AXIAL TURBOMACHINERY FLOW SIMULATIONS WITH DIFFERENT TURBULENCE MODELS

Diego Thomas da Silva, diego.t.silva@gmail.com
Jesuino Takachi Tomita, jtakachi@ita.br
Instituto Tecnológico de Aeronáutica - ITA

Abstract. *Two* differents *turbulence models are compared for turbomachinery flow simulations. The one-equation Spalart-Allmaras and the standard two-equation k-ε turbulence models were set to study the influence of both modeling on the flow within an axial turbine for fully turbulent flows. The first model is a single transport equation model to solve the modified turbulent eddy viscosity and the second is a baseline two transport equation model to solve the turbulent kinetic energy and turbulent dissipation rate. Hence, each turbulence model calculates the turbulent viscosity differently. The results are compared and discussed based on the CFD solution of fluid properties distribution along the blade spanwise at each blade row inlet and outlet streamwise positions.*

Keywords*: CFD, Turbulence models, turbomachinery, axial turbine, gas turbines.*

1. INTRODUCTION

Along the years, Computational Fluid Dynamics (CFD) has been an important tool for turbomachinery design and its internal flow prediction. For a good agreement between numerical simulation and experimental data, it is essential to take account geometry details and the choice of turbulence models for each case (Pecnik *et al.*, 2005). According to Wilcox (1993), turbulence modeling is one of the three key elements of CFD. It involves a series of mathematical complexities and physical nontrivial considerations and remain not fully understood by the scientist and engineers.

For turbomachines, the performance, skin friction and secondary flow effects have strong dependence on how the turbulence is modeled. Turbomachines which operate under adverse pressure gradient and heat transfer is present, turbulence modeling is even more difficult, requiring complex models and specific numerical schemes. The turbulence model must be well selected to avoid incorrect results and unnecessary computational cost, becoming the selection of an appropriate model important to obtain accurate results. To select a suitable model, must be verified the occurrence of curtains phenomena: boundary separation; stagnation zones, laminar to turbulent transition; unsteadiness; shocks and strong swirl components.

With the improvement of computational capability along of 20^{th} Century, the two-equations models became common in industrial application. It solves two equation of transport, accounting the kinetic energy and dissipation rate of turbulent effects. One-equation model are formulated based on just turbulence energy and take some flow dimension to relate the turbulence length. Because of these assumption, one-equation model are likely to be faster than two-equation. In 1992, Spalart and Allmaras (1992) developed an one-equation model for aerospace application and also has been showing good results in turbomachinery associated a low computational cost with good accuracy compared with different two-equation turbulence models.

2. OBJECTIVE

There is not a best turbulence model, but it is an adequate model for each fluid mechanics problem, if internal or external flow, if subsonic or supersonic, with or without separation, with or without free-shear flows and so on. The literature contains many works comparing the results from different turbulence models in engineering application. For turbomachinery application the use of turbulence modeling are important because the nature of the flow within the machine is completely turbulent.

During many years algebraic turbulence model with zero equation was used, but for flows with separation these models not presents an accurate result mainly for flows with boundary-layer separation. By the way, the number of equations not means that the modeling is better. Hence, is strongly important to understand the context of the formulation of each turbulence model to set what is the most appropriate for each engineering problem.

The objective of this work is to analyze the performance of two turbulence models: two-equation *k-ε* standard and one-equation Spalart and Allmaras, for calculation of the flow within a high-pressure single stage axial-flow turbine. The results were compared with experimental data available. For the 3D grid generation and flow calculation a commercial software called AxCent developed by Concepts ETI, Inc was used. All simulations were runs for steady-state regime.

3. LITERATURE REVIEW

A common one-equation model is based on the turbulence energy. Hence, turbulence length scale is indeterminate, and some assumptions are needed (Wilcox, 1993). However, the Spalart-Allmaras model is an one-equation model written in terms of eddy viscosity, with eight closure coefficients and three closure functions. The model solve a modeled transport equation for modified kinematic eddy viscosity, Eq.(1), releasing the need to estimate the length scale (Fluent, 2008).

$$\frac{\partial \tilde{\nu}}{\partial t} + U_j \frac{\partial \tilde{\nu}}{\partial x_i} = c_{b1} \tilde{S} \tilde{\nu} - c_{\omega 1} f_\omega \left(\frac{\tilde{\nu}}{d} \right)^2 + \frac{1}{\sigma_{\tilde{\nu}}} \frac{\partial}{\partial x_k} \left[\left(\nu + \tilde{\nu} \right) \frac{\partial \tilde{\nu}}{\partial x_j} \right] + \frac{c_{b2}}{\sigma} \frac{\partial \tilde{\nu}}{\partial x_k} \frac{\partial \tilde{\nu}}{\partial x_k} \tag{1}$$

In Eq.(1), the c_{b1}, c_{b2}, $c_{\omega 1}$, and σ are closure coefficients, S and f_ω are auxiliary relations . The mean velocity in tensor is represented by U_j; x_i and x_j are position vectors in vector notation. Time, distance from closest wall and kinematic molecular viscosity is give by t, d and ν respectively. The notation "~" is used to relate the Favre-averaged.

The model was created for aerospace application, especially for wall-bounded flows, and has been demonstrating a relative success in prediction flow in turbomachines and aerodynamics problems. The literature pointed that the model has a reasonable accuracy for unsteady and mildly separated flows, realistic wake spreading rates, can be integrated in viscous sublayer and numerically robust, becoming an alternative to two-equation models.

Good agreements of Spalart-Allmaras model in compute skin friction even in flows under adverse pressure gradient are showed by Wilcox (1993). He also observed for a certain case that the predicted separation by the closure is little higher than measured and the reattachment was quite well determined. Wilcox (1993) concluded that the Spalart-Allmaras is attractive model as engineering tool, better than one-equation Baldwin-Lomax, however the results obtained by jet spread test concerns.

Santin (2006) used Spalart-Allmaras model in his work to simulate a DLR cascade with relative successful. The contours of Mach number and pressure agree with experimental measurements except in tip and hub region as the exit flow angles. The behavior of the flow at tip region is very complex and will not be discussed on the present work. The results at the hub may be explained for the use of wall function by the program. Santin (2006) also calculated the performance of an axial turbine, matching with results obtained by the Ainley-Mathieson and Kacker&Okapuu semi-empirical methods used for design purpose.

Tomita (2009) implemented the Spalart-Allmaras turbulence model in an in house turbomachinery three-dimensinal CFD code and obtained results with good agreement compared with the design values from turbomachinery preliminary design program (Tomita, 2003) and streamline curvature program (Barbosa, 1987) using empirical loss correlations from test data.

Pecnic *et al.* (2005) compare three models: the v^2-f, SST k-ω and Spalart-Allmaras. The first two models obtained better results in calculate the efficiency of a transonic turbine component. The v^2-f was more accurate in heat transfer, the other both models predict the exact location of the passage vortex. The Sapalart-Allmaras underpredict the heat transfer at leading edge, the author pointed these behaviors due to low production of eddy viscosity of the closure in stagnations regions.

Richardson (2009), likewise Pecnic *et al.* (2005), compared the Spalart-Almaras model with SST k-ω, applied in a simulation of a high-work axial turbine. Although he also conclude that the SST k-ω is more suitable, mainly because captures better the flow separation, during design process and the Spalart-Allmaras closure is more indicated due to the matching between low computational cost and accuracy.

The standard k-ε turbulence model is a simple and most popular two-equation turbulence closure. It was developed by Jones and Launder (1972) for aerodynamics purpose but has been applied in many other engineering areas. This model make use of two transport equation, to compute the turbulent kinetic energy (k), Eq. (2), and the turbulent dissipation rate(ε), Eq. (2). It make possible to account the turbulence length and velocity independently (Wilcox, 1993). The turbulence kinetic transport equation is obtained exactly, but the equation for turbulence dissipation rate is determined by using physical assumptions.

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{b1} \frac{\partial U_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[\left(\nu + \nu_T / \sigma_k \right) \frac{\partial k}{\partial x_j} \right] \tag{2}$$

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = c_{\varepsilon 1} \frac{\varepsilon}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[\left(\nu + \nu_T / \sigma_k \right) \frac{\partial \varepsilon}{\partial x_j} \right] \tag{3}$$

Where $C_{\varepsilon 1}$, $C_{\varepsilon 1}$, σ_ε and σ_k are closure coefficients and the τ_{ij} is the Reynolds stress tensor. Kinematic eddy viscosity, kinetic energy of turbulence and its dissipation are represented by ν_T, k and ε respectively.

The *k-ε* model show good results for many engineering problems, however has difficulty to predict the behavior of the fluid close the wall. Different versions of this model have been development, as RNG and realizable *k-ε*, trying to obtain better results in certain application and near the walls. The RNG *k-ε* was compare with Spalart-Allmaras and Reynolds Stress Model (RSM) by Pasinato *et al.* (2004) to mitigate the high heat transfer in the vicinity of the vane leading edge and along the endwall. The three models predicted similar flow field and vortical structures nearby leading edge, with slightly advantage to RSM. Although theses flow structures are main responsible for strong heat transfer, differences between the models were found and all models over predicted the highest values of Stanton number along the endwall.

In the standard model, the using of damping functions is necessary to correct the behavior of the fluid close to solid surfaces (Tartinville, 2007). Menter (2003) recommend the use of limiters in *k-ε* formulation when applied in flows with heat transfer and reattachment. Stagnation zones also must receive special attention.

In work of Djouimaa *et al.* (2007), the blade loading and isentropic Mach number of a cascade was calculated using several known turbulence closures. All models represent quite well the experimental measurements. However as the study was lead in 2-D, 3-D and important turbulence phenomena were neglected. According to Ferzinger (1998) considering the flow two dimensional, some turbulence models can calculate exactly the eddy viscosity, becoming an exact model.

Despite the standard *k-ε* model is not suitable in under negative pressure gradient cases (Menter, 2003), Li (2000) applied it to "close" the incompressible Reynolds-averaged equations of a multistage low-speed axial compressor. Accurate results were obtained demonstrating that the two-equation model still provide good approaching in modeling the Reynolds stresses and is numerically robust in negative gradient pressure flows.

Tartinville *et al.*(2007) compared some turbulence models in turbomachinery flows and related cases. They intended to determine the capability of the *v²-f* model comparing with a modified *k-ε* and Spalart-Allmaras models and confronted with experimental data. All the models were able to determine the velocity profile and circumferential flow angle in a cascade test. But in a low-pressure turbine component, the one-equation closure with a transition model predicted the laminar to turbulent transition zone and the heat transfer coefficient better than *v²-f.*

Maybe Dunham (1998) did one of the biggest analyses of turbulence models and numerical schemes for CFD validation. In these AGARD report, several CFD simulations from different working groups are joined and compare with experimental data. Among the many different turbulence models and numerical schemes, is hard to choice the best turbulence closure, however Spalart-Allmaras contrast from others. The *k-ε* closure produced accurate results too, but as the numerical methods differ model to model, a final conclusion cannot be made.

4. SINGLE STAGE HIGH-PRESSURE TURBINE

In the current study, single stage high-pressure turbine component was used to compare the two turbulence closures performance (Thulin *et al.*, 1983). This turbine had been represented at 1983 the state of art in aircraft propulsion, with advanced technologies in materials, aerodynamics and structure. The component was designed to give an efficiency of 88.8% and around 4 atm. of expansion at 10.668 m and Mach 0.8 (design-point). Supplementary information is given in Tab.1.

Table 1. Single-stage axial-flow turbine data.

	Stator	Rotor
Number of blades	24	54
Hub radius	35.26(inlet)	34.74
Tip radius	42.12(inlet)	41.02
Mean chord	10.9	4.96
Mean axial chord	4.36	2.95

The geometrical data from reference was treated to generate the 3D turbine geometry by the use of a CAD tool. Figure 1 shows the axial turbine view and its blade rows.

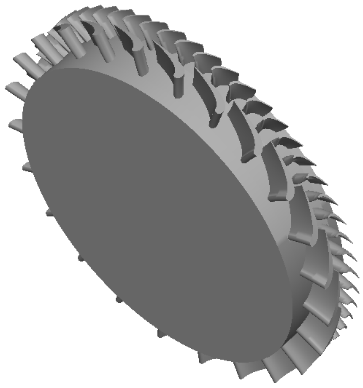

Figure 1. 3D view of the single-stage axial flow turbine

5. GRID GENERATION AND NUMERICAL ISSUES

The structured H-type grid with hexahedron elements was generated around the turbine blades. The final grid size has 876.090 nodes: 57 nodes from blade hub-to-tip, 53 nodes at blade-to-blade section and 290 nodes along the turbine streamwise. The mesh dependence was evaluated for different mesh size. Figure 2 shows the final grid generated around the turbine blade rows.

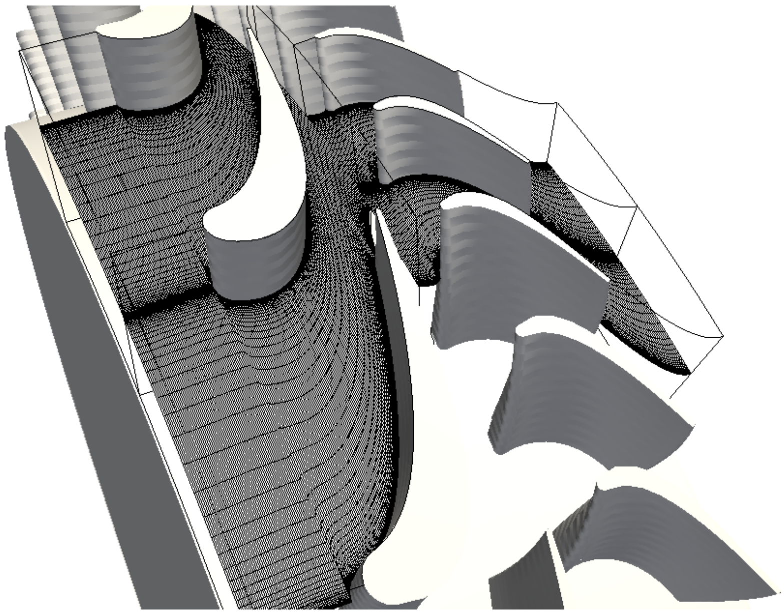

Figure 2. 3D view of the single-stage axial flow turbine

The boundary conditions used are: stagnation conditions at inlet; static pressure at outlet; periodic condition and mixing-plane approach at inter-rows region. At inlet and outlet the non-reflective boundary condition was used (Giles, 1990). The second-order centered scheme was used for the spatial discretization of convective and diffusive terms from Navier-Stokes equations with the addition of second and fourth-order artificial dissipation terms to avoid numerical instabilities in the shock regions. For time-integration, a second-order four-step Runge-Kutta scheme was used. Reference (Jameson *et al*, 1981) describes the details of these numerical methods.

To accelerate the numerical convergence the variable time-step for each control-volume, implicit residual smoothing and multigrid techniques were used. The Courant number was set equal to 1.0 to maintain the numerical stability along the iterations. High values of Courant number was used, but the numerical procedure pitfall and diverges. The same grid was used for all simulations.

For each turbulence model (Spalart-Allmaras and k-ε) several turbine operation points was run varying the outlet static pressure. The mass-flow, efficiency and pressure-ratio were monitored to check the numerical convergence as shown in Figs. 3 and 4. In Fig. 4 the bulk efficiency means that the efficient is calculated based on other averaged variable, in this case mass-flow.

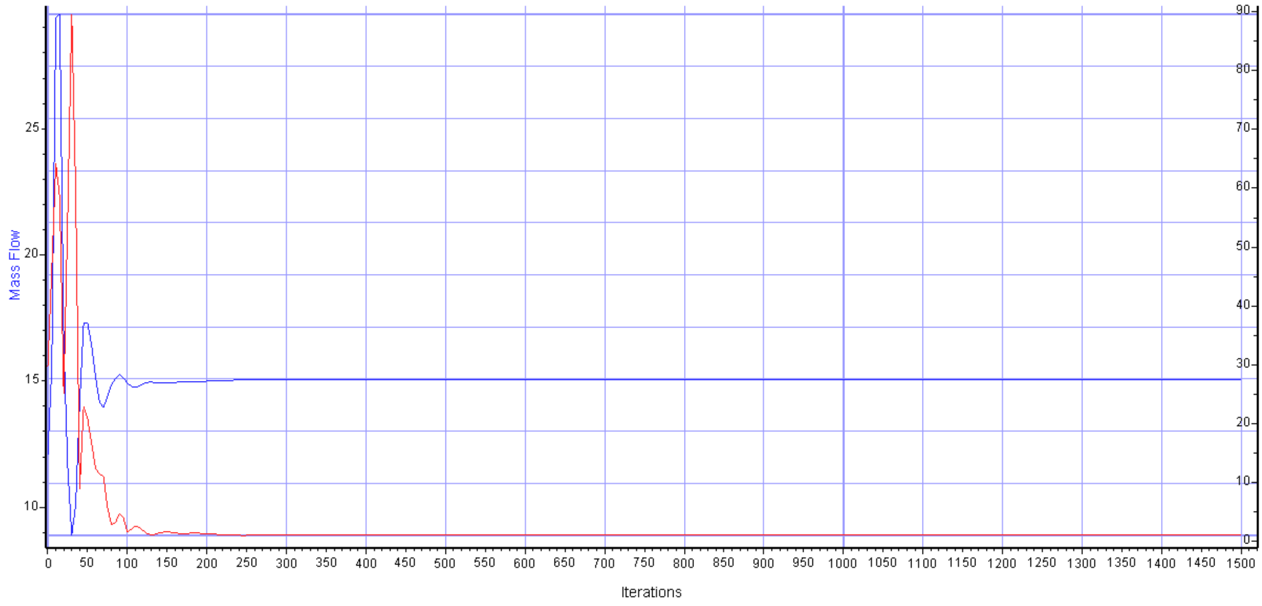

Figure 3. Inlet mass-flow monitoring during numerical iterations.

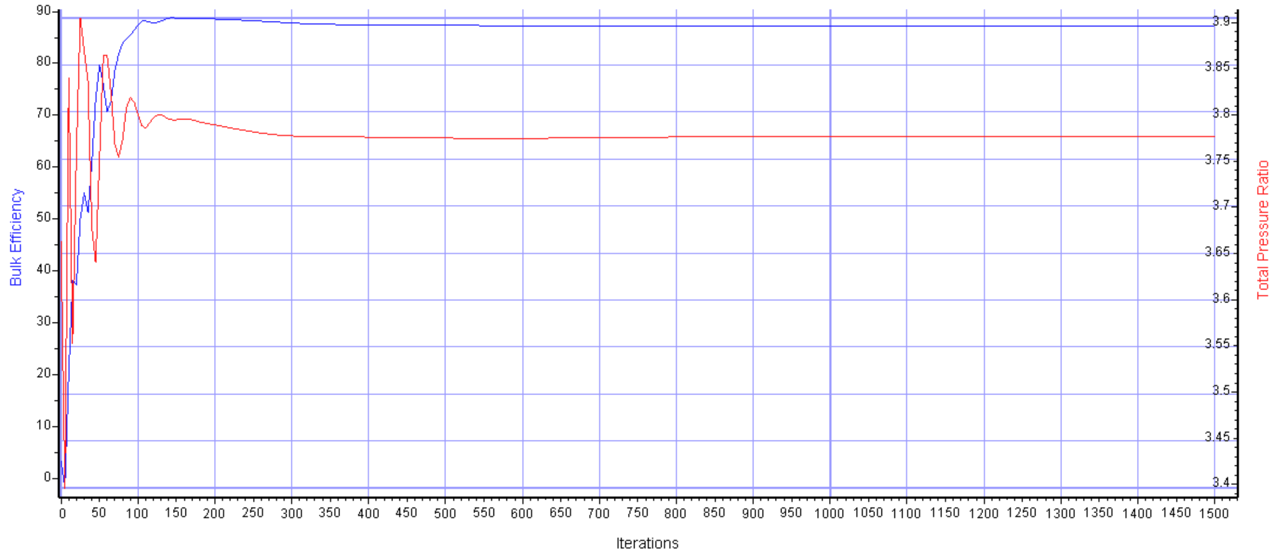

Figure 4. Pressure-ratio and efficiency monitoring during numerical iterations.

6. RESULTS

The results of the simulations were compared with the test data containing information of pressure ratio versus efficiency. Figure 5 shows the CFD results with both turbulence models versus test data. The high pressure turbine stage runs with high velocities mainly in the region close to tip due to the high blade tangential velocity. Hence, it is possible to observe supersonic flows around the blade span as show in Fig. 6.

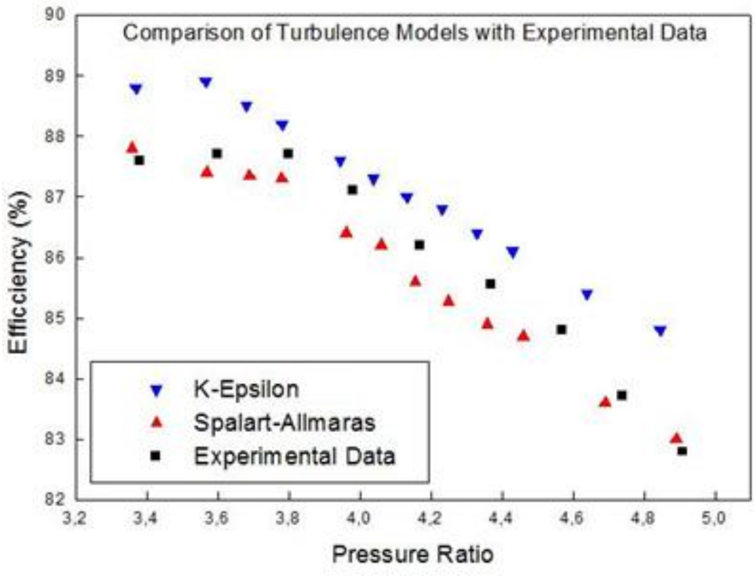

Figure 5. Comparison between turbulence models and experimental data.

For high pressure operation, the flow presents some separation at the turbine rotor suction side as presented in Fig. 7. Reverse flows appear stronger for the simulation using Spalart-Allmaras than *k*-ε turbulence models. But, for Spalart-Allmaras turbulence model the turbulence equation was integrated until the wall and for *k*-ε the wall function was used. The standard *k*-ε turbulence model is not recommended for separated flows. For high pressure ratio operations, it is common the existence of reverse flows for these cases Spalart-Allmaras turbulence model presents better results.

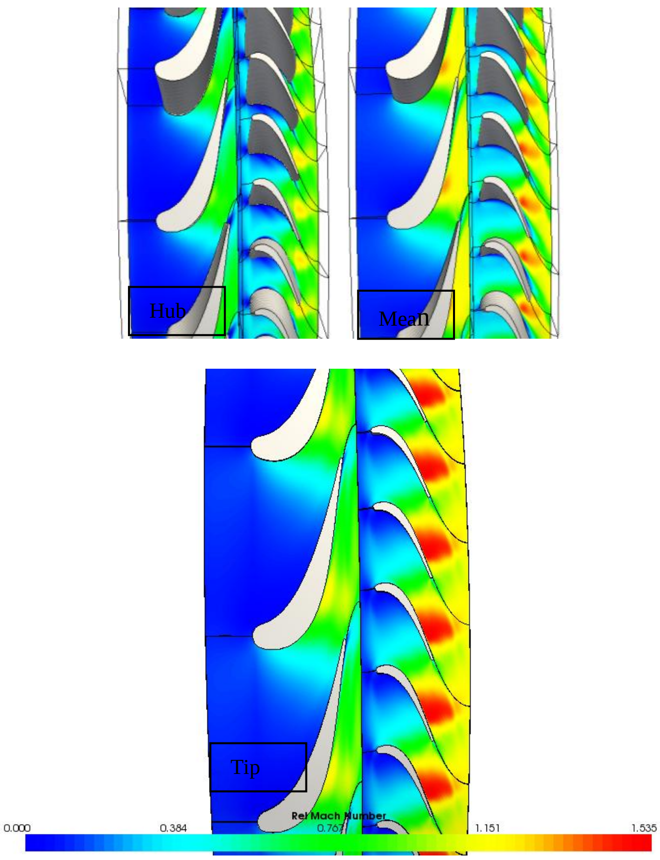

Figure 6. Variation of Mach number at 10%, 50% and 90% blade span for 3.78 pressure ratio.

For high pressure operation, the simulation using *k*-ε model presents different Mach number distribution when compared with Spalart-Allmaras turbulence models. Figure 8 show that the Mach number reaches the value 1.55 for

Spalart-Allmaras turbulence model and 1.62 for *k-ε* model. Of course that, the use of wall functions the flowfield suffers modifications due to the velocity distribution imposed by the wall laws. Many researches are strongly against the use of wall laws because near-wall region controls everything about flow proprieties distributions.

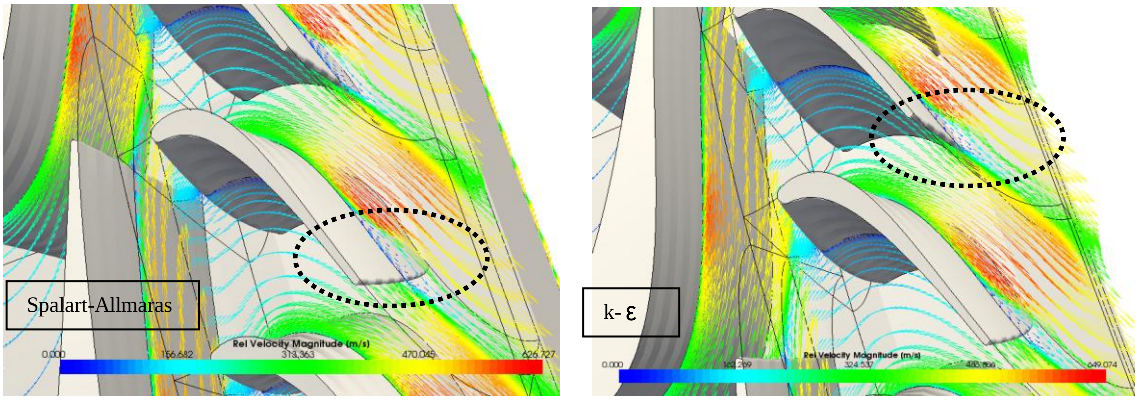

Figure 7. Reverse flow at turbine rotor suction side (Spalart-Allmaras and *k-ε* turbulence models).

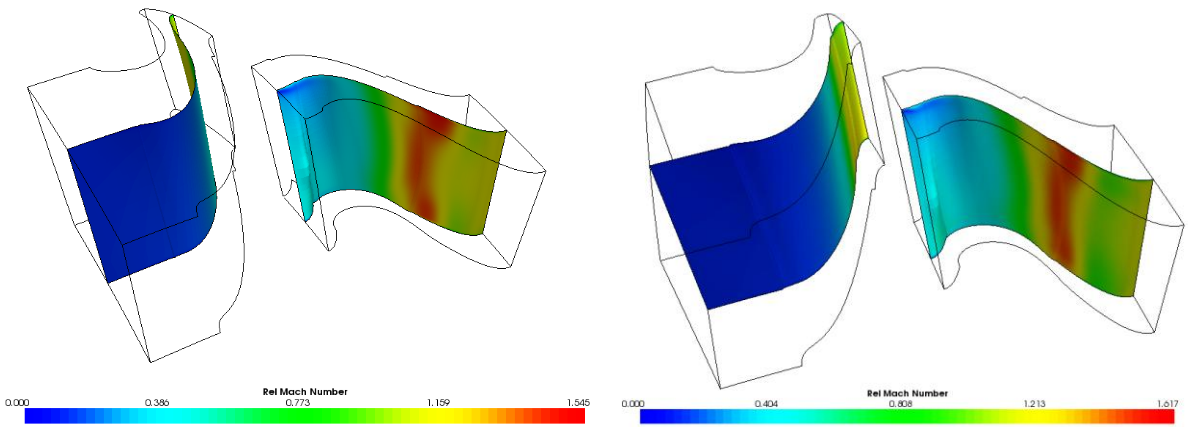

Figure 8. Mach number distribution from hub-to-tip at mid distance between two adjacent blades (Spalart-Allmaras and *k-ε* turbulence models).

Figure 9 shows the comparison of Mach number distribution along the blade span at leading and trailing edges (LE and TE) of the NGV and rotor blades for pressure ratio equal to 3.78 and 4.46.

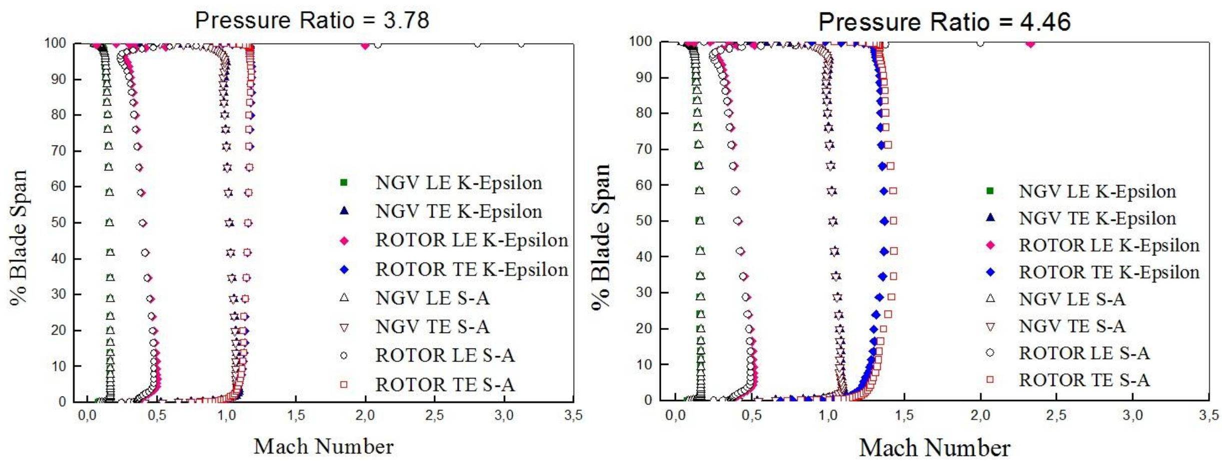

Figure 9. Mach number distribution at leading and trailing edges for each blade row and different operation conditions.

It is possible to observe a good agreement of the results for both turbulence models for the pressure ratio equal to 3.78. For pressure ratio equal to 4.46 the flow at rotor blade suction side, in the outlet region, presents flow separation and different Mach number distribution for each turbulence model. The wall function used for *k-ε* turbulence model is not capable to calculate the flow separation accurately when compared with the Spalart-Allmaras turbulence model. The CFD results for *k-ε* turbulence model are quite different with experimental data for low and high pressure ratio. Unfortunately, there are no experimental data of Mach number distribution for this axial turbine.

7. CONCLUSION AND DISCUSSION

Turbulence modeling still plays an important role in flow field calculations. The prediction of CFD codes are dependent of how the turbulence is modeled, the problem to be solved, strong grid quality dependence and robustness of numerical methods. Flow separation, passage vortex, transition are examples of complex physical phenomena that occur in several engineering problems.

There is no general turbulence model and all problems should be analyzed to set an appropriate modeling to do a good approach for each case. Each model has its limitations and constraints that should be completely understand by the user.

All simulations were run with an H-grid type. The use of an O-grid type can be improving the results due to the better adjustments between grid points distribution and turbine blades geometry. The process of grid generation of this high pressure turbine was not easy due to the complex blade geometry and the adequacy of the grid smoothing from wall to the boundaries. Unfortunately, only the pressure ratio and efficiency experimental results were supplied for this single-stage axial turbine. In the turbomachinery community is very difficult to obtain all geometrical details and test data results due to the proprietary information.

In the case of turbomachinery and aerodynamics problems the Spalart-Allmaras turbulence model is vastly used due to the robustness and accurate results when compared with experimental data. The well- known standard *k-ε* turbulence model is to be used for flows without separations and it is very interesting if the turbulent kinetic energy and turbulence dissipation rate equations if they could be integrated until the wall. But, for this purpose the mesh should be very fine close to the wall due to the y^+ requirements of the model. The results presented in this work it shows that the Spalart-Allmaras still quite better than *k-ε* turbulence model mainly for low and high pressure ratio operation points.

For low pressure ratio operation condition the *k-ε* estimate the efficiency around +1.0% (average value) and the Spalart-Allmaras model estimate the efficiency around 0.28% (average value) compared with test data. For high pressure ratio operation condition the *k-ε* estimate the efficiency around +1.0% (average value) and the Spalart-Allmaras model estimate the efficiency around 0.15% (average value) compared with test data. For the range of pressure ratio between 3.8 – 4.5, both turbulence models presents good results. The *k-ε* turbulence model not presented good agreement with test data when separated flow was found at rotor blade suction side even with the use of wall functions.

For future work, the same high pressure turbine stage will be used for other two-equation turbulence models that can be used for flows with boundary-layer separation.

8. ACKNOWLEDGEMENTS

The authors would like to thank ITA, for supporting this work developed in the Center for Reference on Gas Turbine and Energy in association with Vale Soluções em Energia – VSE.

9. REFERENCES

Baldwin, B. S. and Barth, T. J, 1990, “A One-Equation Turbulence Transport Model for High Reynolds Number Wall-Bounded Flows”, NASA, NASA-TM-102847.

Barbosa, J. R., 1987, “A Streamline Curvature Computer Programme for Performance Prediction of Axial Flow Compressors”. PhD thesis, Cranfield Institute of Technology.

Djouimaa, S., Messaoudi, L. and Giel, P. W., 2007, “Transonic Turbine Blade Loading Calculations Using Different Turbulence Models – Effects of Reflecting and Non-Reflecting Boundary Conditions”, Journal of Applied Thermal Engineering, Vol. 27, pp.779-787, Elsevier.

Dunham, J., 1998, “CFD validation for propulsion system components”, AGARD-AR-355, Neuilly-sur-Seine, France.

Ferzinger, J. H., 1998, “The Physics and Simulation of Turbulence-State of Art 1998”, Advances in Turbulence Modeling, von Karman Institute for Fluid Dynamics, Lecture Series 1998-05.

FLUENT INC., 2008, Fluent 12.0 Theory Guide, Ansys Inc.

Giles, M. B., dez. 1990, “Non-refecting Boundary Conditions for Euler Equation Calculation”, AIAA Journal, Vol. 28, No. 12, pp. 2050-2058.

Jameson, A., Schmidt, W. and Turkel, E., 1981, “Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes”, Fluid and Plasma Dynamics Conference ,AIAA 14, Palo Alto, EUA, pp. 1-14. California

Jones, W. P and Launder, B. E., 1972, “The Prediction of Laminarization with Two-Equation Model of Turbulence”, International Journal of Heat and Mass Transfer, Vol. 15, pp. 301-314

Li, Y., 2000, “Three-Dimensional Flow and Performance Simulation of Multistage Axial Flow Compressors”, PhD Thesis, Cranfield University.

Menter, F. R., 2003, “Turbulence Modelling for Turbomachinery”, QNET-CFD Network Newsletter, Vol. 2, No. 3, Germany, pp. 10-13.

Pasinato, H. D., Squires, K. D. and Roy, R. P., 2004, “Assessment of Reynolds-Averaged Turbulence Models for Prediction of the Flow and Heat-Transfer in an Inlet Vane-Endwall Passage”, Journal of Fluids Engineering, Vol. 126, p.p. 305-315.

Pecnik, R., Pieringer, P. and Sanz, W., 2005, “Numerical Investigation of The Secondary Flow of a Transonic Turbine Stage Using Various Turbulence Closures”, ASME Turbo Expo, Nevada, USA, GT2005-68754.

Richardson, D., 2009, “Unsteady Aerodynamics of High Works Turbines”, Engineering Doctorate Thesis, Cranfield University.

Santin, M. A. B, 2006,“Simulação Numérica de Escoamento em Turbinas de Alto desempenho”, Master’s Thesis, Instituto de Tecnologia de Aeronáutica, São José dos Campos, Brazil.

Soghe, R. D., Innocenti, L., Andreini, A. and Poncet, S., 2010, “Numerical Benchmark of Turbulence Modeling in Gas Turbine Roto-Stator System”, ASME Trubo Expo 2010, Glasgow, UK, GT2010-22627.

Spalart, P. and Allmaras, S., 1992, “A One-Equation Turbulence Model for Aerodynamic Flows” AIAA, 1992. Aerospace Sciences Meeting & Exhibit, pp.1-22, AIAA-92-0439.

Tartinville, B., Lorrain, E. and Hirsch C., 2007, “Application of the v^2-f Turbulence Model to Turbomachinery Configurations”, ASME Turbo Expo 2007, Montreal, Canada, GT2007-27708.

Thulin, R. D, Howe, D. C, and Singer I. D., 1982, “Energy Efficient Engine - High-Pressure Turbine Detailed Design Report”, National Aeronautics and Space Administration(NASA), NASA CR-165608.

Tomita, J. T., 2009, “Three-Dimensional Flow Calculations of Axial Compressors and Turbines Using CFD Techniques”. PhD thesis, Instituto Tecnológico de Aeronáutica, São José dos Campos, Brazil.

Tomita, J., 2003, “Numerical Simulation of Axial Flow Compressors.”, Master’s Thesis, Instituto Tecnológico de Aeronáutica, São José dos Campos, Brasil.

Versteeg, H. K. and Malalasekera, W., 1995, “An Introduction to Computational Fluid Dynamics - The Finite-Volume Method”. Longman Scientic & Technical.

WILCOX, D. C.,1993. “Turbulence Modeling for CFD”, DCW Industries, 2. ed., California, United States of America, pp. 01-21, 103-215.

10. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.