Proceedings of ENCIT 2012

14th Brazilian Congress of Thermal Sciences and Engineering
November 18-22, 2012, Rio de Janeiro, RJ, Brazil

AN AXIAL-FLOW TURBINE PERFORMANCE CALCULATIONS BASED ON CFD TECHNIQUES

Luiz Müller, luizmuller.es@terra.com.br
Jesuino Takachi Tomita, jtakachi@ita.br
Instituto Tecnológico de Aeronáutica – ITA
Praça Mal. Eduardo Gomes, 50 – 12288-900 São José dos Campos – SP, Brasil

Abstract. *A single-stage axial-flow turbine was designed using the meanline approach with the addition of loss models and its 3D geometry was generated to become possible the study and its performance evaluation using CFD techniques. A specialized turbomachine commercial software was used for preliminary design and flow calculations. The three-dimensional turbulent flow calculations was performed based on a 3D CFD software with RANS formulations and the turbine operation map was determined. The results from CFD software were compared with the meanline turbine data including its performance map. The mesh generation processes and numerical issues are discussed and the CFD results are presented based on the fluid properties distribuiton along the blade span (NGV and rotor). The turbine shaft work and its efficiency for several operation points also are calculated and compared.*

Keywords*: turbomachines, CFD, axial turbine, computational grids, gas turbine.*

1. INTRODUCTION

The development of high performance computers in recent decades has allowed growth of an alternative for the study of fluid flow, the use of Computational Fluid Dynamics (CFD). This tool solves numerically the equations describing the flow: mass conservation, Navier-Stokes and energy conservation.

The CFD simulation is especially interesting because it has much lower costs when compared to experimental tests, such as the use of test benches and wind tunnels. With the CFD is possible to reduce the amount of experiments and explore various phenomena not so well understood in the laboratory tests.

The flow calculation in turbomachines still is a major challenge in the area of CFD. The development of CFD techniques and the obtaining of more accurate and reliable mathematical models are extremely important for making the results obtained increasingly closer to reality. We can include in this context, the use of turbulence models.

The equations that mathematically describe the behavior of fluids are known as equations of: mass conservation, Navier-Stokes and energy conservation. In the CFD community it is common to call these equations as governing equations.

Analytical solutions to the Navier-Stokes equations are determined only for a few simple cases. Hence, a good and robust numerical method should be used to solve numerically this non-linear system of equations. The designer is interested in values of pressure, temperature and velocity distributions along the turbine blade-to-blade passage from inlet to outlet of the machine in the streamwise direction.

The use of numerical solutions applied in the Navier-Stokes equations potentially reduces time and cost of a turbine project. Therefore, it is necessary a specialized labor to properly interpret the flow physics and detailed understanding of the numerical techniques used as well as their limitations.

2. OBJECTIVE

The objective of this work is to determine the flow behavior within a single-stage axial turbine using a CFD commercial software developed by CONCEPTS NREC and called AxCent. The AxCent software has a CFD module called PbCFD. This module was used to generate the grid and to calculate the fluid mechanics equations in a 3D environment for a flow in the steady-state regime.

The results from CFD tool were compared with the turbine preliminary design based on a meanline 1D technique with appropriate internal loss model. The values of Mach numbers, total pressure and temperature distributions are presented for both design tools. The turbine operation map was determined using CFD tool and it is compared with the preliminary design results too.

3. LITERATURE REVIEW

Currently, many studies on application of CFD tool in axial turbines have been developed. Some contributions are discussed in this section.

Silva, D.T. and Tomita, J.T. (2011) discuss the influence of two different turbulence models in fully turbulent flow within an axial turbine: the one equation turbulence model of Spalart-Allmaras and the two equations turbulence model k-ε. The results are discussed for each turbulence model based on the solution from CFD tool analyzing the distribution of fluid properties along the turbine blade span.

The main results showed that the Sparlart-Allmaras model shows better results than the k-ε model for high pressure ratios. At high pressure ratios can occur boundary layer separation on the turbine blade suction side. The k-ε turbulence model, which uses wall functions, cannot obtain, in some cases, accurate results under these conditions and engineering problems. With the Sparlart-Allmaras turbulence model the equation is integrated until the wall, respecting the values of y+ following the model recommendations.

An detailed discussion and comparison between the results of a high pressure turbine flowfield using different turbulence models including a seven equations model and the well known two-equation SST turbulence model was described in the work developed by Tomita, J. T., Silva, L. M. and Silva, D. T. (2011). In this work, the models were compared for different mesh types as structured and unstructured and its application for different turbulence equations.

Silva, L.M., Tomita, J.T. and Barbosa, J.R. (2011) discuss the influence of the tip clearance of rotor blades in internal losses, which affect the operation of an axial turbomachine when the pressure ratio and efficiency are analyzed.

Comparing different configurations of the tip clearance, significant differences in the turbine efficiency were found. Thus, it reinforces that the tip clearance has a great influence on the losses and can cause a significant drop in engine efficiency. This parameter should be evaluated since the preliminary design when a small improvement in engine performance means large reduction in operating costs.

Tomita, J.T. and Barbosa, J.R. (2011) discuss the influence of the flow turbulence intensity in an axial turbine inlet in its design-point operation. The flow calculations were performed by an in house 3D CFD code based on RANS (Reynolds Average Navier-Stokes). The turbulence effect was performed using the one-equation turbulence model developed by Sparlat and Allmaras. The flow was considered completely turbulent.

The main results show that the influence in Mach number, varying the turbulence intensity, is small along the NGV, but, to the rotor, the Mach number is slightly different for each turbulent intensity value. The total pressure distribution realizes great differences to the region close to the NGV trailing edge. Similar effect occurs for the rotor blades, but much less intense. The velocity distribution along the trailing edge of the rotor blade is also greatly affected by the existence of strong secondary flow. And, there are large differences in the eddy viscosity ratio (turbulent and molecular viscosity ratio, determined by the turbulence model) for each value of turbulence intensity in the machine inlet. Also, note that the flow characteristics near the turbine are highly influenced by the characteristics of the flow at the outlet of the combustion chamber.

4. THE FLUID MECHANICS EQUATIONS (CONSERVATION LAW) IN THE 3D FORM

It is possible to represent the fluid mechanics equations of partial differential equations in matrix form:

$$\frac{\partial \vec{Q}}{\partial t}+\frac{\partial \vec{E}}{\partial x}+\frac{\partial \vec{F}}{\partial y}+\frac{\partial \vec{G}}{\partial z}=\vec{S} \tag{1}$$

The term $\vec{Q}$ is the vector of conserved variables. The vector $\vec{S}$ is the sources terms in the momentum equations. The flux vectors $\vec{E}$, $\vec{F}$ and $\vec{G}$ can be divided into different components: inviscid, viscous and turbulent terms, as indicated:

$$\vec{E}=\vec{E}_e+\vec{E}_v \tag{2}$$

$$\vec{F}=\vec{F}_e+\vec{F}_v \tag{3}$$

$$\vec{G}=\vec{G}_e+\vec{G}_v \tag{4}$$

The Euler terms (convective components) are represented by the subscript “e”. The viscous terms are represented by the subscript “v”. The matricial representations of these equations are presented below:

$$\vec{Q}=\begin{Bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ e_t\end{Bmatrix} \tag{5}$$

$$\vec{E}_e = \begin{Bmatrix} \rho u \\ \rho u u + P \\ \rho u v \\ \rho u w \\ (e_t + P)u \end{Bmatrix} \tag{6}$$

$$\vec{F}_e = \begin{Bmatrix} \rho v \\ \rho v u \\ \rho v v + P \\ \rho v w \\ (e_t + P)v \end{Bmatrix} \tag{7}$$

$$\vec{G}_e = \begin{Bmatrix} \rho w \\ \rho w u \\ \rho w v \\ \rho w w + P \\ (e_t + P)w \end{Bmatrix} \tag{8}$$

$$\vec{E}_v = \begin{Bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{xx} u + \tau_{xy} v + \tau_{xz} w + \gamma \left(\frac{\mu_l}{Pr_l} + \frac{\mu_t}{Pr_t} \right) \frac{\partial e_i}{\partial x} \end{Bmatrix} \tag{9}$$

$$\vec{F}_v = \begin{Bmatrix} 0 \\ \tau_{yx} \\ \tau_{yy} \\ \tau_{yz} \\ \tau_{yy} v + \tau_{yx} u + \tau_{yz} w + \gamma \left(\frac{\mu_l}{Pr_l} + \frac{\mu_t}{Pr_t} \right) \frac{\partial e_i}{\partial y} \end{Bmatrix} \tag{10}$$

$$\vec{G}_v = \begin{Bmatrix} 0 \\ \tau_{zx} \\ \tau_{zy} \\ \tau_{zz} \\ \tau_{zz} w + \tau_{zx} u + \tau_{zy} v + \gamma \left(\frac{\mu_l}{Pr_l} + \frac{\mu_t}{Pr_t} \right) \frac{\partial e_i}{\partial z} \end{Bmatrix} \tag{11}$$

Where ρ is the density, τ_{ij} are the viscous stresses, e_t is the total energy, P is the total pressure, μ_l is the molecular coefficient of viscosity, μ_t is the turbulent coefficient of viscosity, Pr_l is the laminar Prandtl number, Pr_t is the turbulent Prandtl number and γ is the ratio of specific heats.

The first term of each vector corresponds to the mass equation, the last term to the energy equation and the other three terms to the Navier-Stokes equations.

5. THE AXIAL TURBINE DESIGN PARAMETERS

The turbine to be analyzed is a single-stage axial turbine. The preliminary design of the turbine was calculated using the commercial softwares AXIAL™ and AxCent™ both developed by NREC Concepts ETI.

For more details of the process adopted during the preliminary design and the turbine design requirements see Martins (2011). Basically, this turbine was studied based on a small gas turbine engine with thrust around 5kN in its turbojet version. The main turbine design requirements are shown in Table 1:

Table 1 – Turbine design requirements

Single-Stage Axial Turbine Design Parameters		
Inlet Total Pressure	P_{t0} (Pa)	101,325
Inlet Total Temperature	T_{t0} (K)	288.15
Rotor Blade Tip Speed	U_{tip} (m/s)	439.5
Outlet Static Pressure	P_s (Pa)	164,215
Mass-Flow	$\dot{m}$ (kg/s)	7.95

Total-to-Total Pressure Ratio	PR_{tt}	2.16
Isentropic Efficiency	η (%)	88.50%
Specific Work	$\dot{W}$ (MW/kg)	1.72

The NGV and rotor blade rows are shown in Figures 1 and 2. Figure 3 show the 3D axial turbine geometry.

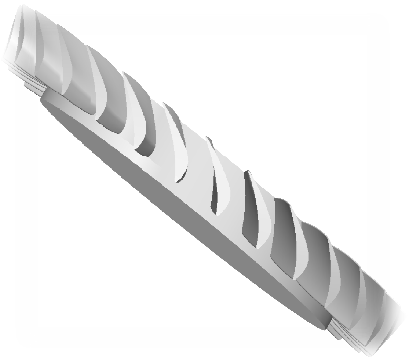
Figure 1. NGV blade row

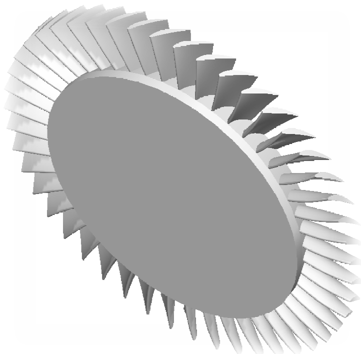
Figure 2. Rotor blade row

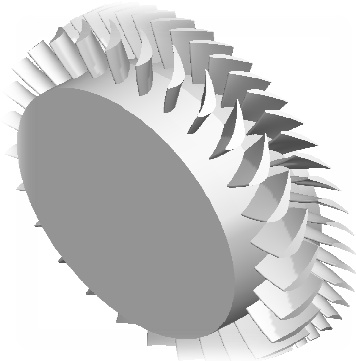
Figure 3. 3D turbine view

6. GRID GENERATION

A good grid quality is strongly desired in the computational domain discretization to ensure a good numerical stability as well as to obtain accurate results of the calculations from partial differential equations discretized using appropriate numerical schemes for convective and diffusion terms of the fluid mechanics equations. Moreover, discontinuous regions, as shock waves, are very aggressive numerically speaking and these regions needs special attention in the domain discretization and numerical schemes treatments applied in the convective terms to avoid the propagation of numerical errors.

The grids type as H and O are widely used on turbomachinery geometries. The H-grid is simpler and easier to generate. But, generally the O-grid is better due to its improved accuracy in representation of leading and trailing edges - complex and high curved regions. The O-grid is best suited to the contours of the blade suction and pressure surfaces including its edges, but is more costly to generate due to the high numbers of polynomial functions and control points to guarantee the quality of elements, for example, their orthogonality and angles. The problem of using H-grid is that the curvature of the leading and trailing edge is partly lost and to reduce this effect would be necessary a very refined mesh.

In general, the O-grid best fits the vicinity of the blade, providing a significant improvement in the computational domain representation. It is also interesting to mention that the O-grid is currently recommended for axial machines. Both of them are shown in Figures 4 and 5.

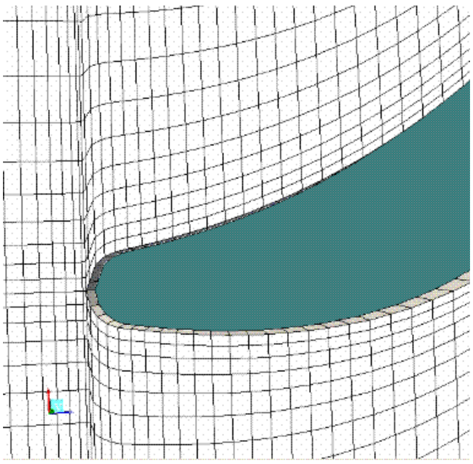
Figure 4. H-grid generated in a turbine blade

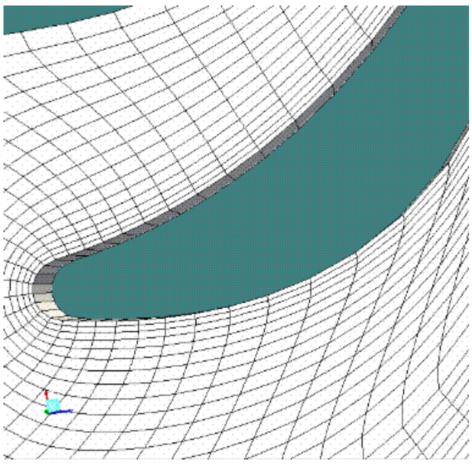
Figure 5. O-grid generated in a turbine blade

In this work, an O-grid was generated, and its dependence was studied based on the different mesh sizes. This mesh type was chosen because this is the mesh configuration that is best suited to the contours of the turbine blade surfaces as afore mentioned. The final mesh size has 802,028 nodes and it is showed in Figure 6.

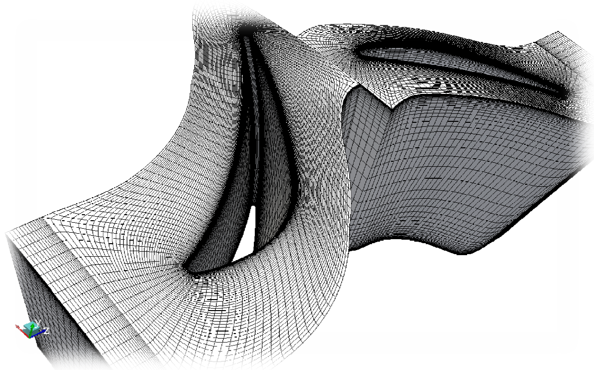

Figure 6. O-grid generated for a single-stage axial turbine

7. NUMERICAL ISSUES

The spatial discretization scheme used in this work for the convective terms is based on the Advection Upwind Split Method ($AUSM^+$) developed by Liou (Chima and Liou, 2003) using a third-order numerical scheme including the Monotone Upstream Schemes for Conservation Laws (MUSCL) and limiter functions as minmod to increase the original discretization order (first order) of upwind schemes. For the time integration, the four-step second-order Runge-Kutta explicit scheme was performed. To accelerate the numerical convergence a W-cycle multigrid, local time-step and the implicit residual smoothing were performed.

The turbulent eddy viscosity was calculated using the one-equation Spalart-Allmaras (Spalart and Allmaras, 1992) turbulence model that determines the flow eddy viscosity based on the modified eddy viscosity term ($\tilde{\nu}$). This turbulence model determines the modified eddy viscosity transport following the equations bellow:

$$\frac{\partial \tilde{\nu}}{\partial \hat{t}} = C(\tilde{\nu}) + D(\tilde{\nu}) = Prod(\tilde{\nu}) + Dest(\tilde{\nu}) + T \tag{12}$$

where the convection term is

$$C(\tilde{\nu}) = -\nabla \cdot (\tilde{\nu}\vec{W})^2 \tag{13}$$

the diffusion term is

$$D(\tilde{\nu}) = \frac{1}{\sigma}\left[\nabla \cdot \left((\nu + \tilde{\nu})\nabla\tilde{\nu}\right) + c_{b2}(\nabla\tilde{\nu})^2\right] - (\tilde{\nu}\vec{W})^2 \tag{14}$$

the production term is

$$Prod(\tilde{\nu}) = c_{b1}[1 - f_{t2}]\tilde{S}\tilde{\nu} \tag{15}$$

the destruction term is

$$Dest(\tilde{\nu}) = \left[c_{w1}f_w - \frac{c_{b1}}{\kappa^2}f_{t2}\right]\left[\frac{\tilde{\nu}}{d}\right]^2 \tag{16}$$

and the transition term is

$$T = f_{t1}\Delta U^2 \tag{17}$$

More details and the model calibration can be found in the references (Tomita, J.T., 2009) and (Spalart and Allmaras, 1992).

8. RESULTS

The values of Mach number, total pressure and temperature distributions calculated using CFD technique were compared with the results from the meanline turbine preliminary design. These values are evaluated along the blade span - NGV and rotor rows.

The results were compared to values obtained in the preliminary design done by Martins (2011). It is important to mention that a more reliable comparison should be done with experimental results, but unfortunately this resource is not yet available.

The Figures 7 to 12 shows the distribution of Mach numbers and total fluid properties for both blade rows. Note that, in the NGV row the results are in the absolute frame of reference and in the rotor row in relative frame of reference.

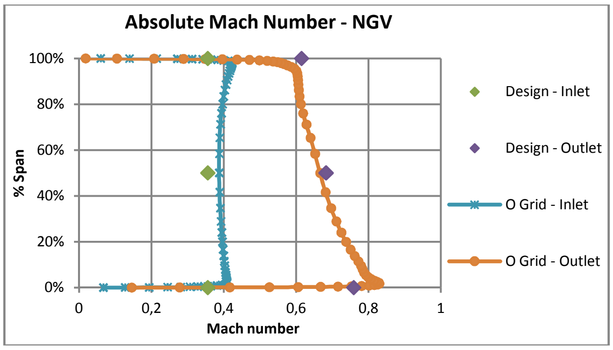

Figure 7. Mach Distribution – NGV

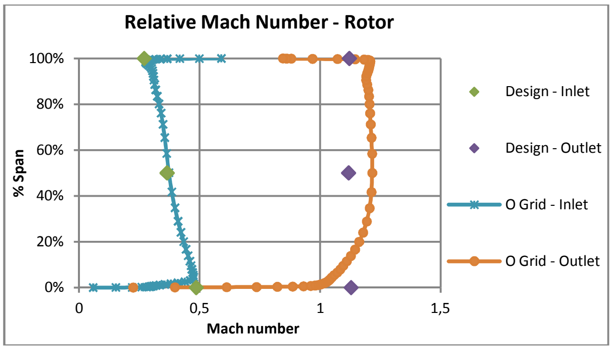

Figure 8. Mach Distribution – Rotor

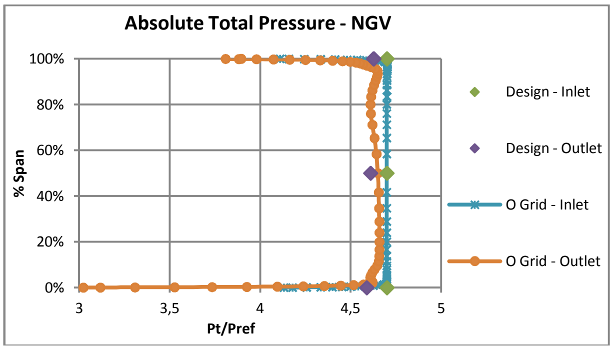

Figure 9. Pressure Distribution – NGV

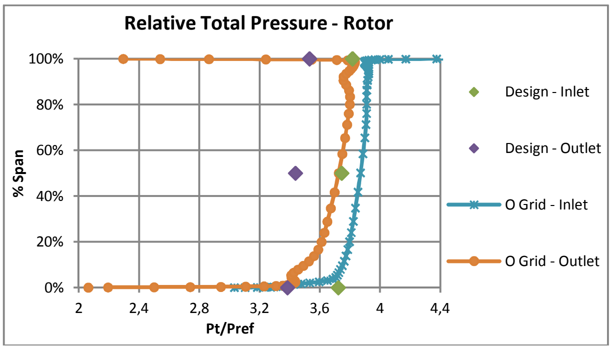

Figure 10. Pressure Distribution – Rotor

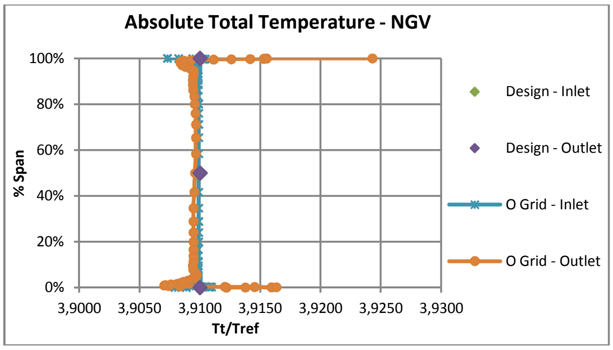

Figure 11. Temperature Distribution – NGV

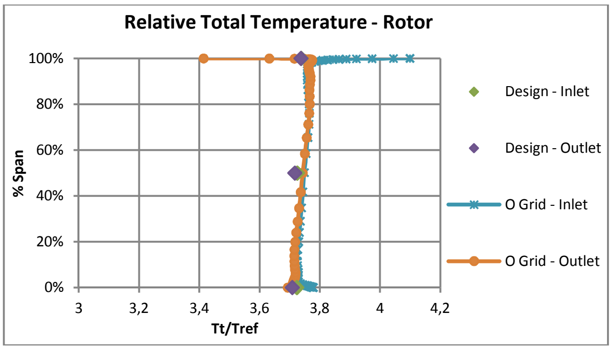

Figure 12. Temperature Distribution – Rotor

Some differences can be observed in Figure 10 in the rotor blade passage. This is due to the fact that in the preliminary design procedure the losses are quantified based on semi-empirical correlations and, in some cases, adjustments are necessary in these loss coefficients. The same behavior can be observed in the wall regions (endwall and casing). The preliminary design tool cannot determine with high accuracy the flow behavior close to the walls where the boundary-layer is acting in the main flow.

Regions with high flow accelerations in the NGV and rotor blades passage are found within turbine flow path. The absolute and relative Mach numbers contours are shown in Figures 13 and 14 for a 50% of blade span. Supersonic flow is common to found in this turbine class and can be observed mainly in the rotor outlet (blade suction side). High performance turbines frequently operate in the choke conditions as presented bellow.

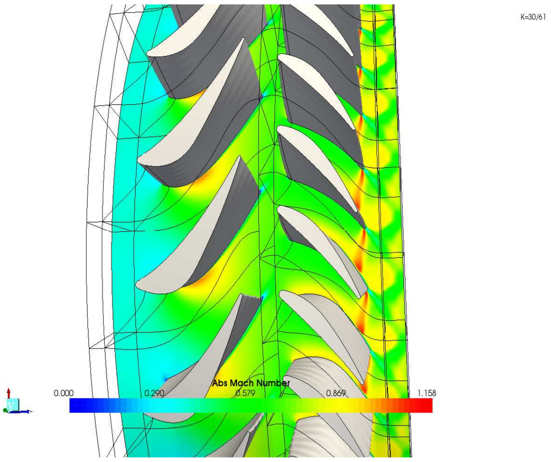

Figure 13. Absolute Mach number

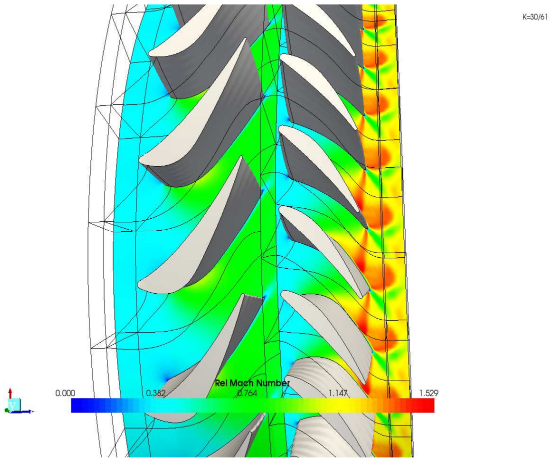

Figure 14. Relative Mach number

To determine the axial turbine operation map is necessary to calculate the operating characteristics of the machine for off-design condition. In this work, nine different operation points were selected according to the turbine outlet static pressure variation.

The turbine map obtained from the results of both techniques (meanline and CFD) is showed in Figure 15.

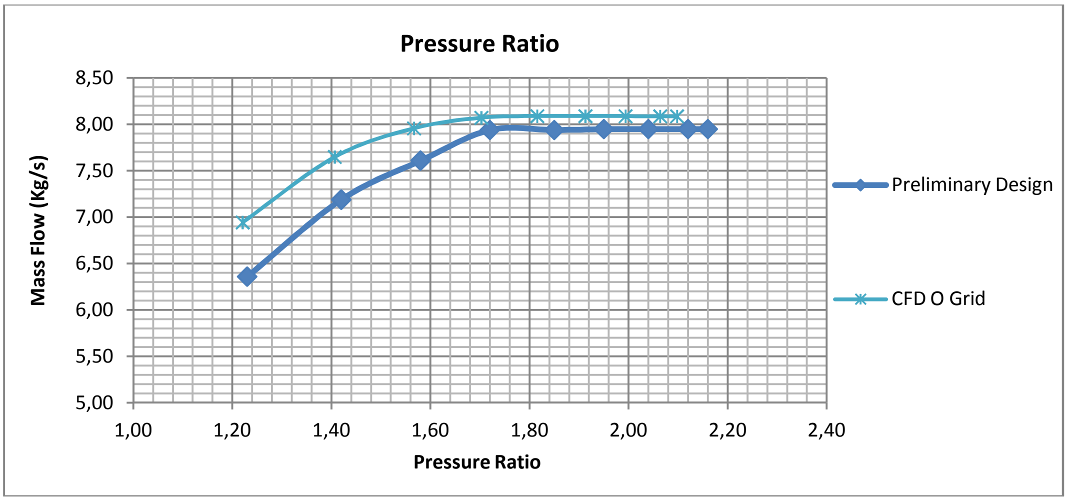

Figure 15. Pressure Ratio Comparison

The turbine shaft work, for several operating points, is also showed and compared in Figure 16.

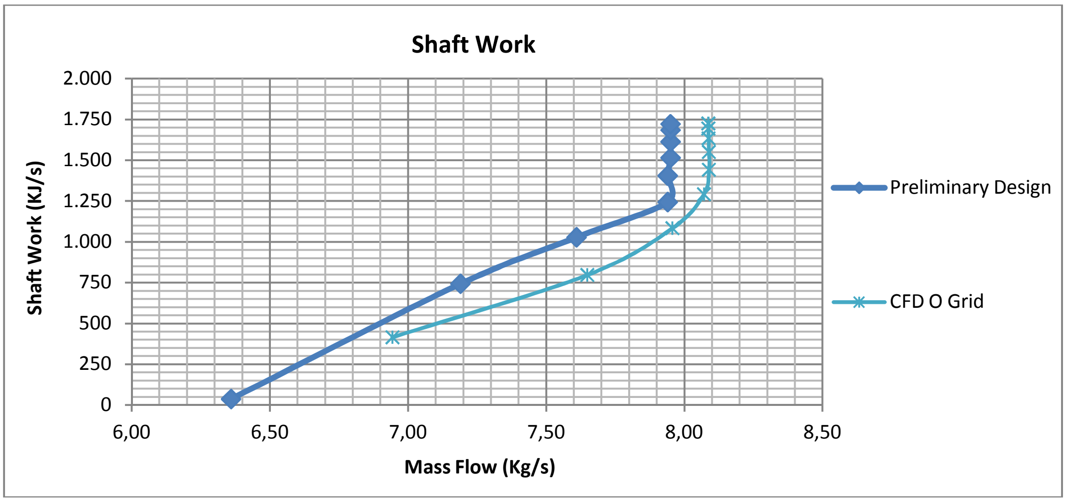

Figure 16. Shaft Work Comparison

The tip clearance of the rotor blades was not modeled in this work. Its influence affects directly the turbine operation as well as pressure ratio. The tip clearance causes an increase in internal losses, which has high interest in turbomachinery community, because the flow that "leaks" in the region of clearance between rotor and casing not participate in the process of energy transfer what consequently will affect the performance of the machine.

9. CONCLUSION

The results of both design tools were close and acceptable values were obtained. Obviously, the 1D technique does not consider some important flow effects as turbulence and boundary layer behavior. These effects are modeled using semi-empirical correlations that sometimes need some re-calibration of its coefficients. But, the preliminary design can provide good results to guide the more detailed studies, in other words, the preliminary results are a good starting point for 3D analysis. Moreover, without the turbine geometry is impossible to perform its 3D flow calculation, and this geometry is generated based on reduced order tools.

CFD can be used as an important tool to check significant design parameters, its values, requirements and also to improve the turbine geometrical characteristics for better performance.

In this work, we studied the flow within an axial turbine, but in a turbomachinery project is also given emphasis to several other important assessments such as: structural, vibration, noise, labyrinth seals, among others. Which ones can be explored in further work.

10. ACKNOWLEDGEMENTS

The authors would like to thank the Turbomachines Department at ITA, for supporting this research.

11. REFERENCES

Chima, R.; Liou, M., 2003. *Comparison of the AUSM+ and H-CUSP Schemes for Turbomachinery Applications*. Glenn Reserch Center. Ohio.

Cohen, H.; Rogers, G.F.C.; Saravanamuttoo, H.I.H., 1996. *Gas Turbine Theory*. 4ª. ed. Essex: Longman.

Fortuna, A.D.O., 2000. *Técnicas Computacionais para Dinâmica dos Fluidos: Conceitos básicos e aplicações*. São Paulo: Editora da Universidade de São Paulo.

Martins, V.A.C., 2011. *Projeto Preliminar de uma Turbina Axial para uso em Turbina a gás de Pequena Potência*. ITA. São José dos Campos, SP, Brazil.

Tomita, J.T., 2009. *Three-dimensional flow calculations of axial compressor and turbines using CFD techniques.* ITA. São José dos Campos, SP, Brazil.

Spalart, P.R.; Allmaras, S.R. 1992. *A One-equation Turbulence Model for Aerodynamics Flows.* Aerospace Sciences Meeting & Exhibit, AIAA-92-0439.

Silva, D.T.; Tomita, J.T., 2011. *Axial Turbomachinery Flow Simulations with Different Turbulence Models.* ITA. São José dos Campos, SP, Brazil.

Silva, L.M.; Tomita, J.T.; Barbosa, J.R., 2011. *A Study of the Influence of the Tip-Clearance of an Axial Turbine on the Tip-Leakage Flow Using CFD Techniques.* ITA. São José dos Campos, SP, Brazil.

Tomita, J.T.; Barbosa, J.R., 2011. *Influence of Inflow Turbulence Intensity Variations in an Axial Turbine Using 3D Rans Computations.* ITA. São José dos Campos, SP, Brazil.

Tomita, J.T.; Silva, L.M.; Silva, D.T., 2011. *Comparison Between Unstructured and Structured Meshes With Different Turbulence Models for a High Pressure Turbine Application*. Proceedings of ASME Turbo Expo 2012, GT2012-6990, June 11-15, 2012, Copenhagen, Denmark.

11. RESPONSIBILITY NOTICE

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