Comparison of Turbulence Models in the Flow over a Backward-Facing Step

Priscila Pires Araujo[1], André Luiz Tenório Rezende[2]

Department of Mechanical and Materials Engineering, Military Engineering Institute, Rio de Janeiro, Brazil

Abstract—*This work presents the numerical simulation and analysis of the turbulent flow over a two-dimensional channel with a backward-facing step. The computational simulation performed in this study is based on the Reynolds equations using a technique denominated Reynolds Average Navier-Stokes (RANS). The main objective of the present work is the comparison of different models of turbulence applied to the turbulent flow over a backward-facing step. The performance of each RANS model used will be discussed and compared with the results obtained through a direct numerical simulation present in the literature. The RANS turbulence models used are k-ω, k-ε, Shear Stress Transport k-ω (SST k-ω) and the second-order closure model called Reynolds Stress Model (RSM). The Reynolds number used in all the numerical simulations constructed in this study is equal to 9000, based on the height of the step h and the inlet velocity U_b. The results are the reattachment length, the mean velocity profiles and the turbulence intensities profiles. The k-ε model obtained poor results in most of the analyzed variables in this study. Among the RANS turbulence models, the SST k-ω model presented the best results of reattachment length, mean velocity profile and contour when compared to results obtained in the literature. The RSM model found the best results of turbulence intensity profile, when compared to the models of two partial differential equations that use the Boussines hypothesis.*

Keywords—*backward-facing step, DNS, RANS, turbulence models.*

I. INTRODUCTION

The flow separation caused by an adverse pressure gradient is a common phenomenon in many practical applications in engineering. The adverse pressure gradient is the increase of the static pressure in the direction of the flow, significantly affecting the flow. In several engineering cases, the adverse pressure gradient is caused by a sudden change in geometry, leading to separation of the flow and subsequent reattachment. Such phenomenon can also be observed in devices such as electronic cooling equipment, combustion chambers, diffusers and valves. In this context, the backward-facing step is one of the most studied cases, in order to understand the effects on the flow caused by a sudden change in geometry using a simple geometry. Therefore, it has been much studied in cases of computational simulation, requiring less computational cost than other cases and presenting satisfactory results in the study of phenomena caused by the flow separation.

The present work deals with the computational simulation and analysis of the turbulent flow over a channel with a backward-facing step by means of the construction of a relatively simple geometry with the great advantage of presenting important characteristics for the scope of the study of turbulent flows with boundary layer separation. The simulated cases in the present work are based on the study carried out by [1] using the same geometry as this one, with the objective of validating the obtained results and comparing different turbulence models, analyzing results such as the reattachment length, profiles of mean velocity, pressure coefficient and Reynolds stress components.

II. MATHEMATICAL MODELING

In this work, the Reynolds Averages Navier-Stokes equations method (RANS) is applied to the government equations shown previously. The method of the average Reynolds equations is based on the decomposition of the velocity instantaneous value in $u_i = \overline{u_i} + u_i'$, where u_i represents the velocity instantaneous value, $\overline{u_i}$ is the mean velocity vector and u_i' represents the velocity fluctuation vector. Therefore, the equations of conservation of mass and conservation of the amount of linear motion obtained by applying the RANS methodology are given by

$$\frac{\partial \overline{u_j}}{\partial x_j} = 0; \ \frac{\partial \overline{u_i u_j}}{\partial x_j} = g_i - \frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\upsilon \frac{\partial \overline{u_i}}{\partial x_j} - \overline{u_i' u_j'}\right) \tag{1}$$

It is observed the appearance of a new term, $\overline{u_i' u_j'}$, denominated Reynolds tensor. The Reynolds tensor can be modeled by an analogy with Stokes law, based on the Boussinesq hypothesis, where the turbulent stresses are proportional to the mean flow velocity gradient, as show in Eq. 2. The constant of proportionality is called turbulent viscosity, υ_t.

$$-\overline{u_i'u_j'} = \upsilon_t \left(\frac{\partial \overline{u_j}}{\partial x_j} + \frac{\partial \overline{u_i}}{\partial x_i} \right) - \frac{2}{3}k\delta_{ij} \tag{2}$$

Three turbulence models of two differential equations using the Boussinesq hypothesis approach are used in the present work: k-ε, k-ω e SST k-ω.

The standard k-ε model [3, 4] is the turbulence model of two partial differential equations most commonly used today. In this model an equation for turbulent kinetic energy k and one equation for the dissipation of turbulent kinetic energy per unit mass ε are solved. In the standard k-ε model developed by [3], the equation for turbulent viscosity is given by Eq. 3.

$$\upsilon_t = \frac{C_\mu k^2}{\varepsilon} \tag{3}$$

The vast use of this model has shown that it has good results for the simulation of simple flow cases, but its most known deficiency is its imprecision close of adverse pressure gradients [5].

In the standard k-ω model an equation for the turbulent kinetic energy k and an equation for the specific rate of dissipation of the turbulent kinetic energy ω are solved. Its rate is determined by the rate of energy transfer over the spectrum of lengths, so ω is defined by the large scales of motion and is closely related to the mean flow properties [6].

The most commonly used k-ω model is called the standard k-ω model, where the turbulent viscosity is given by Eq. 4.

$$\upsilon_t = a^* \frac{k}{\omega} \tag{4}$$

The standard k-ω model shows good performance for free shear flows and flow on flat plates with boundary layer, as well as for more complex flows with adverse pressure gradients and separate flows. The main negative point of this model is that it presents a strong dependence of the boundary condition on the free current for ω [7, 8].

The SST k-ω model (Shear-Stress Transport k-ω) [9] is widely used in cases with high adverse pressure gradients and boundary layer separation, by means of a combination of k-ε and K-ω turbulence models. The SST k-ω model is the robust and precise combination of the k-ω model in the region near the walls with the independence of the free current of the k-ε model outside the boundary layer. For this, the k-ε model is written in terms of the specific dissipation rate, ω. Then, the standard k-ω model and the modified k-ε model are multiplied by a mixing function and summed [2]. The mixing function F1 is defined as a unite value (considering the standard k-ω model) in the inner region of the turbulent boundary layer and is zero (considering the standard k-ε model) at the outer edge of the layer, given by

$$F_1 = \tanh(\text{arg}_1{}^4); \ \text{arg}_1 = \min\left[\max\left(\frac{\sqrt{k}}{\beta^*\omega d}; \frac{500\upsilon}{d^2\omega}\right); \frac{4\sigma_\omega 2k}{CD_{k\omega}d^2}\right]; \ CD_{k\omega} = \max\left(2\rho\sigma_D \frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}; 10^{-10}\right) \tag{5}$$

In this model the turbulent viscosity is formulated through Eq. 6.

$$\upsilon_t = \frac{a_1\omega}{\max(a_1\omega; SF_2)}; \ F_2 = \tanh(\Phi^2); \ \Phi = \max\left(\frac{2\sqrt{k}}{\beta^*\omega d}; \frac{500\upsilon}{d^2\omega}\right) \tag{6}$$

The turbulent kinetic energy k and the specific dissipation rate ω of this model can be obtained by the solution of its conservation equations, where the closing set ϕ for the SST k-ω are calculated by the use of a mixing function between the constants ϕ1 , of the standard k-ω model and ϕ2 of the model k-ε, making $\phi = F_1\phi_1 + (1 - F_1)\phi_2$.

The models discussed above are RANS models based on the Boussinesq Hypothesis. These models present a good solution to the problem of turbulence closure, but they present some faults, generally related to the limitations imposed by the concept of turbulent viscosity. The RSM model (Reynolds Stress Model) is an alternative to the models shown, based on the determination of direct equations for Reynolds transport. The RSM model is usually referred to as the direct closure model or the second order model. The transport equations for the Reynolds stress can be determined by the Navier-Stokes equations and are given by the Eq. 7.

$$\frac{\partial \overline{u'_i u'_j}}{\partial t} + \frac{\partial \overline{u'_i u'_j u_l}}{\partial x_l} = D_{ij} + P_{ij} + \Phi_{ij} - \varepsilon_{ij} \tag{7}$$

The left side of Eq. 7 concerns the convective transport of the Reynolds tensor over the average flow. The first term on the right side is called diffusive transport term, the term P_{ij} is the term of stress production, Φ_{ij} is the pressure term and the term ε_{ij} refers to the dissipation rate of the Reynolds tensor caused by the viscosity.

III.   METHODOLOGY

All cases constructed in the present work have the same geometry, with an expansion ratio $E_R = 2$. The expansion ratio refers to the relation between the height of the outlet channel (L_y) and the height of the step (h), given by $E_R = L_y/(L_y - h)$. The inlet channel has a width of 4h and height equal to h and the outlet channel has width 29h and height 2h. In the simulations performed in the current study, the value of h used is 1 m. Fig. 1 shows the constructed geometry.
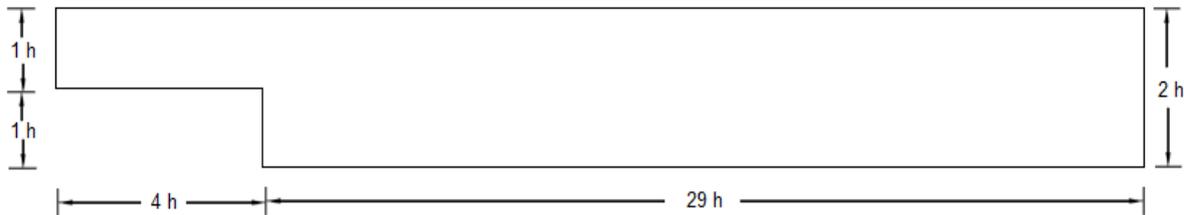

FIGURE 1. GEOMETRY BUILT.

The Reynolds number is defined by the step height, the kinematic viscosity of the fluid and the mean inlet velocity, given by $Re = U_b h/v$. The Reynolds number used in the cases constructed in the current work is Re = 9000. The mean inlet velocity set is equal to 1 m/s. The top and bottom wall regions present wall boundary conditions with the non-slip condition on the wall. The outlet region has a gauge pressure equal to zero. The mesh constructed and used in all simulated cases has 155000 elements and 156751 nodes. The region near the step received a treatment of most refined mesh, being the region of greater interest of the study realized.

The numerical method called Finite Volume Method [10] was used to discretize the equations of government. The interpolation scheme chosen for the simulations was QUICK [11] and the SIMPLE scheme [12] was used in the velocity-pressure coupling. The Multigrid technique [13] was chosen to solve the system of linear equations. The problem was considered converged when all residues were less than 10^{-6}.ANSYS FLUENT® software was used for geometry construction, mesh construction, support for numerical methods and government equations, and post-processing of built cases.

IV.   RESULTS AND DISCUSSION

The reattachment length (X_R) refers to the position where, after separation and recirculation of the flow, the reattachment happens. It is an important quantity being analyzed in the flow on a backward-facing step.Tab. 1 presents the values of reattachment length obtained in each of the cases analyzed.

TABLE 1
REATTACHMENT LENGTHS.

Case	X_R
DNS [1]	8.62
k-ω	9.83
k-ε	6.34
SST k-ω	8.50
RSM	5.86

When analyzing the Tab. 1, it is noted that the k-ω model overestimates the value of the reattachment length and the k-ε model has a significantly short reattachment length when compared to the result found in the DNS case. The SST k-ω model, as discussed above, is a RANS turbulence model recommended for cases with adverse pressure gradient and flow separation, addressing the advantages of k-ω and k-ε models. This model presents a better treatment in the regions close to the wall than the other RANS models treated in this case, thus, it was expected that this model would obtain the best result of reattachment length. The RSM model is the model that presents the lowest value of reattachment length. In spite of being a second-order closure model and solving the government equations directly, this model does not present a good treatment for the regions near the wall, which justifies the poor result for the value of reattachment length, measured in wall of the channel.

The velocity profiles were analyzed in four different flow positions, obtained in the simulations with the RANS models and compared with the results obtained by [1]. Fig. 2 presents the velocity profiles in the following positions: $x/h = 0.5$, $x/h = 4$, $x/h = 8$ and $x/h = 20$.
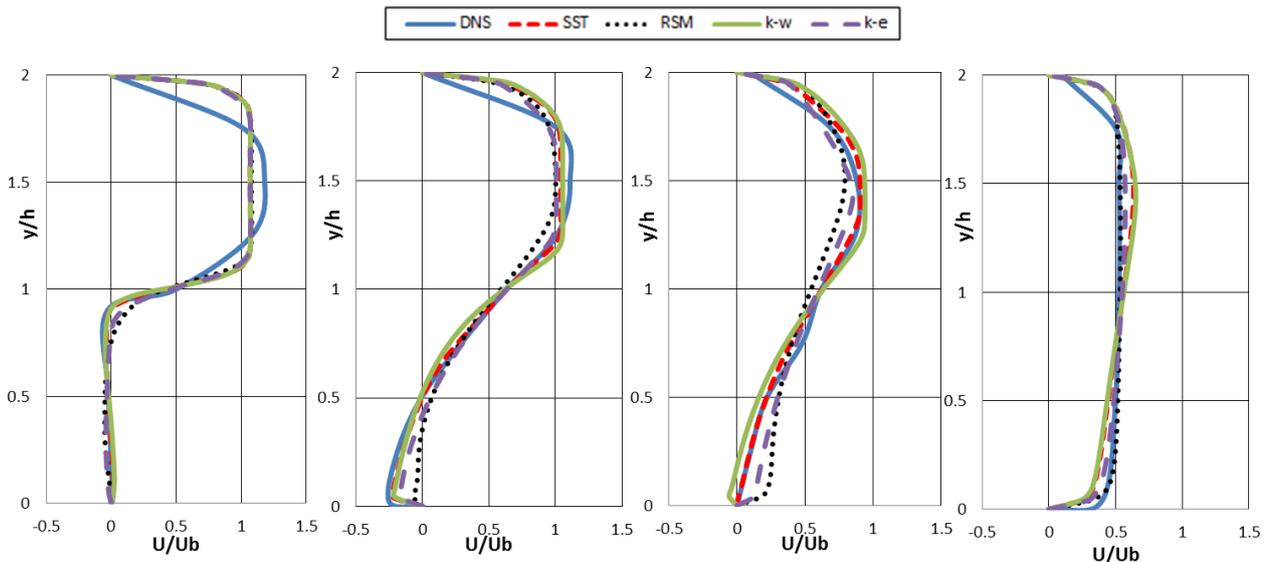

FIGURE 2. VELOCITY PROFILES AT FOUR POSITIONS.

The first position is located just after the step and it can be seen that the fully developed flow extends freely in all cases shown in Fig. 2. In $x/h = 4$, where the recirculation occurs, it is noted that there are negative values of velocity in the region near the bottom wall in all cases simulated in this work, as well as in the result found for the DNS case. The negative velocity values are due to the flow separation and represent the presence of inverse flow in the recirculation zone. In this position it can be observed that the velocity values near the bottom wall found with the RSM model are higher than the values obtained by the other models analyzed. Consequently, the presence of reverse flow is lower in this position for the case simulated with the RSM model. This result is explained by the fact that the reattachment length of the RSM model is considerably smaller than the reattachment length of the others turbulence models analyzed, so in the position $x/h = 4$ the reattachment is closer in the RSM model case. When comparing the velocity profile of k-ε model with the profiles found by k-ω, SST k-ω and DNS models, it is also possible to observe that the velocity values of this model are larger than the others since the reattachment occurs before in the k-ε model. In the position $x/h = 8$, it is observed that the model k-ω is the only one that presents negative values of velocity. The presence of the inverse flow in this position was already expected for this model, since the reattachment length of the k-ω model is the largest among the analyzed models, with a value equal to $x/h = 9.83$, while in the other models, has already occurred or is about to happen. It is also noted that the models RSM and k-ε present the highest velocity values in the region near the bottom wall, a result justified by the fact that in both models the reattachment has already happened, while it is close to occurring in the SST k-ω and DNS models. In $x/h = 20$ it can be observed that, although not fully developed, the flow tends to equilibrium and the velocity profiles of all models behave in a similar way.

Are analyzed the velocity profiles u^+ as a function of y^+. The term u^+ is the velocity u normalized by the friction velocity, $u\tau$. The friction velocity is obtained as a function of the wall shear stress, τ_w, and the specific mass of the fluid. These terms and the term y+ are given below.

$$u^+ = \frac{u}{u_\tau}; \ y^+ = \frac{u_\tau}{\nu}y; \ u_\tau = \sqrt{\frac{\tau_w}{\rho}} \tag{8}$$

For all positions that the u+ profiles were obtained numerically the theoretical curve of the wall law for the laminar boundary layer and the logarithmic region was also constructed in order to compare the results found by each turbulence model with the expected results by theory. The curve for the region of the laminar boundary layer is created according to the relation $u^+ = y^+$, while the logarithmic region profile, called log, is constructed by means of $u^+ = 2.5 \ln y^+ + 5$. Fig. 3 show the velocity profiles u^+ obtained by the numerical simulations performed with different turbulence models at four different flow positions: $x/h = 14$, x h = 18, $x/h = 26$ and $x/h = 28$.
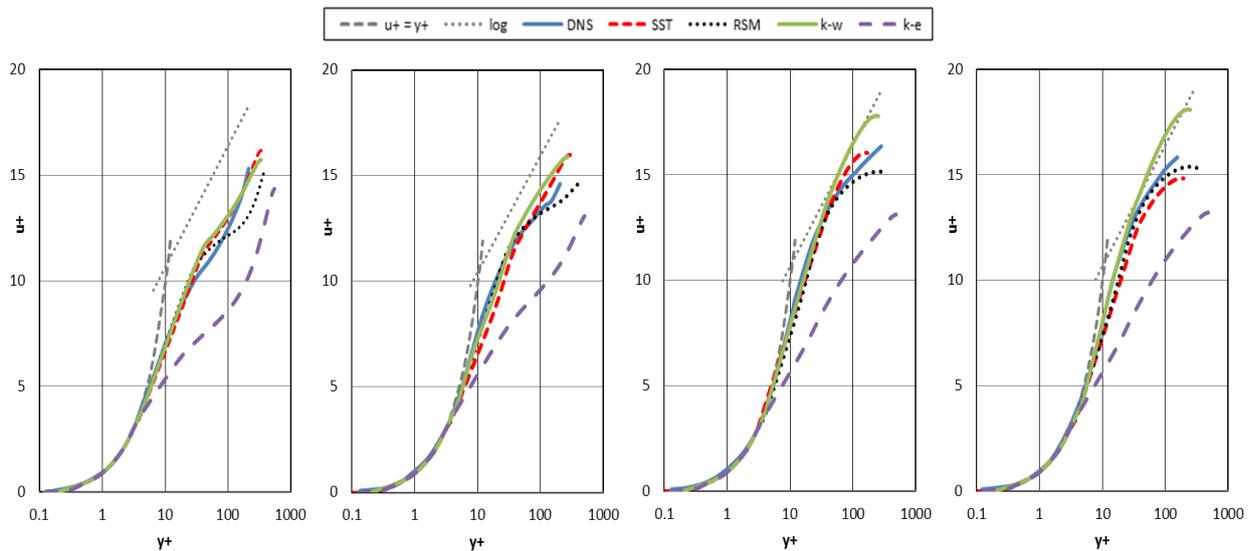
FIGURE 3. VELOCITY PROFILE u^+ AT DIFFERENT POSITIONS.

By analyzing the u^+ velocity profiles found, it can be observed that insofar as the flow occurs the results obtained by the turbulence models discussed in the present work become closer to the results obtained by means of the theoretical curves of the law of the wall. In addition, further downstream the velocity profile tends to the fully developed velocity profile, which is not fully achieved even at long distances for the case of the descending rung. It should be noted that all the RANS turbulence models used had u^+ velocity curves with behaviors very similar to the curves obtained by direct numerical simulation, with the exception of the k-ε model. The results found by the k-ε model diverge significantly from the results obtained by other turbulence models analyzed and by the theoretical curves of the wall law in the turbulent region. This result corroborates the fact that the turbulence model of two differences equations k-ε does not present good performance in cases with adverse pressure gradient and flow separation, especially in those regions near the wall are regions of great interest.

Fig. 4 presents the profiles of $\sqrt{u'u'}/U_b$ obtained in the simulations carried out in the present work, compared with the results found by [1].
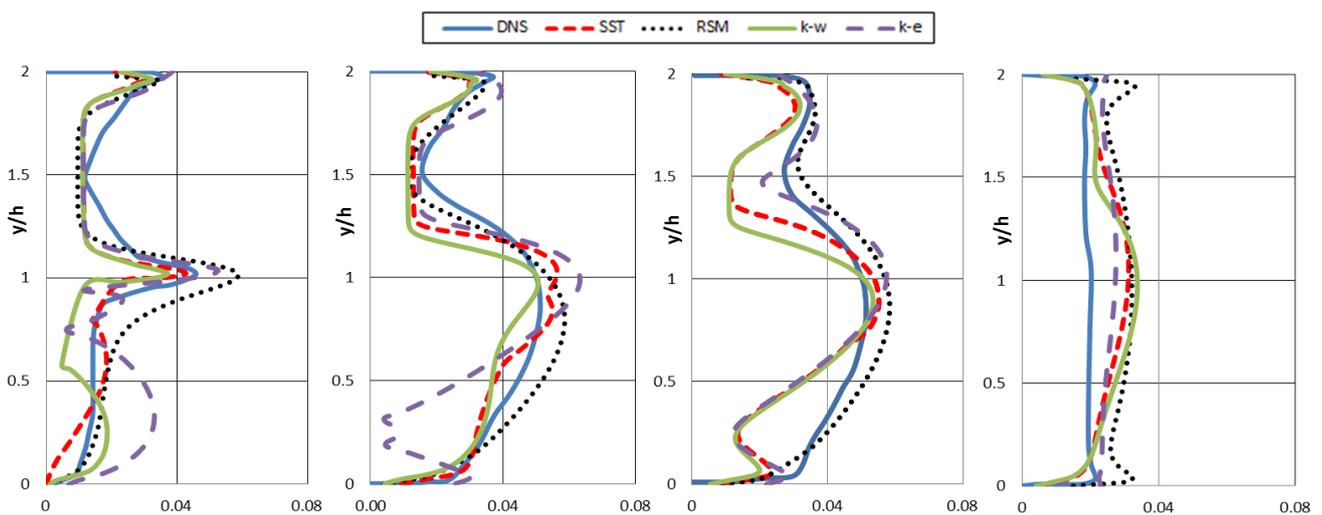

FIGURE 4. $\sqrt{u'u'}/U_b$ PROFILES AT FOUR POSITIONS.

In the turbulent intensity profile of the first position, $x/h = 0.5$, the presence of a sharp peak at the height of the step, $y/h = 1$, is observed in all models analyzed. This location refers to the region of mix layer. The $\sqrt{u'u'}/U_b$ profile obtained by means of the k-ε model presents peaks in the region below the step not predicted in the result obtained in the DNS model and not seen in the other RANS models. All models predict a peak near the top wall, according to the profile obtained by [1]. In the recirculation region, at $x / h = 4$, the k-ω, k-ε and SST k-ω models still show sharp peaks in the region near the step height,

whereas the RSM and DNS models show peaks in y/h = 1.5. As in the first position, the presence of peaks located in the top wall is observed in all models. The k-ε model again presents peaks in the regions near the bottom wall that are not predicted by any of the other models discussed in the present work. This fact is justified by the deficit of performance of this model in regions near the wall. At x/h = 8, the region close to the reattachment in most of the analyzed models, the peaks located near the top wall are regenerated in all the profiles found and peaks close to the bottom wall appear in all models of turbulence analyzed, except for the model RSM. These peaks located near the bottom wall are caused by the reattachment of the flow and cannot be seen in the RSM model because the reclosing in this model already happened in x/h = 5.86. As the flow occurs, the peaks are softened, as the flow tends to equilibrate. In the last location, at x/h = 20, we note the presence of peaks close to the bottom and top wall in all models and the behavior of the second-order quantity $\sqrt{u'u'}/U_b$ occurs more uniform. In all positions analyzed, it was possible to observe that the behavior of the $\sqrt{u'u'}/U_b$ profiles obtained by the RSM model shows greater agreement with the results found by [1] through direct numerical simulation.

V.   CONCLUSION

The present work carried out computational simulations of the turbulent flow over a channel with the presence of a backward-facing step with different models of turbulence that approach the methodology of Reynolds averaged equations (RANS). The model SST k-ω obtained a result for the reattachment length equal to X_R = 8.50 and it was the model RANS that found the result closest to the value obtained by [1] through the direct numerical simulation, with reattachment length of X_R = 8.62. The second-order RSM closure model presented the lowest value of reattachment length (X_R = 5.86) and results different from that found by DNS in velocity profiles, especially in the near regions to the wall at the positions closest to the region of the collection, x/h = 4 and x/h = 8. The model k-ε obtained a value of reattachment length significantly low equal to 6.34, fact already expected since this model does not perform well in cases with pressure gradient and boundary layer separation. The RSM model obtained the best results related to the second order quantity discussed in the present study, a conclusion made when observing its profiles. This result was already expected, since it is a second-order closure model that calculates this quantity directly, unlike the other RANS models used, which use the Boussinesq Hypothesis and the turbulent viscosity modeling to calculate the quantities associated with the components of the Reynolds tensor. The k-ε model presented remarkably weak results for the analyzed second order quantity.

REFERENCES

[1] Kopera, M. A. "Direct numerical simulation of turbulent flow over a backward-facing step." University of Warwick, 2011, 178p.

[2] Rezende, A. L. T. "Análise Numérica da Bolha de Separação do Escoamento Turbulento sobre Placa Plana Fina Inclinada." Tese (Doutorado em Engenharia Mecânica) - Pontifícia Universidade Católica, Rio de Janeiro, 2009.

[3] Jones, W. P., Launder, B. E. "Prediction of laminarization with a two-equation model of turbulence." International Journal of Heat and Mass Transfer, v. 5, n.2, p. 31-34, 1972.

[4] Launder, B. E.; Sharma, B. I. "Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disk." Letters in Heat and Mass Transfer, v. 1, n. 2, p. 131-138, 1974.

[5] Wilcox, D. C. "Reassessment of the Scale-Determining Equation for Advanced Scale Models." AIAA Journal. v. 26, n. 11, p. 1299-1310, 1988.

[6] Miranda, W. R. "Simulação Numérica de uma Bolha de Separação em Bordo Arredondado Utilizando Equações Médias de Reynolds." Dissertação (Mestrado em Engenharia Mecânica) - Instituto Militar de Engenharia, Rio de Janeiro, 2014.

[7] Wilcox, D. C. "A half-century historical review of the k-w model." AIAA paper 91-0615, 1991.

[8] Menter, F. R. "Influence of free stream values on k-ω turbulence model predictions." AIAA Journal, v. 30, n. 6, p. 1657-1659, 1992.

[9] Menter, F. R. "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications." AIAA Jounal, v. 32, n. 8, p. 1598-1605, 1994.

[10] McDonald, P. W. "The computation of transonic flow through two-dimensional gas turbine cascades." ASME, p. 71-89, 1971.

[11] Leonard, B. P."A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation." Computer Methods in Applied Mechanics and Engineering, p. 59-98, 1979.

[12] Patankar, S. "Numerical Heat Transfer and Fluid Flow." John Benjamins Publishing, 197p, 1980.

[13] Hutchinson B. R., Raithby G. D., 1986. "A Multigrid Method Based on the Additive Correction Strategy." Numerical Heat Transfer, p. 511-537, 1986.