PAPER • OPEN ACCESS

The Study on the Influence of Boundary Condition Applied to the Case of Flow Past through Bump Model.

To cite this article: Nurhayati Rosly *et al* 2019 *J. Phys.: Conf. Ser.* **1150** 012055

View the article online for updates and enhancements.

You may also like

- Modeling shallow water waves
  D Lannes

- On the global existence and blowup of smooth solutions to the multi-dimensional compressible Euler equations with time-depending damping
  Fei Hou and Huicheng Yin

- Madelung, Gross–Pitaevskii and Korteweg
  Rémi Carles, Raphaël Danchin and Jean-Claude Saut

The Study on the Influence of Boundary Condition Applied to the Case of Flow Past through Bump Model.

Nurhayati Rosly[1*], Bambang Basuno[1], Sofian Mohd[1], Mohd Shalahuddin Adnan[2]

[1]Aircraft System and Design Research Focus Group, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Batu Pahat,Johor, Malaysia
[2]Faculty of Civil and Environmental Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, Malaysia

*Email: nurhayati@uthm.edu.my

Abstract. The present work focused on solving two dimensional compressible Euler equation based on cell-cantered finite volume discretization technique with the convective terms appear in the Euler equation were evaluated by using Roe scheme [4]. These methods used for solving the flow past through bump with different setting on their boundary conditions. Six different setting of boundary conditions had been applied; three of them are able to provide a convergent solution, while three others tent to diverge. It is clear the result shows that boundary condition plays an important role in determining the flow solution.

1. Introduction

The way how to solve the external flow problem can be done in various manners, since there are various form of governing equation of fluid motion such as the Navier Stokes equation, Time Averaged Navier Stokes Equation, Parabolized Navier Stokes, Thin Shear Layer equation, Euler Equation, Full Potential equations, Boundary layer equation etc. Each of governing equation has its own method in the way how to solve it. If the flow problem in hand is showing that viscous effects can be ignored, may one can solve the flow problem by solving the Euler Equation, or Full Potential equation or Transonic Small Perturbation equation or by solving the governing equation of fluid motion in the form of Laplace Equation. The last equation represents the governing equation of fluid motion for the case of inviscid, irrotational and incompressible flow problem. It is the lowest of mathematical flow model, while the Navier Stokes equations represent the highest level of the governing equation of fluid motion. All physical flow phenomena may appear in the flow field surrounding the immersed body can be captured if one able to solve it. In line with the advancement of computer technology and the progress in numerical method had made all mentioned type of governing equation of fluid motion are now numerically solvable. The Navier Stokes equations can be solved by Direct Numerical Simulation method (DNS) [1] or by Large Eddy simulation method (LES) [2]. The Time Averaged Navier Stokes Equation including its derivatives (Parabolized Navier Stoke equation or Thin Shear Layer Equation) can be solved by the method such as McCormack Scheme, Beam and Warming Scheme or TVD scheme [3]. These mentioned methods can also be used for solving the Euler equation. While for the lowest level of the governing equation of fluid motion described by the

Laplace equation including the compressibility effects can be solved by the numerical method called as Panel Method. In this respect, there are various Panel Methods had been developed such as Source Panel Method, Doublet Panel Method, Vortex Panel Method, the Combined Source-Vortex panel method [4] etc.

The present work focused on solving two dimensional compressible Euler equation based on cell-centered finite volume discretization technique with the convective terms appear in the Euler equation were evaluated by using Roe scheme [4]. These methods used for solving the flow past through bump with different setting their boundary conditions. Six different setting boundary conditions had been applied, the most of them are able to provide a convergent solution but the other three tent to diverge. It is clear the result shows that boundary condition plays an important role in determining the flow solution.

2. Methodology

2.1 Governing Equation of Fluid Motion

The governing equation of compressible inviscid two dimensional flows can be written in the integral form for a given region Ω and boundary S as

$$\frac{\partial}{\partial t} \int_\Omega Q \, d\Omega + \int_S F ds = 0 \tag{1}$$

where the vector of conserved variables Q and the vector convective fluxes F are:

$$Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho e \end{bmatrix}; \quad F = \begin{bmatrix} \rho u \vec{V} \\ \rho u \vec{V} + n_x p \\ \rho v \vec{V} + n_y p \\ \rho H \vec{V} \end{bmatrix}; \tag{2}$$

Where;

$$\vec{V} = n_x u + n_y v \tag{3}$$

In the above equations, u and v are the Cartesian components of the velocity vector $\vec{V}$ in the x and y directions, respectively; ρ is density and p represents the static pressure. e and H are the total energy and total enthalpy per unit of mass. The Notation Ω and S represent the volume and surface area respectively, while n_x and n_y are the Cartesian components of the exterior surface unit normal vector on the boundary ds. In addition, that the air flow behaves as an ideal gas, one can define the relationship between pressure p and total energy e and H as:

$$p = (\gamma - 1)\left[e - \frac{1}{2}\rho(u^2 + v^2)\right] \text{ and } H = (e + p) \tag{4}$$

where γ is the ratio of specific heats.

2.2 Roe Scheme

Equation (1) describes a relationship where the time rate of change of the state vector Q, within the domain Ω, is balanced by the net flux F across the boundary surface ∂S. Discretization of the Euler equations in integral form is obtained by subdividing the computational domain Ω into separate quadrilateral cell Ω_{ij}, i =1,2,…,Ni, and j = 1,2,…, Nj, and by requiring the conservation laws for each finite volume separately. For particular cell i,j, Eq. (1) can be written as [6]:

$$\frac{dQ_{i,j}}{dt} = -\frac{1}{\Omega_{i,j}}\sum_{m=1}^{4} F_m \Delta S_m = R_{i,j} \tag{5}$$

In above equation the notation $R_{i,j}$ is called as residual, $\Omega_{i,j}$ is the area of cell i,j , while Δs is the length of segment of the mth face. In obtaining the residual term R_{ij}, Roe [5] use an approximate Riemann solver, in which the convective flux F_m at the face of a control volume can be written as:

$$\left(F_m\right)_{i+\frac{1}{2},j} = \frac{F(Q_R)+F(Q_L)}{2} - \frac{1}{2} A_{i+\frac{1}{2},j}\left(Q_R - Q_L\right) \tag{6}$$

Where:

$$A\left(Q_R - Q_L\right) = \sum_{k=1}^{4}\left|\Delta F_k\right| \quad ; \quad \left|\Delta F_1\right| = \left|\overline{\overline{\vec{V}}} \cdot \overline{\overline{c}}\right| \left(\frac{\Delta p - \overline{\overline{\rho}}\,\overline{\overline{c}}\Delta\overline{\overline{\vec{V}}}}{2\overline{\overline{c}}^2}\right)\begin{bmatrix} 1 \\ \overline{\overline{u}} - \overline{\overline{c}}n_x \\ \overline{\overline{v}} - \overline{\overline{c}}n_{yx} \\ \overline{\overline{H}} - \overline{\overline{c}}\overline{\overline{\vec{V}}} \end{bmatrix}$$

$$\left|\Delta F_{2,3}\right| = \left|\overline{\overline{\vec{V}}}\right|\left\{\left(\Delta p - \frac{\Delta\rho}{\overline{\overline{c}}^2}\right)\begin{bmatrix} 1 \\ \overline{\overline{u}} \\ \overline{\overline{v}} \\ \frac{\overline{\overline{q}}}{2} \end{bmatrix}\right\} + \left(\overline{\overline{\rho}}\left|\overline{\overline{\vec{V}}}\right|\right)\begin{bmatrix} 01 \\ \Delta u - \Delta\vec{V}n_x \\ \Delta u - \Delta\vec{V}n_y \\ \overline{\overline{u}}\Delta u + \overline{\overline{v}}\Delta v - \overline{\overline{\vec{V}}}\,\Delta\vec{V} \end{bmatrix}$$

$$\left|\Delta F_4\right| = \left|\overline{\overline{\vec{V}}} + \overline{\overline{c}}\right|\left(\frac{\Delta p - \overline{\overline{\rho}}\,\overline{\overline{c}}\Delta\overline{\overline{\vec{V}}}}{2\overline{\overline{c}}^2}\right)\begin{bmatrix} 1 \\ \overline{\overline{u}} + \overline{\overline{c}}n_x \\ \overline{\overline{v}} + \overline{\overline{c}}n_{yx} \\ \overline{\overline{H}} + \overline{\overline{c}}\overline{\overline{\vec{V}}} \end{bmatrix} \tag{7}$$

The Roe – averaged variables in above equation denoted by superscript double bar are computed by the following formula:

$$\overline{\overline{\rho}}_{i+\frac{1}{2}} = \sqrt{\rho_R \rho_L} \;\; ; \qquad\qquad \overline{\overline{u}}_{i+\frac{1}{2}} = \frac{\sqrt{\rho_R}\,u_R + \sqrt{\rho_L}\,u_L}{\sqrt{\rho_R} + \sqrt{\rho_L}}$$

$$\overline{\overline{v}}_{i+\frac{1}{2}} = \frac{\sqrt{\rho_R}\,v_R + \sqrt{\rho_L}\,v_L}{\sqrt{\rho_R} + \sqrt{\rho_L}} \;\; ; \qquad \overline{\overline{H}}_{i+\frac{1}{2}} = \frac{\sqrt{\rho_R}\,H_R + \sqrt{\rho_L}\,H_L}{\sqrt{\rho_R} + \sqrt{\rho_L}}$$

$$\overline{\overline{c}}_{i+\frac{1}{2}} = \sqrt{(\gamma - 1)\left(\overline{\overline{\overline{H}}} - \frac{\overline{\overline{u}}^2 + \overline{\overline{v}}^2}{2}\right)_{i+\frac{1}{2}}} \quad ; \quad \overline{\overline{\overrightarrow{V}}}_{i+\frac{1}{2}} = \overline{\overline{u}}_{i+\frac{1}{2}}n_x + \overline{\overline{v}}_{i+\frac{1}{2}}n_y \tag{8}$$

In view of cell centered finite volume method, where the flow properties at k^{th} cell, the Eq. (5) can be integrated, by using a Fourth order Runge Kutta scheme in the form [5]:

$$
\begin{aligned}
Q_k^{(0)} &= Q_k^n \\
Q_k^{(1)} &= Q_k^{(0)} - \frac{1}{4}\frac{\Delta t_k}{\Omega_k}\left[R\left(Q_k^{(0)}\right) - D\left(Q_k^{(0)}\right)\right] \\
Q_k^{(2)} &= Q_k^{(0)} - \frac{1}{6}\frac{\Delta t_k}{\Omega_i}\left[R\left(Q_k^{(1)}\right) - D\left(Q_k^{(1)}\right)\right] \\
Q_k^{(3)} &= Q_k^{(0)} - \frac{3}{8}\frac{\Delta t_k}{\Omega_k}\left[R\left(Q_k^{(2)}\right) - D\left(Q_k^{(2)}\right)\right] \\
Q_k^{(4)} &= Q_k^{(0)} - \frac{1}{2}\frac{\Delta t_k}{\Omega_k}\left[R\left(Q_k^{(3)}\right) - D\left(Q_k^{(3)}\right)\right] \\
Q_k^{n+1} &= Q_k^{(4)}
\end{aligned}
\tag{9}
$$

The second term $D(Q_k)$ is called as dissipation term, the manner how to evaluate this term one can refer to the Ref. 5 for its detail derivation. The calculation procedure as given by Eq. (9) is repeated from one-time step to other time step which finally ended to converge solution. In this respect, one may use convergence criteria by measuring the difference of the density variable from two consecutive time steps follows the following a relationship as given below:
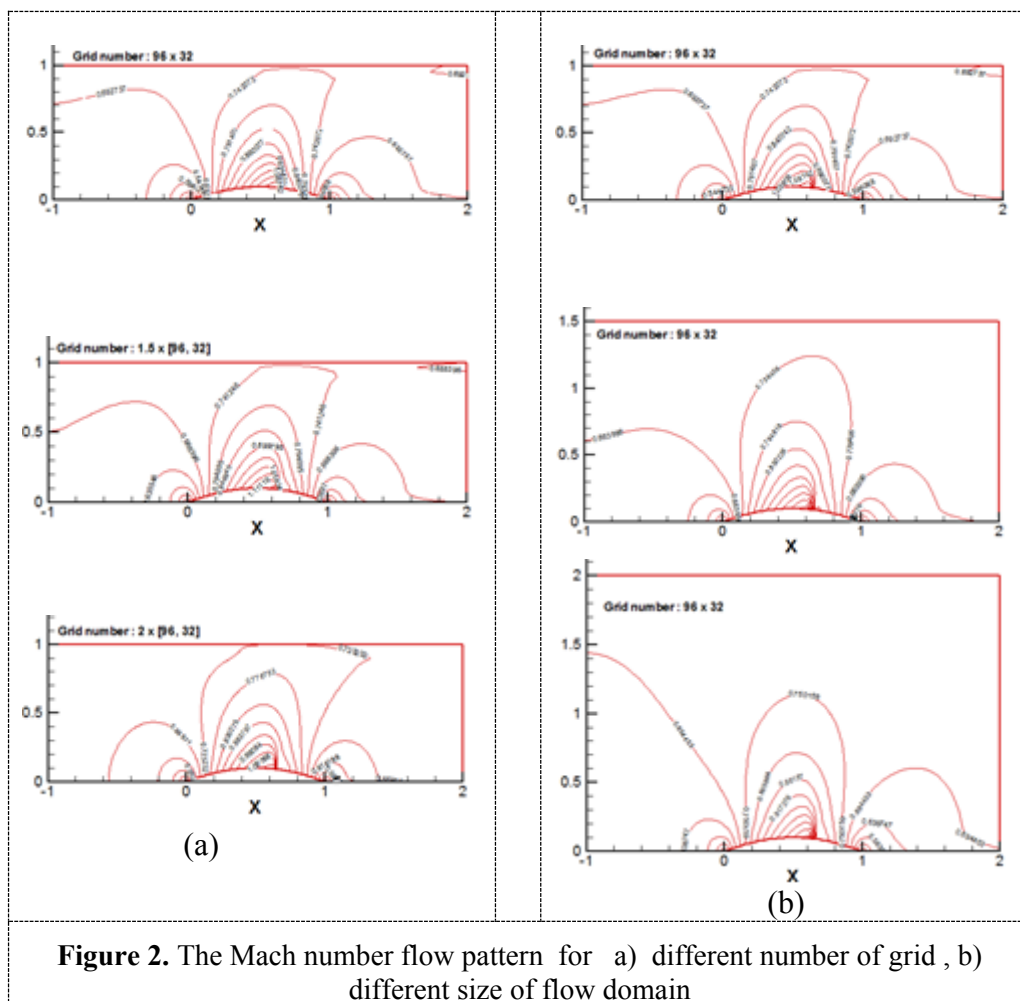
$$\frac{\Delta\rho_k^{n+1}}{\Delta\rho_k^1} = \left[\frac{\sqrt{\sum_{k=1}^{N}\left(\rho_k^{n+1} - \rho_k^n\right)^2}}{\sqrt{\sum_{i=1}^{N}\left(\rho_k^n - \rho_k^0\right)^2}}\right] \le 0.01 \tag{10}$$

3. Results and Discussion

The flow past bump model had been studied by various researchers such as Uygun [6] and Chima [7]. In implementing the Finite Volume method one firstly define the topology of the flow domain. For external flow problems, the grid topology of flow domain may in the form of O-grid, C-grid or H-grid [8]]. However for the case of flow past through bump, one can simply introduce an appropriate grid topology is in rectangular form. Figure 1 shows the meshing of the flow domain in structured grid with each cell has a quadrilateral shape. The size of flow domain is 3 unit of length in x-direction and 1 unit in y – direction. The line AB, BC, CD, DE, EF and FB represent the line where the boundary condition may be imposed. Line BC represent the bump with the maximum bump thickness is 10 % of chord

Figure 1. The Mesh Flow Domain in the flow past through a bump model.
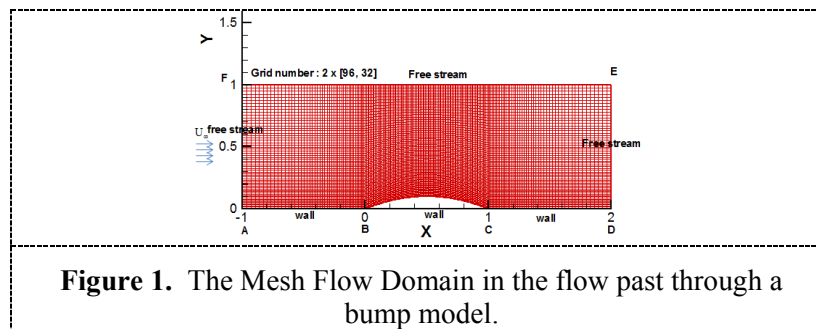
As external flow problem, the incoming velocity is set to have the Mach number of free stream M_∞ = 0.7 and the flow properties defined according to standard atmospherics at sea level. First analysis carried out by considering the flow problem in has a bottom line AD considered as the wall while for other boundary lines [DE, EF and FA] act as far field. The implementation of cell centered Roe scheme finite volume to this flow problem for three different number of cells, 96 x 32, 1.5 x 96 x 32 and 2 x 96 x 32, give the result in term of the Mach number over the flow field as shown in Fig. 2.a While Figure 2.b shows the comparison result, if the flow problem treated with different size of flow domain but having the same number of cells. Three size of flow domains under investigation are a rectangular 3 x 1 unit of length, 3 x 1.5 and 3 x 2 unit of length with number of cell is 96 x 32 cells.

(a)

(b)

Figure 2. The Mach number flow pattern for a) different number of grid , b) different size of flow domain

Considering result as shown in these two figures, one can identify the size of flow domain as well as the number of cells do not give a significant difference result. Here one may conclude the setting of the size of flow domain 3 x 1 unit of length combined with 92 x 32 cells number are sufficient enough for solving this flow problem. Using this setting, the effects boundary conditions are carried out.

There various boundary condition can be applied in solving the external flows such as a wall boundary condition, a symmetrical boundary condition, cut off boundary condition or free stream (far field) boundary condition. Detail in manner how the boundary condition can be defined mathematically, one may refer to Blazek [9]. The result as shown in Fig. 2a in which the boundary conditions for the flow problem consist of wall (along line AD) and far field (along line DE, EF and FA) taken as references and called as the flow problem case – 1. For other setting boundary condition as given in Table 1. Hence there six cases of flow problem.

Table 1. Different type of setting boundary conditions used in solving the flow past through bump model.

Flow problem	AB	BC	CD	DE	EF	FA
			Line			
Case - 1	Wall	Wall	Wall	Far field	Far field	Far field
Case - 2	Symmetrical	Wall	Symmetrical	Far field	Far field	Far field
Case – 3	Far field	Wall	Far field	Far field	Far field	Far field
Case – 4	Cut off	Wall	Cut off	Far field	Far field	Far field
Case - 5	Symmetrical	Wall	Symmetrical	Far field	Symmetrical	Far field
Case - 6	Symmetrical	Wall	Symmetrical	Far field	Wall	Far field

Table 2 show the iteration number required for the converge solution. The required number of iteration 10000 means that the developed computer by using this Roe scheme for the corresponding flow problem not yet converges. The first three setting boundary conditions give a convergent solution while the other three produce diverge solutions.

Table 2. The required number of iteration for a convergent solution.

Case no	Flow problems					
	1	2	3	4	5	6
Iteration no	972	971	936	10000	10000	10000
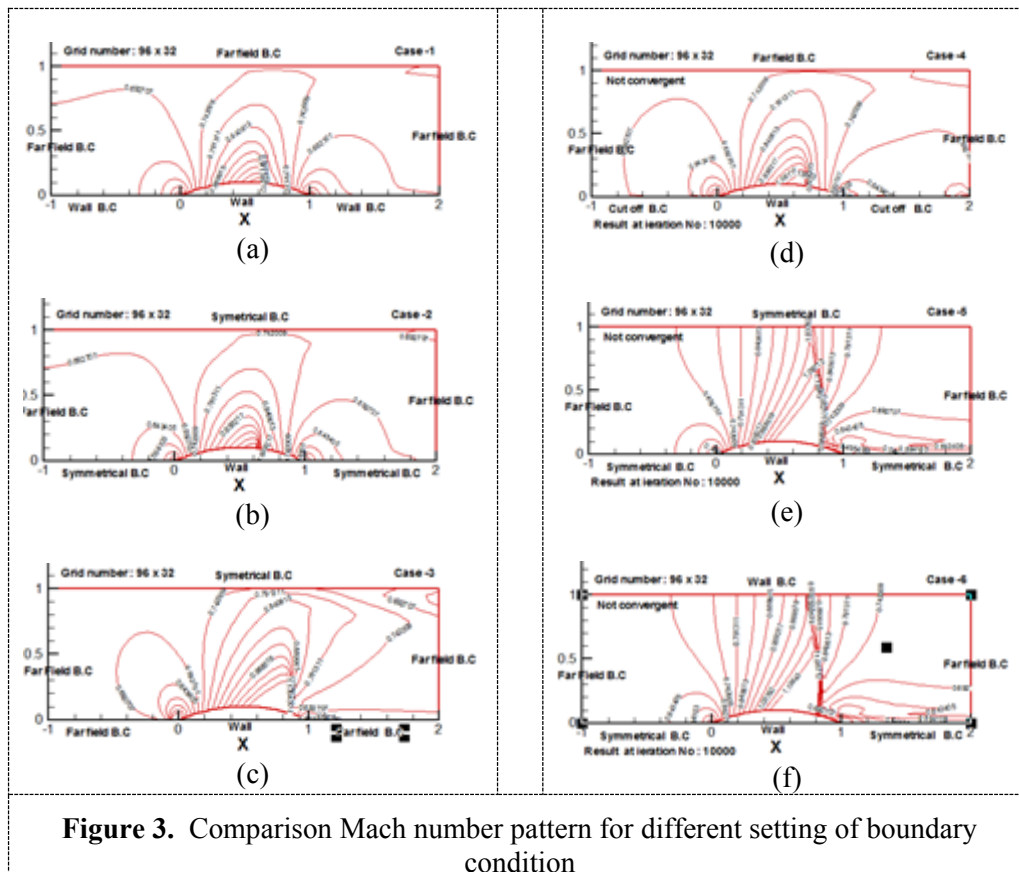
Figure 3. Comparison Mach number pattern for different setting of boundary condition

4. Conclusions

The boundary condition plays an important role in solving flow problem. The Roe scheme had been recognized as a high accurate and good resolution for capturing shock wave. However, this scheme would not be able to produce a proper flow solution if the boundary conditions were not well defined.

Acknowledgements

References

[1]   Ferziger, Direct J.H. and *Large Eddy Simulation of Turbulence. In "Numerical Methods in Fluid Mechanics"*, A. Vincent (ed.), Centre de Recherches MathCmatiques Universitk de MontrCal, Proceedings and Lecture Notes, **16**, pp. 53-97.

[2]   Piomelli, U 1999 *Large-Eddy Simulation of Turbulent Flows*, VKI Lecture Series 1998-05,

[3]   Hirsch C 1988 *Numerical Computation of Internal and External Flows* Vols. **1 and 2**, (John

Wiley and Sons)

[4] Katz J, and Plotkin A 1991 *Low Speed Aerosynamics* (Cambridge University Press)

[5] Maciel E S D 2005 *Comparison Between a Centered and a Flux Difference Split Schemes Using Unstructured Strategy* (Journal of the Brazilian Society of Mechanical Sciences) Vol. **XXVII**

[6] Uygun M, Kirkköprü K 2005 *Numerical solution of the euler equations by finite volume methods*: central versus upwind schemes (Journal of Aeronautics and Space Technologies) vol. **2**: p 47-55

[7] Chima R.V, Turkel E, Steve S S. Comparison of three explicit mulligrid methods for the euler and navier-stokes equations "*NASA TM 88878*, **19878**.

[8] Thompson J F, Warsi Z U A, Mastin C W 1985 *Numerical Grid Generation:* Foundations and Applications (Elsevier Science)

[9] Blazek J 2015 *Computational fluid dynamics*: principles and applications. 3rd (Butterworth Heinemann)