Journal of Engineering and Development, Vol. 10, No. 2, June (2006) ISSN 1813-7822

CFD Simulation of Incompressible Steady 2-D Laminar and Turbulent Flows Over a Flat Plate Using Two Equation Turbulence Model

Dr. Fouad Alwan Saleh
Mechanical Eng. Dept., College of Engineering
Al-Mustansiriya University, Baghdad, Iraq

Asst. Lect. Ahmed kadhim Hussein
Mechanical Eng. Dept., College of Engineering
Babylon University, Hilla, Iraq

Abstract

The aim of the present work is to develop the computer program to simulate the steady two-dimensional laminar and turbulent flows. The finite volume method is used to solve the flow governing equations numerically. The Navier-Stokes equations are solved for the velocity flow field. Since all the variables are stored at the center of each control volume. The correct velocity field is then used to solve k-epsilon equations. The eddy viscosity, that represents the influence of turbulence on the mean flow field, can be calculated from those values of k and epsilon obtained. The boundary layer on a flat plate is employed as a test case because it is one of the standard famous problems for the validation of CFD program and a Reynolds number is chosen at 1400 as a case study. It is found that the computed results are in good agreement with the available published data. Moreover, the governing equations are solved by using body-fitted coordinates system so that this computer program can be developed further for the simulation of laminar and turbulent flows over any object of complex geometry in the future.

الخلاصـــــــة

الهدف من الدراسة الحالية هو تطوير برنامج حاسوبي معد سلفا،لغرض نمدجة جريان طباقي واضطرابي مستقر ثنائي البعد استخدمت في الدراسة الحالية طريقة الحجوم المحددة لحل المعادلات الرياضية المستخدمة عدديا.تم أيجاد سرع الجريان بواسطة حل معادلات نافير-ستوكس الثنائية الابعاد.الطبقة المتاخمة على صفيحة مستوية تم استخدامها كحالة اختبار لتحقيق دقة النتائج حيث تم مقارنة النتائج المستحصلة مع نتائج سابقة وأبدت توافق جيد في النتائج.في الدراسة الحالية تم استخدام نظام إحداثيات الجسم المتوافقة بحيث تساعد على استخدام برنامج الدراسة الحالية لنمدجة الجريان الطباقي والاضطرابي لأشكال معقدة مستقبلا.

1. Introduction

CFD simulation of fluid mechanics flow problems covers many important applications in mechanical engineering. Understanding, the flow behavior is therefore important for the design and development of engineering applications. In fluid dynamics, the flow behavior is governed by the continuity equation, the Navier-Stokes equations, the energy equation and the equation of state. For incompressible flow, where the free-stream Mach number is lower than 0.5, the effects of temperature variation on fluid properties are very small, so that the fluid properties can be practically treated as constant. Therefore, the continuity and Navier-Stokes equations are adequate for the prediction and study of viscous incompressible flow. To study turbulent flow, the continuity and Navier-Stokes equations can be solved directly by any numerical method. However, the solution requires the large number of grid points, volumes, elements or other form of sub-domain to capture the characteristics of turbulent flow. In general, the turbulent flow is predicted and studied on the basis of mean quantities. By this way, the continuity and Navier-Stokes equations are time-averaged. From the other hand, turbulence models have been developed and widely used with success over a wide range of engineering applications [1].

The current work is aimed to develop the computer program to simulate the steady two-dimensional turbulent flow using two-equation turbulence model.

2. Mathematical Analysis

Incompressible steady flow is governed by the continuity equation and the Navier-Stokes equations where all the fluid properties are constants. For turbulent incompressible flow, these governing equations are essentially time-averaged and the resulting solution is the mean flow field. This technique gives rise to the additional unknown terms which cause an important problem. This problem can be solved using a two equation turbulence model [2]. The following equations are used in the analysis:

2-1 Continuity Equation

$$\frac{\partial}{\partial \mathbf{x}_j}\left(\rho \bar{\mathbf{u}}_j\right) = 0.0 \quad \text{.......... (1)}$$

where, ρ is the fluid density and $\bar{u}_j$ is the flow velocity.

2-2 Navier-Stokes Equations

$$\frac{\partial}{\partial \mathbf{x}_j}\left(\rho \bar{\mathbf{u}}_j \bar{\mathbf{u}}_i\right) = \frac{\partial}{\partial \mathbf{x}_j}\left(\bar{\mathbf{t}}_{ij} + \tau_{ij}\right) - \frac{\partial \bar{\mathbf{P}}}{\partial \mathbf{x}_i} \quad \text{.......... (2)}$$

where, $\overline{P}$ is the pressure, and $\bar{t}_{ij}$ and τ_{ij} are the laminar and turbulent flow shear stresses respectively with the following definitions [3]:

$$\bar{t}_{ij} = \mu\left[\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) - \frac{2}{3}\theta_{ij}\frac{\partial \bar{u}_k}{\partial x_k}\right] \quad \text{.......} (3)$$

where, μ is the fluid dynamic viscosity and θ_{ij} is the Draic delta function where, $\theta_{ij} = 0$ for $i \neq j$ and $\theta_{ij} = 1$ for $i = j$, and the turbulent shear stress is defined by [4]:

$$\tau_{ij} = \mu_t\left[\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) - \frac{2}{3}\theta_{ij}\frac{\partial \bar{u}_k}{\partial x_k}\right] - \frac{2}{3}\theta_{ij}\rho k \quad \text{.......} (4)$$

where, μ_t is the eddy viscosity and k is the kinetic energy of turbulence.

2-3 Two-Equation Turbulence Model

The formula of this model is given by [5]:

$$\mu_t = \rho\lambda\beta_\mu \frac{k^2}{\varepsilon} \quad \text{.......} (5)$$

where, λ is the model constant, β_μ is the turbulence damping function, and ε is the dissipation rate of k.

The Transport Equations for k and ε can be determined from [6]:

$$\frac{\partial}{\partial x_j}\left(\rho \bar{u}_j k\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \tau_{ij}\frac{\partial \bar{u}_i}{\partial x_j} - \rho\varepsilon + \rho G \quad \text{.......} (6)$$

where, σ_k is the model constant and G is the additional turbulence term.

$$\frac{\partial}{\partial x_j}\left(\rho \bar{u}_j \varepsilon\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + c_{\varepsilon 1} f_{\varepsilon 1}\tau_{ij}\frac{\partial \bar{u}_i}{\partial x_j}\frac{\varepsilon}{k} - \rho c_{\varepsilon 2} f_{\varepsilon 2}\frac{\varepsilon^2}{k} + \rho E \quad \text{.......} (7)$$

where, (σ_ε, $c_{\varepsilon 1}$, $c_{\varepsilon 2}$) are the model constants, and ($f_{\varepsilon 1}$, $f_{\varepsilon 2}$) are the damping functions, and E is the additional term.

For the $k-\varepsilon$ turbulence model [7], the model constants, damping functions and additional terms are provided as follows:

$$\lambda = 0.11,\ \sigma_k = 1.1,\ \sigma_\varepsilon = 1.27,\ c_{\varepsilon 1} = 1.42,\ c_{\varepsilon 2} = 2.0 \quad \text{(8)}$$

$$\beta_\mu = \exp\left[\frac{-3.4}{\left(1 + \frac{R_t}{50}\right)^2}\right] \quad \text{(9)}$$

$$f_{\varepsilon 1} = 1.11 \quad \text{and} \quad f_{\varepsilon 2} = 1 - 0.3\exp\left(-R_t^2\right) \quad \text{(10)}$$

$$G = -2\frac{\mu}{\rho}\left(\frac{\partial\sqrt{k}}{\partial x_i}\right)^2 \quad \text{(11)}$$

and

$$E = 2\frac{\mu}{\rho}\frac{\mu_t}{\rho}\left(\frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_k}\right)^2 \quad \text{(12)}$$

where,

$$R_t = \frac{\rho k^2}{\mu\varepsilon} \quad \text{(13)}$$

3. Numerical Scheme

The finite volume method [8] is used numerically to solve the governing equations which can be written in a general form as follows:

$$\frac{\partial}{\partial x_i}\left(\rho \bar{u}_i \phi\right) = \frac{\partial}{\partial x_i}\left(\Gamma \frac{\partial \phi}{\partial x_i}\right) + S^\phi \quad \text{(14)}$$

where, ϕ is the general dependent variable, Γ is the effective diffusion coefficient, and S^ϕ is the source/sink term of ϕ. To simulate the external flow past a body of complex shape, the general form of the governing equations is essentially transformed from the physical domain (x, y) into the computational domain (ξ, η) as in the following equation [9]:

$$\frac{\partial}{\partial \xi}(\rho U \phi) + \frac{\partial}{\partial \eta}(\rho V \phi) = \frac{\partial}{\partial \xi}\left[\frac{\Gamma}{J}\left(\alpha \frac{\partial \phi}{\partial \xi} - \beta \frac{\partial \phi}{\partial \eta}\right)\right] + \frac{\partial}{\partial \eta}\left[\frac{\Gamma}{J}\left(\gamma \frac{\partial \phi}{\partial \eta} - \beta \frac{\partial \phi}{\partial \xi}\right)\right] + JS^\phi \quad \text{(15)}$$

where,

$$U = \bar{u}\frac{\partial y}{\partial \eta} - \bar{v}\frac{\partial x}{\partial \eta}, \qquad V = \bar{v}\frac{\partial x}{\partial \xi} - \bar{u}\frac{\partial y}{\partial \xi}, \quad \text{(16)}$$

$$\alpha=\left(\frac{\partial x}{\partial \eta}\right)^2+\left(\frac{\partial y}{\partial \eta}\right)^2, \qquad \beta=\frac{\partial x}{\partial \xi}\frac{\partial x}{\partial \eta}+\frac{\partial y}{\partial \xi}\frac{\partial y}{\partial \eta}, \qquad \gamma=\left(\frac{\partial x}{\partial \xi}\right)^2+\left(\frac{\partial y}{\partial \xi}\right)^2, \text{ and}$$

$$\mathbf{J}=\frac{\partial x}{\partial \xi}\frac{\partial y}{\partial \eta}-\frac{\partial y}{\partial \xi}\frac{\partial x}{\partial \eta} \qquad \textbf{(17)}$$

Using the finite volume method, the computational domain is divided into a number of control volumes. The transformed equations can be integrated as follows:

$$[(\rho U\Delta\eta)\phi]_w^e+[(\rho V\Delta\xi)\phi]_s^n=\left[\frac{\Gamma\Delta\eta}{J}\left(\alpha\frac{\partial\phi}{\partial\xi}-\beta\frac{\partial\phi}{\partial\eta}\right)\right]_w^e +\left[\frac{\Gamma\Delta\xi}{J}\left(\gamma\frac{\partial\phi}{\partial\eta}-\beta\frac{\partial\phi}{\partial\xi}\right)\right]_s^n+(J\Delta\xi\Delta\eta)\bar{S}_P^\phi \qquad \textbf{(18)}$$

where, $\bar{S}_P^\phi$ is the average value of S^ϕ at the center P of each control volume, and (e, w, n, s) are the east, west, north and south faces of each control volume. The convection terms are approximated by the first-order upwind differencing scheme and the diffusion terms are estimated by the second-order central differencing scheme. Therefore, the standard form of the finite volume equation can be obtained as:

$$A_P^\phi\phi_P=A_E^\phi\phi_E+A_W^\phi\phi_W+A_N^\phi\phi_N+A_S^\phi\phi_S+b^\phi \qquad \textbf{(19)}$$

where,

$$A_E^\phi=\left(\frac{\Gamma}{J}\alpha\frac{\Delta\eta}{\Delta\xi}\right)_e+\max[0,-(\rho U\Delta\eta)_e],\quad A_W^\phi=\left(\frac{\Gamma}{J}\alpha\frac{\Delta\eta}{\Delta\xi}\right)_w+\max[0,(\rho U\Delta\eta)_w], \qquad \textbf{(20)}$$

$$A_N^\phi=\left(\frac{\Gamma}{J}\gamma\frac{\Delta\xi}{\Delta\eta}\right)_n+\max[0,-(\rho V\Delta\xi)_n],\quad A_S^\phi=\left(\frac{\Gamma}{J}\gamma\frac{\Delta\xi}{\Delta\eta}\right)_s+\max[0,(\rho V\Delta\xi)_s], \qquad \textbf{(21)}$$

$$A_P^\phi=A_E^\phi+A_W^\phi+A_N^\phi+A_S^\phi,$$

$$\text{and } b^\phi=(J\Delta\xi\Delta\eta)\bar{S}_P^\phi-\left[\frac{\Gamma\Delta\eta}{J}\left(\beta\frac{\partial\phi}{\partial\eta}\right)\right]_w^e-\left[\frac{\Gamma\Delta\xi}{J}\left(\beta\frac{\partial\phi}{\partial\xi}\right)\right]_s^n \qquad \textbf{(22)}$$

The continuity equation is not solved directly with other governing equations. The p'-equation is solved instead to obtain the pressure correction p' and its value is used to correct the values of pressure and velocities to satisfy the conservation law of mass. The p'-equation can be written in a standard form as follows [10]:

$$A_P^p p'_P = A_E^p p'_E + A_W^p p'_W + A_N^p p'_N + A_S^p p'_S + m_P \quad \text{.......................................} \quad (23)$$

where,

$$A_E^p = \left(\rho B \frac{\Delta\eta}{\Delta\xi}\right)_e,\ A_W^p = \left(\rho B \frac{\Delta\eta}{\Delta\xi}\right)_w,\ A_N^p = \left(\rho C \frac{\Delta\xi}{\Delta\eta}\right)_n,\ A_S^p = \left(\rho C \frac{\Delta\xi}{\Delta\eta}\right)_s,$$

$$A_P^p = A_E^p + A_W^p + A_N^p + A_S^p,$$

$$\text{and } m_P = \left(\rho U^* \Delta\eta\right)_w - \left(\rho U^* \Delta\eta\right)_e + \left(\rho V^* \Delta\xi\right)_s - \left(\rho V^* \Delta\xi\right)_n \quad \text{..........................} \quad (24)$$

U^*, V^* are calculated from the resulting velocities of the Navier -Stokes equations, whereas

$$B = B^u \frac{\partial y}{\partial \eta} - B^v \frac{\partial x}{\partial \eta}, \quad \text{and } C = C^v \frac{\partial x}{\partial \xi} - C^u \frac{\partial y}{\partial \xi} \quad \text{.....................................} \quad (25)$$

where,

$$B^u = -\frac{\Delta\xi\Delta\eta}{A_P^u}\frac{\partial y}{\partial \eta},\ B^v = \frac{\Delta\xi\Delta\eta}{A_P^v}\frac{\partial x}{\partial \eta},\ C^u = \frac{\Delta\xi\Delta\eta}{A_P^u}\frac{\partial y}{\partial \xi},$$

$$\text{and } C^v = -\frac{\Delta\xi\Delta\eta}{A_P^v}\frac{\partial x}{\partial \xi} \quad \text{..} \quad (26)$$

However, the current grid system is technically rather complicated for programming and requires a large amount of computer storage. This point becomes clear when the computer program is developed further for real-world applications. The collocated grid system is employed in this work so that all the variables are stored at the center of each control volume gradient [11].

In the current work, the boundary layer on a flat plate is chosen as a test case. Physically, the pressure field of this flow is constant and known throughout the flow domain. Therefore, the pressure correction p' obtained is used to correct the velocities only, not to correct the pressure because the pressure itself is already known and constant.

4. Computer Program Algorithm

The algorithm for the simulation of turbulent two-dimensional flow can be summarized as follows:

1. Using the p'-equation for the pressure correction.
2. Correct the velocities by the pressure correction.
3. Calculate the k-equation for the turbulence kinetic energy.
4. Start the computation with an initial value of velocities, pressure correction, turbulence kinetic energy and dissipation rate of turbulence kinetic energy.

5. Calculate the Navier-Stokes equations for the velocities.
6. Calculate the ε-equation for the dissipation rate of turbulence kinetic energy.
7. Repeat from step (2) until the solution converges the steady state.

5. Results and Discussion

Calculations are used for laminar and turbulent incompressible flows and program input data are summarized in **Table (1)** below:

Table (1) Program input data

Parameter	Laminar Flow	Turbulent Flow
ξ_{max}	105	250
η_{max}	105	160
Re_L	2 x 10 ^ 4	6 x 10 ^6
P_∞ (Pa)	101325	101325
Relaxation Factor	0.31	0.31

where, ξ_{max} and η_{max} are the numbers of grid lines used in the computational domain, Re_L is the Reynolds number based on the length of the flat plate and the free-stream velocity , P_∞ is the free-stream pressure, and the relaxation factor is used to stabilize the numerical scheme used.

Figures (1) and **(2)** show the velocity distribution and skin friction coefficient of the laminar boundary layer on a flat plate in which the computed results are compared with the analytical solution. In **Fig.(1)**, $u^* = u/U_\infty$ and $y^* = y\sqrt{\rho U_\infty/\mu x}$. It is found that the computed results are in good agreement with the analytical solution so that the numerical method used is accurate for the simulation of the laminar boundary layer on a flat plate. It is found that the computed results are in good agreement with the analytical solution [12] so that the $k-\varepsilon$ turbulence model is capable of simulating the turbulent boundary layer on a flat plate. **Figures (3)** to **(6)** show the distributions of the velocity, Reynolds stress, turbulence kinetic energy and dissipation rate of turbulence kinetic energy of the turbulent boundary layer on a flat plate. They are normalized in dimensionless units .As problem size increases, the speed up also increases rapidly. Also, as more grids are included, more speed is required.

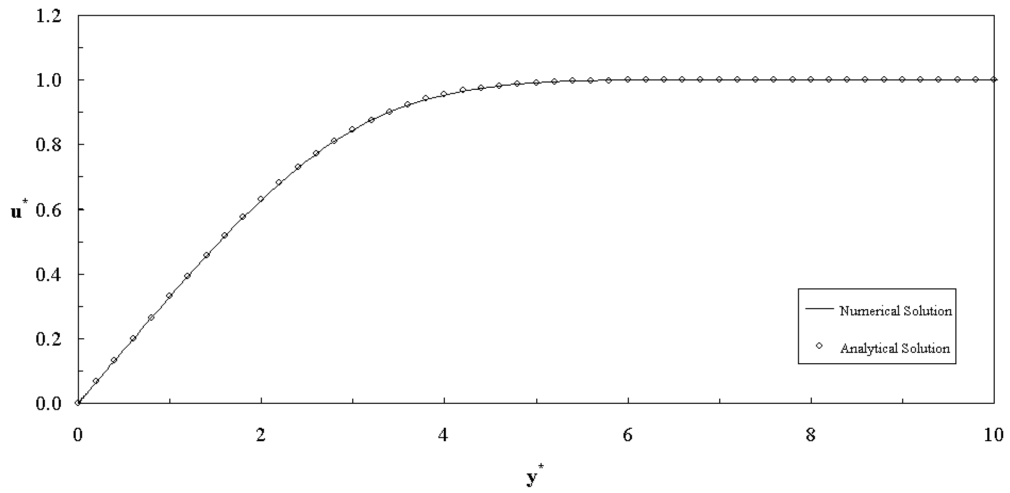

Figure (1) Velocity distribution of the laminar boundary layer on a flat plate

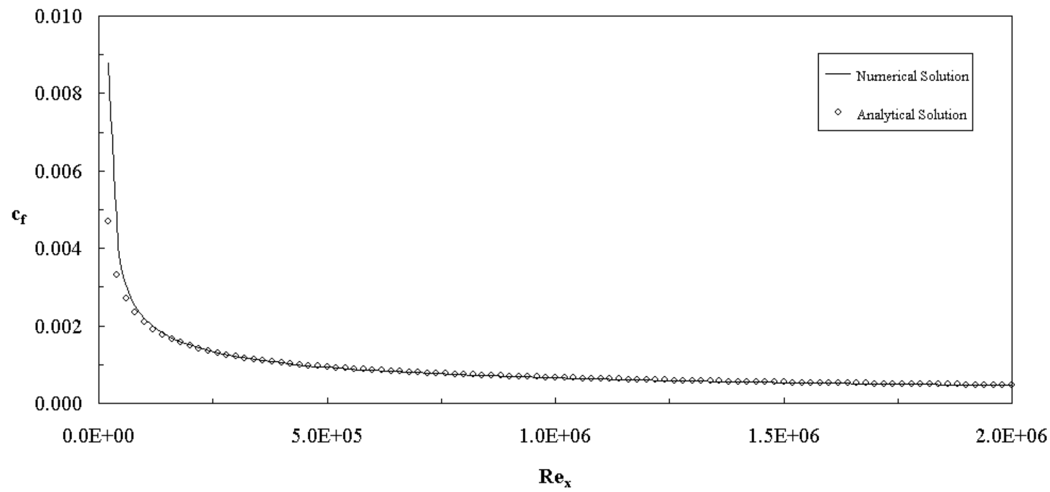

Figure (2) Skin friction coefficient of the laminar boundary layer on a flat plat

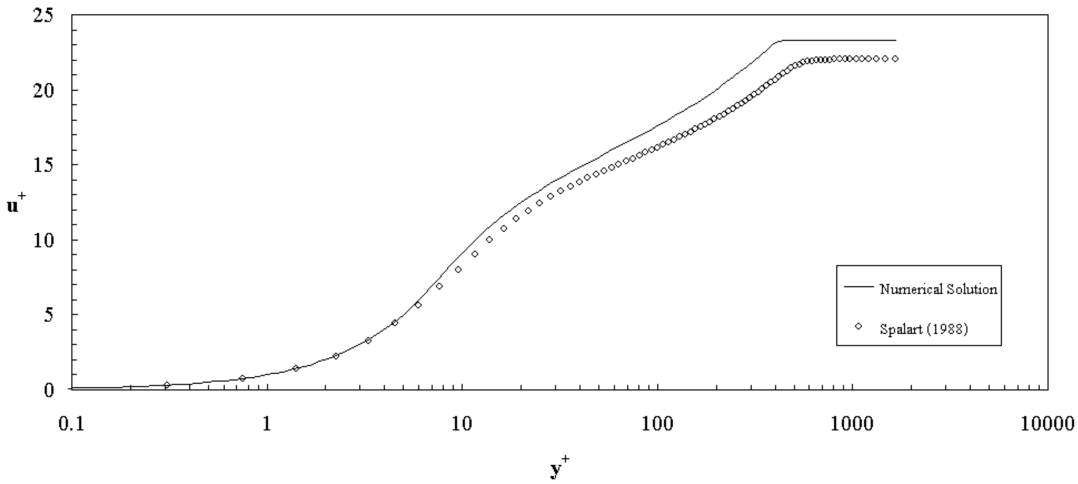

Figure (3) Velocity distribution of the turbulent boundary layer on a flat plate at Re_θ = 1400

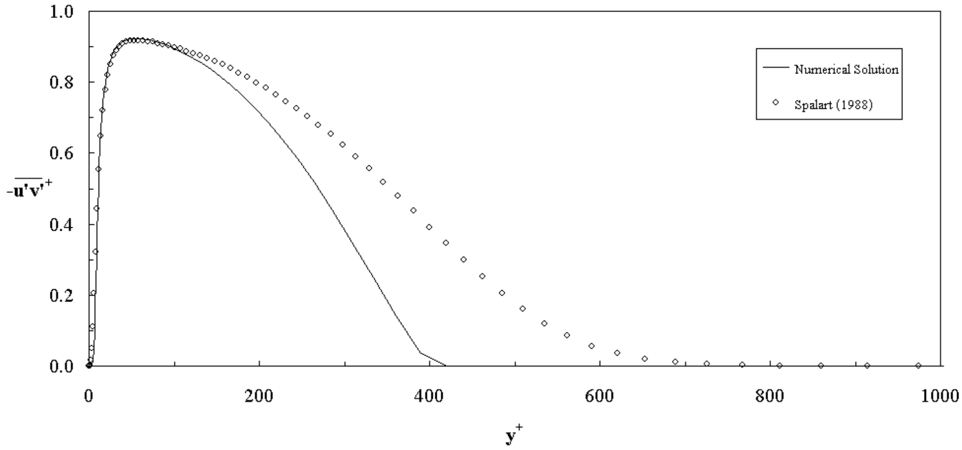

Figure (4) Reynolds-stress distribution of the turbulent boundary layer on a flat plat at Re_θ = 1400

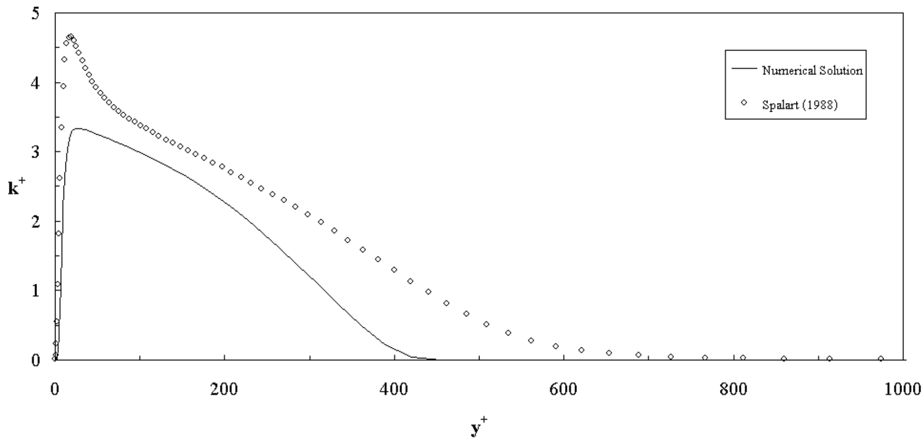

Figure (5) Distribution of the turbulence kinetic energy of the turbulent boundary layer on a flat plate at Re_θ = 1400

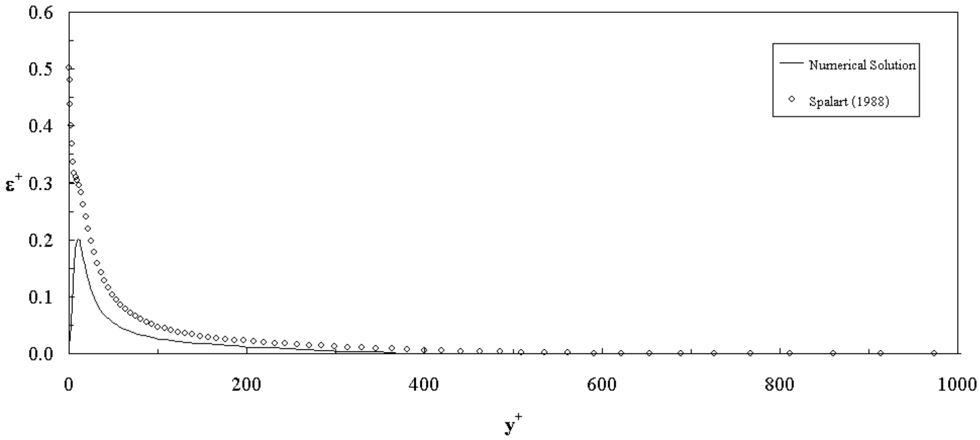

Figure (6) Distribution of the dissipation rate of k of the turbulent boundary layer on a flat plate at Re_θ = 1400

6. Conclusions

Both laminar and turbulent incompressible steady viscous flows are simulated in the current work. The numerical method used is capable of accurately predicting the laminar and turbulent boundary layers on a flat plate. The data of the turbulent boundary layer on a flat plate at $\mathbf{Re_\theta = 1400}$ is used as a test case to yield the performance of the $k-\varepsilon$ turbulence model of [7] .It is found that the computer program in general provides good results. Also, the use of body fitted coordinate system seems to be very effective in simulating this problem.

7. References

1. Majewski, R., and Wieteska, R., ***"Investigation of Weno Schemes for 3D Unstructured Grids"***, XXI ICTM, Warsaw, Poland, 2004, pp. 1-2.

2. Hoffmann, K. A., ***"Computational Fluid Dynamics for Engineers"***, Engineering Education System Publications, Texas, U.S.A., 1989.

3. Nemec, M., Aftosmis, A., and Pullian, T., ***"CAD-Based Aerodynamic Design of Complex Configurations using a Cartesian Method"***, NASA Technical Report, NASA-04001, 2004, pp. 1-13.

4. Gibson, M. M., Jones, W. P., and Whitelaw, J. H., ***"Turbulence Models for Computational Fluid Dynamics"***, Course Lecture Notes, Department of Mechanical Engineering, Imperial College of Science, Technology and Medicine,1992..

5. Hinze, J. O., ***"Turbulence"***, 2nd Edition, McGraw-Hill Book Company, 1975.

6. Karki, K. C., and Patankar, S. V., ***"Pressure Based Calculation Procedure for Viscous Flows at All Speeds in Arbitrary Configurations"***, AIAA Journal, Vol. 27, No. 9, 1989, pp.1167-1174.

7. Launder, B. E., and Sharma, B. I., ***"Application of the Energy-Dissipation Model of Turbulence to the Calculation of a Flow Near a Spinning Disk"***, Letters in Heat and Mass Transfer, Vol. 1, 1974, pp. 131-138.

8. Anderson, J. D., ***"Computational Fluid Dynamics, the Basic with Applications"***, McGraw-Hill Book Company, U.S.A., 1995.

9. Sorensen, N., ***"3D Background Aerodynamics using CFD"***, Riso National Laboratory Publications, Roskilde, Denmark , 2002, pp. 1-18.

10. Hosseini, R., Rahimian, M., and Mirzaei, M., ***"Performance of High-Accuracy Schemes in Inviscid Fluxes Calculation"***, Unpublished Paper, By e-mail from: hoseinis @ me.ut.ac.ir.

11. Lang, N. J., and Shih, T. H., ***"A Critical Comparison of Two-Equation Turbulence Models"***, NASA Technical Memorandum 105237, 1991.

12. Spalart, P. R., ***"Direct Simulation of a Turbulent Boundary Layer"***, Journal of Fluid Mechanics, Vol. 187, 1988, pp. 61-98.