Improvement of a solution of inviscid compressible flows using a mixed wall boundary condition

A. Madadi*, M.J. Kermani and H. Khazaei

Department of Mechanical Engineering, Amirkabir University of Technology (Tehran Polytechnic), Tehran, Iran, P. Code 15875-4413

(Received 6 March 2014; final version received 6 October 2014)

Boundary conditions have a vital role in the numerical solution of the partial differential equations governing fluid flows, and they have a great influence over the numerical stability and accuracy of the final solutions. One of the most important physical boundary conditions in flow field analysis, especially for inviscid flows, is the wall boundary condition imposed on body surfaces. To solve a three-dimensional compressible Euler equation (with five PDE’s), a total of five boundary conditions on the body surface should be prescribed. The wall’s velocity magnitude is one of the parameters to be determined, and the way this velocity magnitude is calculated affects the accuracy and stability of the numerical approach. In this paper, four different methods for calculation of the wall velocity magnitude are introduced, tested and compared against several test cases of subsonic and supersonic flows. Since there are many problems where both subsonic and supersonic flows coexist, a mixed boundary condition takes into account the flow Mach number is proposed. The mixed boundary condition is applied to several test cases and the stability of the method is examined.

Keywords: wall boundary condition; inviscid; mixed boundary condition

1. Introduction

Design and analysis of systems which deal with fluid flows are of interest to many researchers. With the increased calculation speed of new computers, CFD has become an important tool for fluid engineers. This increased speed has facilitated the research of flow patterns in a number of studies. Yu, Shademan, Barron, and Balachandar (2012) used CFD to study the effect of nozzle geometry on incompressible flow through four nozzles with different turbulence models and compared the results to previous experimental data. Other researchers applied CFD to design and optimization of geometries used in industrial systems. Idahosa, Golubev, and Balabanov (2008) developed and automated design tools for turbo machinery parts of propulsion systems. They used CFD coupled with optimization tools to improve the efficiency of a low speed fan. Liu, Dang, and Xi (2008) used CFD to improve the performance of a centrifugal fan. They optimized the profile of linkage between the inlet duct and impeller.

The accuracy and stability of CFD models are very important especially in optimization systems. Several researchers have focused on development of new numerical methods and CFD models to improve the accuracy of simulations. For example, Qamar, Hasan, and Sanghi (2010) presented a new spatial discretization method for convective terms in the hyperbolic equations of fluid dynamics.

It is obvious that boundary conditions have a significant role in the stability and accuracy of numerical methods. An improper boundary condition may lead to a non-physical solution and also the divergence of the numerical method. The body surface is one of the most important boundary conditions which should be implemented carefully. Kim, Jeon, Kim, Kwon, and Lee (2001) showed that the solution accuracy and stability of Roe’s FDS scheme is affected by the wall boundary condition.

The effect of different wall boundary conditions in two-dimensional coordinates with three types of upwind schemes; Stager-Warming’s flux vector splitting, Roe’s flux difference splitting, and Liou’s advection upstream splitting method (AUSM) (Liou & Steffen, 1993), was investigated in Kim and Kim (2000). Wang and Yuzhi (2002) developed a curvature-based wall boundary condition for inviscid flows. They used the projected adjacent node velocity vector of the wall, and using the surface curvature and computed velocity magnitude on the wall, obtained the wall pressure.

Marshall and Ruffin (2004) presented a new wall boundary condition for subsonic and supersonic flows using an embedded boundary Cartesian solver. For subsonic flows, they also used the projected velocity from the adjacent node, and for computation of pressure and temperature on the wall the gas dynamic relation between static and total pressures and temperatures was applied. In the supersonic cases, they used the oblique shock and expansion fan gas dynamic relationship regarding the angle between the adjacent velocity vector and the surface direction.

*Corresponding author. Email: ali.madadi@gmail.com

© 2015 The Author(s). Published by Taylor & Francis. This is an Open Access article distributed under the terms of the Creative Commons Attribution License (http://creativecommons.org/licenses/by/4.0/), which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.
Khazaei, Madadi, and Kermani (2011) applied four types of wall boundary conditions on subsonic and supersonic test cases. They used four methods to modify velocity magnitude on the walls and found that the method of correction of the velocity magnitude greatly affected the accuracy and stability of the solution. They proposed two different methods for subsonic and supersonic flows.

The objective of this paper is to suggest an accurate, stable, and physical wall boundary condition for three-dimensional, compressible, and inviscid subsonic and supersonic flow analysis. On the wall, five boundary conditions should be imposed. The momentum equation in the normal direction is used to compute the pressure on the wall (Hoffmann & Chiang, 2000). For the no-heat source and sink conditions, the total temperature through the domain remains constant. Hence, as a second condition, the total temperature on the wall is set equal to the inlet $T_0$.

To compute the velocity components, the no penetration condition through the wall is imposed as the third equation. To complete the equations, the velocity vector from an adjacent node is projected onto the wall and the velocity direction on the wall is obtained. Four different methods for correcting the wall velocity magnitude introduced by Khazaei et al. (2011) are investigated here. These four methods are applied to six benchmarks which consist of the following subsonic and supersonic test cases. Two consecutive $90^\circ$ elbows, NACA65-410 airfoil, and subsonic flow over a bump are used to study subsonic flows; and a converging diverging nozzle with supersonic outflow, transonic, and supersonic flow over a bump are used as supersonic test cases.

The results show that the third method is more convenient for subsonic flows while for supersonic problems the second type gives better accuracy and stability. Since both subsonic and supersonic flows can be presented in the same simulation, it is important to provide the correct boundary conditions on the wall. Therefore, in this work, we derive a new wall boundary condition that results from a combination of methods two and three (explained before), taking into account the Mach number on the adjacent node. This mixed boundary condition was applied to different test cases and produced results with acceptable accuracy and stability.

NOMENCLATURE

| Symbol | Definition                                      |
|--------|------------------------------------------------|
| $C_p$  | Specific heat in constant pressure [J/kg.K]     |
| $E$    | x-Direction flux vector                        |
| $e_t$  | Total energy [J/kg]                            |
| $F$    | y-Direction flux vector                        |
| $G$    | z-Direction flux vector                        |
| $h_t$  | Total enthalpy [J/kg]                          |
| $n$    | Unit vector                                    |
| $NC$   | Not Corrected                                  |

2. Governing equations

The Euler equations in three-dimensional, unsteady, inviscid and conservative form for compressible flow are given as follows:

$$\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = 0$$

where,

$$Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho e_t \end{bmatrix}$$,

$$E = \begin{bmatrix} \rho u \\ \rho u^2 + P \\ \rho uv \\ \rho uw \\ \rho e_t \end{bmatrix}$$,

$$F = \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + P \\ \rho vw \\ \rho vh_t \end{bmatrix}$$,

$$G = \begin{bmatrix} \rho w \\ \rho uw \\ \rho vw \\ \rho w^2 + P \\ \rho wh_t \end{bmatrix}$$

Q represents the conservative vector; and E, F, and G represent the flux vectors in the x, y, and z-directions, respectively. The equations are completed with equation of state,

$$P = \rho RT$$

3. Solver description

To solve the inviscid flow field, a recently developed in-house code based on the flux difference splitting (FDS) scheme of Roe (1981) is used. The governing equations
are discretized in the computational domain using formulations presented by Kermani and Plett (2001a). In this paper, the primitive variables are extrapolated to the cell faces by the MUSCL idea using the third order upwind biased scheme. Time integration is done using a fourth order explicit Runge–Kutta scheme with the local time stepping method.

The van Albada, van Leer, and Roberts (1982) flux limiter is used to prevent spurious numerical oscillations. Metrics of transformation are used to convert the equations from the physical domain to the computational domain. The Roe scheme gives non-physical expansion shocks in the regions where the eigenvalues of the Jacobian matrix vanish, e.g., the sonic regions and the stagnation points. To obtain a physical solution under the abovementioned conditions, an entropy correction formula from Kermani and Plett (2001b) is used.

4. Grid generation

In the present work an in-house combined algebraic-elliptic algorithm is developed and used for grid generation (Hoffmann & Chiang, 2000). To impose the grid orthogonality on the body surfaces and clusterings near the wall, the corresponding control functions are considered in the elliptic algorithm for complicated geometries. Because the governing equations are discretized in the computational domain, metrics of transformation are needed to transform equations from the physical domain to the computational domain (Kermani & Plett, 2001a). These metrics of transformation are computed using a second-order finite difference formula.

5. Inviscid wall boundary conditions

To solve three-dimensional Euler equations five boundary conditions on each surface, such as a wall, are needed. For an inviscid flow velocity components on the wall can be calculated using the slip condition. Therefore, the contravariant velocity components, except normal to the surface, can be extrapolated to the wall from the interior. This is shown in Figure 1. The equations for contravariant velocity components on the wall are given here:

\[
\begin{align*}
\text{on the Wall} \quad & u_{c,w} = (\xi_x u + \xi_y v + \xi_z w)_w \\
& v_{c,w} = (\eta_x u + \eta_y v + \eta_z w)_w \\
& w_{c,w} = (\zeta_x u + \zeta_y v + \zeta_z w)_w
\end{align*}
\]

where \( \xi_x, \xi_y, \xi_z, \eta_x, \eta_y, \eta_z, \zeta_x, \zeta_y, \) and \( \zeta_z \) are the metrics of transformation. Because no penetration through the solid wall is permitted, the normal component of velocity vector is set to zero, i.e:

\[
\vec{V} \cdot \hat{n}_W = 0
\]

where \( \hat{n}_W \) is the unit vector normal to the wall. Hence, one of the following cases will occur:

(i) the \( \xi \)-constant surface is a wall \( u_{c,w} = 0 \)
(ii) the \( \eta \)-constant surface is a wall \( v_{c,w} = 0 \)
(iii) the \( \zeta \)-constant surface is a wall \( w_{c,w} = 0 \)

For example, consider a \( \eta \)-constant wall, the velocity equations on the wall are rewritten as:

\[
\begin{align*}
& u_{c,w} = (\xi_x u + \xi_y v + \xi_z w)_{adj} \\
& v_{c,w} = (\eta_x u + \eta_y v + \eta_z w)_w = 0 \\
& w_{c,w} = (\zeta_x u + \zeta_y v + \zeta_z w)_{adj}
\end{align*}
\]

The Crammer formula can be applied to the system of linear equations to compute the velocity components on the wall surface. These equations force the flow vector to be tangent to the wall surface.

Since there are no heat sources or sinks in the test cases studied here and the flow is steady, the total temperature on the wall is equal to the incoming flow. So the wall static temperature can be computed from the stagnation temperature as:

\[
T_W = T_{0,\text{In}} - \frac{u_w^2 + v_w^2 + w_w^2}{2C_p}
\]

Because of the wall surface curvature the zero pressure gradient normal to the wall could not be used here. In Hoffmann and Chiang (2000), using the momentum equation in the direction normal to the solid wall the relation for the pressure in two-dimensional coordinates is given. Similarly, the computations for three-dimensional cases are performed and applied here.

With the slip condition, no-penetration, constant total temperature, and pressure relation described above, the boundary conditions on the wall can be determined. The
wall velocity magnitude (Equation (8)) is an important parameter which affects the flowfield results.

\[ V_t = \sqrt{u_w^2 + v_w^2 + w_w^2} \]  
(8)

Four different methods for correction of the velocity magnitude on the wall are investigated in this paper. To consider the velocity magnitude on the wall, a correction factor is defined as:

\[ \alpha_{corr} = \frac{V_{t,corr}}{V_{t,NC}} \]  
(9)

where, \( V_{t,NC} \) refers to the uncorrected velocity magnitude computed from Equations (6) and (8), and \( V_{t,corr} \) is the corrected velocity magnitude. In the first method, the projected velocity vector with no correction is used. In this method one of the contravariant velocity components is set to zero; therefore the computed velocity magnitude on the wall is less than the adjacent node which may not be compatible with the flow physics in some flow fields.

In the second method, the same velocity magnitude of the adjacent node is applied to the wall. The velocity magnitude on the curved wall would not be remained constant; hence, this method has non-physical results in test cases with curved walls.

The constant mass flux condition for correcting velocity magnitude is used in the third method. In this method the following constraint is applied:

\[ \rho_{Adj}V_{t,Adj} = \rho_wV_{t,w} \]  
(10)

or

\[ V_{t,w} = \frac{\rho_{Adj}V_{t,Adj}}{\rho_w} \]  
(11)

The correction factor for this type can be computed from the following equation:

\[ \alpha_{III} = \frac{V_{t,w}}{V_{t,NC}} = \frac{\rho_{Adj}V_{t,Adj}}{\rho_wV_{t,NC}} \]  
(12)

Applying constant total pressure is the fourth method considered here. If the grid size on the wall is fine, the Bernoulli equation for steady frictionless (\( P_0 = \text{cte} \)) flow along a streamline can be written as:

\[ dV_t + \frac{dP}{\rho V_t} = 0 \]  
(13)

so, the increment of velocity magnitude of the two neighbor nodes on the wall is,

\[ V_t = \sqrt{u_w^2 + v_w^2 + w_w^2} \]  
(14)

In this case, according to the velocity direction (upwinding scheme) along each grid line on the wall, for example \( \xi \)-Line and \( \zeta \)-Line on a \( \eta \)-Constant wall, the velocity magnitude is computed.

\[ \begin{align*}
V_{t,\xi} &= \left( V_{t,Adj} - \frac{P_w - P_{Adj}}{\rho V_{t,Adj}} \right) \xi \\
V_{t,\zeta} &= \left( V_{t,Adj} - \frac{P_w - P_{Adj}}{\rho V_{t,Adj}} \right) \zeta
\end{align*} \]  
(15)

and,

\[ V_{t,Ber} = \frac{|u_c|V_{t,\xi} + |w_c|V_{t,\zeta}}{|u_c| + |w_c|} \]  
(16)

And \( \alpha_{IV} \) is computed as,

\[ \alpha_{IV} = \frac{V_{t,Ber}}{V_{t,NC}} \]  
(17)

Since the correction factor is computed, the velocity vector can be modified as below,

\[ \vec{V}_{W,Cor} = \alpha \vec{V}_{W,NC} \]  
(18)

It should be noted that the corrected velocity should be used to compute the wall static temperature and pressure. These four methods of velocity magnitude correction are summarized in Table 1.

6. Results and discussion
Here, the effects of the introduced wall boundary conditions on the accuracy and convergence rate of the flow field solutions are investigated. To do so, four subsonic and supersonic test cases are studied:

- Subsonic flow through two consecutive 90° elbows.
- Isentropic flow through a converging-diverging nozzle with an outlet-throat area ratio of 16.

| No. | The Correction Method of the Wall Velocity Magnitude | Correction Factor |
|-----|----------------------------------------------------|-------------------|
| I   | Not Corrected (NC)                                 | \( \alpha = 1 \) |
| II  | Constant Velocity Magnitude (\( V_t = \text{cte} \)) | \( \alpha = \frac{V_{t,Adj}}{V_{t,NC}} \) |
| III | Constant Mass Flux (\( \rho V_t = \text{cte} \))    | \( \alpha = \frac{\rho_{Adj}V_{t,Adj}}{\rho_wV_{t,NC}} \) |
| IV  | Constant Total Pressure (\( P_0 = \text{cte} \))   | \( \alpha = \frac{V_{t,Ber}}{V_{t,NC}} \) |

Table 1. The correction factors for each calculation method of the wall velocity magnitude.
Subsonic, transonic and supersonic flow over a bump. Subsonic flow through a NACA65-410 compressor cascade.

The flow field of the test cases is solved using all introduced types of wall boundary conditions and the performance of the approach is studied. Here, the results for each test case are presented and compared.

Each test case has been studied using different initial conditions. If the solution converges for all initial conditions the method is called “stable” in the following sections. On the other hand, if the solution diverges for all initial conditions it is called “unstable,” and if the method converges for some initial conditions and diverges for other ones it is called “sensible to initial conditions.”

### 6.1. Subsonic flow through two consecutive 90° elbows

In this section, the subsonic flow field through two consecutive 90° elbows is studied. For the elbow test case, all types of wall boundary conditions are imposed and the solution is achieved. A sample of the results of the elbows in Figure 2 is presented by contours of static pressure and Mach number containing velocity vectors. As shown, the velocity on the inner surface is greater than the outer surface which leads to higher static pressure on the outer wall.

In Figure 3, the velocity magnitude contours for four types of wall boundary conditions are presented. As it is seen in Figure 3(a) for the uncorrected method, the trend of velocity magnitude toward the wall is reversed. For the inner wall, the velocity magnitude on the wall is
smaller than the adjacent interior node. Similarly, for the outer wall, the velocity magnitude on the wall is greater than the adjacent interior node which is incompatible with the expected physical trend. Here, there is a discontinuity in the constant velocity magnitude lines. In the second type of wall boundary conditions, the velocity magnitude is extrapolated from the adjacent interior node; hence in Figure 3(b) the constant velocity lines are normal to the wall. Similar to Figure 3(a), a nonphysical solution on the wall is achieved. In Figure 3(c) and (d), the physical trend is continued to the wall and the velocity contours are continuous. So, the third and fourth types of wall boundary conditions make a better prediction of the velocity magnitude on the wall. It should be noted that the solution with the fourth type of wall boundary condition was sensitive to the initial conditions, i.e., using some initial conditions the method diverges. In summary the third type of wall boundary condition, \((\rho V_t = \text{cte})\), gave the best accuracy and stability for the elbow test case.

6.2. Isentropic flow through a converging-diverging nozzle

In this section, flow through a converging-diverging nozzle is investigated. For this test case, the third method of wall boundary conditions \((\rho V_t = \text{cte})\) diverges and the results for the other three methods are presented here.

In Figure 4, samples of the results are presented by static pressure and Mach number contours. In Figure 5, the numerical results on the nozzle center line are compared with the analytical solution of isentropic flow. The results show good agreement. In the diverging section of the nozzle, the flow is supersonic and at the outlet the Mach number is about 4.5.

In Figure 6, contours of velocity magnitude for the nozzle are shown. As it is seen, the first and second type of wall boundary conditions make smoother contours, the
third type results in divergence of the solution, and for the fourth type there are some oscillations in the results.

As a conclusion, it can be noted that for the converging-diverging nozzle, the first and second types of wall boundary conditions gives better stability and continuity of the results.

6.3. Subsonic channel flow over a bump

The subsonic channel flow over a bump is a benchmark for validation of numerical methods and boundary conditions for inviscid flow regimes. A 10% thick bump at the bottom of the channel is studied for subsonic and transonic flow. If the inlet Mach number is less than a certain (or critical) value, the flow is subsonic in the channel. If a decrease in back pressure results in the increase of the inlet Mach number to the critical value, the flow is transonic and a sonic line appears over the bump. So increasing the inlet Mach number causes a supersonic pocket to grow on the top of the bump. For the channel flow over the bump, the results are compared with numerical solutions presented by Ni (1982).

Here, the back pressure to the inlet total pressure ratio is selected in such a way that the inlet Mach number is equal to 0.5. The flow through the duct is subsonic. The solution with all types of wall boundary conditions converged. In Figure 7, the Mach number contours for the four types of boundary conditions are shown. As can be seen in Figure 7(c), the third type (Figure 7(c)) gives smoother contours, while a discontinuity occurred near the wall for the others. The discontinuity is more obvious in Figure 7(d) where the disturbances are expanded through the domain.

In Figure 8, the Mach number distribution on the upper and lower walls for all four types of boundary conditions are shown and compared with the results of Ni (1982). As seen, there is a major difference between the results of the fourth type and the others that is consistent with Figure 7. The oscillation in the results for the fourth type of wall boundary condition is more obvious here.

Finally, it can be concluded that for a subsonic flow over the bump, the third type of wall boundary conditions shows better accuracy, while its stability is as good as the first and second types.

6.4. Transonic channel flow over a bump

The transonic flow over the bump is studied in this section. As mentioned before, by decreasing the back pressure, the inlet Mach number will increase and reach a critical Mach number. In this case, a normal shock appears on the bump. The ratio of back pressure to the inlet total pressure is selected to have an inlet Mach number of 0.675.
In Figure 9, the Mach number contours of transonic flow over the bump for all types of boundary conditions are presented. As you can see, a normal shock occurs over the bump and the flow become subsonic. In Figure 9(a), the trend of the Mach number near the wall is reversed. Physically, the velocity magnitude near the wall must be increased and a constant total temperature leads to a lower static temperature; so, a higher Mach number near the bump surface is anticipated. As mentioned above, in the first type of boundary condition the velocity magnitude on the wall is less than the adjacent node and in this case it is not consistent with the physical trend.

In Figure 9(b), as expected, the iso-Mach number line is normal to the wall. Although the trend seen in Figure 9(b) does not match with the physics, with respect to the first type of boundary conditions it gives a better solution. In Figure 9(c), the best compatibility with the physical trend is observed and the iso-Mach number lines are continuous through the wall. In Figure 9(d), the Mach number on the wall is over estimated and again a discontinuity occurs in the Mach number contours.

In Figure 10, comparison between Mach number distribution over the upper and lower walls for the four types of boundary conditions are presented and compared with the results of Ni (1982). As expected, the first type gives the lowest Mach number over the walls. If the wall is straight or its curvature radius is very large, the difference between velocities obtained using the four types
of boundary conditions studied in this paper is not sensible. Because the upper wall is straight and the lower wall has a curved surface, the difference between Mach number distributions is more visible over the lower wall on the bump section.

In conclusion, for this test case, the third type has the best prediction of velocity on the wall surfaces.

### 6.5. Supersonic channel flow over a bump

In this section, the supersonic flow over the bump is studied. For the supersonic flow, the thickness of the bump is selected as 4% of the channel height. In this case the oblique shock waves and the reflections from the walls are investigated. The inlet Mach number is set at 1.4 and the outlet is set as supersonic outflow boundary conditions.

For the supersonic flow, the solution with the third type of boundary conditions does not converge; hence, its results are not presented here.

In Figure 11, the Mach number contours are presented. For all types of wall boundary conditions, the Mach number contours near the wall are discontinuous. The discontinuity for the first and second types is in the reversed direction and the Mach number on the wall is underestimated, while for the fourth type the predicted Mach number on the wall is greater than the physically expected value. To summarize, it can be said that the
second type of boundary conditions with the least deviation gives the best trend of Mach number through the wall.

In Figure 12, the comparison between Mach number distributions on the upper and lower walls for three types of boundary conditions is presented and compared with the numerical results of Ni (1982). As seen, the fourth type gives a non-smooth distribution especially near the outlet.

We concluded that for the supersonic flow over the bump, the second type of wall boundary conditions has the best stability and accuracy.

### 6.6. Flow through a NACA65-410 compressor cascade

To assess the performance of the wall boundary conditions for an applicable test case, inviscid flow through a NACA65-410 compressor cascade, is studied here. In Figure 13, a sample of the results for a NACA65-410 cascade is shown by Mach number contours. As can be seen in the figure, the flow is subsonic and the Mach number decreases from inlet to outlet of the cascade as expected for a compressor airfoil. For the NACA65410 test case, the solution with the fourth type of boundary conditions does not converge. For previous test cases this method was sensible for the initial conditions.

In Figure 14, the velocity magnitude contours around the leading edge of the blade are shown. As shown, the third type ($\rho V_t = cte$) makes a smoother distribution of
the velocity magnitude. In Figure 15, the Mach number distributions on the blade surfaces for three types of wall boundary condition are presented and compared with the isentropic Mach number calculated from experimental results of Emery, Herrig, Erwin, and Felix (1957). As seen in the previous test cases (channel flow over the bump), the results for the Mach number of the first, second, and third types of boundary conditions are close together.

From the continuity of velocity magnitude contours, it is concluded that for subsonic flow through the NACA 65-410 cascade, the third type of velocity correction gives better accuracy.
7. Mixed boundary conditions

In previous sections, four types of wall boundary conditions are investigated and their stability and accuracy are compared. It has been concluded that for the subsonic test cases, the third type, which is the $\rho V_t = cte$ condition, has better accuracy and produces more compatible results with flow physics. On the other hand, for the supersonic test cases, the third method was unstable (diverged) and the second, $(V_t = cte)$, shows better accuracy and stability.

Since there are many problems where some part of the domain is subsonic and some other parts are supersonic, one needs a boundary condition which supports both subsonic and supersonic flows. Hence, a mixed wall boundary condition is introduced and implemented here. To do so, for a node on the wall, the Mach number for the adjacent node is considered and in regard to the flow regime, the appropriate type of boundary condition is imposed. If the Mach number is less than unity, the third type ($\rho V_t = cte$) is applied and if the Mach number is greater than unity the second type ($V_t = cte$) is imposed.

To evaluate the mixed boundary condition, two transonic test cases are studied. In these cases both subsonic and supersonic flow regimes are observed. Based on the Mach number, in parts of the walls where the flow is subsonic the third method is applied and in the supersonic parts the second method is used. The first test case is the transonic flow over a bump, this case was has been previously studied for all the methods. In Figure 16, Mach number distribution over the walls for the second, third, and mixed types of boundary conditions are shown. As seen, the results of the mixed boundary conditions show good agreement with the second and third method.

![Figure 16. Comparison between Mach number over the walls obtained using 2nd, 3rd and mixed types of boundary conditions for transonic bump problem.](image)

8. Conclusions

Four types of wall boundary conditions are introduced. To assess the performance of each type, several subsonic and supersonic test cases are selected, the boundary conditions are implemented for each test case, and the flow fields are investigated using a recently developed inviscid flow solver.

In Table 2, the studied test cases are summarized for stability and accuracy. For subsonic and transonic flow the first, second and third conditions are robust while the fourth type of wall boundary conditions is more sensitive to the initial conditions. In comparison with the others, the third type or ($\rho V_t = cte$) has better accuracy.

For supersonic flows, the third method is unstable and results in divergence of the solution; the fourth method has less stability and is more sensitive to initial conditions. The second method ($V_t = cte$) shows better accuracy for supersonic test cases.

To have a convenient boundary condition, a combination of the second and third type of flow regimes called the mixed boundary condition is introduced and tested in two
Table 2. Summary of test cases.

| Test Case       | Flow Regime | Not Corrected (NC) | \( V_1 = \text{cte} \) | \( \rho V_1 = \text{cte} \) | \( P_0 = \text{cte} \) | Best accuracy |
|-----------------|-------------|--------------------|--------------------------|--------------------------|--------------------------|--------------|
| 90° Elbows      | Subsonic    | Robust             | Robust                   | Robust                   | Sensible                 | 3\textsuperscript{rd} type |
| Nozzle          | Supersonic  | Robust             | Robust                   | Diverged                 | Sensible                 | 1\textsuperscript{st} and 2\textsuperscript{nd} types |
| Bump            | Subsonic    | Robust             | Robust                   | Robust                   | Robust                   | 3\textsuperscript{rd} type |
|                 | Transonic   | Robust             | Diverged                 | Sensible                 | 2\textsuperscript{nd} type |
| NACA 65-410     | Subsonic    | Robust             | Robust                   | Robust                   | Diverged                 | 3\textsuperscript{rd} type |

The results show that the mixed boundary condition is applicable for both subsonic and supersonic flow regimes.

**ORCID**

A. Madadi [http://orcid.org/0000-0002-6257-5454](http://orcid.org/0000-0002-6257-5454)

H. Khazaei [http://orcid.org/0000-0001-7094-1605](http://orcid.org/0000-0001-7094-1605)

**References**

Emery, J. C., Herrig, L. J., Erwin, J. R., & Felix, A. R. (1957). Systematic two dimensional cascade tests of NACA 65-series compressor blades at low speeds. NACA Reports 1368.

Hoffmann, K. A., & Chiang, S. T. (2000). Computational fluid dynamics (Vol. I, 4th ed.). Wichita, KS: Engineering Education System.

Idahosa, U., Golubev, V. V., & Balabanov, V. O. (2008). An automated optimization of fan blade using an integrated CFD/MDO computer environment. Journal of Engineering Applications of Computational Fluid Mechanics, 2(2), 141–154.

Kermani, M. J., & Plett, E. G. (2001a, January 8–11). Roe scheme in generalized coordinates: part I-formulations. 39th AIAA Science Meeting and Exhibition, Reno, NV, 2001–0086.

Kermani, M. J., & Plett, E. G. (2001b, January 8–11). Modified entropy correction formula for the Roe scheme. 39th AIAA Science Meeting and Exhibition, Reno, NV, 2001–0083.

Khazaei, H., Madadi, A., & Kermani, M. J. (2011, July 24–29). Assessment of various inviscid-wall boundary conditions: applications to NACA65 compressor blade. ASME-JSME-KSME Joint Fluids Engineering Conference, AJK2011-FED, Hamamatsu, Shizuoka, Japan.

Kim, M. S., Jeon, B. W., Kim, Y. N., Kwon, H. B., & Lee, D. H. (2001). An implementation of the robust inviscid wall boundary condition in high-speed flow calculations. KSME International Journal, 15(5), 671–680.

Kim, M. S., & Kim, H. K. (2000). An investigation of characteristics of Roe’s FDS by using the compressible Euler equation. Journal of the Korean Society for Aeronautical and Space Science, 28(7), 1–9.

Liou, M. S., & Steffen, C. (1993). A new flux splitting scheme. Journal of Computational Physics, 107(1), 23–39.

Liu, X., Dang, Q., & Xi, G. (2008). Performance improvement of centrifugal fan by using CFD. Journal of Engineering Applications of Computational Fluid Mechanics, 2(2), 130–140.

Marshall, D. D., & Ruffin, S. M. (2004, January 5–8). A new inviscid wall boundary condition treatment for embedded boundary Cartesian grid schemes. 42\textsuperscript{nd} AIAA Science Meeting and Exhibition, Reno, NV.

Ni, R. H. (1982). A multiple-grid scheme for solving the Euler equations. AIAA Journal, 20(11), 1565–1571.

Qamar, A., Hasan, N., & Sanghi, S. (2010). A new spatial discretization strategy of the convective flux term for the hyperbolic conservation laws. Journal of Engineering Applications of Computational Fluid Mechanics, 4(4), 593–611.

Roe, P. L. (1981). Approximate Riemann solvers, parameter vectors and difference schemes. Journal of Computational Physics, 43, 357–372.

Van Albada, G. D., van Leer, B., & Roberts, W. W. (1982). A comparative study of computational methods in cosmic gas dynamics. Astronomy and Astrophysics, 108(1), 76–74.

Wang, Z. J., & Yuzhi, S. (2002, January 14–17). A curvature based wall boundary condition for the Euler equations on unstructured grids. 40\textsuperscript{th} AIAA Science Meeting and Exhibition, Reno, NV.

Yu, Y., Shademian, M., Barron, R. M., & Balachandar, R. (2012). CFD study of geometry variations on flow in a nozzle. Journal of Engineering Applications of Computational Fluid Mechanics, 6(3), 412–425.