured grid solution algorithm for compressible flow.* International Journal of Numerical Methods for Heat & Fluid Flow, 1993. **3**(4): p. 283-304.

[17] Zadorin, A.I. and S.V.e. Tikhovskaya, *Analysis of a difference scheme for a singular perturbation Cauchy problem on refined grids.* Numerical Analysis and Applications, 2011. **4**(1): p. 36-45.

[18] Kabanikhin, S. and O. Krivorot'ko, *A numerical algorithm for computing tsunami wave amplitudes.* Numerical Analysis and Applications, 2016. **9**(2): p. 118-128.

[19] Palymskii, I., *Simulation of complex regimes of Rayleigh-Benard convection.* Numerical Analysis and Applications, 2011. **4**(2): p. 145-166.

[20] Munir, A., M. Rizwan and A. Shah, *Simulation of incompressible flow in two sided lid driven cavity using upwind compact scheme.* CFD Letters, 2013. **5**(3): p. 57-66.

[21] Mostafa, N.H. and M. Adel, *Unsteady numerical simulation of cavitation in axial turbine.* CFD Letters, 2012. **4**(3): p. 140-149.

[22] Bagabir, A.M., J.A. Khamaj and A.S. Hassan, *Turbulent periodic flow and heat transfer in a square channel with different ribs.* CFD Letters, 2013. **5**(3): p. 67-80.