Simulations of curved turbulent boundary layers: a progress report

By P. Orlandi[1]

1. Introduction

The objective of this work is to develop a space-time accurate numerical method for the solution of incompressible Navier-Stokes equations in generalized coordinates. The resulting code is to be used for direct and large-eddy simulation of turbulence in complex geometries. In a previous paper (Orlandi 1989), the system of Navier-Stokes equations in general curvilinear coordinates was solved by a second-order accurate finite-difference scheme. Satisfactory results were obtained for several flows in two and three dimensions. The system of Navier-Stokes for the fluxes are given in Orlandi (1989). The main deficiency of the numerical scheme was the large CPU time required for the solution of the Poisson equation for the "pressure" field. The point SOR relaxation, in conjunction with a multigrid scheme, was used for the Poisson equation. In some cases, particularly with very fine grids, it was impossible to obtain a divergent-free flow.

The "pressure" solver has been improved by introducing a Fourier transform in the direction with Cartesian coordinates, and a line zebra iterative scheme has been used for the relaxation scheme. The line sweeps are used for the direction of stretched coordinates. The zebra scheme allows a very efficient vectorization of the "pressure" solver and reduces the required CPU time. This improvement resulted in a higher convergence rate and the divergence-free condition was always obtained. The Fourier transform leads to a series of 2-D Helmholtz equations with the reduced wave number $\mathbf{k}$ as the parameter. At high $\mathbf{k}$, as expected, the convergence rate of the relaxation scheme is very high because the negative eigenvalues of the matrix have large magnitudes. Slower convergence was obtained for the zero-wave number, however. In all cases, tested a very small reduction factor was found for the first cycle $O(10^{-2})$, and in the subsequent cycles the reduction factor was approximately 0.3. The maximum residual of 10^{-6} is reached at $\mathbf{k} = 0$ by few $\mathbf{W}$ multigrid cycles with two iteration sweeps at each line zebra relaxation. At high $\mathbf{k}$, the convergence was obtained by a single multigrid cycle. This efficient "pressure" solver has been incorporated into the direct simulation of flows over riblets by Choi (1990). As a consequence, he could employ a much finer grid than the grid previously used by Orlandi (1989). A very fine grid is required to represent the physics of drag reduction by the riblets. The entire numerical method and the associated computer program have reached a level of robustness that permits its use as a tool to obtain reliable flow fields.

1 Dipartimento di Meccanica ed Aeronautica Universita' di Roma "La Sapienza"

In practical applications, one often encounters turbulent flows along curved boundaries with inflow and outflow conditions. In these cases, the conventional Reynolds averaged turbulence models have not been very satisfactory, as found in the 1980-81 Stanford conference on complex turbulent flows (Kline *et al.* 1981). A full numerical simulation, e.g. for a flow around a turbine blade or an airfoil, requires an enormous amount of computer resources, mainly because these flows develop along the streamwise direction for hundreds of boundary layer thicknesses. Considering that usually in the direct simulation it is necessary to have at least ten grid points per boundary layer thickness, at least 10^3 grid points are required in the streamwise direction. A grid of 128×128 is necessary along the vertical and spanwise directions to describe the interaction of large and small structures at moderately low Reynolds numbers. With such large grid requirements, it appears to be difficult to do a direct simulation of the flow considered by Barlow and Johnston (1988), consisting of a boundary layer growing on a flat plate subjected to a relatively strong curvature followed by a recovery section. The direct simulation of this flow requires inflow conditions for the instantaneous turbulent velocity field similar to those currently tested for other flows in the CTR.

The use of a large eddy simulation, with an affordable grid of say $64 \times 64 \times 64$, could give satisfactory results in simulating the Barlow & Johnston flow at a $Re_\theta = 1300$. On the other hand, it is unlikely that two-dimensional Reynolds averaged computations of such flows can give good predictions if the dominant role of the three dimensional structures is not incorporated somehow in the turbulence model.

Enroute to large-eddy simulation of curved turbulent flows, we investigated whether the aforementioned numerical scheme can predict the well-known features of flows in presence of curved boundaries. In presence of concave walls, centrifugal instabilities cause the generation of Gortler vortices which enhance transition to turbulence. For this case, a well-documented experiment of Swearingen & Blackwelder (1987) exists together with the numerical simulation of Liu & Domaradzki (1990). In the numerical simulation of Liu & Domaradzki, it was assumed that the boundary layer is parallel; this assumption allows the application of periodic boundary conditions in the streamwise direction. This is not a realistic assumption, and in the past it was criticized by Hall (1983), who showed that the assumption could give erroneous results. Liu & Domaradzki computed a time developing flow rather than a spatially evolving flow. They were able to reproduce qualitatively the results of Swearingen & Blackwelder, but the transition to turbulence in the numerical simulation occurred later than in the experiment. A reason for the disagreement could reside on the assumption of parallel flow.

In the present paper, a preliminary attempt is made to compute the spatially evolving flow of Swearingen & Blackwelder. To reduce the streamwise distance, the inflow was at a distance $x = 60cm$ from the leading edge, as done by Liu & Domaradzki. See the sketch of the geometry in Fig.1. In the experiments of Swearingen & Blackwelder, Gortler vortices were not present at $x = 60cm$, but they were present at $x = 90cm$. At this position, the displacement thickness δ^* in between the vortices reaches the maximum value, and the δ^* outside the vortices

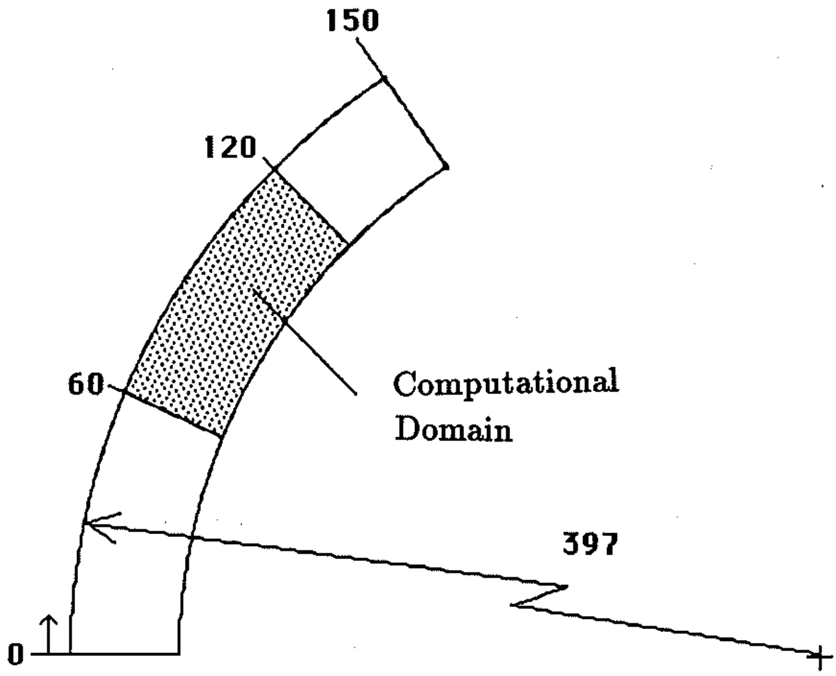

FIGURE 1. Sketch of the geometry.

reaches the minimum. We do not intend to compute the transition to turbulence, thus the outflow condition has been located at $x = 110cm$.

The parameters of the present simulation are those of the experiment. The initial Gortler number is $Go = (U_0\theta/\nu)\sqrt{\theta/R} = 5.6$, the radius of curvature of the wall is $R = 320cm$, the kinematic viscosity $\nu = 0.15cm^2/sec$, and the free-stream velocity $U_0 = 500cm/sec$; from these values it follows that the initial momentum thickness θ is $0.095cm$. For the sake of simplicity, at the inflow the Polhausen rather than the Blasius profile was used. In this case, $\theta/\delta = 37/315$. The initial boundary layer thickness is $\delta = 0.81cm$ and $Re = U_0\delta/\nu = 2685$. At $Go = 5.6$, the theoretical neutral stability limit corresponds to the non-dimensional wave number $k\theta = 2\pi\theta/\lambda = 0.24$; the spanwise dimension of the domain is then $\lambda = 3.0175\delta$. The "free-stream" was located at $y/\delta = 5$. The computational domain has been discretized by 65×79 points in the streamwise and normal directions, respectively. In the spanwise direction, two grids were used, the first with 33 and the second one with 65 points. The coarse grid, used particularly in the streamwise direction, is not sufficient for obtaining very accurate results. However, in this preliminary study, we intend to examine whether and in what degree the numerical method captures the growth of the Gortler vortices. These vortices modify the distribution of the streamwise velocity in (y, z)-planes, generating low-speed regions separated from high speed regions. These vortices enhance the transition to turbulence, and

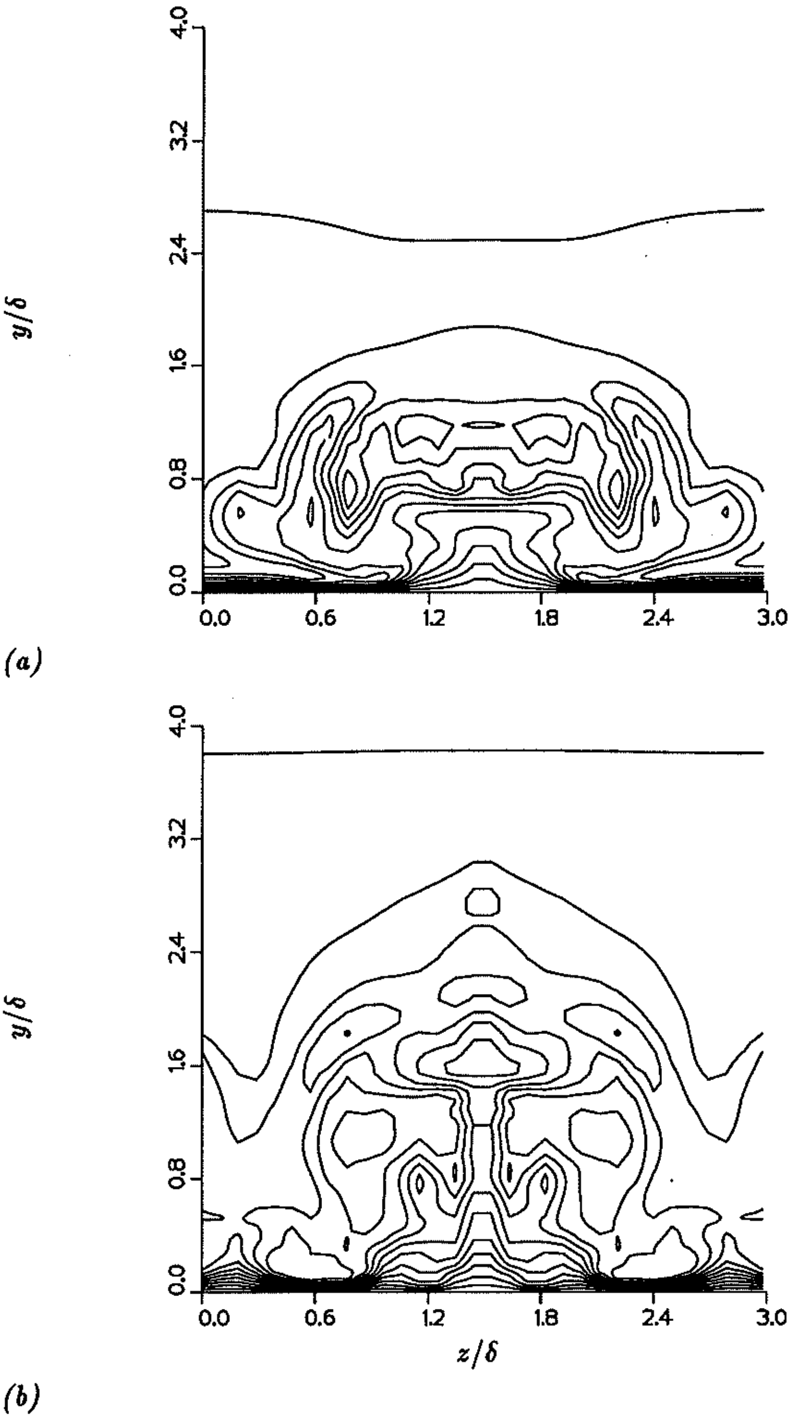

FIGURE 2. Contours of streamwise velocity in (y, z)-planes, grid $65 \times 79 \times 33$: a) $x/\delta = 98.25$; b) $x/\delta = 121.5$.

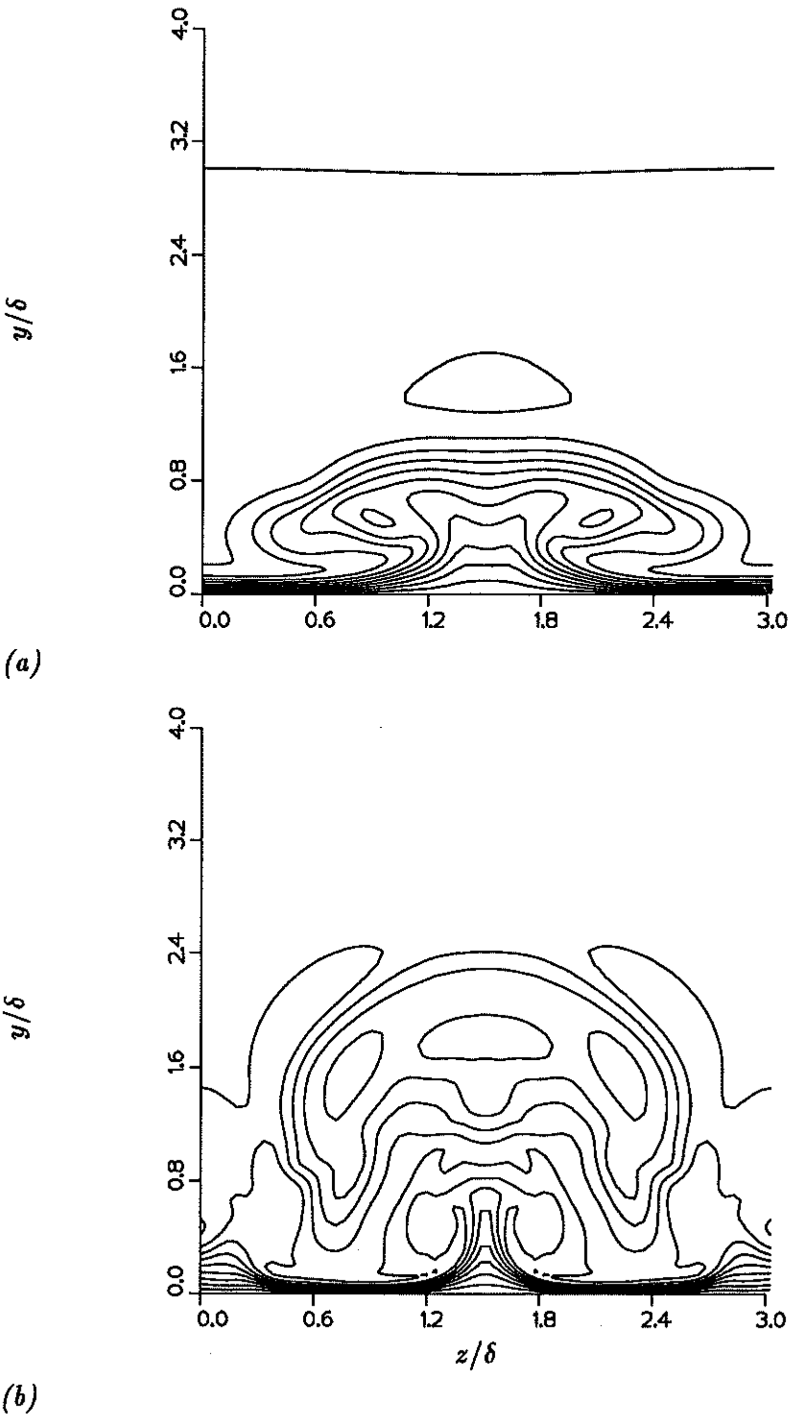

FIGURE 3. Contours of streamwise velocity in (y,z)-planes, grid $65 \times 79 \times 65$: a) $x/\delta = 98.25$; b) $x/\delta = 121.5$.

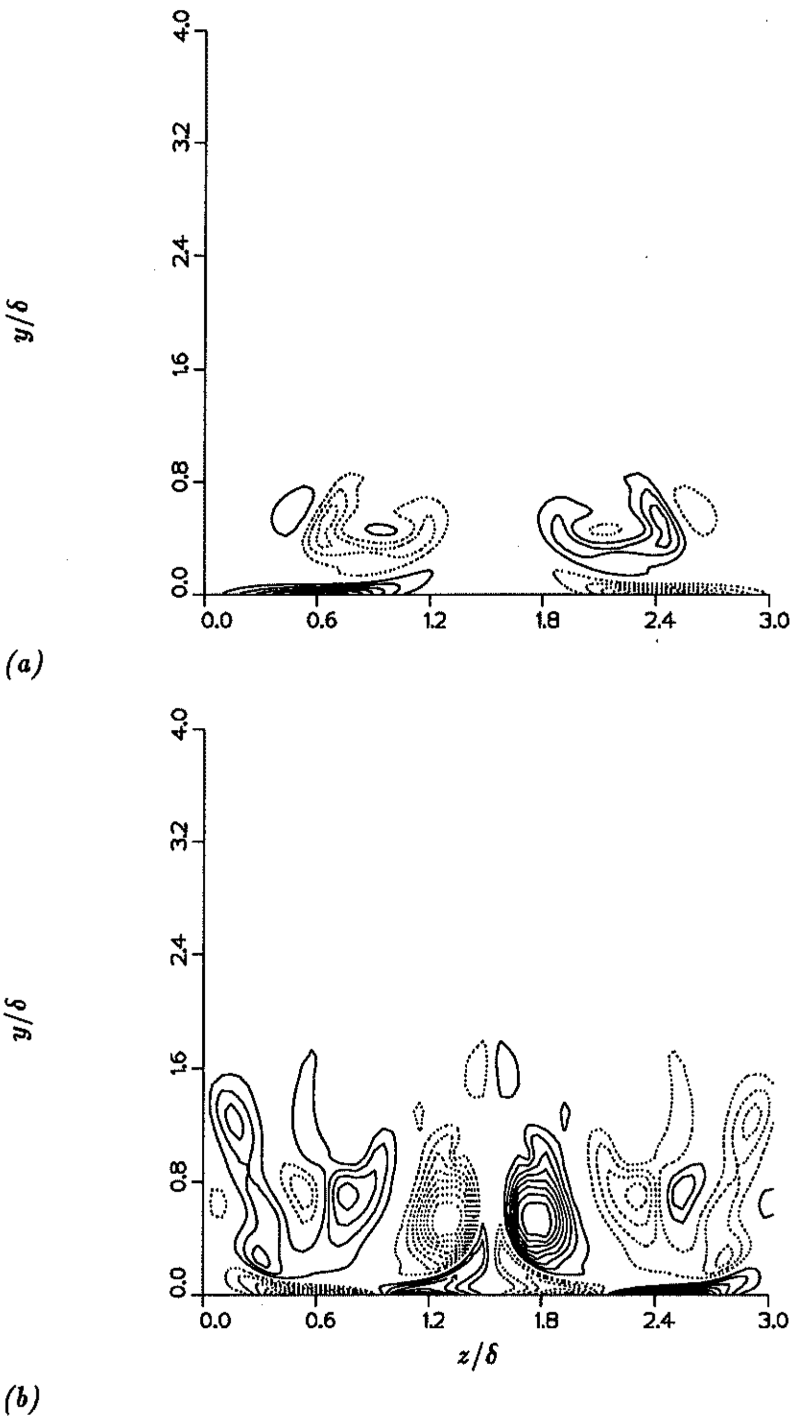

FIGURE 4. Contours of streamwise vorticity in (y, z)-planes, grid $65 \times 79 \times 65$: a) $x/\delta = 98.25$; b) $x/\delta = 121.5$.

as pointed out by Swearingen & Blackwelder, inflectional spanwise profiles of the streamwise velocity are much more effective.

Liu & Domaradzki used the following initial velocity distribution

$$u(x,y,z) = u_0(y) + \epsilon u_1(y) cos(2\pi z/\lambda)$$

$$v(x,y,z) = \qquad \epsilon v_1(y) cos(2\pi z/\lambda)$$

$$w(x,y,z) = \qquad \epsilon w_1(y) sin(2\pi z/\lambda)$$

where $u_0(y)$ is the laminar Blasius profile and $u_1(y), v_1(y)$ and $w_1(y)$ are the component of the eigenmode obtained by the linear stability theory. We used a much simpler relation for the perturbations

$$w(x,y,z) = \epsilon sin(2\pi z/\lambda) sin(2\pi y/\delta), \quad y/\delta < 1.$$

and $w = 0$ elsewhere. In order to have a faster growth in time, $\epsilon = 0.05$ was used. The perturbation $u(y,z)$ was set to 0 and $v(y,z)$ was obtained from continuity. At the outflow, convective boundary conditions were imposed.

2. Results

In this section, we present some qualitative results which will demonstrate the formation and growth of the Gortler vortices. Quantitative comparisons with the experiment of Swearingen & Blackwelder will be made only after checks of grid independence has been completed. At present, the results strongly depend on the grid.

We wish to point out that the number of grid points used in the present finite difference simulation (up to $65 \times 79 \times 65$) is much smaller than that used in the numerical simulation of Liu & Domaradzki. Considering that they were using a spectral method, the present grid is too coarse to lead to reliable results. The present scheme, however, has the advantage that it can be extended to different boundary shapes. Fig. 2 shows contour plots of streamwise velocity in the (y,z)-planes at the streamwise locations $x/\delta = 98.25$ and $x/\delta = 121.5$ obtained using a $65 \times 79 \times 33$ grid. The larger growth of the boundary layer between the vortices is apparent. The numerical solution also shows that the spanwise extent of the Gortler vortices grows with the downstream direction.

Fig.3 shows that the solution improves with doubling the number of grid points in the spanwise direction; a more defined structure is observed in the low-speed region. The numerical diffusion in the spanwise direction, although reduced in respect to the previous grid, is still quite large. The effect of the numerical diffusion is observed also in the contour plot of streamwise vorticity (Fig. 4), where two very strong vortices appear at the center, with some scattered vorticity on the side. The numerical simulation predicts that the boundary layer in the low-speed region grows by a factor of 1.8 at $x/\delta = 98.25$ and 3.4 at $x/\delta = 121.5$. The experiments reports factors of 2.0 and 2.6, respectively. A further grid refinement has been performed by using 128 points along the streamwise direction; the results improve, but still the high-speed regions are contaminated by the low-speed regions. The reason of this is the very coarse mesh used in the streamwise direction.

We have presented in this paper the work in progress to simulate turbulent boundary layers along concave walls. The numerical scheme requires 10 secs of CPU for the grid $65 \times 79 \times 65$ for a full-third order Runge-Kutta step, which is slightly larger than the typical times required by a spectral method. However, with finite-difference schemes, larger time steps can be taken as compared to spectral methods. The complete grid convergence test has not been finished. When this check is done, quantitative comparison with the experiments will be presented.

Acknowledgements

I am grateful to H. Choi, P. Moin, and J. Kim for many fruitful discussions.

REFERENCES

BARLOW, R. S. & JOHNSTON, J. P. 1988 Structure of turbulent boundary layer on a concave surface. *J. Fluid Mech.* **191**, 137-176.

CHOI, H. Private communication.

HALL, P. 1983 The linear development of Gortler vortices in growing boundary layers. *J.Fluid Mech.* **130**, 41-58.

KLINE, S. J., CANTWELL, B. J. & LILLEY, G. M.(EDS) 1981 The 1980-81 AFOSR-HTTM Stanford Conference on Complex Turbulent Flows: A Comparison of Computation and Experiment Volumes I, II and III. Stanford University.

LIU, P & DOMARADZKI, J. 1990 Direct numerical simulation of transition to turbulence in Gortler flow. *AIAA paper 90-0114*. 28th Aerospace Science Meeting, Reno.

ORLANDI, P. 1989 A numerical method for direct simulation of turbulence in complex geometries. *CTR Annual Research Briefs.* 215-228.

SWEARINGEN, J. D. & BLACKWELDER, R. F. 1987 The growth and breakdown of streamwise vortices in the presence of a wall. *J. Fluid Mech.* **182**, 255-290.