VOL. 11, NO. 4, FEBRUARY 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

NUMERICAL EVALUATION OF GALERKIN FINITE VOLUME SOLVER FOR LAMINAR/TURBULENT FLOW OVER FLAT PLATE

F. Namazi-Saleh, K.V. John and Z. Bt. Mustaffa
Department of Civil and Enviromental Engineering, Universiti Teknologi Petronas, Perak Darul Ridzuan, Malaysia
E-Mail: fa_namazi@yahoo.com

ABSTRACT

In this paper, attempts are made to use combination of numerical techniques to simulate fluid flow over a flat plate. The objective was not to investigate the physical phenomenon of flow in detail but to study numerical method as well as modeling aspects, which influence the quality of solutions. The incompressible Navier Stokes equations with large eddy simulation (LES) turbulence model were numerically solved to estimate velocity profile over surface of flat body exposed to current. The results are obtained by solving the incompressible form of the mass and momentum conservation equations using finite volume method. The near wall model and the subgrid scale (SGS) model plays an important role on modeling. Accordingly, proper boundary layer condition based on logarithmic velocity profile was imposed to capture turbulent velocity near to the wall. Several techniques such as local time steeping, residual smoothing and unstructured multigrid mesh were used to increase convergence acceleration. Results from large eddy simulation with Smagorinsky subgrid scale model are presented in two different types of flow such as laminar ant turbulent flow. All computed results are compared with Blasius solution or experimental data represented in literature. The results show good agreement with the aforementioned experimental and computational data. Imposing logarithmic law for velocity profile normal to the wall provide more accurate velocity profile in general especially for relatively coarse mesh.

Keywords: galerkin finite volume method, artificial dissipation, method large eddy simulation, runge-kutta method, smagorinsky.

INTRODUCTION

Direct numerical simulation of the Navier-Stokes equations has been the subject of many intensive studies. In this paper, we focus on numerical simulations of incompressible flow over a flat plate. To organize the behavior of fluid flow over flat plate, relative parameters such as velocity or pressure offered by CFD are required. Therefore, choosing a suitable computational simulation to solve fluid equation especially in complicated flow cases, has become one of the challenging areas in numerical research works (Bitsuamlak *et al.* 2004) (Tamimi, 2012).

In recent decades, comprehensive literature reviews on the use of CFD for modeling flow over flat surface have been published. Kraichnan used turbulent boundary layer (TBL) flows to model wall pressure and the noise for turbulent flows (Kraichnan, 1956). He mentioned the pressure fluctuation presented within a turbulent boundary layer would exert a force on the boundary surface resulting in transmissions of noise. This effect may make an important contribution to the noise levels encountered within high-speed aircraft. (Liao, 1999) presented new kind of analytic technique named homotopy analysis method (HAM), to give a valid explicit and uniform solution for two-dimensional laminar viscous flow over a semi-infinite flat plate. He successfully applied HAM to solution of Blasius equation. Later on, Lenormand used large eddy simulations to simulate compressible wall-bounded flows (Lenormand *et al.* 2000). The convective terms are discretized according to "skew-symmetric" method and fourth order of finite deference scheme was used in this flow simulation. Good agreement with respect to the experimental reference was obtained. For near wall region simulation, Glockner et al. used Reichardt's velocity profile to apply near wall turbulence formulation in turbulent convection analysis (Glockner and Naterer, 2005). They proved that in contrast with the standard law-of-the-wall, this method could simulate all parts of the near wall region; and Reichardt's velocity profile is proper for simulation of turbulent flows in near wall layer.

To calculate any characteristics of flow, related to any new design through the application of numerical simulation in CFD problems is becoming indeed cheaper than measuring these characteristics in field or wind tunnel. Moreover, in numerical simulation, the results of the flow characteristics at every point of model are accessible simultaneously. However, accurate simulation of flow in the computational domain is imperative to obtain correct and reliable predictions of the fluid behavior. As mentioned by Blocken et al. and Franke *et al.* important parameter, such as grid resolution, iterative convergence loops, selected turbulence models and also near wall treatment have significant impact on the results (Blocken *et al.* 2007), (Franke *et al.* 2007). There are varieties of turbulence models, which can affect the results in different modeling. Kawai et al. used Large Eddy Simulation (LES) turbulence method to modeled wall-bounded flow simulations; he showed that by moving away from the wall the numerical errors significantly reduced. In addition, they illustrated that by using wall model, the errors due to friction can be removed (Kawai and Larsson, 2012). So choosing appropriate numerical solver, proper technique and accurate boundary condition are very important to reach accurate and reliable results with lowest cost in time and computational price.

The purpose of this report is to present a numerical analysis that yield continuous velocity and shear distribution for laminar and turbulent flows over a flat

plate. To reach these, the Navier-Stokes equations were solved for an incompressible fluid flow with an unstructured finite volume mesh. In this work, the artificial compressibility method was used to solve the equation of continuity and motion simultaneously. According to(Hino *et al.* 1993) and(Namazi-Saleh *et al.* 2014), this technique makes the system of equations to hyperbolic form and efficient numerical solution method can be applied.

The large eddy simulation (LES) model was used to compute the turbulent eddy viscosity coefficient in diffusion terms of the momentum equations. The assumption of incompressibility is valid for common civil, environmental, and offshore engineering problems. Therefore, for the incompressible flow condition, the time derivative of the density vanishes from the continuity equation. Thus, to solve the two equations of motion and continuity simultaneously in a coupled manner, using pseudo compressibility technique for the steady state problems was helpful. On the other hand, if the boundary layer thickness is negligible in the flow domain, the inviscid form of the equations of motion can be used in desired dimensions. These set of equations that consists of time-independent velocity and the time-dependent equations of motion, mathematically represent the behavior of fluid flow. As mentioned by (Sabbagh-Yazd *et al.* 2008), the effects of some parameters for instance artificial compressibility parameters, multistage time stepping limit and artificial dissipation coefficient, would affect the converge behavior of numerical model.

For the first step, Navier-Stokes equations were used to simulate two-dimensional incompressible inviscid flow over flat plate. The accuracy of the modeling was presented by existing analytical value of velocity profile for flat plate in laminar flow solved by Blasius (Ahmad and Al-Barakati, 2009). In second step, turbulent flow over flat plate at high Reynolds number (Re=1.44×〖10〗^5) is simulated and the results are discussed by comparison of computed results with available experimental measurements (Gete and Evans, 2003).

GOVERNING EQUATIONS AND NUMERICAL METHOD

The Navier-Stokes equations for an incompressible fluid can be described by conservation of mass and momentum equations:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \vartheta \frac{\partial}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{\partial \overline{\acute{u}_i \acute{u}_j}}{\partial x_j} \tag{2}$$

where x_j and x_i point out the Cartesian coordinate system for horizontal and vertical directions. In mentioned equations u_j and u_i implies the velocities in vertical and horizon direction, t involve time marching, p illustrate pressure, υ indicates kinematics viscosity and $\overline{\acute{u}_i \acute{u}_j}$ denotes Reynolds stress tensor. According to Boussinesq assumption, the Reynolds stress tensor can be modeled by introducing ϑ_t as eddy viscosity, following formulation is obtained:

$$-\overline{\acute{u}_i \acute{u}_j} = \vartheta_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3} k \delta_{ij} \tag{3}$$

where k is dimensionless turbulent kinetic energy and δ_{ij} is the Kronecker delta. The variables of mentioned equations need to be converted to non-dimensional form by dividing x and y by L, a reference length u and v by upstream wind velocity, U_0 , and p by ρU_0^2.

In this research, the subgrid scale (SGS) model is utilized to modeling turbulence flow. To find value of eddy viscosity, $\vartheta_t = \vartheta_{SGS}$, the Smagorinsky model are used as follow (Li and Cheng, 2001):

$$\vartheta_{SGS} = (C_s \Delta)^2 [1/2 \overline{s_{ij}}\ \overline{s_{ij}}]^{1/2} \tag{4}$$

$$(\overline{s_{ij}}\ \overline{s_{ij}})^{1/2} = \sqrt{2\left(\frac{\partial \bar{u}}{\partial x}\right)^2 + \left(\frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x}\right) + 2\left(\frac{\partial \bar{v}}{\partial y}\right)^2} \tag{5}$$

where, x_j and x_i are used for two-dimensional computation. The Smagorinsky model is utilized for definition of ϑ_{SGS}, where Δ is the area of a triangular cell and $C_s = 0.10$ is assumed. In equation (5), $\bar{u}, \bar{v}$ are mean values of velocity in each edge of the triangular element (Sohankar, 2008), (Ferziger and Perić, 2002).

The non-dimensional form of the governing equations in Cartesian coordinates can be written as:

$$\frac{\partial W}{\partial t} + \left(\frac{\partial F^c}{\partial x} + \frac{\partial H^c}{\partial y}\right) + \left(\frac{\partial F^d}{\partial x} + \frac{\partial H^d}{\partial y}\right) = 0 \tag{6}$$

where:

$$W = \begin{pmatrix} p/\rho_0 \\ \beta^2 \\ u \\ v \end{pmatrix},$$

$$F^c = \begin{pmatrix} u \\ u^2 + p/\rho_0 \\ uv \end{pmatrix}, H^c = \begin{pmatrix} v \\ uv \\ v^2 + p/\rho_0 \end{pmatrix},$$

$$F^d = \begin{pmatrix} 0 \\ \vartheta_T \frac{\partial u}{\partial x} \\ \vartheta_T \frac{\partial v}{\partial x} \end{pmatrix}, H^d = \begin{pmatrix} 0 \\ \vartheta_T \frac{\partial u}{\partial y} \\ \vartheta_T \frac{\partial v}{\partial y} \end{pmatrix} \tag{7}$$

W represents the conserved variables, F^c , H^c are indicate the components of convective flux vectors and F^d , H^d are denote the components of viscous flux vectors in shape of non-dimensional form in Cartesian coordinates. W is consist of three dependent variables of velocity , u and v, and pressure, p. ϑ_T is the summation of eddy viscosity, ϑ_t, and kinematic viscosity, ϑ.

In equation (7), β^2 is the artificial compressibility parameter used in pseudo compressible technique. This parameter is applied to establish the correlation of equations with sound movement instead of compressible flow. Due to the lack of diffusive effects, hyperbolic conservation laws admit discontinuous and, possibly, non-unique weak solutions. By Use of this pseudo compressible technique, the elliptic system of incompressible flow equations converted to hyperbolic equations (Hino *et al.* 1993). To find value of pseudo compressibility parameter, some algorithms have been developed on basis of constant coefficients and some others developed based on complex algorithms. However, the values of pseudo compressibility parameter can affected by local velocity. The following formula proposed by (Wendt, 2008) provide this ratio:

$$\beta^2 = \max(\beta_{min}^2, C|U^2|) \tag{8}$$

As discussed by (Sabbagh-Yazdi *et al.* 2007) the optimum value of C is considered in the range of 1 to 5 and value of β_{min}^2 is suggested between 0.1 to 0.3.

Velocity gradients in vicinity of the wall, because of molecular viscosity of the fluid, are formed sharply. Therefore close to the wall, boundary layer is developed. Moreover, the effects of wall surface, e.g. wall roughness, on shear stress are considerable. In addition, at high Reynolds number, especially in turbulent flow modeling, LES cannot resolve eddies near wall region exactly unless a very fine mesh is used. Considering all the above cases, this requirement may be relaxed by use of wall-law approximation to correct instantaneous velocity at the wall nearest computational nodes. In this method, the value of velocity for grid points near to the wall need to follow the law-of-the-wall velocity profile. For flow over smooth walls, this law is formulated using dimensionless velocity and distance normal to wall surface as:

$$u^+ = \frac{u}{u_\tau} = f(y^+)\,; \quad y^+ = \frac{u_\tau y}{\vartheta} \tag{9}$$

where u represents the resolved velocity tangential to the wall at the wall-nearest point and ϑ is the fluid kinematics velocity. The dimensionless distance from the wall, y^+, depends on the friction velocity, $u_\tau = \sqrt{\tau_w/\rho}$, τ_w is near wall shear stress, and normal distance of this point from the wall, y. In conventional turbulence models:

$$u^+ = y^+\,; \quad y^+ < 5 \tag{10}$$

$$u^+ = \frac{1}{\kappa}\ln(Ey^+) = \frac{1}{\kappa}\ln(y^+) + 5.5\,; \quad y^+ > 30 \tag{11}$$

where $\kappa = 0.41$ is the von Karman constant and $E = 9.793$ (Durst *et al.* 1996), (Temmerman *et al.* 2003), (Blocken *et al.* 2007). As shown in Figure-1, for turbulent flow over flat wall, the boundary layer is usually divided in to two major zones of viscous sub-layer or laminar sub-layer, $y^+ < 5$, and turbulent boundary layer, $y^+ > 30$. The buffer layer is the region between laminar and turbulent layer. As mentioned by (Glockner and Naterer, 2005) the viscous and buffer layers are very thin

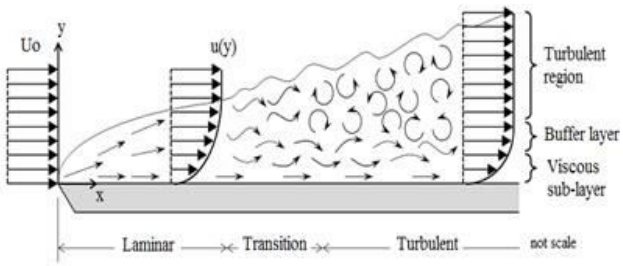

Figure-1. Boundary layer for turbulent flow over flat plate.

In this study, to drive the discrete form of governing equations, the Galekin Finite Volume Method based on unstructured triangular mesh is used. Consequently, the following 2D formulation can be obtained:

In this study, to drive the discrete form of governing equations, the Galekin Finite Volume Method based on unstructured triangular mesh is used. Consequently, the following 2D formulation can be obtained:

$$\frac{\Delta W}{\Delta t} = -\frac{P}{\Omega}\left[\sum_{k=1}^{N_{face}}[\overline{F_c}(\Delta y) - \overline{H_c}(\Delta x)]\right] - \frac{P}{A}\left[\frac{2}{3}\sum_{k=1}^{N_{cells}}[\overline{F_d}(\Delta y) - \overline{H_d}(\Delta x)]\right] \tag{12}$$

Here, $\overline{F_c}$, $\overline{H_c}$ are the mean values of convective fluxes at the control volume boundary faces and $\overline{F_d}$, $\overline{H_d}$ are the mean values of viscous fluxes which are computed at each triangle cell.

The main problem is growing up numerical errors. These errors usually disturb the explicit solution of formulations and will be overcome by using artificial dissipation terms. This method is suitable for the triangular meshes. Actually these extra terms, artificial dissipation terms, are utilized to reduce and overcome unwanted errors to achieve stability in numerical simulation to conserve the accuracy of the results. To obtain this numerical dissipation terms the following Laplacian operator are used:

$$\nabla^4 W_i = \varepsilon_4 \sum_{k=1}^{N_{edge}} \lambda_{ij}(\nabla^2 W_j - \nabla^2 W_i) \tag{13}$$

$$\nabla^2 W_i = \sum_{k=1}^{N_{edge}}(W_j - W_i) \tag{14}$$

λ_{ij}, is the scaling factors of the edges associated with the end nodes i of the edge k and ε_4 is an empirical

coefficient discussed by (Mavriplis, 1990), (Hino *et al.* 1993).

$$\lambda_i = \sum_{k=1}^{N_{edge}} \left[|U_k(\Delta S)_k| + \sqrt{(U_k(\Delta S)_k)^2 + (\Delta S)_k^2} \right] \quad (15)$$

$$\lambda_{ij} = min(\lambda_i, \lambda_j) \quad (16)$$

To estimate the velocity of nodes near the wall, accurate calculation of near wall shear stress and its modification based on law-of-the-wall velocity profile is necessary. The shear stress will be achieved based on equation (9) and (11) as follow:

$$\tau_{wall} = \rho \left[\frac{u}{\frac{1}{\kappa} \ln\left(\frac{y}{\vartheta}\sqrt{\frac{\tau_{wall-G}}{\rho}}\right) + 5.5} \right] \quad (17)$$

which u present tangential velocity at node i and y is normal distance of node i to the wall. τ_{wall-G} is shear stress according to Green's Theory , $\tau_{wall-G} = \Delta u / \Delta y$.

Having calculated wall shear stress, τ_{wall}, the value of tangential velocity, u, at wall nearest node can be updated using following relation:

$$u = \sqrt{\frac{\tau_{wall}}{\rho}} \left[\frac{1}{\kappa} \ln\left(\frac{y}{\vartheta}\sqrt{\frac{\tau_{wall}}{\rho}}\right) + 5.5 \right] \quad (18)$$

To improve the computational efficiency using various numerical technique like edge-base algorithm, Runge-Kutta multi-stage time stepping and the residual smoothing are employed in this study (Hino *et al.* 1993).

By using described formulations, despite Cell Centre Finite Volume Methods and similar to Cell Vertex Finite Volume Methods, variables are explicitly computed at the nodal points. Thus, there is no need to use reconstruction method to transfer computed value to the nodal points (Sabbagh-Yazdi *et al.* 2007). Additionally, explicit nature of formulations, despite Galerkin Finite Element Methods, pave the way for matrix free computations procedure (Iskandarani *et al.* 2005).

In order to apply boundary condition, unit free stream velocity is imposed for inflow boundaries and unit pressure is imposed at outflow boundaries. Additionally, for initial conditions the unit inflow velocity and unit outflow pressure are imposed at every computational node (Sabbagh-Yazd *et al.* 2008).

The errors in this numerical simulation are computed as following:

$$Error = |W_{comp} - W_{exp}| / W_{exp} \times 100 \quad (19)$$

NUMERICAL RESULTS AND DISCUSSIONS

Incompressible Laminar Flow

In order to assess the changes of velocity distribution over flat plate with standard geometrical feature, the flow solver is applied to solve the incompressible viscous flow on a mesh of unstructured triangles. The accuracy of the developed incompressible viscous flow solver is examined by solving case with Blasius analytical solution (Ahmad and Al-Barakati, 2009). The boundary condition for the velocity at the solid wall nodes can be considered as zero at normal velocity and tangential components are computed. At inflow boundaries, unit free stream velocity and at outflow boundaries unit pressure is imposed. The free stream flow parameters, outflow pressure and inflow velocity, are set at every computational node as initial conditions (Hino *et al.* 1993).

In case of flat plate, the verification of computed results proved the accurate performance of the algorithm without any numerical conflict for velocity components. The general and close views of the applied mesh are shown in Figure-2. The computations filed are performed on a triangular mesh containing 19,200 grid points, 37,818 triangular elements and 57,017 faces. At the solid wall nodes no slipping velocity are considered.

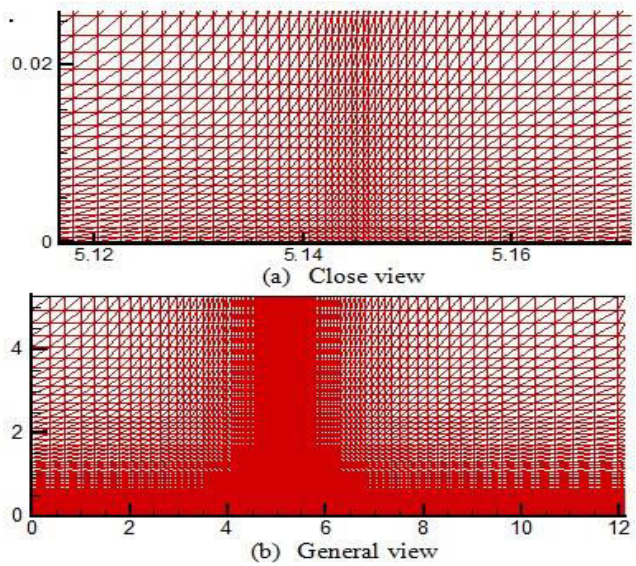

Figure 2. (a) Close view (b) General view of regular mesh for Flat Plate in laminar test.

The computed results for velocity value over flat plate at laminar flow were plotted in terms of colored map with velocity vectors, as shown in Figure-3.

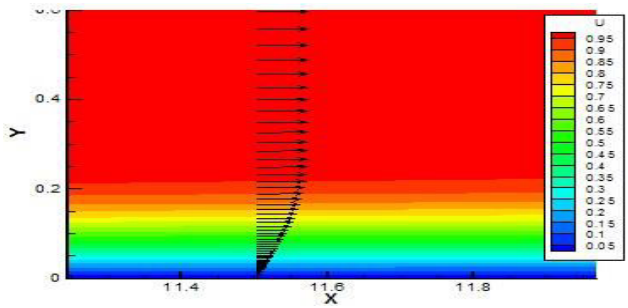

Figure-3. Colored map velocity with vectors at subcritical Reynolds number.

In order to present the independency of the results from the choice of Reynolds number in laminar flow, the result of (u/U) over flat plate, in this test case x=11.5 was selected, are compared with Blasius similarity solution for various Reynolds number. Present simulation was examined for five different Reynolds number with value of 2000, 3000, 5000, 8000 and 10,000. Figure-4 shows comparison of velocity profile at mentioned Reynolds numbers and Blasius solution. In general, good agreement between computational result and the parameters of Blasius similarity solution are obtained.

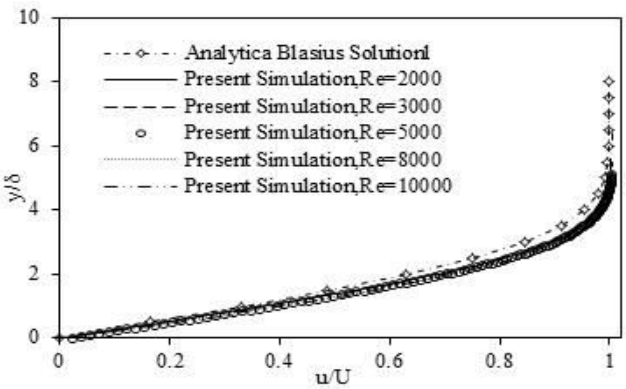

Figure-4. Comparison of computed u/U with analytical blasius solution.

Table-1. Average and maximum errors of velocity in comparison with analytical solution.

Location	Maximum error	Average error
x=11.5	5.15%	3.14%

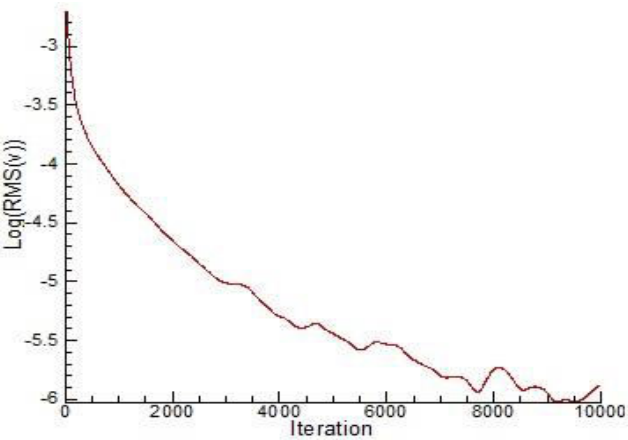

Figure-5. Convergence behavior of computer errors for velocity (Log (RMS (V)) for laminar incompressible viscous flow.

According to Figure-4, the present computed results are reasonably acceptable in comparison with Analytical Balasius results for this case study. The model's ability in the prediction of the velocity profile was quantified with the root mean square error (RMSE) described in Table-1.

The convergence behavior of velocity error is shown in term of logarithmic function. As shown in Figure-5, the convergence error for laminar incompressible viscous flow simulation over flat plate is reached with adequate accuracy (1×10^{-6}).

Incompressible Turbulent Flow

Accuracy of the developed turbulent flow solver is examined by solving the case with available experimental measurements which is done in University of British Columbia (Gete and Evans, 2003). The test section of the wind tunnel has 400 mm length and 250 mm cross section width, and the maximum free stream velocity is proposed on $20\ m/s$ and the Reynolds number, based on distance along the plate, was changed between $Re = 0.144 \times 10^5$ to $Re = 1.44 \times 10^5$. The applied mesh for this verification is considered as triangular mesh and containing 19,200 grid points, 37,818 triangular elements and 57017 faces, which is shown in Figure-2.

Figure-6 shows the obtained values of velocity over flat plate at turbulent flow ($Re = 1.44 \times 10^5$) in terms of colored map a long velocity vectors.

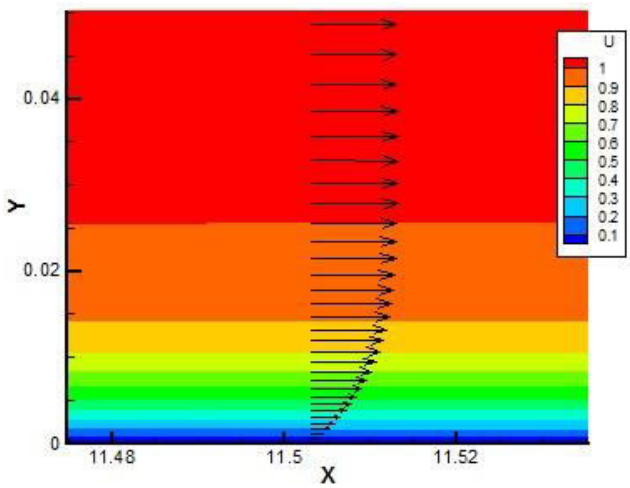

Figure-6. Colored map velocity with vectors at supercritical Reynolds number.

To verify the accuracy of model in turbulent flow, at $Re = 1.44 \times 10^5$, the obtained velocity profile over flat plate with and without imposing logarithmic law is compared with existing experimental test case done by (Gete and Evans, 2003). As shown in Figure-7, the results at the certain position of plate, x=11.5, are compared with experimental results. Although not all the velocity values are agrees with within experimental uncertainly, the results from logarithmic law condition are generally in better agreement.

As Table-2 illustrates, the errors of obtained velocity over flat plate in turbulent flow for two type of wall condition which compared with existing experimental

results by (Gete and Evans, 2003). It is seen that the model slightly over predicts the velocity in boundary layer region. However, it should be noted that in this study the flat plate surface has been assumed as smooth wall; therefor the effect of friction is neglected in this study.

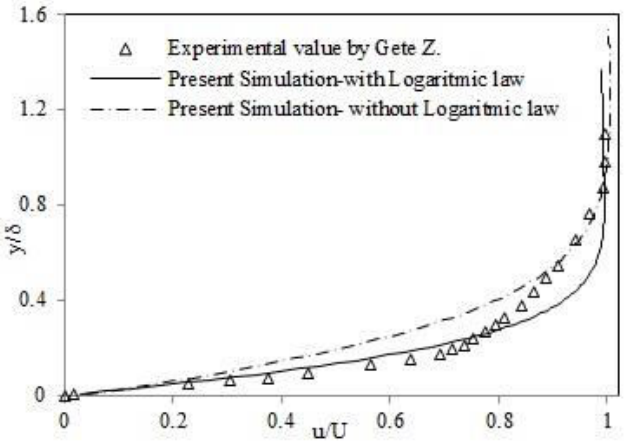

Figure-7. Comparison of computed u/U with experimental data reported by (Gete and Evans, 2003), $Re = 1.44 \times 10^5$.

Table-2. The average and maximum errors of velocity, compared with experimental data reported by (Gete and Evans, 2003).

Type of wall condition	Maximum error	Average error
with Logarithmic law	8.5%	5.6%
Without Logarithmic law	23.7%	10.2%

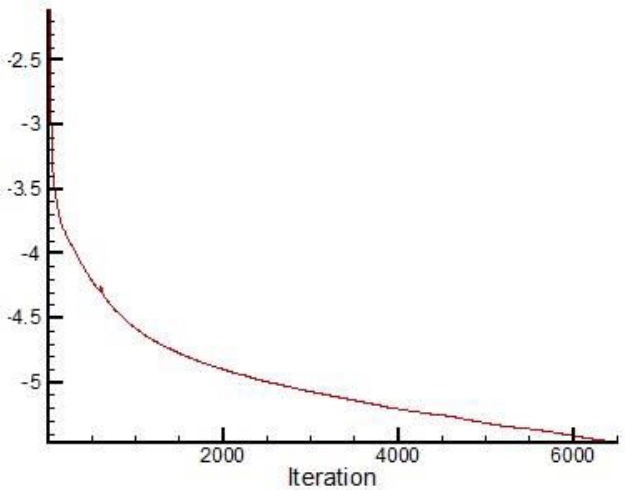

Figure 8. Convergence behavior of computer errors for velocity (Log (RMS (V)) for incompressible turbulent flow.

Figure 8 indicates the convergence behavior of velocity in term of logarithmic function. As can be seen, the convergence error for turbulent incompressible flow is reached with good accuracy.

DISCUSSION AND CONCLUSIONS

In this paper, the flow simulation over flat surface in laminar and turbulent flow were numerically investigated in terms of non-dimensional form of velocity profile. The Navier-Stokes equations with Smagorinsky turbulence model employed to simulate the flow pattern over flat plate. The equations were discretized based on artificial compressibility method for unstructured triangular mesh and explicitly solved by use of Galerkin finite volume method.

This matrix free computational model calculates value of pressure and velocity over flat surface. The Galerkin finite volume method solver was verified by comparison of obtained values with validated data. In this case, verification was done based on available analytical and experimental data reported in literature. The findings of the study are in excellent agreement with numerical and experimental data. In addition, the velocity profile over flat plate was reported which shows a good agreement with existing results. To prevent fine mesh near the wall, adding logarithmic law as boundary layer condition generally improved velocity profile especially for turbulent flow.

It concluded that present mathematical model has been well developed to simulate numerically fluid flow over flat plate like submarine flat surface exposed to current. The results show that the artificial dissipation method is well adapted to limit unwanted errors in computational simulation of flow over flat surface in both laminar and turbulent condition.

ACKNOWLEDGEMENTS

The authors are grateful for the funding and facilities support by Universiti Teknologi Petronas.

REFERENCES

[1] Ahmad, F. and W. H. Al-Barakati (2009). "An approximate analytic solution of the Blasius problem." Communications in Nonlinear Science and Numerical Simulation 14(4): 1021-1024.

[2] Bitsuamlak, G., T. Stathopoulos and C. Bedard (2004). "Numerical evaluation of wind flow over complex terrain: Review." Journal of Aerospace Engineering 17(4): 135-145.

[3] Blocken, B., T. Stathopoulos and J. Carmeliet (2007). "CFD simulation of the atmospheric boundary layer: wall function problems." Atmospheric environment 41(2): 238-252.

[4] Durst, F., H. Kikura, I. Lekakis, J. Jovanović and Q. Ye (1996). "Wall shear stress determination from near-wall mean velocity data in turbulent pipe and channel flows." Experiments in Fluids 20(6): 417-428.

[5] Ferziger, J. H. and M. Perić (2002). Computational methods for fluid dynamics, Springer Berlin.

[6] Franke, J., A. Hellsten, H. Schlünzen and B. Carissimo (2007). Best practice guideline for the CFD simulation of flows in the urban environment, COST Action 732: Quality Assurance and Improvement of Microscale Meteorological Models.

[7] Gete, Z. and R. Evans (2003). "An experimental investigation of unsteady turbulent-wake/boundary-layer interaction." Journal of fluids and structures 17(1): 43-55.

[8] Glockner, P. and G. Naterer (2005). "Near-wall velocity profile with adaptive shape functions for turbulent forced convection." International communications in heat and mass transfer 32(1): 72-79.

[9] Hino, T., L. Martinelli and A. Jameson (1993). A finite-volume method with unstructured grid for free surface flow simulations. Sixth International conference on numerical ship hydrodynamics.

[10] Iskandarani, M., J. Levin, B.-J. Choi and D. Haidvogel (2005). "Comparison of advection schemes for high-order h–p finite element and finite volume methods." Ocean Modelling 10(1): 233-252.

[11] Kawai, S. and J. Larsson (2012). "Wall-modeling in large eddy simulation: Length scales, grid resolution, and accuracy." Physics of Fluids (1994-present) 24(1): 015105.

[12] Kraichnan, R. H. (1956). "Pressure fluctuations in turbulent flow over a flat plate." The Journal of the Acoustical Society of America 28(3): 378-390.

[13] Lenormand, E., P. Sagaut, L. T. Phuoc and P. Comte (2000). "Subgrid-scale models for large-eddy simulations of compressible wall bounded flows." AIAA journal 38(8): 1340-1350.

[14] Li, F. and L. Cheng (2001). "Prediction of lee-wake scouring of pipelines in currents." Journal of waterway, port, coastal, and ocean engineering 127(2): 106-112.

[15] Liao, S.-J. (1999). "A uniformly valid analytic solution of two-dimensional viscous flow over a semi-infinite flat plate." Journal of Fluid Mechanics 385: 101-128.

[16] Mavriplis, D. J. (1990). "Accurate multigrid solution of the Euler equations on unstructured and adaptive meshes." AIAA journal 28(2): 213-221.

[17] Namazi-Saleh, F., V. Kurian and Z. Mustaffa (2014). "Investigation of Vortex Induced Vibration of Offshore Pipelines near Seabed." Applied Mechanics and Materials 567 256-270.

[18] Sabbagh-Yazd, R., N. Mastorakis, F. Meysami and F. Namazi-Saleh (2008). "2D Galerkin Finite Volume Solution of Steady Inviscid/Viscous/Turbulent Artificial Compressible Flow on Triangular Meshes." International Journal of Computers 2(1): 39-46.

[19] Sabbagh-Yazdi, S., N. Mastorakis and F. Meysami (2007). "Wind flow pressure load simulation around storage tanks using SGS turbulent model." International Journal of Mechanics 1: 39-44.

[20] Sohankar, A. (2008). "Large eddy simulation of flow past rectangular-section cylinders: Side ratio effects." Journal of wind engineering and industrial aerodynamics 96(5): 640-655.

[21] Tamimi, S. (2012). "On Turbulent Flow near a Wall." Journal of Communication and Computer 9: 1104-1109.

[22] Temmerman, L., M. A. Leschziner, C. P. Mellen and J. Fröhlich (2003). "Investigation of wall-function approximations and subgrid-scale models in large eddy simulation of separated flow in a channel with streamwise periodic constrictions." International Journal of Heat and Fluid Flow 24(2): 157-180.

[23] Wendt, J. (2008). Computational fluid dynamics: an introduction, Springer Science & Business Media.