Available online at www.sciencedirect.com
ScienceDirect
Journal of Computational Physics 227 (2008) 7077–7082
JOURNAL OF COMPUTATIONAL PHYSICS
www.elsevier.com/locate/jcp

Short Note

A novel outflow boundary condition for incompressible laminar wall-bounded flows

G. Fournier *, F. Golanski, A. Pollard

Department of Mechanical and Materials Engineering, McLaughlin Hall, Queen's University, Kingston, Ontario, Canada K7K 3N6

Received 23 May 2007; received in revised form 19 February 2008; accepted 25 March 2008
Available online 11 April 2008

Keywords: Outflow conditions; Laminar boundary layer; Wall-bounded flows

1. Introduction

The aim of this note is to propose an alternative to the classical convective outlet boundary condition heavily used for wall-bounded laminar/turbulent flows.

Despite the huge improvements in computational power in the last few years, numerical simulations (DNS or LES) of boundary layers are still expensive and any simulation may take many weeks of processing time to achieve statistically stationary results. Hence, any methodology that reduces the computational time is welcome. Concerning the outflow boundary, many conditions can be applied, such as a streamwise periodicity [11] or a zero-gradient condition on the velocity [6]. However, in most of the studies that deal with numerical simulations of a turbulent flow, a convective condition is used as the exit boundary condition. This exit condition is a solution to the linearised convective equation (Eq. (1)), where the convection velocity U_c is chosen to be either the maximum streamwise velocity [8], the mean streamwise velocity at this plane [4,7] or the local velocity [2,5] and x is the streamwise direction:

$$\frac{\partial u_i}{\partial t} + U_c \frac{\partial u_i}{\partial x} = 0 \tag{1}$$

In the case of a boundary layer, however, the implementation of Eq. (1) can create a singularity in the shear stress distribution. The shear stress error in the context of a fractional step algorithm [3] will lead to an overcorrection of the velocity field by the pressure, which will contaminate a large part of the computational domain because of the elliptic nature of the Poisson equation. Moreover, the effect of this singularity contaminates an increased portion of the computation domain with decreasing Reynolds number. Fig. 1 represents the pressure field obtained for a laminar boundary layer developing over a flat plate obtained using the outflow boundary condition (Eq. (1)), U_c being the local velocity. The contamination, as made evident in the non-uniform pressure contours in the figure, extends approximately one fifth of the way into the simulation domain. The "polluted" pressure field clearly is wrong. The main consequence of having this kind of polluted

* Corresponding author.
E-mail address: fournier@me.queensu.ca (G. Fournier).

0021-9991/$ - see front matter © 2008 Elsevier Inc. All rights reserved.
doi:10.1016/j.jcp.2008.03.038

Fig. 1. Non-dimensional pressure field for a laminar boundary layer developing over a flat plate obtained with the classical outlet condition.

zone is that the domain size is larger than it should be (with the concomitant increase in computational cost) assuming that there is no impact farther upstream and that it is acceptable to ignore this region.

The linearised form of the Navier–Stokes equations (i.e. Eq. (1)) at the outlet implies that the first- and second-order wall-normal derivatives are negligible. This may be true for some flows; however, it is demonstrated below that in the case of laminar boundary layers this is not true. The velocities are small close to the wall; however, they increase rapidly away from the wall. The magnitude of the wall-normal derivatives are, therefore, large, as can be observed in Fig. 2.

Fig. 2 demonstrates that the classical term $u_x \frac{\partial u_x}{\partial x}$ is much larger in magnitude than other terms. Moreover, for higher Reynolds numbers, the magnitude of these terms remain non-negligible (see Fig. 3). These results clearly are in accordance with the boundary layer theory since the boundary layer equations directly imply that all these terms are of the same order of magnitude [10].

Consider the following equations as boundary conditions:

$$\frac{\partial u_i}{\partial t} + u_x \frac{\partial u_i}{\partial x} + u_y \frac{\partial u_i}{\partial y} - \nu \frac{\partial^2 u_i}{\partial y^2} = 0 \qquad (2)$$

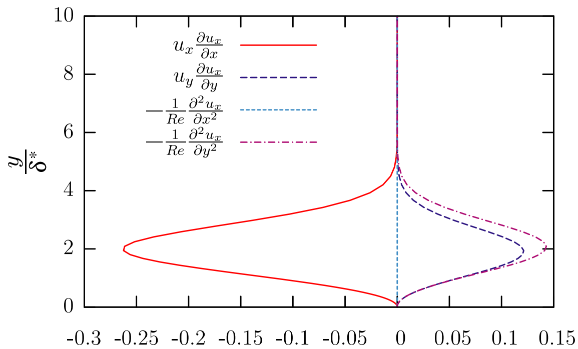

Fig. 2. Order of magnitude of the different terms in the streamwise Navier–Stokes equation for a laminar boundary layer developing over a flat plate at $Re_\theta = 320$.

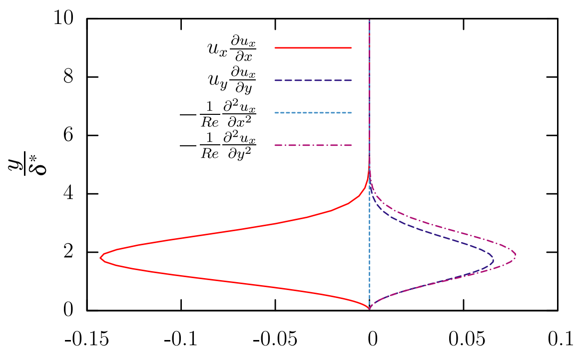

Fig. 3. Order of magnitude of different terms in the streamwise Navier–Stokes equation for a laminar boundary layer developing over a flat plate at $Re_\theta = 700$.

This boundary condition takes into account the terms $u_y \frac{\partial u_i}{\partial y}$ and $\nu \frac{\partial^2 u_i}{\partial y^2}$, the importance of which has been underlined in Figs. 2 and 3.

The literature on exit boundary conditions used in CFD is large and issues and solutions are quite different when dealing with compressible or incompressible flows; however, there have been attempts to develop boundary conditions for compressible flows in the limit as the Mach number approaches zero. This is the case in [1] where the authors performed a long wave asymptotic expansion of the Navier–Stokes equations. The resulting eigensystems provide the decay rate and group velocity of the long wave disturbances in the far-field. An evolution equation for the pressure in the far-field that can be used as a boundary condition could then be defined. In the case of incompressible flow, the eigenvalues of the zeroth and first order eigensystems (providing the decay rate and group velocity) of the incompressible viscous boundary layer equations were found to be the same as the one obtained by the compressible Navier–Stokes equation at a vanishing Mach number, proving that the use of Eq. (2) is well-founded from a mathematical perspective.

Fig. 4. Non-dimensional pressure field for a laminar boundary layer developing over a flat plate obtained with the modified outlet boundary condition, Eq. (2).

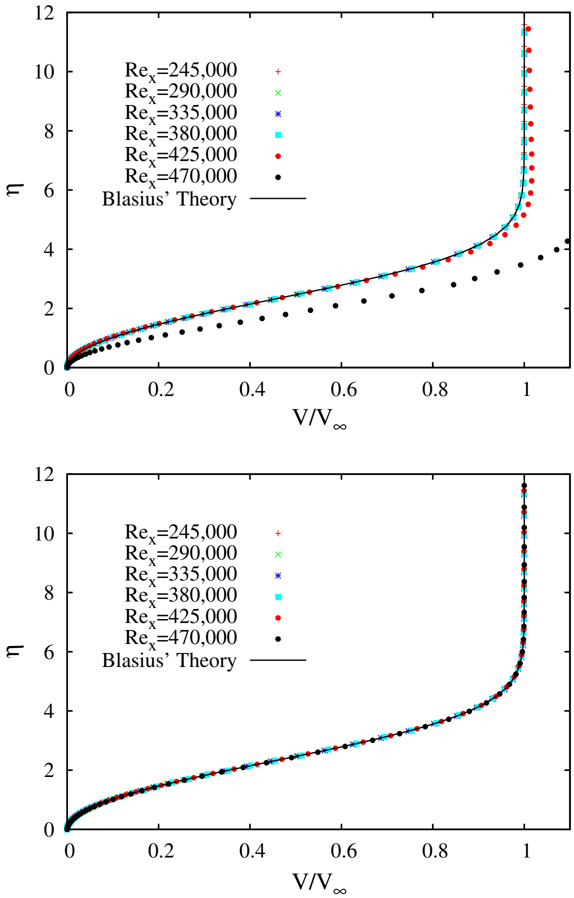

Fig. 5. Similarity profiles of the wall-normal component of the velocity for a laminar boundary layer obtained with the classical (top) and modified (bottom) outflow condition.

Thus, the incompressible boundary layer equations themselves accommodate the perturbations at the outflow boundary. This is particularly interesting in the case of a Navier–Stokes solver based on the fractional step method for which a boundary condition obtained in terms of an equation for the evolution of the pressure is not well suited. On the other hand, the boundary layer equations can be easily implemented as boundary conditions as they constitute evolution equations for the whole velocity vector.

2. Steady flow boundary layer

This new boundary condition was tested and validated for the flow developing along a flat plate at $Re_\theta = 320$, based on the free-stream velocity and on the momentum thickness at the inlet plane. DNS in two dimensions was performed on a finite-volume, staggered grid solver developed at the Center for Turbulence Research [9]. Around 50,000 control volumes were used with stretching in the wall-normal direction such that the mesh size in that direction varies from $\Delta y^+ = 0.45$ close to the wall to $\Delta y^+ = 25$ in the free flow region. At the inlet, Blasius' profiles were used in both the streamwise and wall-normal directions, a classical convective condition was used at the upper boundary and a no-slip condition was applied at the lower boundary to simulate the flat plate. This case was considered because it is known to be extremely sensitive to the choice of the outlet boundary condition. Finally, a reference case with exactly the same parameters except for the outlet condition was also run. For this reference simulation, the convective velocity was chosen to be the local velocity at the exit plane, so that the two terms of this boundary condition, derived from (1), are exactly included in the new condition (2). Note that several convective velocities were applied but as they were not as good as the local velocity, the results obtained are not presented here.

The pressure field computed with the modified outflow condition is given in Fig. 4 and should be compared to Fig. 1, obtained with the classical condition and plotted with the same intensity levels.

As expected, the singularity at the outlet plane vanishes and there is no polluted zone. Hence, use of this modified outflow condition enables the whole domain and thus each grid point to be useful for physical flow analysis. The superiority of this outflow condition over the more common convective condition is made evident by comparing the similarity profiles for the wall-normal velocity (Fig. 5). The profiles obtained from the inlet ($Re_x = 245{,}000$) to the outlet ($Re_x = 470{,}000$) boundaries are all in accordance with Blasius' theory throughout the whole computational domain.

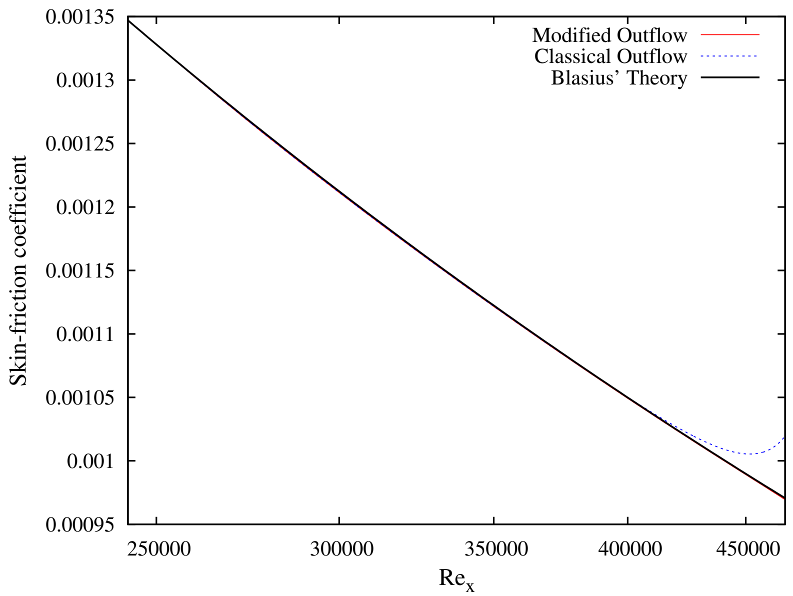

Fig. 6. Streamwise evolution of the skin-friction coefficient as a function of Re_x.

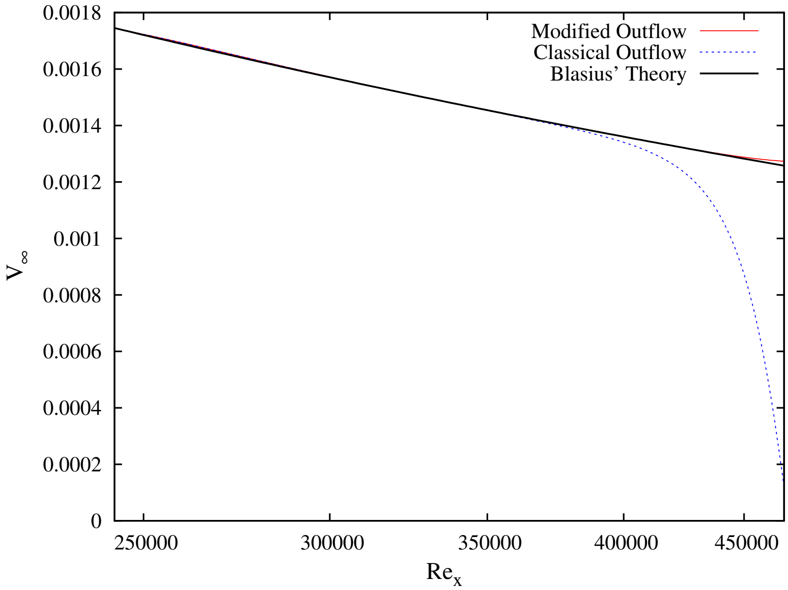

Fig. 7. Streamwise evolution of the free-stream wall-normal velocity (bottom) as a function of Re_x.

The skin-friction coefficient as well as the free-stream wall-normal velocity are plotted as functions of Re_x and compared to the theoretical evolution predicted by Blasius' theory (Figs. 6 and 7).

It is readily apparent from these figures that the modified outlet condition respects the Blasius solution, while the skin-friction and the free-stream velocity obtained using the traditional convective outlet boundary condition agree with the Blasius solution only up to $Re_x \simeq 400,000$ and $Re_x \simeq 360,000$, respectively.

3. Conclusion

A new outflow boundary condition for wall-bounded laminar flows was demonstrated to reproduce exactly the Blasius solution over the full computational domain, contrary to those results obtained using the traditional convective outlet boundary condition. Given that the boundary layer equations represent a good approximation to a large number of wall-bounded and free-shear flows, and that the eigensystem analysis of [1] showed that they lead to the same estimation as the zero Mach number limit of the compressible equations, this condition should also be useful for other types of turbulent flows.

Acknowledgements

The authors would like to thank the Natural Science and Engineering Research Council of Canada, Bombardier Aerospace and Cray Canada Corp. for their financial support.

References

[1] J.S. Danowitz, S.A. Abarbanel, E. Turkel. A far-field non-reflecting boundary condition for two-dimensional wake flows. Technical Report ICASE 95-63, 1995.

[2] T.Y. Han, J.C.S. Meng, G.E. Innis, An open boundary condition for incompressible stratified flows, J. Comput. Phys. 49 (1983) 276.

[3] H. Le, P. Moin, An improvement of fractional step methods for the incompressible Navier–Stokes equations, J. Comput. Phys. (1991) 369–379.

[4] H. Le, P. Moin, J. Kim, Direct numerical simulation of turbulent flow over a backward facing step, J. Fluid Mech. 330 (1997) 349–374.

[5] T.S. Lund, X. Wu, K.D. Squires, Generation of turbulent inflow data for spatially-developing boundary layer simulations, J. Comput. Phys. 140 (1998) 233–258.

[6] M. Manhart, R. Friedrich, DNS of a turbulent boundary layer with separation, Int. J. Heat Fluid Flow 23 (2002) 572–581.
[7] Y. Na, P. Moin, Direct numerical simulation of a separated turbulent boundary layer, J. Fluid Mech. 374 (1998) 379–405.
[8] J.S. Paschkewitz, C.D. Dimitropoulos, Y.X. Hou, V.S.R. Somandepalli, M.G. Mungal, E.S.G. Shaqfeh, P. Moin, An experimental and numerical investigation of drag reduction in a turbulent boundary layer using a rigid rodlike polymer, Phys. Fluid 17 (2005).
[9] C. Pierce, Progress-variable approach for large-eddy simulation of turbulent combustion, PhD thesis, Stanford University, June 2001.
[10] H. Schlichting, Boundary-Layer Theory, McGraw-Hill, 1979.
[11] P.R. Spalart, Direct simulation of a turbulent boundary layer up to $R_\theta = 1410$, J. Fluid Mech. 187 (1988) 61–98.