Characteristic-based formulation of boundary conditions for preconditioned or non-preconditioned flow equations

Characteristic-based formulation of boundary conditions for preconditioned or non-preconditioned flow equations

Cite as: AIP Conference Proceedings **2116**, 030030 (2019); https://doi.org/10.1063/1.5114014
Published Online: 24 July 2019

Jinguang Yang, Michele Ferlauto, and Jiri Blazek

Characteristic-Based Formulation of Boundary Conditions for Preconditioned or Non-Preconditioned Flow Equations

Jinguang Yang[1,a)], Michele Ferlauto[2] and Jiri Blazek[3]

[1]*School of Energy and Power, Dalian University of Technology, Dalian, China*
[2]*DIMEAS, Politecnico di Torino, Turin, Italy*
[3]*CFD Consulting & Analysis, Sankt Augustine, Germany*

[a)]Corresponding author: jinguang_yang@dlut.edu.cn

Abstract. A numerical implementation of characteristic boundary conditions for Euler and Navier-Stokes equations is presented. The method combines the specified boundary conditions and the outgoing characteristic variables according to the wave propagation directions. In the general case, the proposed boundary conditions update the primitive variables by solving a small system of linear equations (4×4 in 2D, 5×5 in 3D) at each boundary point/cell. The method can be used for both preconditioned and non-preconditioned equations. For a perfect gas without preconditioning, a closed analytical solution is provided. Two possible methods of extrapolating the outgoing characteristic variables are discussed. Finally, the numerical approach is validated for the 2D internal flow in a channel with a bump.

INTRODUCTION

The numerical enforcement of the Boundary Condition (BC) plays a very important role in CFD code development. In steady state calculations, an effective implementation of BCs can enhance the convergence of the flow solver and allows for smaller computational domains with CPU time savings. Stronger issues may arise in unsteady calculations, since physical acoustic waves or spurious perturbations may lead to large flow instabilities and to incorrect results [1]. Since the flow equation system (i.e. compressible Euler or Navier-Stokes equations) we consider is hyperbolic in nature, flow perturbations propagate as waves across the computational domain along characteristic directions. Well-posedness considerations suggest that boundary conditions should also reflect this aspect. The numerical BC enforced according to the wave propagation theory is known as a Characteristic Boundary Condition (CBC). These considerations still apply to low Mach number flowfields computed by means of compressible solvers via preconditioning. Within this method, a preconditioning matrix is used to scale the speed of sound when Mach number is low, so as to decrease the condition number (CN) of the problem [2–7] . Numerical CBCs have a low spurious reflectiveness [8] and are the milestone for the derivation of totally non-reflecting boundary conditions (NRBC) [9]. The NRBC has been proposed for many years. A simple implementation for perfect gas usually involves the so-called *Riemann invariant*. One example of such implementation is detailed in Ref. [10]. Giles and coworkers [9, 11] made a great contribution on the mathematical properties of the NRBCs for perfect gas. Anker et al. [2] had successfully extend Giles theory for preconditioned governing equations.

In present paper a method of imposing CBC in center schemes is proposed. The method can be applied to both preconditioned and non-preconditioned flow equations. For the non-preconditioned case, an analytical expression is obtained. Two different method of extrapolating the characteristic variables are evaluated and their performances are compared. The proposed method is applied to the solution of the internal flow in a channel over a bump.

International Conference of Numerical Analysis and Applied Mathematics (ICNAAM 2018)
AIP Conf. Proc. 2116, 030030-1–030030-4; https://doi.org/10.1063/1.5114014
Published by AIP Publishing. 978-0-7354-1854-7/$30.00

FLOW EQUATIONS PRECONDITIONING

The basic principle of the precondition method can be explained as follows for one-dimensional (1D) flow problem.

$$\frac{\partial \boldsymbol{W}}{\partial t} + \frac{\partial \boldsymbol{F}}{\partial x} = 0 \tag{1}$$

with

$$\boldsymbol{W} = \{\rho, \rho u, \rho E\}^T, \quad \boldsymbol{F} = \{\rho u, p + \rho u^2, \rho H u\}^T$$

As usual, ρ, p, T, u are the density, pressure, temperature and fluid velocity, respectively. The total energy is $E = c_v T + u^2/2$ and the enthalpy is $H = E + p/\rho$. The perfect gas law $p = \rho R_g T$ holds. By defining as primitive variables the vector $\boldsymbol{W}_p = \{p, u, T\}^T$ the preconditioned Euler equation are expressed as

$$\frac{\partial \boldsymbol{W}_p}{\partial t} + \Gamma^{-1} \boldsymbol{A}_p \frac{\partial \boldsymbol{W}_p}{\partial x} = 0 \tag{2}$$

where $\boldsymbol{W}_p$ is the primitive variable vector, Γ the preconditioning matrix, and $\boldsymbol{A}_p$ is the Jacobian matrix of the fluxes with respect to the primitive variables. The following equation holds

$$\Gamma^{-1} \boldsymbol{A}_p = \boldsymbol{T}_p \Lambda_p \boldsymbol{T}_p^{-1} \tag{3}$$

and finally, the governing equation can be transformed to a characteristic form as

$$\frac{\partial \boldsymbol{C}}{\partial t} + \Lambda_p \frac{\partial \boldsymbol{C}}{\partial x} = 0 \tag{4}$$

Let us note that the proposed one-dimensional approach 2D/3D finite volume methods still maintain a 1D splitting treatment in the computation of the fluxes at the cell interfaces and in the imposition of the BCs [1, 8].

BOUNDARY CONDITION ENFORCEMENT

The CBCs we propose follow the guidelines of Ref. [9]. However, simpler residual variables are chosen, which are suited for preconditioned flow equation system. The boundary condition can be transferred to a residual form, and connected with the extrapolated characteristic variables. A linear system is then constructed and solved. Take the subsonic inlet boundary condition as an example. As mentioned above, usually total pressure, total temperature and flow angle are specified at subsonic inlet. It is easy to transfer the specified total pressure and temperature to total enthalpy and entropy according to thermodynamic relations $h = h(p, T), s = s(p, T)$. For 2D case, there are 3 data specified at the inlet boundary condition, so a residual variable vector can be defined as

$$\boldsymbol{R} = \begin{bmatrix} R_1 \\ R_2 \\ R_3 \end{bmatrix} = \begin{bmatrix} H_t - H_{to} \\ s - s_o \\ v - u \tan\beta \end{bmatrix} \tag{5}$$

where H_{to} and s_o are the total enthalpy and entropy specified, and H_t and s are the values calculated respectively. Furthermore, β denotes the specified flow angle and u and v are the velocity components. vector $\boldsymbol{R}$ is dependent on the primitive variables at the boundary, and should vanish when convergence is achieved. By using the Newton's method, it results

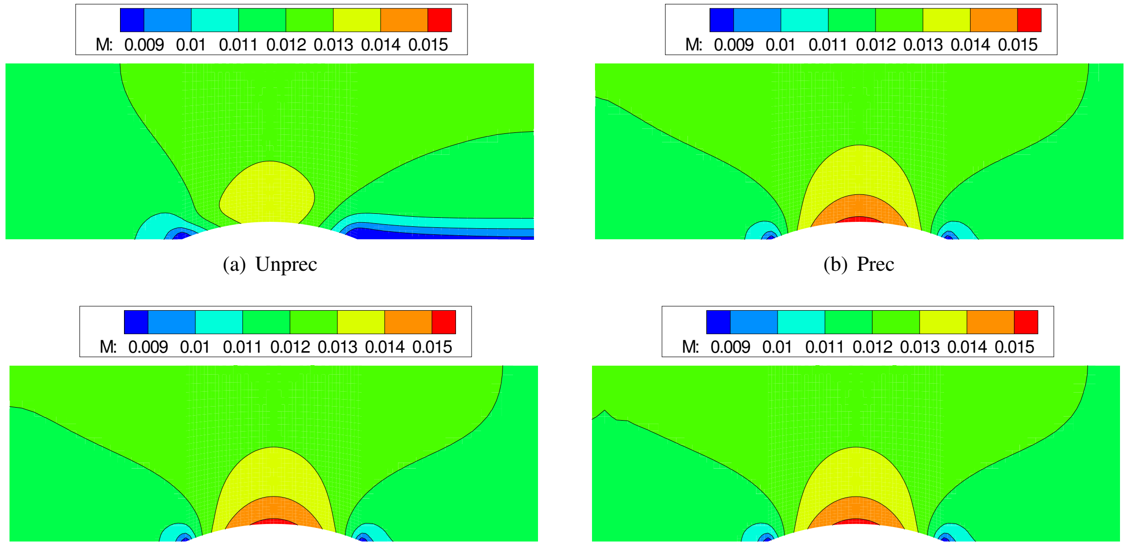
$$\frac{\partial \boldsymbol{R}}{\partial \boldsymbol{W}_p} d\,\boldsymbol{W}_p = -\boldsymbol{R} \tag{6}$$

Considering the out-going characteristic variable, a linear system can be constructed as

$$\boldsymbol{A} \cdot d\,\boldsymbol{W}_p = \boldsymbol{B}, \qquad \text{with } \boldsymbol{B} = \{-R_1, -R_2, -R_3, dC_4\}^T \tag{7}$$

and then $d\,\boldsymbol{W}_p$ can be solved. The system is solved for $d\,\boldsymbol{W}_p$, which is used to update the values in the ghost cells. For a perfect gas without preconditioning,an analytical solution can then be obtained as

$$dp = \frac{\rho dH - \frac{p}{R} ds - \frac{\rho V_t V_n}{\cos\theta} + \rho(u^2 + v^2)}{1 + \frac{V_t}{c\,\cos\theta}} \qquad dT = \frac{1}{c_p}\left(T ds + \frac{dp}{p}\right) \tag{8}$$



(a) Unprec (b) Prec

(c) Prec,CBC,Ext1 (d) Prec,CBC,Ext2

FIGURE 1. Bump test-case. Mach number isolines for different boundary conditions and extrapolation methods. Prec = Preconditioned, UnPrec = UnPreconditioned, Ext1 = extrapolation 1, Ext2 = extrapolation 2, RBC = Reflecting BC

$$dV_n = \frac{dp}{\rho c}, \qquad du = dV_n n_x, \qquad dv = dV_n n_y \tag{9}$$

where $V_t = \boldsymbol{V} \cdot \boldsymbol{s}$, $V_n = \boldsymbol{V} \cdot \boldsymbol{n}$, $\boldsymbol{s} = \cos\beta\, \boldsymbol{i} + \sin\beta\, \boldsymbol{j}$ and $\cos\theta = \boldsymbol{n} \cdot \boldsymbol{s}$. For subsonic outlet boundary, a similar process can be applied.

At subsonic outlet, usually static pressure is specified, so the residual vector is defined as

$$dR_4 = p - p_b \tag{10}$$

where p_b stands for the specified static pressure. And the matrix system is

$$\boldsymbol{B} = \{\delta C_1, \delta C_2, \delta C_3, -dR_4\} \tag{11}$$

for perfect gas A is reduced to an analytical solution is got as follows

$$dp = p_b - p\,, \qquad du = -\frac{dp}{\rho c} n_x\,, \qquad dv = -\frac{dp}{\rho c} n_y\,, \qquad dT = \frac{dp}{\rho c_p} \tag{12}$$

In the proposed CBC method, the out-going characteristic variable is extrapolated according to well posedness requirements of the hyperbolic system. This condition can be satisfied numerically with different interpolation methods as investigated in the section of numerical results.

NUMERICAL RESULTS

The flow in a 2D channel with a circular bump is proposed as test-case. The bump height is 10% of the channel width. The inviscid case has been preferred because the smearing effect of viscosity may hide possible flow perturbations. In the first test, the inlet Mach number is has been set to $M = 0.01$. Four simulations with different BCs were conducted on a computational grid of 97×33 nodes. The Mach number isolines of all cases are shown in Figure 1. The positive effect of preconditioning is evident. Concerning the extrapolation methods one can adopt the boundary value as a reference state (method 1) or extrapolate the conservative solution update (method 2). The CBC case with extrapolation method 2 shows some numerical oscillations at inlet. The convergence histories are compared in

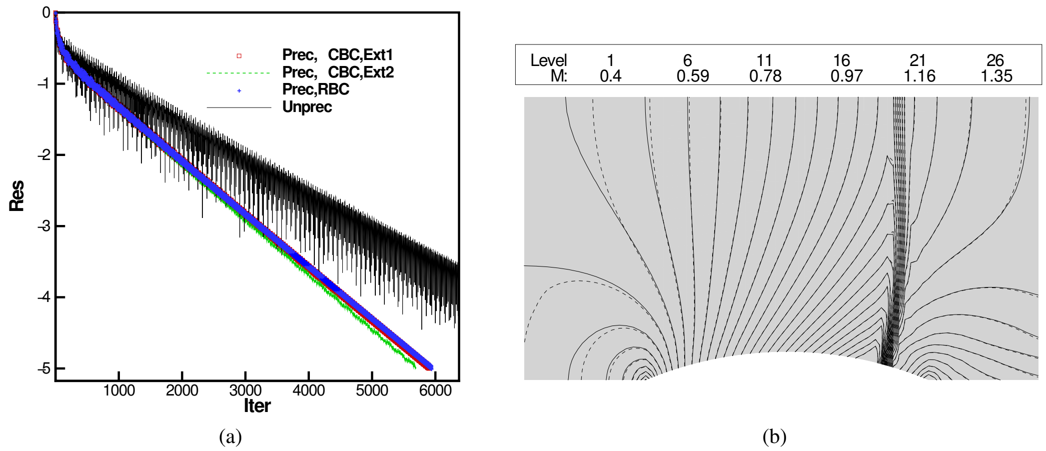


FIGURE 2. Bump testcase. (a) Convergence history of the low Mach number test and (b) Mach isolines of the transonic test.

Fig. 2(a) . The convergence history of the non-preconditioned calculation exhibits large wiggles, whereas the three preconditioned cases converge smoothly.

A transonic test with an embedded shock has been then carried out. The computational domain has been shortened in order to enhance the effect of boundaries on the solution. The Mach number contour lines are compared in Fig.2(b). The solid lines denote the simulation with purely extrapolated, reflective inlet and outlet boundary condition, whereas the dashed line refers the CBC case. The mismatch between the isolines, especially in the inlet part before the bump, is evident. By varying the boundary distance, it is observed that CBCs give a flowfield that is less influenced by the inlet position, a typical behavior of low or non-reflecting BCs.

CONCLUSIONS

A characteristic based approach to the boundary condition enforcement in centered difference schemes for the preconditioned and non-preconditioned Euler and Navier-Stokes equations has been proposed. After the validated against the channel flow with a bump, the following conclusions are drawn: (i) the developed method is robust and applicable to both preconditioned and non-preconditioned flow equations; (ii) for a perfect gas without preconditioning, an analytical solution exists for the developed method. (iii) Concerning the computation of the outgoing characteristic variables, method of utilizing the boundary value as a reference state (method 1) should be preferred.

ACKNOWLEDGMENTS

The financial support by the China Scholarship Council (CSC) and by the National Science Foundation of China (N^o 51606026) is gratefully acknowledged.

REFERENCES

[1] T. Poinsot and S. Lele, J. Comp. Phys. **101**, 104–129 (1992).
[2] J. Anker, J. Mayer, and H. Stetter, Task Quarterly **6**, 143–176 (2002).
[3] H. Kazem and R. Kamali-Moghadam, Journal of Computational Physics **231**, 4384–4402 (2012).
[4] J. Weiss and W. Smith, AIAA J **33**, 2050–7 (1995).
[5] Y. Choi and C. Merkle, J Comput Phys **105**, 207–23 (1993).
[6] M. Mulas, S. Chibbaro, G. Delussu, and M. Talice, Int J Numer Meth Heat Fluid Flow **12**, 637–57 (2002).
[7] E. Turkel, Journal of Computational Physics **72**, 277–298 (1980).
[8] M. Ferlauto, A. Iollo, and L. Zannetti, AIAA J **42**, 1582–1592 (2004).
[9] M. Giles, Gas Turbine Laboratory Report, No. 205, MIT. Cambridge, Massachusetts (1991).
[10] J. Blazek, *Computational Fluid Dynamics: Principles and Applications* (Elsevier, 2005).
[11] D. Darmofal, P. Moinier, and M. Giles, Journal of Computational Physics **160**, 369–384 (2000).