Investigation The Behaviour of Cell Centered Finite Volume Scheme to The Convergent Divergent Nozzle Flow Problems.

Hasan Taher M.elkamel[1], Dr. Ir. Bambang Basuno [2], and Dr. Norzelawati [3]

Faculty of Mechanical and Manufacturing Engineering,Tun Hussein Onn University of Malaysia (UTHM), Parit Raja, Batu Pahat, 86400 Johor, Malaysia

[1]hassan.elkamel@gmail.com, [2] Bambangb@uthm.edu.my, [3]norzela@uthm.edu.my

Abstract

The present work focuses on the development of computer code which allow one investigate the flow behavior through a convergent - divergent nozzle. The flow is assumed as steady quasi one dimensional inviscid compressible flow. In other word, the flow problem is governed by compressible Euler equation. The Euler equation in steady form may behaves as elliptic partial differential equation or as hyperbolic type of partial differential equation depending on the local Mach number. If the local Mach number $M < 1$, the Euler Equation behaves as elliptic equation while for $M > 1$, it will behave as hyperbolic equation. In one flow domain, with the boundary which separation between two flow domains are not clear, make the Euler equation becomes very difficult to be solved. To avoid such difficulty, one may solve the Euler equation in unsteady form. The present work uses unsteady Euler equation as its governing equation of fluid motion inside the nozzle and cell centered finite volume scheme for their numerical solution. This numerical scheme applied for the case of flow past through two type nozzle models. The first nozzle model follow the nozzle model introduced by Blazek and the second nozzle according to Anderson. Here use the Cell Centered Finite Volume scheme for the purpose of flow analysis on the nozzle. The flow analysis carried out over various flow condition applied to the inlet and outlet section of the nozzle. The result shows that this numerical approach has a limitation. One can not impose arbitrary boundary conditions in solving divergent – convergent nozzle numerically, especially if one use a Cell Centered scheme.

Keywords: Convergent Divergent Nozzle, Cell Centered Scheme, Finite Volume, Computatipnal Fluid Dynamics.

1. Introduction

The convergent divergent nozzle represent a device for expanding flow speed starting from a low subsonic speed to a supersonic speed at the exit station of the nozzle. In manner how to predict the flow behavior along the nozzle can be done by analytical method or numerical method. In assumption that there are no friction effects and the flow is adiabatic flow and in chocked one can formulate the relationship between cross section area and the Mach number. So for a given a convergent divergent nozzle geometry and the flow condition at the reservoir, one can define the Mach number distribution along the nozzle. If the pressure at the nozzle exit station at the time the flow speed at that station is supersonic is equal to P_e. A normal shock wave will occur at a point some where in divergent part of the nozzle if the setting of back pressure P_b is higher than P_e. The position of the normal shock wave is depended on the value P_b. If P_b is equal to P_e, there is no shock wave. If P_b is becoming a higher than P_e the position of normal shock wave will move forward to the throat of the nozzle. The manner how to predict flow behavior along the convergent divergent nozzle with and without a normal shock wave analytically can be found in various compressible flow text books such as given in Ref. 1,2 and 3. In view of solve the flow problem past through a convergent divergent nozzle numerically, one has to start from the governing equation of fluid motion. In assumption that the flow is inviscid and the nozzle cross section area along the main nozzle axis is varying slowly. The governing equation of fluid motion of the flow past through nozzle can be presented in the form of quasi one dimensional compressible flow. This governing equation is known as a Compressible Euler equation and consist of three non linear differential equations which coupling each to other. There are various method had been developed for solving the compressible Euler equation. This equation allows to capture a discontinuity flow phenomena due to a shock wave in their solution if such flow phenomena appear in the flow field. There are various method had been developed for solving the compressible Euler equation such as MacCormack scheme , Steger Warming Scheme, Harten Yee TVD scheme. Such numerical scheme is developed based on finite difference approach. Here the numerical approach for solving the compressible Euler Equations is developed based on a Finite Volume method. The analysis carried out over two nozzle models. The first nozzle model is the nozzle provided by Anderson while the second nozzle model is defined according to Blazek. In view of analytical approach there are no any problems for what ever values of boundary condition provided. Their solution in term of Mach number, pressure, density or velocity distribution along the nozzle can be obtained. However in the case of solving problem use of numerical approach, there are some constraints. In the case of Blazek nozzle, the presence of the normal shock can be captured if the ratio between the exit pressure P_e to P_b has to be greater than 0.5. While for the Anderson nozzle nozzle, such ratio quantity can only be provided for no more than 0.97. If the $\left(\frac{P_e}{P_b}\right) > 0.97$, the numerical scheme diverge and no solution can be obtained. However if the length of the Anderson nozzle is reduced, so the computation starting at station x = 0.75 instead of x=0, the numerical scheme able to produce their solution up the pressure ratio $\left(\frac{P_e}{P_b}\right)$ reach a value of 0.73. Below that value of pressure ration the numerical scheme fail. However for the case of fixed value of pressure ratio applied to this nozzle with different defined inlet station, this numerical scheme give the same result. In other word the starting point of the inlet does not give influence to the solution.

2. Governing Equation Flow Past Through Nozzle.

The derivation of the principal equations of fluid dynamics is based on the fact that the dynamical behavior of a fluid is determined by the following conservation laws, namely: (1). the conservation of mass, (2). the conservation of momentum, and (3). the conservation of energy.

In transforming from those three conservation statements into a mathematical model can be done in two manners, one can express them based on integral approach or differential approach. However in expressing the conservative law into mathematical model, may one introduces some of assumptions which can be imposed due physical flow consideration of the flow problems in hand. The flow passes through a streamline body at relative angle of attack can approximated as the flow problem with no viscous effects. The same situation can be applied as well for the case of the passes through convergent divergent nozzle. Under condition of slowly varying cross section along the main nozzle axis, the air viscosity can be ignored. The governing equation of fluid flow with out viscous effect is known as Euler equation. For the case of flow passes through a convergent divergent nozzle, the Euler equations which can be derived from the conservation of law are written in term of the conservative dependent can be given as:

Continuity equation,

$$\frac{\partial}{\partial t}(\rho S) + \frac{\partial}{\partial x}(\rho u S) = 0 \qquad (2.1)$$

Momentum equation,

$$\frac{\partial}{\partial t}(\rho u S) + \frac{\partial}{\partial x}[(\rho u^2 + p)S] - p\frac{dS}{dx} = 0 \qquad (2.2)$$

Energy equation,

$$\frac{\partial}{\partial t}(\rho e_t S) + \frac{\partial}{\partial x}[(\rho e_t + p)uS] = 0 \qquad (2.3)$$

Where S is the cross-sectional area assumed independent of time, i.e. ,
$S = S(x)$ and the total internal energy e_t can be defined as:

$$e_t = e + \frac{1}{2}u^2$$

Equations (2.1) through (2.3) are expressed in a flux vector and conservation form which in vector notation and compact form as :

$$\frac{\partial}{\partial t}(SQ) + \frac{\partial E}{\partial x} - H = 0 \qquad (2.4)$$

Where

The vector of conserved variable $\vec{Q} = \begin{bmatrix} \rho \\ \rho u \\ \rho e_t \end{bmatrix}$ (2.5a)

Flux vector $E = S\begin{bmatrix} \rho u \\ \rho u^2 + p \\ (\rho e_t + p)u \end{bmatrix}$ (2.5b)

Source term : $H = \frac{dS}{dx}\begin{bmatrix} 0 \\ p \\ 0 \end{bmatrix}$ (2.5c)

Above equation represents a system equation which each equation are coupling to other.
For arbitrary nozzle geometry may the analytical solution for solving such system equations can not be done and so a numerical approach may be required.

3. Cell-centred scheme

If the control volumes are identical with the grid cells and if the flow variables are located at the centroids of the grid cells as indicated in Fig. 3.2, that approach called as Cell-centered scheme . When evaluate the discretised flow equations (3.11), must supply the convective fluxes at the faces of a cell (6). They can be approximated in one of the three following ways:

1: By the average of fluxes computed from values at the centroids of the grid cells to the left and to the right of the cell face, but using the same face vector (generally applied only to the convective fluxes);

2: By using an average of variables associated with the centroids of the grid cells to the left and to the right of the cell face;

3: by computing the fluxes from flow quantities interpolated separately to the left and to the right side of the cell face (employed only for the convective fluxes).

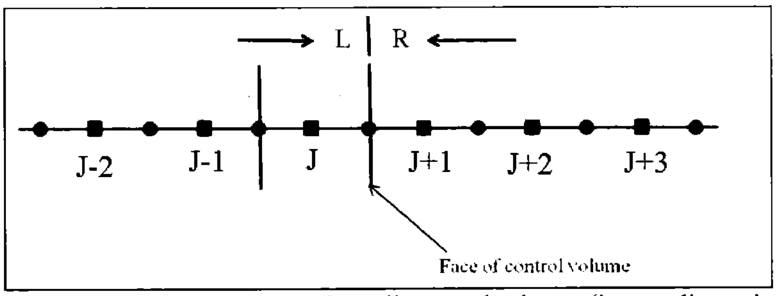

Figure (3.1) Control volume of a cell-centred scheme (in one dimension)

Thus, taking the cell face n_{J+1} ,Fig. 3.1 as an example, the first approach - average of fluxes – can be approximated as:

$$\vec{F}_{AB} = \frac{1}{2}\left(\vec{F}_J + \vec{F}_{J+1}\right) \tag{2.6a}$$

Where

$$\vec{F}_J = \vec{F}(U_J) \tag{2.6b}$$

The second possible approach - average of variables - can be formulated as follows

$$\vec{F}_{AB} = \vec{F}\left(\frac{U_J + U_{J+1}}{2}\right) \tag{2.7}$$

The third methodology starts with an interpolation of flow quantities (being mostly velocity components, pressure, density and total enthalpy) separately to both sides of the cell face. The interpolated quantities - termed the left and the right state.

For the current stage of this study, a computer code of cell-centered scheme finite volume method for convergent-divergent or CD nozzle has been done to get more familiarization on the computer code development of Euler solver (Finite Volume method).

3.1. The algorithm of Cell Centered Scheme Applied For the Nozzle Flow Problems

The cell centered scheme adopted here can be said as the result of combination between time integration Fourth Order Runge Kutta Scheme and spatial discretization based Finite Volume Centered scheme. Such combination gives the algorithm for solving the flow problem passed through convergent divergent nozzle can be summarized as shown in the Flow chart Figure 3.2.

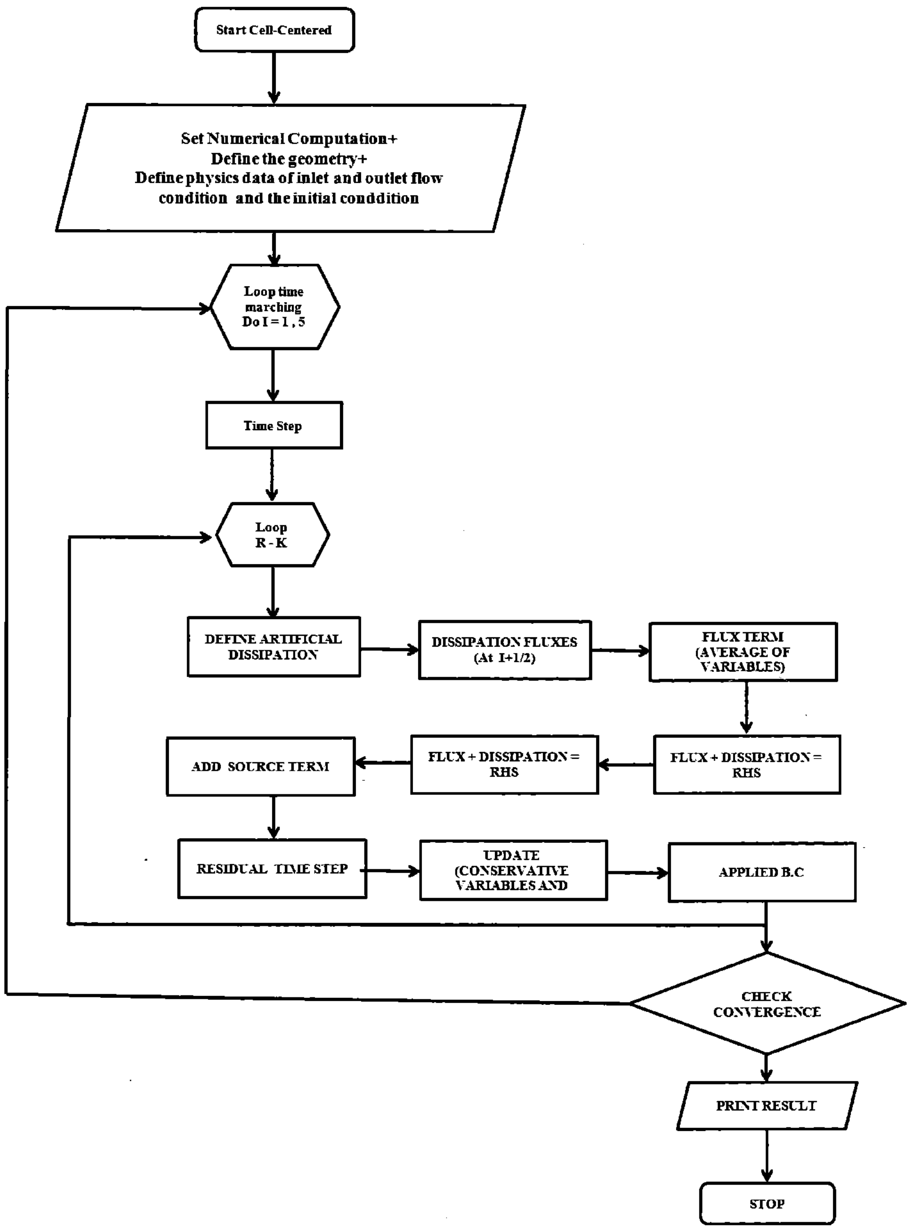
Start Cell-Centered
Set Numerical Computation+
Define the geometry+
Define physics data of inlet and outlet flow
condition and the initial conddition
Loop time marching Do I = 1 , 5
Time Step
Loop R - K
DEFINE ARTIFICIAL DISSIPATION
DISSIPATION FLUXES (At I+1/2)
FLUX TERM (AVERAGE OF VARIABLES)
FLUX + DISSIPATION = RHS
FLUX + DISSIPATION = RHS
ADD SOURCE TERM
RESIDUAL TIME STEP
UPDATE (CONSERVATIVE VARIABLES AND
APPLIED B.C
CHECK CONVERGENCE
PRINT RESULT
STOP

Fig 3.2 Flow chart of cell centered scheme FVM computer code.

Following such flow chart, computer code is written by use of standard computer programming language FORTRAN-77. The developed computer code designed to consist of main program and accompanied with several subroutine and sub functions in order to simplify process of computer code development. the code starts with setting the numerical computation and define the geometry of the nozzle. Figure 3.3 shows the sketch for the grid in the i-direction while Figure 3.5 illiterates the geometry of the nozzle. The shape of nozzle defined according to shape of Nozzle from Blazek .

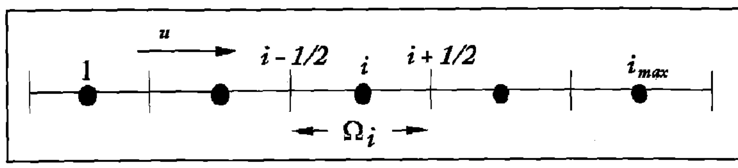

Figure 3.3 : sketch for the grid in the i-direction

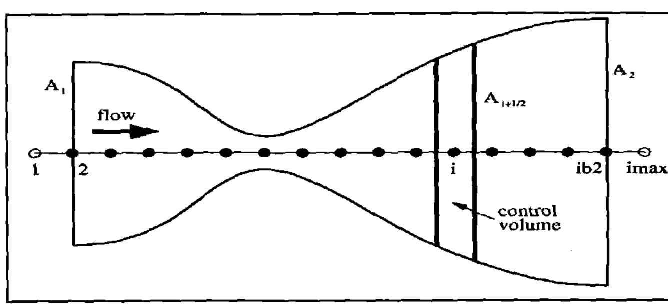

Figure (3.4) Grid and control volume for the 1-D Euler solver; points ***i*** = 1 and ***i*** = *imax* are dummy points

Here along the nozzle divided into N number of segments. The control point X_i is located at the mid element. With the nozzle cross section area denoted by A, volume of the element i with respect to the nozzle cross section area is approximated by :

$$V_i = [X_{i-1} + X_{i+1}]\frac{\Delta A}{8} \quad (3.1)$$

Where :

$$\Delta A = A_{i-1} + 2A_i + A_{i+1}$$

Air properties in view of universal gas constant denoted as R_g and the heat coefficient ratio as γ. The flow condition at the nozzle are depended on the setting up of the flow condition at the entry station and at the exit flow condition. At the entry condition, the flow condition can be described to follow the flow setting condition at the reservoir. So the stagnation pressure P_{01} and stagnation temperature T_{01} are represent the known flow condition, while at the exit station, the known flow condition may be through the setting of the back pressure P_2 . Using those three flow quantities (P_{01} , T_{01} and P_b) and accompanied by knowing air properties γ and R_{gas}, one can define the initial condition along the nozzle. Here the conserved variable $\overrightarrow{C_V}$ is defined :

$$\overrightarrow{C_V} = \begin{bmatrix} \rho A \\ \rho u A \\ \rho e_t A \end{bmatrix} \tag{3.2}$$

The required initial condition basically can be set up arbitrary. For a given the reservoir flow condition P_{01}, T_{01}, and the back pressure P_2, the conserved variable of density ρ, temperature T and the total internal energy can be initialized as follow:

$$\rho = \left(\frac{P_2}{R_q T}\right) \tag{3.3}$$

$$T = T_{01}\left(\frac{P_2}{P_{01}}\right)^{\frac{\gamma-1}{\gamma}} \tag{3.4}$$

$$e = (C_p - R)T_{01} \tag{3.5}$$

To carry out step by step calculation in time, the time incremental ΔT, defined in accordance to:

$$\Delta T = V_i / S_d$$

Where

$$S_d = c * \sqrt{\Delta x^2 + A_i^{\ 2}} + |u| * A_i \tag{3.6}$$

Where c is the speed of sound , u is the local velocity .and A_i is the cross section area at station x_i.

In the loop of the Runge Kutta scheme involves determination of the artificial dissipation Δp , flux averaging $D_{i,j}$, and the total residual time step $RHS_{i,j}$. These three quantities can be defined respectively as :

The artificial dissipation Δp

$$\Delta P = \left|\frac{P_{i+1} - 2*P_i + P_{i-1}}{P_{i+1} + 2*P_i + P_{i-1}}\right| \quad ; \ i=1, 2, 3, \ldots\ldots i_{max-1} \tag{3.7}$$

The flux averaging $D_{i,j}$ defined as:

$$D_{i,1} = \varepsilon_2 * C_{v\,i+1,1} - C_{v\,i,1} - \varepsilon_4 * C_{v\,i+2,1} - 3 * C_{v\,i,1} - C_{v\,i-1,1}$$

$$D_{i,2} = \varepsilon_2 * C_{v\,i+1,2} - C_{v\,i,2} - \varepsilon_4 * C_{v\,i+2,2} - 3 * C_{v\,i,2} - C_{v\,i-1,2}$$

$$D_{i,3} = \varepsilon_2 * C_{v\,i+1,3} - C_{v\,i,3} - \varepsilon_4 * C_{v\,i+2,3} - 3 * C_{v\,i,3} - C_{v\,i-1,3}$$

Where

$$\varepsilon_2 = e_{val} * v_2 * P_{max}$$

$$\varepsilon_4 = e_{val} * v_4$$

While the total residual time step as:

$$RHS_{i,1} = F_{ac} * RHS_{i,1}^{n}$$
$$RHS_{i,2} = F_{ac} * RHS_{i,2}^{n}$$

$$RHS_{i,2} = F_{ac} * RHS_{i,2}^{n}$$

Where

$$F_{ac} = A_{rK} * C_{FL}$$

$$RHS_{i,1} = RHS_{i,1}^{n} - \left(P_i \frac{A_{i+1} - A_i}{2}\right) \tag{3.8}$$

3.2. Result and Discussion.

The investigation on the capability of the Cell Centered Finite Volume Scheme applied to the two nozzle models. The first nozzle model is the nozzle with the nozzle geometry as given by Blazek while the second one is the Nozzle from Anderson . The Blazek nozzle is having a distribution of cross section area along the main nozzle axis **A** defined as:

$$A(x) = 1 + \frac{1}{2}(A_1 - 1)\left\{1 + \cos\left(\frac{\pi x}{0.35}\right)\right\} \qquad for\ 0 \le x \le 0.35 \tag{3.9a}$$

$$A(x) = 1 + \frac{1}{2}(A_2 - 1)\left\{1 - cos\left(\frac{\pi(x - 0.35)}{0.65}\right)\right\} \qquad for\ 0.35 \le x \le 1 \tag{3.9b}$$

In above equation, the constant A_1 is set equal to 1.5 and $A_2 = 2.5$.

The air flow assumed behave as a perfect gas, with heat coefficient ratio $\gamma = 1.4$ and the universal gas constant is equal $R = 1716$ ft $Ib_f/Slugs^0R$ The flow at the entry station is supersonic with the flow condition in term of Mach number, static pressure and temperature at that station are given as:

$P_1 = 1*10^5$ Pa $\quad T_1 = 288$ K

The result for different value of back pressure P_b had been carried out. Different value of back pressure will give different position of the normal shock wave location will occurred. Figure (3.5) shows the result of distribution Mach number along the Nozzle for a given pressure back $P_b = 0.9\ 10^5$ N/m^2 0.7 10^5 N/m^2 and 0.5 10^5 N/m^2. The problem with this nozzle is the Cell Centered Finite Volume does not able to produce the result if the back pressure is set below than 0.5 10^5 N/m^2. Decreasing value of back pressure meant that solution would produce the location shock will goes to near the exit station. This unsuccessful solution may due to the nozzle geometry can be able the flow over the whole flow domain in isentropic flow condition.

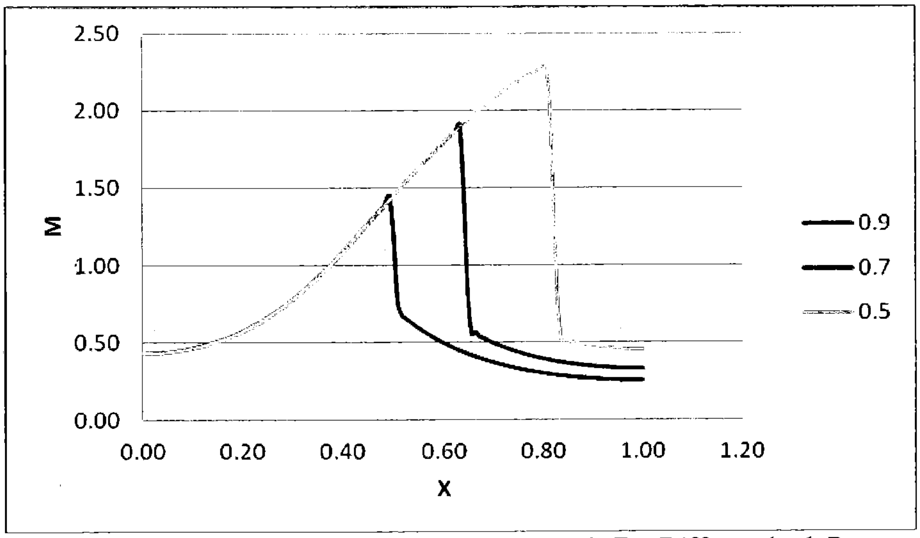

Figure 3.5 Mach Number Distribution over Blazek Nozzle For Different back Pressure ratio.

The second nozzle geometry according to Anderson is given by the distribution of the cross section area A according to :

$$A = 1 + 2.2\,(x - 1.5)^2 \qquad (3.2)$$

Here the nozzle has a 3 units length with the geometry from side view as depicted in Figure (3.6),

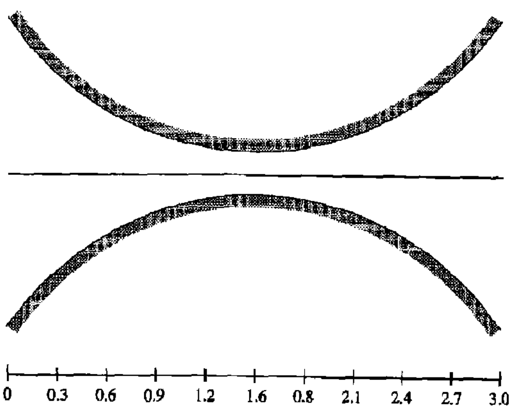

Figure. 3.6: The Anderson Nozzle Geometry.

The entrance condition had been applied as it had been applied to the Blazek nozzle. If the Blazek nozzle the flow analysis can be done up the back pressure reach the value of 0.5 10^5 N/m^2. However for this case, the back pressure cannot be set up less than 0.97 10^5 N/m^2. Below this value, the cell centered scheme fail to produce their results. Figure (3.7) shows the distribution of Mach number along the Nozzle for the back pressure at 0.99 10^5 N/m^2, 0.98 10^5 N/m^2, and 0.97 10^5 N/m^2.

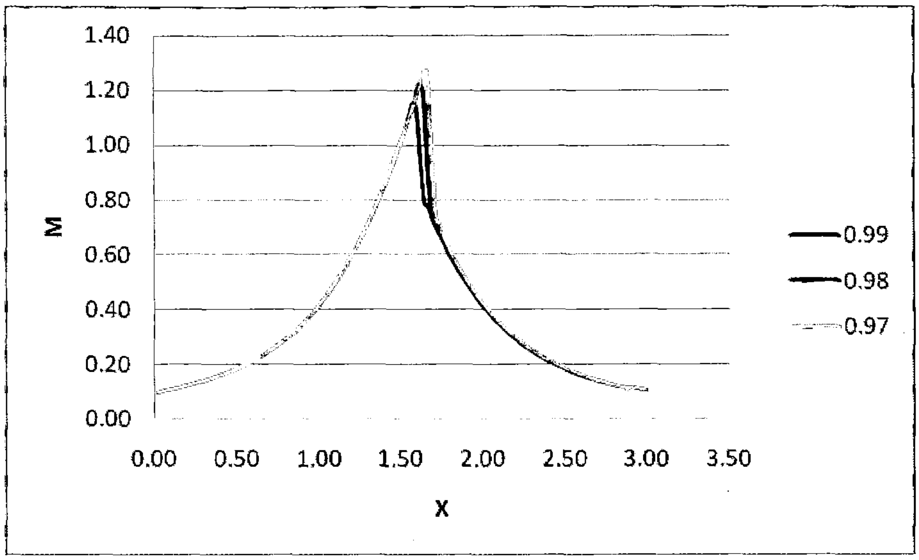

Figure 3.7 Mach Number Distribution over Anderson Nozzle For Different back Pressure ratio.

The previous calculation is carried out over the whole length of the nozzle starting from x = 0 to x = 3. For a given entrance flow condition as used in Blazek nozzle, it had been found that the Cell Centered Scheme is only be able solve this nozzle problem if the back pressure is greater than 0.97 10^5 N/m^2.

However it the nozzle is shortened so the nozzle problem under investigated only for the flow domain in between x = 0.75 to x = 3, with the flow condition at entry station x = 0.75 and the Mach number M = 0.25, the flow analysis can be done up to back pressure P_b is equal to 0.73 10^5. The result for different back pressure greater than to 0.73 10^5 in term of Mach number distribution along the nozzle as shown in the Figure 3.8.

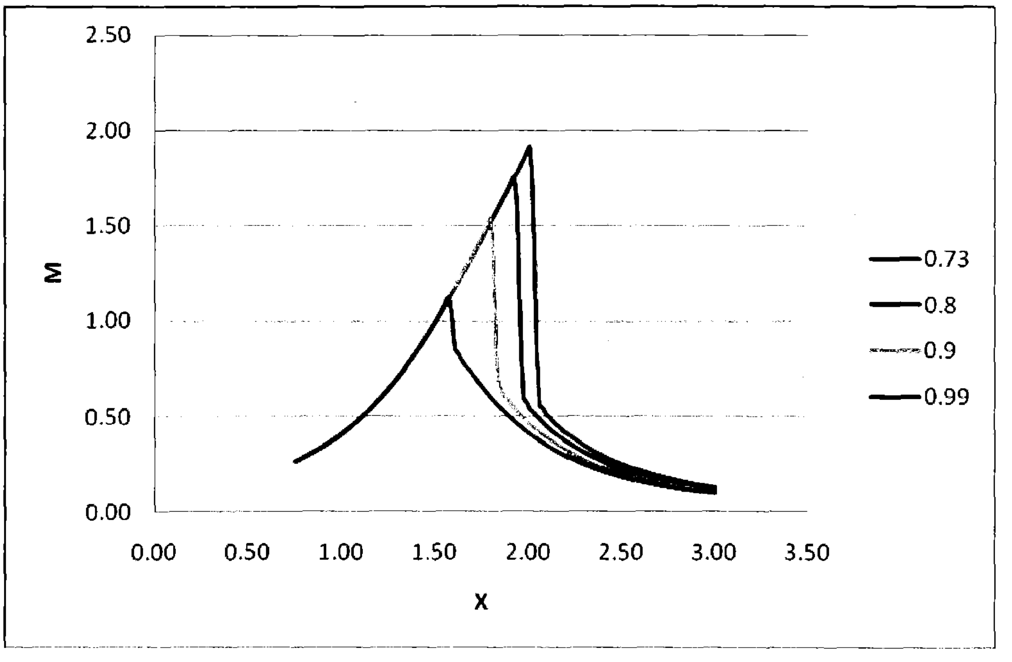

Figure 3.8 Mach Number Distribution if the Back pressure P_b kept a fixed value at 0.94 10^5 while the length of the Anderson nozzle is varied.

Figure 3.8 shows the result if the Back pressure P_b kept a fixed value at 0.94 10^5 while the length of the Anderson nozzle is varied. The investigation found that the cell centered work well if the starting point for defining the entry station should start from x = 0.2. Starting point for the entry station less than that position will make the cell centered scheme fail to produce their solution.

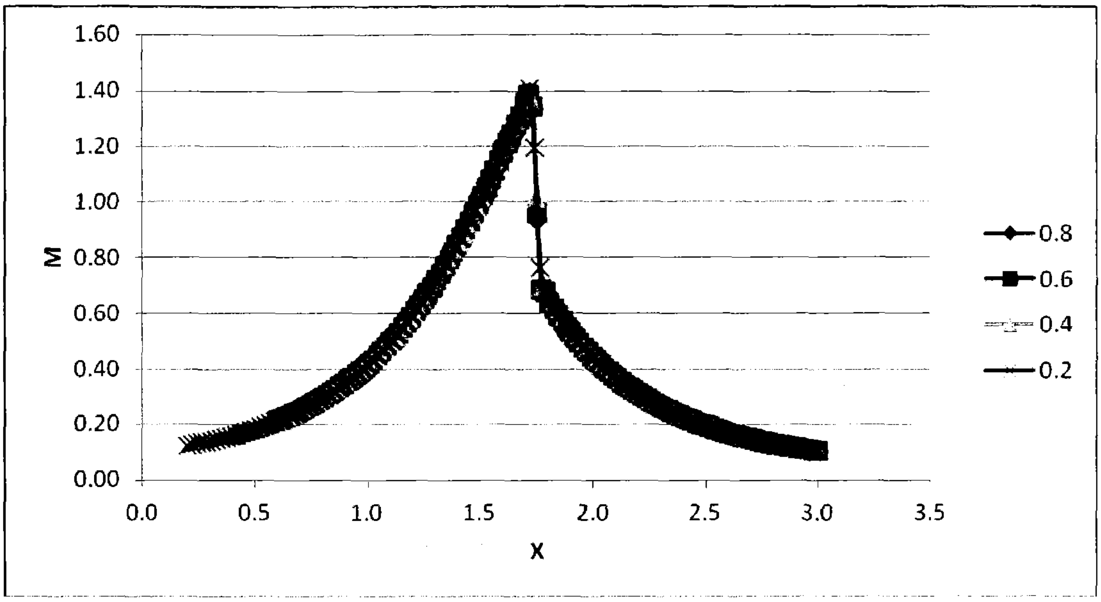

Figure 3.8 Effect on the length of the nozzle.

Conclusion

Considering the result as presented in the discussion and result, it can be concluded that one can not choose a nozzle geometry and their boundary condition arbitrary. In view of analytical approach one· will able to carry out a convergent divergent nozzle flow analysis for any prescribed boundary condition and get their result. Such approach may cannot be done if one try to solve that problem by use of numerical approach such as use of Cell Centered Finite Volume Scheme as it had been done in the present work.

References

1. Anderson J. D., Jr (2002). *Modern Compressible Flow: With Historical Perspective,* McGraw-Hill Series in Aeronautical and Aerospace Engineering.

2. Anderson J. D. (1995). *Computational Fluid Dynamics, the basics with applications*, McGraw-Hill.

3. Blazek J. (2008) *Computational Fluid Dynamics: Principles and Applications*, Elsevier science.

4. Hirsch C.(2007), *"Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics*, 2nd Edition.

5. Hoffmann A. K. and Chiang T.S.(2000) *Computational Fluid Dynamics, Vol. 2,* Engineering Education System; 4th edition .

6. Hodge B. K. and Keith Koenig (1995). *compressible fluid dynamics with personal computer applications*, Prentice Hall College Div.

7. James E.A. John , Theo G. Keith(2006). *Gas Dynamics*, Prentice Hall; 3 edition.

8. Pletcher R. H. , Tannehill J.C, and Anderson D (2011). *Computational Fluid Mechanics and Heat Transfer",* Taylor & Francis; 3rd edition.