Numerical Outflow Boundary Condition for Navier-Stokes Flow Calculations by a Line Iterative Method

Wei Shyy*
General Electric Research and Development Center, Schenectady, New York

Abstract

THIS study investigates the use of explicit/implicit, step/linear space extrapolation as numerical outflow boundary conditions for turbulent flow calculations. The emphasis is on the interplay between the use of curvilinear coordinate systems for complex flow calculation and the semi-implicit line iterative procedure. Both a planar single-channeled and an annular multichanneled flow configuration were used as test models. It was found that implicit linear space extrapolation needs more computing time and may not yield a unique solution. On the other hand, explicit zero-order space extrapolation appears to be a robust outflow boundary condition since it has the largest stability limit with respect to underrelaxation factors and initial conditions.

Contents

The present study investigates the use of outflow boundary condition for subsonic viscous flow calculations. The emphasis is on the complex geometry configuration arising from, e.g., modeling gas turbine combustor flows. Modeling these flows requires the use of the curvilinear body-fitted coordinate system to allow proper treatment of the near-wall region as well as to reduce the skewness between the streamlines and the coordinate lines, as discussed in Refs. 1 and 2. This study concentrates on one category of outflow boundary condition, extrapolation in space, since it is easier to implement into a computer code. The merits between the explicit and implicit implementation with respect to the iteration level are investigated. Both single- and multichanneled geometry configurations are used for the numerical experiments. Four outflow boundary conditions have been implemented and tested:

1) Explicit step space extrapolation—using an explicit one-sided difference to approximate a zero first derivative along the main flow direction, i.e.,

$$\phi^{n+1}_{NI,j} = \phi^{n}_{NI-1,j} \tag{1}$$

where NI denotes the ouflow boundary points on the ξ coordinate (i.e., streamwise direction) and j and n the position on the η coordinate and the iterative level, respectively.

2) Implicit step space extrapolation—using an implicit one-sided difference to approximate the zero first derivative along the main flow direction, i.e.,

$$\phi^{n+1}_{NI,j} = \phi^{n+1}_{NI-1,j} \tag{2}$$

3) Explicit linear space extrapolation—using an explicit one-sided difference to approximate the zero second derivative along the main flow direction, i.e.,

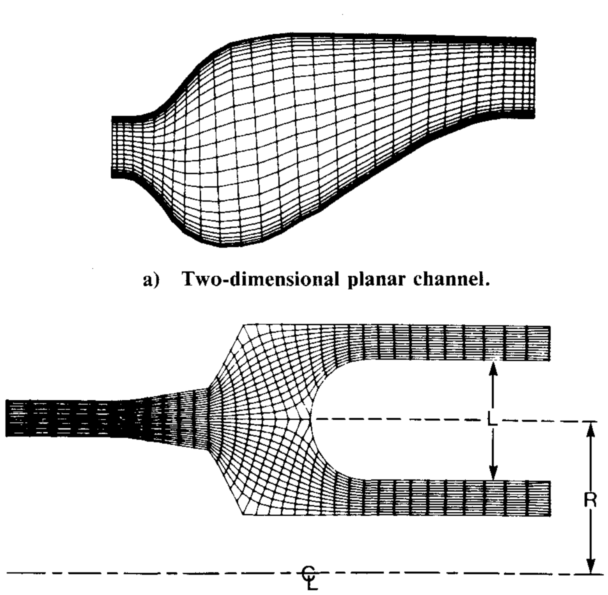
$$\phi^{n+1}_{NI,j} = 2\phi^{n}_{NI-1,j} - \phi^{n}_{NI-2,j} \tag{3}$$

4) Implicit linear space extrapolation—using an implicit one-sided difference to approximate zero second derivative along the main flow direction, i.e.,

$$\phi^{n+1}_{NI,j} = 2\phi^{n+1}_{NI-1,j} - \phi^{n+1}_{NI-2,j} \tag{4}$$

Two different flow configurations have been used to conduct the numerical experiments, as shown in Fig. 1. The Reynolds number based on inlet height is 10^5. The flow is steady-state, incompressible, isothermal and the standard k-ϵ model[3] is adopted to close the Reynolds-averaged equations. The mean inlet velocity is taken as uniform. Since there are no measured data available and since the main purpose of the calculation is to compare the relative performances among the four different outflow boundary conditions, the inlet turbulence kinetic energy (k) and the dissipation rate (ϵ) are uniformly taken as $0.003\bar{u}^2$ and $k^{1.5}/(0.03H)$, respectively, where H is the height of the channel inlet. The method of the wall function[3] for the velocity vector is applied to the nodal points next to the solid boundary. The results were obtained by using the second-order upwind scheme[4,5] to approximate the convection terms. The details of the numerical algorithm employed to solve the Navier-Stokes equations have been given in Ref. 1.

Numerical experiments indicate that while outflow boundary condition 3 is generally the slower one to yield the convergent solution, conditions 1, 2, and 4 closely follow one



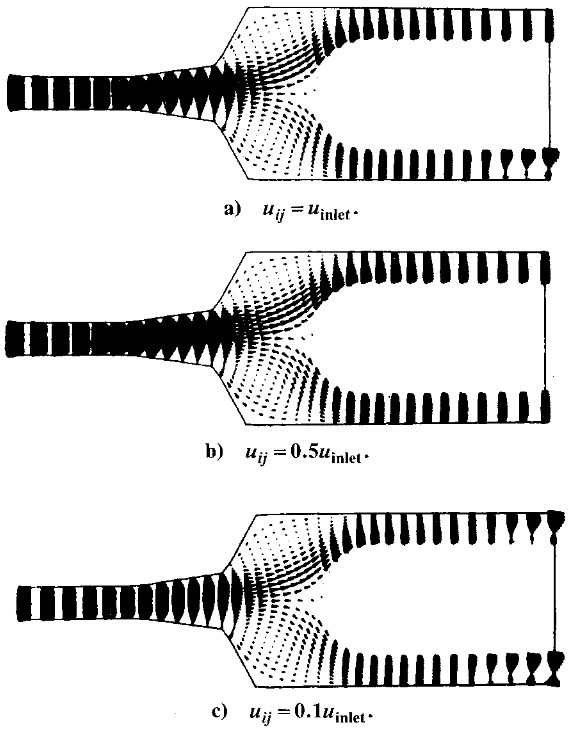
a) Two-dimensional planar channel.

b) Two-dimensional annular dump diffuser ($L/R = 0.57$).

Fig. 1 Configurations used to study numerical outflow boundary conditions.

Received Oct. 30, 1984; synoptic received March 1, 1985. Full paper available from National Technical Information Service, Springfield, VA 22151, at the standard price (available upon request).

*Combustion Modeling Scientist, Fluid Mechanics Branch. Member AIAA.



a) $u_{ij} = u_{\text{inlet}}$.

b) $u_{ij} = 0.5u_{\text{inlet}}$.

c) $u_{ij} = 0.1u_{\text{inlet}}$.

Fig. 2 Calculated velocity fields of using outflow boundary condition 3 with respect to different initial conditions.

another along the convergence paths. The differences between the explicit and the implicit treatments and between the step and linear space extrapolations do not exhibit a systematic trend. Yee et al.[6] have conducted numerical experiments for the one-dimensional gasdynamic equations applied to nozzle flows using the block implicit time marching method. In their study, it was found that the explicit and implicit step space boundary extrapolations may lead to global accuracy problems. They found that the solution profiles obtained by step space extrapolation tend to be more oscillatory, and in one case they lead to an incorrect steady-state solution. This phenomenon was not observed in the present flow calculations. For the two-dimensional planar channel shown in Fig. 1a, the numerical solutions obtained by using the four different outflow boundary conditions agree with one another to the fourth digit.

For the two-dimensional annular dump diffuser shown in Fig. 1b, the presence of the multichanneled flow configuration requires careful numerical treatment. For the outflow boundary conditions, besides the extrapolations mentioned above, the mass-split ratio, i.e., the ratios of total mass fluxes through the outer and inner annuli, must also be specified; otherwise no unique convergent solution can be obtained.[7] In the present study, the total mass fluxes out of the outer and inner settling annuli were monitored at every iteration step and then uniformly modified, through the contravariant velocity components, to satisfy the desired mass-split ratio in the two annuli. The results presented here are for the case of $D/H = 1.2$, where D and H denote the distance between the prediffuser outlet and the head of the centerbody and the height of the prediffuser outlet, respectively. The mass-split ratio is set to 2.3. It was found that the flowfield calculated by using condition 3 is not unique. Three caculated results based on three different initial assumptions for using condition 3 as the outflow boundary condition are shown in Fig. 2. The flow characteristics of these three calculations are all different. This phenomenon was not observed for the other three outflow boundary conditions. The solutions obtained by using outflow boundary conditions 1, 2, and 4 are indistinguishable; they are also unique with respect to the initial flowfield.

With regard to the effects of underrelaxation factors on the computing time needed to yield the convergent solutions, two observations can be made. First, outflow boundary conditions 1 and 3 were found to have the largest and smallest stability bound, respectively, in terms of the relaxation factors being employed. Second, the convergence rate for all cases appears largely insensitive to the relaxation factors. This suggests that the choice of the suitable relaxation factors is not a very critical task in the present algorithm. This finding is important since for the coupled equations, such as those governing the Navier-Stokes flows, optimum relaxation factors are not known a priori.

In conclusion, in the present numerical framework, the explicit step space extrapolation appears to be the most robust outflow boundary condition. This is different from the findings of Yee et al.[6] It appears that the optimum choice of the outflow boundary condition should be judged in the context of the numerical algorithm, as well as flow regime.

References

[1]Shyy, W., Tong, S. S., and Correa, S. M., "Numerical Recirculating Flow Calculation Using a Body-Fitted Coordinate System," *Numerical Heat Transfer*, Vol. 8, 1985, pp. 99-113.

[2]Shyy, W., Correa, S. M., and Tong, S. S., "Demonstration of a New Body-fitted Coordinate Code for Modeling Gas-Turbine Combustor Flows," AIAA Paper 84-1381, 1984.

[3]Launder, B. E. and Spalding, D. B., "The Numerical Calculation of Turbulent Flows," *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, 1974, pp. 269-289.

[4]Shyy, W., "A Study of Finite Difference Approximations to Steady-State, Convection-Dominated Flow Problems," *Journal of Computational Physics*, Vol. 57, 1985, pp. 415-438.

[5]Shyy, W. and Correa, S. M., "A Systematic Comparison of Several Numerical Schemes for Complex Flow Calculations," AIAA Paper 85-0440, Jan. 1985.

[6]Yee, H. C., Beam, R. M., and Warming, R. F., "Stable Boundary Approximations for a Class of Implicit Schemes for the One-Dimensional Inviscid Equations of Gas Dynamics," *Proceedings of AIAA 5th Computational Fluid Dynamics Conference*, 1981, pp. 125-135.

[7]Shyy, W., "A Numerical Study of Annular Dump Diffuser Flows," *Computer Methods in Applied Mechanics and Engineering*, in press.