A CFD model for turbulent and laminar boundary layer momentum and heat transfer on a flat plate

N. Ghiasi [1*], **A. Azimi** [2], **H. Pahlevanzade** [3]
1-PhD Student of mechanical engineering, Tarbiat Modares University
2-PhD Student of chemical engineering, Tarbiat Modares University
3-Associate Professor, chemical engineering, Tarbiat Modares University

ABSTRACT

A CFD $\kappa-\varepsilon$ model is presented for turbulent momentum and heat transfer on a flat plate. The layer is simulated for both compressible and incompressible flow. Momentum and heat boundary layer thickness, velocity and heat profile are compared for turbulent and laminar flow. The equations are solved by Finite Volume technique in a Cartesian frame using non-uniform grid in the vicinity of viscous sub-layer. Also a fine grid size was chosen to have less CPU time. In this research we showed that the boundary layer thickness and friction factor is grater in compressible flow. The results are in good agreement with other proposed empirical equations and also Prandtl & Blasius Theory.

Key word: Turbulent boundary layer, Momentum and heat transfer, Compressible flow, Flat plate, $\kappa-\varepsilon$ model

INTRODUCTION

RANS equations is now almost routinely used to investigate fundamental aspects of turbulence mechanics, to help validate statistical closures and to obtain predictions for flows in which unsteady events associated with turbulence are of major interest or influence [1,2]. Although RANS continues to be an expensive approach at practically relevant Reynolds numbers, the expense is tolerable when the flow being simulated is remote from walls. However, flows which are substantially affected by near-wall shear and turbulence pose serious challenges as a consequence of the need to increase the near-wall grid resolution boundary with $N \propto Re^{1.8}$, in order to restrict the distance between the wall and the nodes closest to the wall to around $y^{+} = 2$ [3]. So a non-uniform grid should be used to capture the details of the near-wall flow. To solve the turbulent boundary layer equations, a turbulent model is needed to represent the Reynolds shear stress, i.e. $\overline{u'v'}$ in the boundary layer equations.

The improvement of turbulence models in such flow regimes requires a deep understanding of the dynamics and mechanisms of the flow and its behavior, some physical phenomena such as separation and reattachment. CFD is a valuable tool to improve our corresponding physical insight because it provides accurate two or three-dimensional and time-dependent information of the flow variables. Up to now, only a limited number of numerical simulations of turbulent boundary layers are available. The dynamics of turbulent boundary layers is fundamentally different from that of laminar boundary layers that undergo transition between separation and reattachment.

The physics of a turbulent boundary layer has first been numerically analyzed by Coleman and Spalart [4]. Recently, Skote et al. [5] and Skote and Henningson [6] performed a DNS of

* *Nima_ghiasi_te@yahoo.co.uk*

a separated turbulent boundary layer. An extensive study has been performed by Na and Moin [7] at a low Reynolds number (Re=300-3000, based on inlet free-stream velocity and momentum thickness). In their study, separation/ reattachment have been enforced by a strong adverse/ favourable pressure gradient in stream-wise direction.

Some studies also aim at analyzing an actually performed experiments of a heat transfer turbulent boundary layer (Kalter and Fernholz, [8]). In comparison to Na and Moin's study, it has a significantly higher Reynolds number and the flow is studied in a region with strong favourable pressure gradient. Therefore, Reynolds stresses play a crucial role in the momentum balance of the presented flow. This study aims at providing data for improving turbulence models and physical insight into the dynamics of compressiblity of this flow.

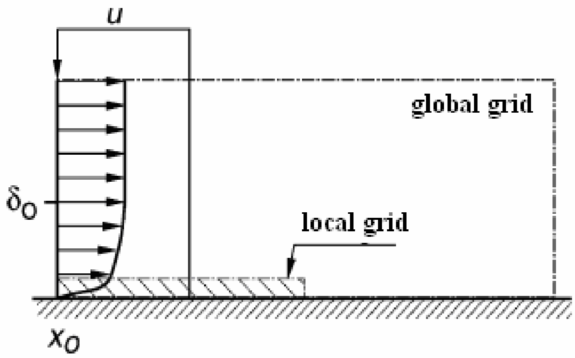
Objective of the current research is to simulate the heat and momentum transfer in boundary layer using finite volume technique. Non-uniform grid plus $\kappa-\varepsilon$ turbulent model is used to capture the details of the flow. Results are compared for turbulent and laminar compressible flow and Blasius solution.

GEOMETREY & FIELD EQUATIONS

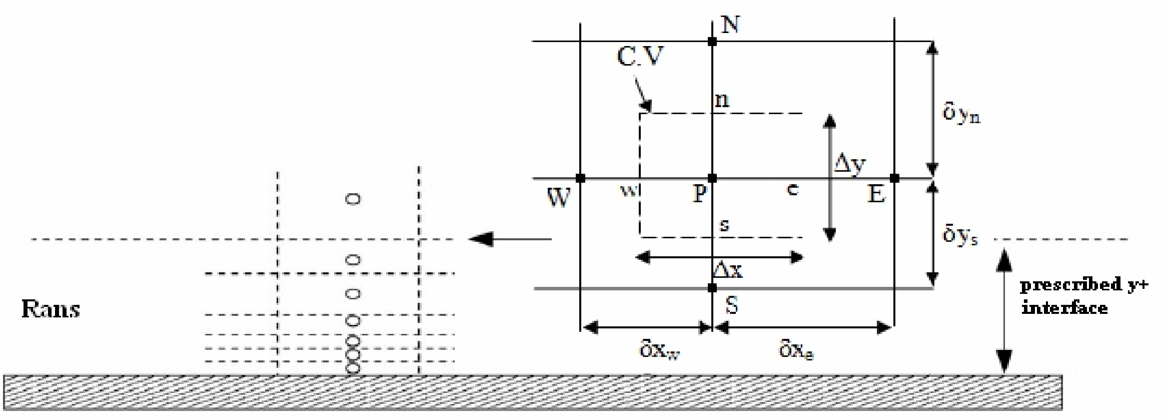
A schematic of boundary layer and grid generation is sketched in Fig. 1 and Fig. 2. As it is shown below the grid is finer in near-wall region to have better resolution.

The flow variables, velocity components and pressure, are defined on a non-uniform Cartesian mesh in a staggered arrangement. Principally, velocity components are stored in the centres of cell faces, while pressure and temperature is stored in the cell centres. The specific discrete formulations are derived by integrating the Navier–Stokes equations for an incompressible and compressible fluid over the corresponding control cells surrounding the definition points of the individual variables. We are using the midpoint rule for approximating the fluxes by the variables. The required interpolations and the approximation of the first derivatives are performed by linear interpolation and second order central finite difference formulations, respectively. This altogether ensures second order accuracy of the spatial discretisation (e.g., Ferziger and Peri_c, [9]).

The conservation equations (1) to (3) are discretised on staggered grid and solved using FV technique respectively. The convective terms are discretised using upwind scheme and implicit temporal scheme is also used [10]. Also density variation due to temperature change is calculated in every step using the equation of state for ideal gas. Because of the flow considered compressible so Favor-averaging method is used for the terms including (ρ) parameter. The advantage of the present algorithm is the easy treatment of boundaries, at which only velocity boundary conditions have to be specified.



Fig(1): Geometry of boundary layer



Fig(2): A schematic of non-uniform grid and staggered mesh

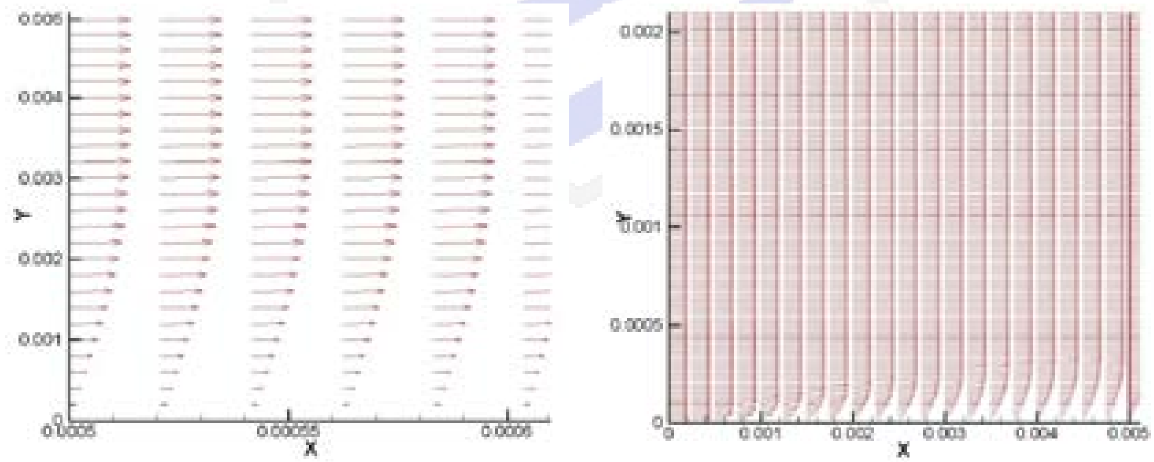
$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)=0 \quad (1)$$

$$\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}=-\frac{dp}{dx}+\frac{\partial}{\partial y}(\mu\frac{\partial u}{\partial y}-\rho\overline{u'v'}) \quad (2)$$

$$\rho\frac{\partial H}{\partial t}+\rho u\frac{\partial H}{\partial x}+\rho v\frac{\partial H}{\partial y}=\frac{\partial}{\partial y}(\frac{\mu}{pr}\frac{\partial H}{\partial y}-\rho cp\overline{v'T'}+u((1-pr)\mu\frac{\partial u}{\partial y}-\rho\overline{u'v'})) \quad (3)$$

RESULTS AND DISCUSSION

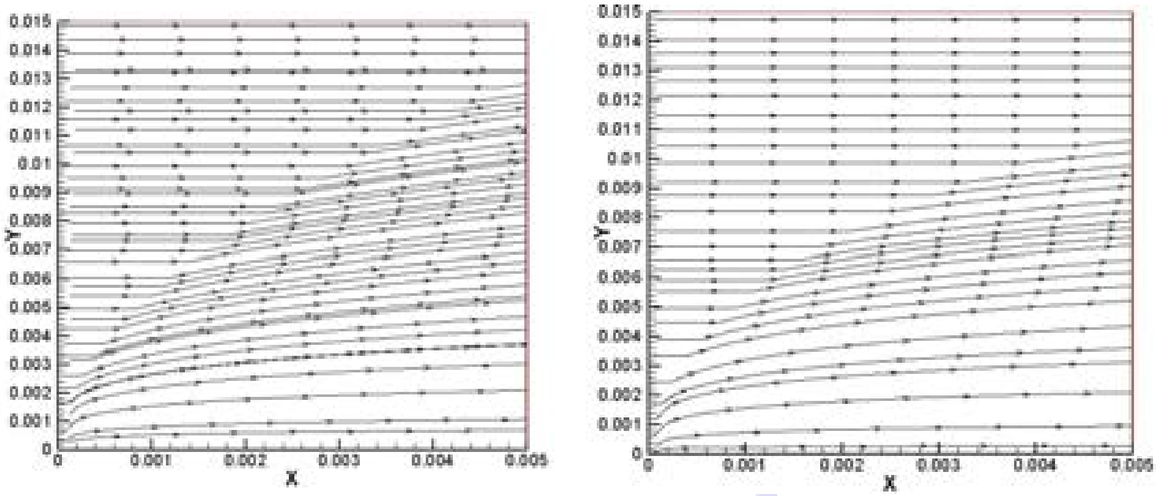
The compressibility and density variation is just because of the temperature gradient (the wall temperature is considered more then the flow temperature) in the results. The first set of results is a comparison between laminar and turbulent velocity vector profile in compressible flow:



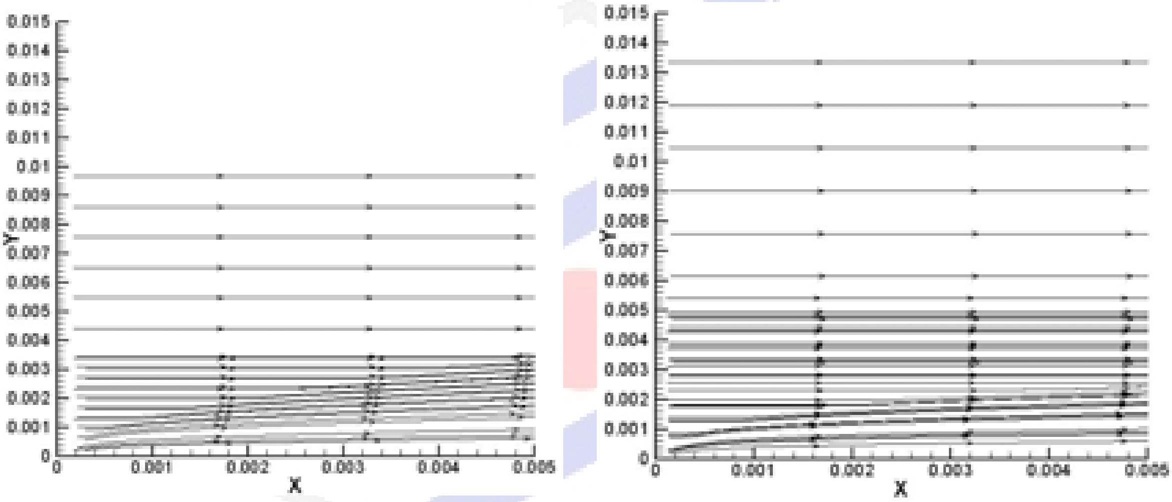
Fig(3): Schematic of velocity vector for laminar (left figure) and turbulent (right figure) compressible flows

It is shown that the velocity magnitude reaches to the free stream velocity in a closer distant for turbulent flow as it is for incompressible flow; this is because of the turbulent viscosity which is due to the nature of the turbulent flow.

Stream lines for compressible and incompressible flow is also obtained for laminar and turbulent flow in Fig. 4 and Fig. 5 respectively; as the density decreased due to temperature rise in compressible flow, the Reynolds number will decrease too, so the boundary layer thickness will increase.

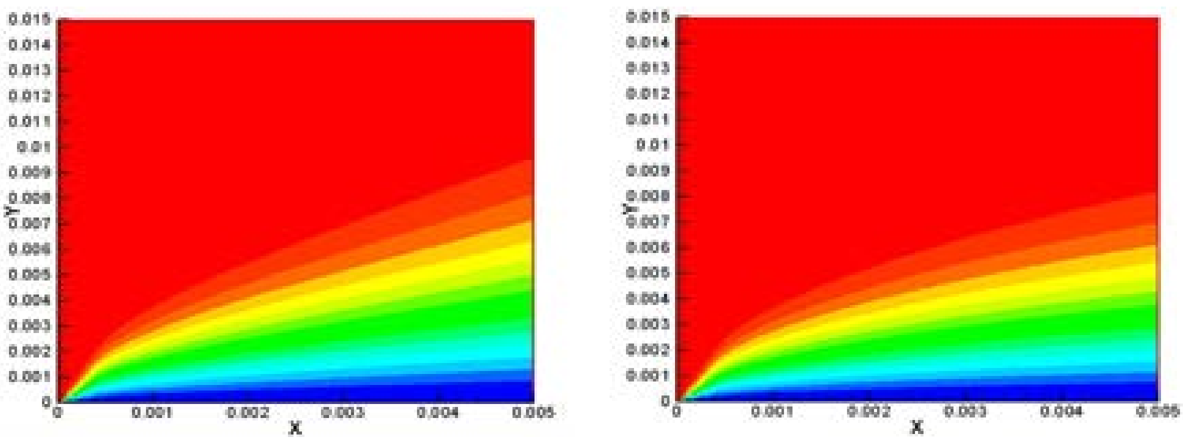


Fig(4): Schematic of stream-lines for laminar flow in compressible (left figure) and incompressible (right figure) flows

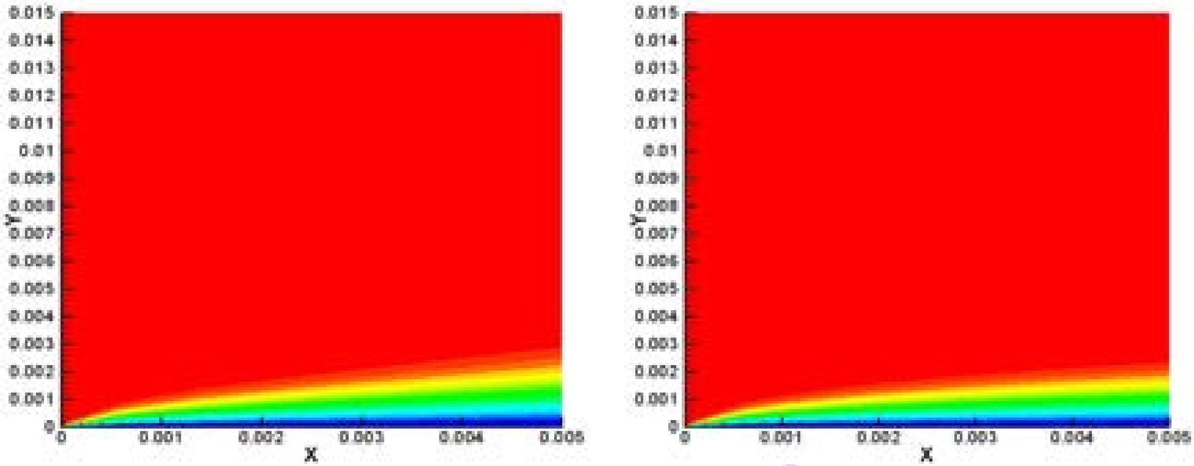


Fig(5): Schematic of stream-lines for turbulent flow in compressible (left figure) and incompressible (right figure) flows

The temperature distribution in compressible and incompressible flow is obtained for laminar and turbulent flow in Fig. 6 and Fig. 7 respectively; as the heat transfer coefficient increases in turbulent flow so the temperature reaches the free stream temperature in a closer distant than laminar flow.



Fig(6): Temperature contours for laminar flow in compressible (left figure) and incompressible (right figure) flows



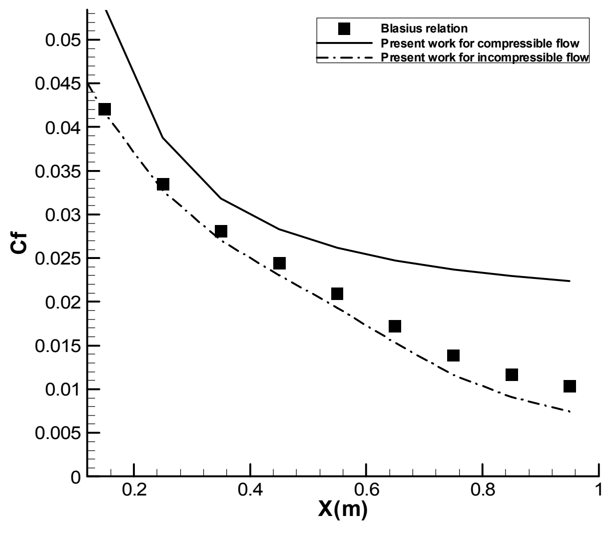
Fig(7): Temperature contours for turbulent flow in compressible (left figure) and incompressible (right figure) flows

Other important phenomena can be considered from these figures by making comparison between compressible and incompressible heat boundary layer. As the prandtl number is constant, so the increase in momentum boundary layer thickness will cause an increase in heat boundary layer thickness too, which is verified by our results.

The friction factor variation for compressible and incompressible flow along the X axis is shown in Fig. 8. The graph is sketched for turbulent flow and is compared with Blasius relation for friction factor in turbulent incompressible flow specified by:

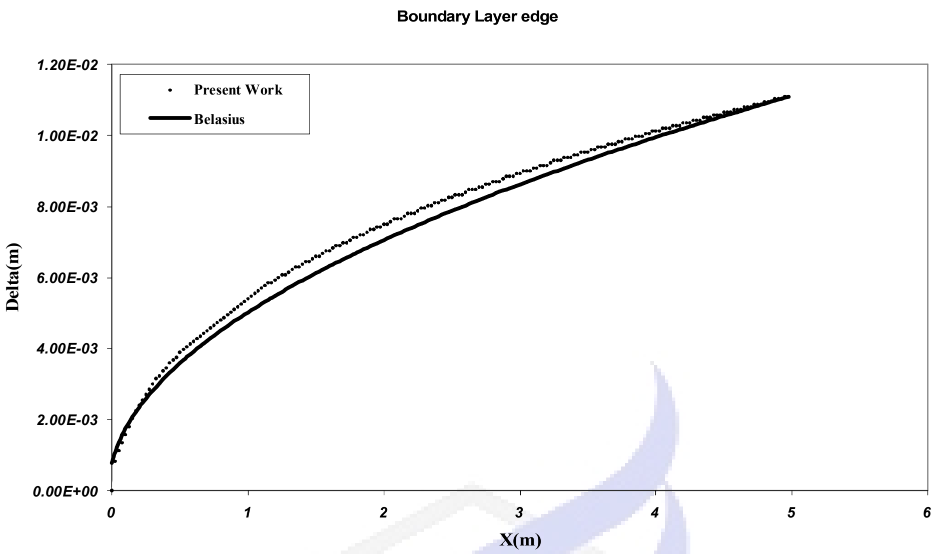
$$C_f \approx \frac{0.027}{\mathrm{Re}_x^{\ 1/7}} \qquad (4)$$

The result for the incompressible turbulent boundary layer is almost coinciding with Blasius relation for friction factor but it shows a decrease in friction factor in comparison with the compressible flow. The discrepancy can be explained as follow: as the density decreases in compressible flow due to temperature rise, the Reynolds number will decrease consequently; so the friction factor will increase according to the relation (4). The results show that the discrepancy increases by getting farther along the plate.



Fig(8): Friction factor variation for compressible and incompressible turbulent flow and comparison with Blasius relation

The last graph is a comparison between momentum boundary layer thickness obtained from present work and Blasius solution. It is shown that our results are in good agreement with Blasius work.



Fig(9): A comparison between momentum boundary layer thickness of present work and Blasius solution

CONCLUSION

Simulation of heat and momentum boundary layer using finite volume technique is well done in this work. Non-uniform grid plus $\kappa - \varepsilon$ turbulent model is used to capture the details of the flow. The main results are as follows:

1- The momentum boundary layer thickness will increase in compressible flow in comparison with incompressible flow due to decrease in Reynolds number.
2- As the prandtl number is constant, so the increase in momentum boundary layer thickness will cause an increase in heat boundary layer thickness.
3- The friction factor will increase in compressible flow in comparison with incompressible flow due to decrease in Reynolds number according to the Blasius relation.

REFERENCES

1- Tessicini, F., Temmerman, L., "Approximate near-wall treatments based on zonal and hybrid RANS–LES methods for high Reynolds numbers", International Journal of Heat and Fluid Flow 27 (2006) 789–799.

2- Anderson, D. A., Tannahill, J. C., Pletcher, R. H., "Computational fluid Mechanics and heat Transfer", Hemisphere (1984).

3- Cebeci, T., Smith, A.M.O., "Analysis of Turbulent Boundary Layers", Academic Press, New York (1974).

4- Coleman, G., Spalart, P., "Direct numerical simulation of a small separation bubble" In: Speziale, C., Launder, B. (Eds.), Near-Wall Turbulence Flows. Elsevier, Amsterdam, pp. 277–286(1993).

5- Skote, M., Henningson, D., Hirose, N., Matsuo, Y., Nakamura, T., "Parallel DNS of a separating turbulent boundary layer" In: Proceedings of the Parallel CFD 2000. NTNU, Trondheim, Norway (2000).

6- Skote, M., Henningson, D.S., "DNS of a separating turbulent boundary layer" In: Lindborg, E. et al. (Eds.), Turbulence and shear flow phenomena. Second International Symposium, KTH, Stockholm (2001).

7- Na, Y., Moin, P., "Direct numerical simulation of a separated turbulent boundary layer" J. Fluid Mech. 370, 175–201(1998).

8- Kalter, M., Fernholz, H., "The influence of free-stream turbulence on an axisymmetric turbulent boundary layer in and relaxing from, an adverse pressure gradient" In: 5th European Turbulence Conference, Siena, 1994.

9- Ferziger, J., Peri_c, M., "Computational Methods for Fluid Dynamics", second ed. Springer, Berlin (1997).

10- Patankar, S. V., "Numerical Heat transfer and fluid Flow", second ed. Mc Graw Hill (1981).