AIAA Journal
Vol. 30, No. 8, August 1992

Flow Separation in the Vicinity of a Moving Boundary

Bashar S. AbdulNour*
University of Wyoming, Laramie, Wyoming 82071

Abstract

THE effect of boundary motion on the laminar, incompressible flow in the entry region of a two-dimensional channel is considered in this study. The formation of a separated region during the upstream motion of a section of the lower boundary is of particular interest. The size of the separated region depends on the Reynolds number and the velocity of the moving boundary. The solution is obtained numerically by considering the continuity and the Navier-Stokes equations using the vorticity-stream function formulation. A computational scheme, which consists of the alternating-direction implicit method for the vorticity equation and the successive over-relaxation method for the stream function equation, is used to solve the finite difference equations.

Contents

Specifying solutions for flow problems comprising moving boundaries has potential importance in aerospace applications such as oscillating airfoils, impellers, helicopter rotors, and compressor blades. The primary objective of this work is to numerically simulate the steady, laminar, incompressible, confined flow in the presence of boundary motion.[1] As a preliminary configuration, the problem is modeled as viscous flow past a moving boundary section placed within the entry region of a two-dimensional channel, with uniform velocity at the inlet and Poiseuille flow at the downstream end. Interest is focused on the development of a separated region in the vicinity of a lower boundary section permitted to move with a constant velocity in its own plane in the direction opposite to the inlet velocity. Laminar separation and reattachement occur in regimes of low Reynolds number flow around airfoils. Also, separated regions can be generated by the flow near the sharp leading edge of a thin airfoil. Unpredictable rapid growth of such separated regions can cause stall.

Viscous, incompressible flows are governed by the Navier-Stokes equations. For two-dimensional flows, it is often desirable to introduce the vorticity and the stream function as dependent variables. The x- and y-component Navier-Stokes equations yield the z-component vorticity equation:

$$\frac{\partial \zeta}{\partial t} + \frac{\partial \Psi}{\partial y}\frac{\partial \zeta}{\partial x} - \frac{\partial \Psi}{\partial x}\frac{\partial \zeta}{\partial y} = \frac{1}{Re}\left(\frac{\partial^2 \zeta}{\partial x^2} + \frac{\partial^2 \zeta}{\partial y^2}\right) \tag{1}$$

The Reynolds number is $Re = HU/\nu$, where H is the channel height, U is the uniform inlet velocity, and ν is the kinematic viscosity. The z-component vorticity is defined as $\zeta = \partial v/\partial x - \partial u/\partial y$, and the velocity components u and v in the x and y directions, respectively, are expressed in terms of the stream function Ψ through the following expressions: $u = \partial\Psi/\partial y$ and $v = -\partial\Psi/\partial x$. This definition automatically satisfies the continuity equation. The stream function is then related to the vorticity through the Poisson-type stream function equation:

$$\frac{\partial^2 \Psi}{\partial x^2} + \frac{\partial^2 \Psi}{\partial y^2} = -\zeta \tag{2}$$

Equations (1) and (2) are the resulting governing equations and are subject to the following boundary conditions:

$$\Psi(0,y) = y, \qquad \Psi(x \geq L_e, y) = 3y^2 - 2y^3$$
$$\Psi(x,0) = 0, \qquad \Psi(x,1) = 1 \tag{3}$$

$$\zeta(0,y) = 0, \qquad \zeta(x \geq L_e, y) = 12y - 6$$
$$\zeta(x,0) = -\frac{\partial^2 \Psi}{\partial y^2}, \qquad \zeta(x,1) = -\frac{\partial^2 \Psi}{\partial y^2} \tag{4}$$

where L_e is the entrance length defined as the distance from the inlet to the point where the centerline velocity reaches 99% of the fully developed value.

The alternating-direction implicit (ADI) method is used to solve the vorticity equation whereas the successive over-relaxation (SOR) method is used to solve the stream function equation. The ADI method provides a reduction in computing time since it allows larger time steps. In Eq. (1), the time t is fictitious and the term $\partial\zeta/\partial t$ is retained because marching methods require the presence of a time derivative. Thus, the solution for this steady flow is obtained as the asymptotic solution for large time of the unsteady equations. Each time step, considered equivalent to an iteration, is split into two halves, resulting in two finite difference equations. The first equation is implicit in the x direction whereas the second is implicit in the y direction. Boundary conditions can destroy the unconditional stability of the ADI method. In this case, a limitation on the time step is necessary to insure the condition of diagonal dominance of the coefficient matrix. Conversely, to avoid a limitation on the time step, the first-order derivatives of ζ in the nonlinear convective terms of Eq. (1) are represented with first-order forward/backward differences instead of the second-order central differences. This technique helps avoid the numerical instability of the solution at high Reynolds numbers. However, the second-order accuracy is preserved at steady state[2] by alternating the direction of the differencing at each time step. This is achieved by considering upwind differencing for the implicit part and downwind differencing for the explicit part.

The iterative SOR by blocks method is used to implicitly solve Eq. (2) for the stream function. The speed of convergence of the method is improved by using a relaxation formula in which the values of the stream function from two successive iterations are combined using an over-relaxation factor, with value ranging between 1.0 and 2.0. For the present Dirichlet problem in a rectangular domain with uniform grid, the optimal value of the over-relaxation factor is calculated from an

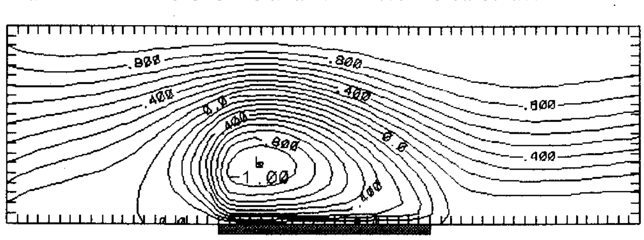
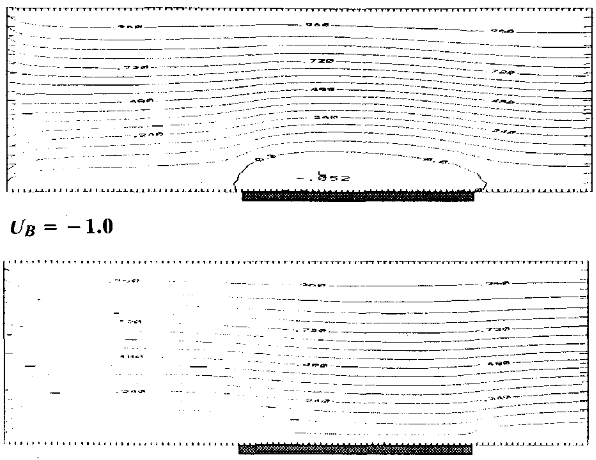


Fig. 1 Flow streamlines with the lower boundary section moving opposite to the inlet velocity at $U_B = -10.0$; $X_1 = 1.0$, $X_2 = 2.0$, and $Re = 20$.

Presented as Paper 90-3047 at the AIAA 8th Applied Aerodynamics Conference, Portland, OR, Aug. 20–22, 1990; received June 24, 1991; synoptic received Aug. 12, 1991; accepted for publication Aug. 19, 1991. Full paper available from AIAA Library, 555 W. 57th St.

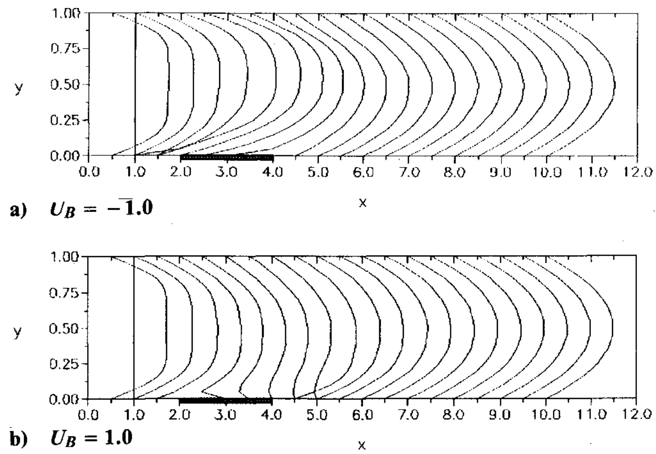
*Assistant Professor, Department of Mechanical Engineering. Member AIAA.



a) $U_B = -1.0$

b) $U_B = 1.0$

Fig. 2 Streamlines of the flow with the lower boundary section moving at U_B; $X_1 = 2.0$, $X_2 = 4.0$, and $Re = 200$.



a) $U_B = -1.0$

b) $U_B = 1.0$

Fig. 3 Development of streamwise velocity profiles along the channel with the lower boundary section moving at U_B; $X_1 = 2.0$, $X_2 = 4.0$, and $Re = 200$.

available analytical expression. The solution is improved with successive iterations until it converges to within specified accuracy after a finite number of inner iterations. All difference equations yield a tridiagonal system of algebraic equations and are solved by Gaussian elimination.

An explicit vorticity boundary condition is necessary to replace the second-order derivatives in Eq. (4). The second-order Woods' condition is used, i.e.,

$$\zeta_B^{(n+1)} = \frac{3[\Psi_B^{(n+1)} - \Psi_{B+1}^{(n)}]}{h^2} - \frac{\zeta_{B+1}^{(n)}}{2} + \frac{3U_B^{(n+1)}}{h} \tag{5}$$

where n is the index for the time step, h is the mesh size equal in the x and y directions, and U_B is the boundary velocity; the subscripts B and $B+1$ represent, respectively, a boundary node and a neighboring internal node in a normal direction. The preceding relationship gives the vorticity on the upper and lower boundaries at each time step in terms of the stream function and vorticity obtained from the previous time step.

To modify the values of the vorticity and the stream function in the interior region, weighted averages at the end of each time step are taken as linear combinations of the new values and the weighted values of the previous time step. Separate weighting factors for the vorticity and the stream function are employed; their values depend on the mesh size and the Reynolds number and are in the range 0.0–1.0. The function of the weighting factors is to slow down the solution in the interior region to reduce the lag in the value of boundary vorticity. Optimum values of the accelerating parameters, which include the time step for the vorticity equation, the over-relaxation factor for the stream function equation, and weighting factors for both the vorticity and stream function, are used to further minimize the time of computation and provide numerical stability.

The vorticity ζ, stream function Ψ, and streamwise velocity u are calculated at each node for a flow with boundary velocity U_B ranging from -10.0 to 5.0. Reynolds numbers are between 20 and 1000. The moving boundary is defined to exist between $x = X_1$ and $x = X_2$. A uniform grid with spacing size of 0.05×0.05 is used. The effect of grid size on the accuracy of the solution is found to be minimal. Convergence is achieved by advancing the solution in time until it no longer varies significantly. An error criterion, with relative tolerance error $\epsilon = 10^{-6}$, is selected to decide whether additional time steps are necessary. The present numerical solution is second-order accurate since both the numerical scheme and boundary conditions have second-order accuracy. Results demonstrated by plots of streamlines, velocity profiles, and vorticity contours show the formation of a separated region when the boundary section is moving opposite to the flow. The size of the separated region is a function of the Reynolds number and boundary velocity. Also, the number of time steps needed to attain convergence to a steady-state solution increases with the increase in Re and the magnitude of U_B.

Streamlines of the flow for $Re = 20$ and $U_B = -10.0$ are shown in Fig. 1, indicating the recirculating pattern of the flow inside the large separated region. For $Re = 200$, streamlines of the flow with boundary velocity $U_B = -1.0$ and 1.0 are displayed in Fig. 2. The separated region becomes larger with an increase in the magnitude of boundary velocity. Also, no separation occurs when the boundary section is moving in the direction of the flow because the fluid in the vicinity of the moving boundary section is accelerated due to the motion of the boundary. At high Reynolds number the separated region is reduced in size since it is apparently suppressed by the relatively high velocity incoming flow. Therefore, the size of the separated region is inversely proportional to Re. In addition, the size of the separated region is related to the length and location of the moving boundary section. The separated region extends along the length of the moving boundary with no apparent increase in its height due to the increase in the length of the moving section of the boundary. Placing the moving boundary section further downstream, including the fully developed region of the flow, produces no noticeable difference in the shape of the streamlines or the size of the separated region. Development of streamwise velocity profiles along the channel are illustrated in Fig. 3 for $U_B = -1.0$ and 1.0 and $Re = 200$. The velocity of the flow at the boundary between X_1 and X_2 is equal to the boundary velocity as required by the no-slip boundary condition. Figure 3 also reveals that the downstream parabolic velocity profile is attained at some distance, equivalent to the entrance length L_e, which is beyond the influence of the moving boundary; this result validates the choice of the downstream boundary location. Furthermore, from vorticity contours with and without the presence of a separated region, it is evident that changes in the vorticity are concentrated in the immediate vicinity of the moving boundary section as well as near the inlet section.

Finally, the aforementioned results are useful in investigating the effect of surface motion and fluid viscosity on the velocity field and the drag. They can be used to calculate the drag in a laminar flowfield due to surfaces undergoing large deformation with respect to the mesh.

References

[1]AbdulNour, B. S., "Laminar Separation in a Two-Dimensional Flow," *Proceedings of the AIAA 8th Applied Aerodynamics Conference*, Pt. 1, AIAA, Washington, DC, 1990, pp. 411–420; also AIAA Paper 90-3047, 1990.

[2]Peyret, R., and Taylor, T. D., *Computational Methods for Fluid Flow*, Springer-Verlag, New York, 1983.