3D TURBULENT FLOW ANALYSIS IN A TURBOCHARGER RADIAL TURBINE OPERATING AT DESIGN-POINT

Diego Silva de Carvalho
Jesuíno Takachi Tomita
ITA, São José dos Campos - SP - Brasil.
diegoferrarisc@gmail.com
jtakachi@ita.br

Abstract. *The design of radial turbomachines still depends on simplified equations from fluid mechanics (mass conservation, Navier-Stokes and energy conservation). The use of loss models improve the preliminary machine sizing and are very important mainly to determine the blade and flow angles at leading and trailing edges. But, to study the details of the flow in critical regions where there are complex flow phenomenon as recirculation, separation and tip leakage a sophisticated methodology should be applied. In this study, the Computational Fluid Dynamics (CFD) technique was applied to determine the operation characteristics of a turbocharger radial turbine operating at design-point. The 3D flow was evaluated from turbine volute inlet until the rotor outlet. The design-point condition results from CFD calculations are compared with experimental data and are in good agreement. The mesh generation, the numerical methods settings and the flow characteristics are presented and discussed.*

Keywords: Turbomachinery, Propulsion, CFD, Turbocharger, Gas Turbines, Radial turbine

1. INTRODUCTION

The turbomachines are designed using reduced order methods with the addition of appropriate loss modeling obtained from experimental data and correlations. Due to the simplified characteristic of these models, to study, analyze and improve some machine operation condition the 3D flow calculations become an interesting tool. Hence, the velocity triangles, pressure and temperature distributions can be compared between different design tools. The 3D effects can be determined and better understood using the set of equations formed by mass conservation, momentum and energy conservation.

The Computational Fluid Dynamics (CFD) is an important tool for turbomachines design process, allowing the analysis and optimization of the performance even in the design phase, greatly reducing the time and cost of project.

In this paper, it was performed a first study of the design characteristics of a existing radial turbine for use in turbochargers for diesel engines, based on 3D turbulent flow calculations. The turbine geometry and its experimental data were provided by a brazilian turbocharger company, Master Power. The mesh generation and the flow calculations were performed using a commercial software developed by ANSYS ICEM-CFD and CFX v.13, respectively.

2. OBJECTIVE

The main goal of this work is to compare the results between experimental data and numerical calculations based on 3D turbulent flow using CFD techniques. The geometry of a complete radial turbine configuration (volute and rotor) was supplied by Master Power including the experimental data. Adequate numerical issues and mesh generation process can be understood to obtain good numerical approaches. Hence, all details of work fluid and turbine operation characteristics should be detailed in the numerical solver to set the real problem and appropriate boundary conditions.

3. RADIAL TURBINE GEOMETRY TREATMENT AND ITS MESH GENERATION PROCESS

3.1 Radial Turbine Geometry

The first step to perform the flow calculation is the geometry generation and its treatment. Depending on the CAD file format, the geometry suffer several modifications in the curves and surfaces, mainly on its intersections. Some problems as surface holes, leading and trailing edges defects, intersections between surfaces must be solved using CAD software before the mesh generation. In this work, the turbine geometry was handle using the commercial software ICEM-CFD v. 13 developed by ANSYS.

Generally, complex geometries must be take care due to the high influence of geometrical details and mesh characteristics in CFD solution. Figure 1 shows the volute geometry, witch has no stator, and Fig. 2 shows the turbine rotor geometry.

Diego Silva de Carvalho, Jesuíno Takachi Tomita
3D Turbulent Flow Analysis In A Turbocharger Radial Turbine Operating At Design-Point
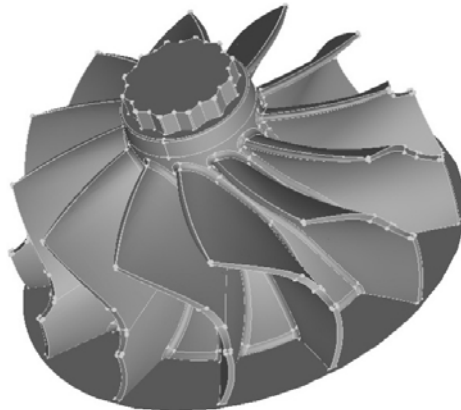

Figure 1. Geometry of studied turbine

Figure 2. Rotor

Several CAD work were performed aiming geometrical enhancements. In the most cases, this failures are related with surfaces disagreements, problems in the matching between curves and surfaces and surface holes. Figures 3 and 4 show examples of holes in volute surfaces. Due to the complexity of the surfaces and methods for their generation used in CAD software, it was necessary a laborious step of correction of such failures before mesh generation.
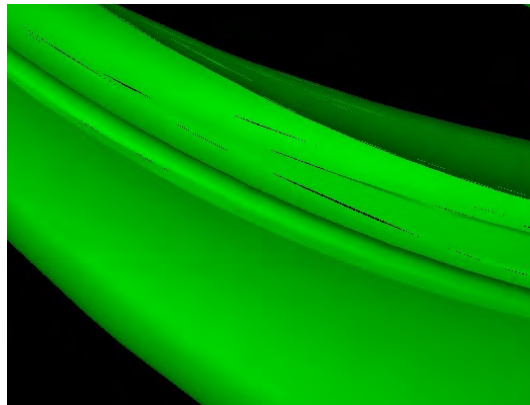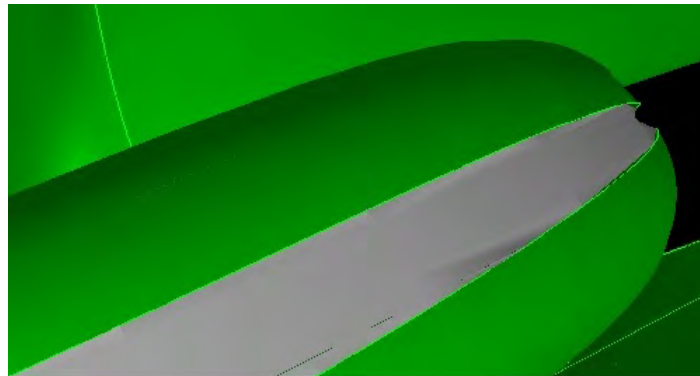

Figure 3. Holes on the surface

Figure 4. Disagreements in the boundaries between surfaces and curves

3.2 Mesh Generation

Despite the existence of different mesh types for turbomachines components (rotor, stator, volutes, return channel, ducts and so on), such as H and O types (generally for control volumes with hexahedral elements), it was used a tetrahedral elements for volute and rotor, which is better suited to complex geometries.

The mesh was carefully refined based on the curvatures of the surfaces and the distance of first control volume from the wall following the turbulence model requirements. The mesh quality, its smoothing and parameters to control a accepted level of control volumes edge angles were made based on theoretical recommendations (including ANSYS manuals).

The radial turbine was disassembled in two parts for mesh generation. The first part is the non-rotating frame of reference (volute) and the second the rotating frame of reference (rotor). This procedure is more appropriate for the current software used because became the process of mesh generation more flexible working in two different domains. For the rotor domain it is necessary to take into account the tip gap between rotor tip surface and its casing. This region needs a good mesh elements distribution to quantify the effects of leakage flow.

The mesh dependence was made starting with a mesh size around 2 million elements. A final mesh was chosen based on the results from CFD solver in which the solution is unchanged. The meshes size performed during the mesh independency study are shown in Tab 1:

Table 1. Mesh independency study.

Mesh	Rotor (mesh size)	Volute (mesh size)
1	1,2 million elements	0,8 million elements
2	2,5 million elements	2,2 million elements
3	5,6 million elements	2,8 million elements

The mesh chosen is the case 2 indicated in Tab. 1. Figures 5, 6, 7 and 8 show the mesh generated at volute inlet sectional area, volute surface, rotor domain and blade trailing edge mesh, respectively.
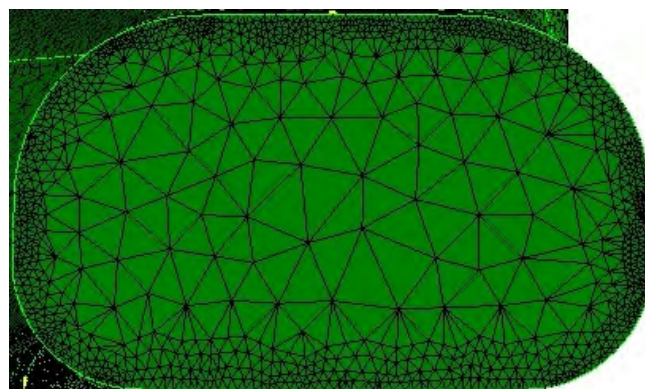

Figure 5. Refined mesh near the walls at the entrance of the volute

Figure 6. Volute domain mesh (2,2 millions of control volumes)
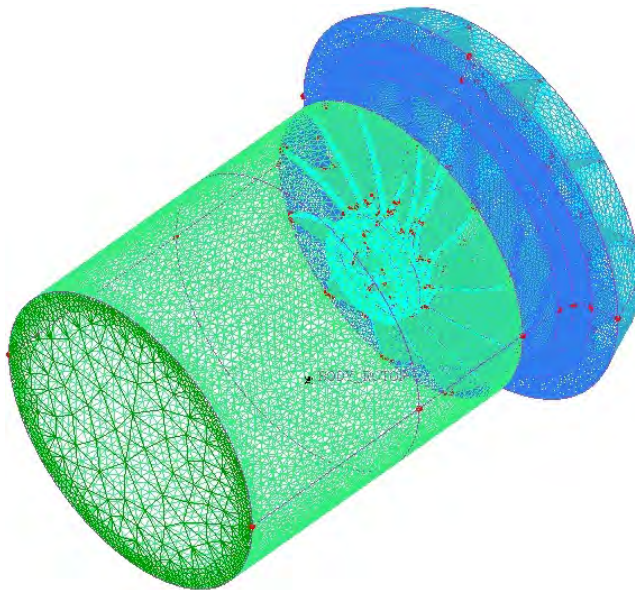

Figure 7. Rotor domain mesh (2,5 millions of control volumes)

Figure 8. Refined mesh at the trailing edge of the rotor

These meshes were exported to CFD solver, in which, for each surface an appropriate boundary condition should be set.

4. BOUNDARY CONDITIONS AND NUMERICAL ISSUES

To calculate the fluid mechanics equations numerically, it is necessary to set boundary conditions on the radial turbine frontiers. Note that, in this work only steady-state regime was calculated. It was set the following boundary conditions:

- In the rotor domain: operating speed (rotating-frame of reference)
- At volute inlet: P_t, T_t, turbulence intensity (5%) and velocity vector angles (normal to surface)
- At rotor outlet: Static pressure
- At inter reference frame surfaces: mixing plane approach
- Walls: no slip condition

The values of operating speed and the boundary conditions at the inlet and outlet correspond to the turbine design point operation.

4.1   Working fluid

It is important to use in the simulation the same working fluid used in experimental tests for the results comparison. The tests were performed using gas from combustion products of Compressed Natural Gas (CNG).

In the commercial CFD solver used in this work, it is necessary to create the fluid library, because there is no automatic selection of this fluid, thus, it was necessary to obtain its composition. According to Windawi, 1992, the composition of the exhaust gas from an engine burning natural gas (CNG) with the stoichiometric mixture is:

Table 2. Composition of the exhaust gas of a CNG

Component	%
HC	0,085
CO	0,25
H_2	0,25
NOx	0,14
O_2	0,325
CO_2	9,45
H_2O	15,0
N_2	74,5

The components CO2, H2O and N2 represent the dominant fraction of the exhaust gas of CNG. Thus, for reasons of simplicity, only they will be considered in the simulation, gas composition being:

Table 3. Gas composition for simulation in CFX

Component	CO_2	H_2O	N_2
%	9,45	15	75,55

It has been used a variable c_p variable (with temperature) according to a linear model in the range of working temperature (676-873 K), based on the values obtained in the tables in *engineeringtoolbox* website. The behavior of c_p in this temperature range is very close to linearity, being this model even more accurate than the models available in CFX (zero pressure polynomial and NASA format). For that were created functions in the language used by CFX, called CEL (CFX Expression Language) functions.

It was also used variable viscosity (with temperature), according to the Sutherland model:

$$\mu = \mu_{ref} \left(\frac{T}{T_{ref}} \right)^{3/2} \frac{T_{ref}+S}{T+S} \tag{1}$$

Where: T_{ref} is the reference temperature; μ_{ref} is the viscosity at the reference temperature T_{ref}; S is the Sutherland temperature (or constant) that is a characteristic of each gas. These coefficients were obtained in Federal Register (2005).

Due to unidentified problems in CFX solver, the simulations using the gas composition did not performed well and the numerical procedure diverged. To solve this problem, it was created a pure gas equivalent to the mixture. Thus, was necessary to implement CEL functions for calculating the properties of the mixture. Some parameters as molar mass (M), c_p, viscosity and thermal conductivity are necessary. The molar mass and thermal conductivity are calculated using a weighted by the mole fraction average of these properties of each component, and the c_p is calculated using a weighted by the mass fraction average. For the viscosity the rule is more involved. The complete model for the calculation of the viscosity of gas mixtures is presented in Brokaw, 1968. In this study, it was used a simplified version of this model, called Wilke correlation, found in *Chemical Professionals* website:

$$\mu_{mixture} = \sum_{i=1}^{n} \frac{y_i \mu_i}{\sum_{j=1}^{n} y_i \phi(i,j)} \tag{2}$$

Where: y_i – mole fraction of gas i in the mixture; μ_i – viscosity of pure gas i; $\phi(i,j)$ is determined by:

$$\phi(i,j) = 1 \; if \; i = j$$

$$\phi(i,j) = \frac{\left[1 + \sqrt{\frac{\mu_i}{\mu_j} \cdot \frac{M_j}{M_i}} \right]^2}{\sqrt{8 \left(1 + \frac{M_i}{M_j} \right)}} \; if \; i \neq j \tag{3}$$

Where: M_i and M_j are the molar mass of the components i and j, respectively.

The viscosities in the above equations were replaced by CEL functions to calculate the variable viscosity with temperature for each component according to the Sutherland model.

4.2   Numerical Issues

The simulation was performed for steady state regime. For the time step, the solver uses an implicit method, where the user can set the value of the time step or the solver calculates this value automatically. It was chosen the first option, with the step value equal to half of the period of revolution, defined from an extrapolation of the value found in the CFX manual, which recommends the time step equal to the period (half was chosen to guarantee numerical robustness).

Since the initial flow and fluid distributions along the radial turbine (pressure, temperature and velocities) are not known the initial condition to start the numerical iterations is not very appropriate. Thus, numerical instabilities can be found during the first iterations. To improve the numerical stability, for the first 100 iterations a more dissipative scheme was set for the discretization of convective terms from Navier-Stokes equations, in this case a first order discretization scheme based on a upwind method. After 100 iterations a second order upwind scheme was set. With this procedure it was possible to guarantee a good numerical stability.

The turbulence equations used is the two-equation SST (Shear Stress Transport) turbulence model as recommended by *Tomita et. al*. In CFX solver, this model turn on automatically the wall functions if the y^+ value is greater than 2. It is recommended by ANSYS, that the y^+ value not exceed 300. This value can be estimated during the mesh generation process based on the distance of the first control volume node (or centroid) from the wall.

The simulation was performed using 16 processors in an AMD Magny-Cours with 1.9GHz CPU and 192 Gb of RAM. The CPU time for the simulation was around 1h. The numerical convergence was monitored through the

conserved residues of the variables, shown in Figure 9, and the stabilization of the desired variables (inlet flow and total-to-static efficiency).
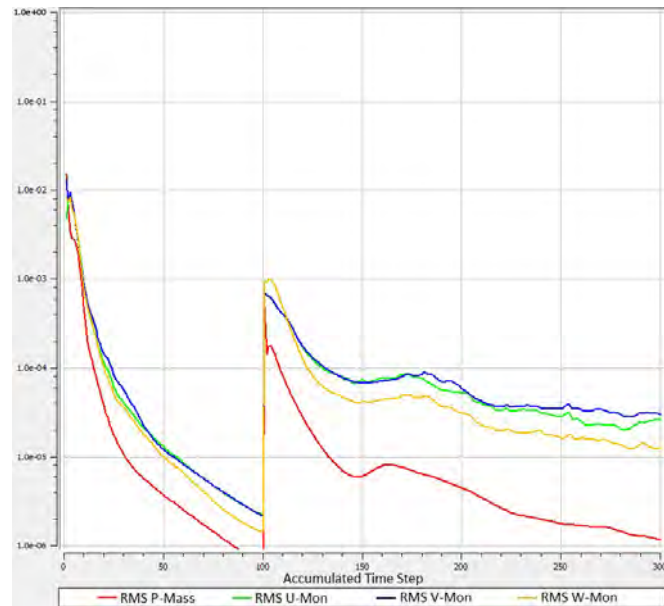

Figure 9. RMS residues monitor

5. RESULTS

The desired variables can be monitored during the solution. The values of the variables in the last iteration of the simulation are the solution for the given boundary conditions, and can be read directly from the monitor. The figures below show the monitors cited for the simulation of the turbine at the design point (set as previously described).
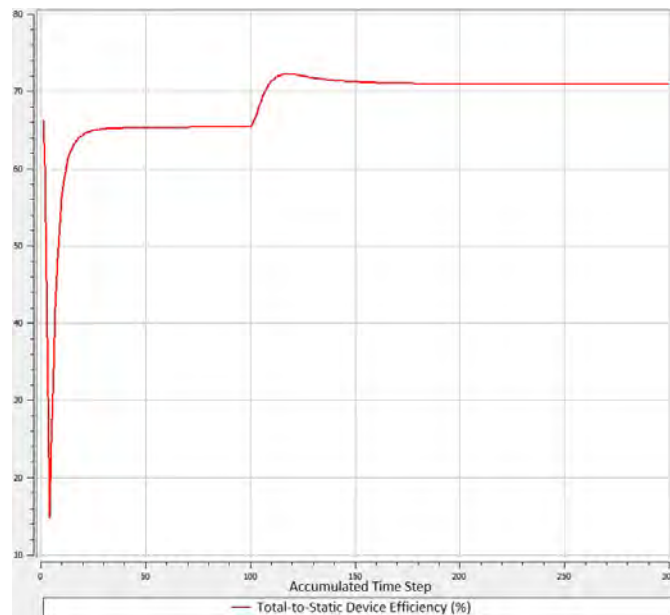
Figure 10. Efficiency monitor
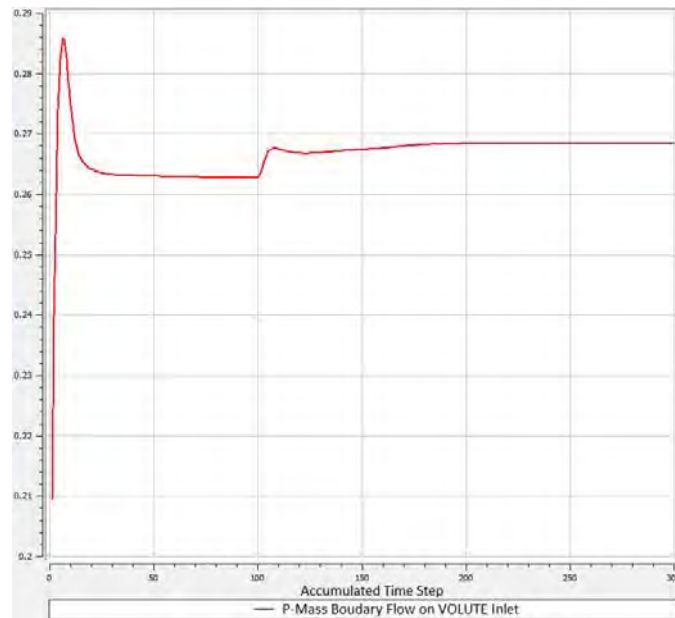
Figure 11. Mass flow monitor

To facilitate the visualization of flow phenomena, the post-processing tool CFX-post allows three-dimensional plot of the contours of fluid properties distributions. The Figures 12, 13, 14, 15 and 16 show some plots of total pressure, total temperature, relative Mach number, streamlines and the parameter y^+, respectively.
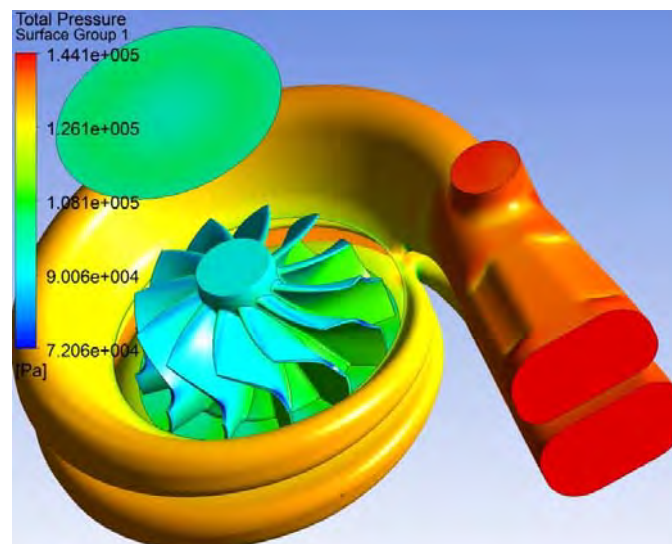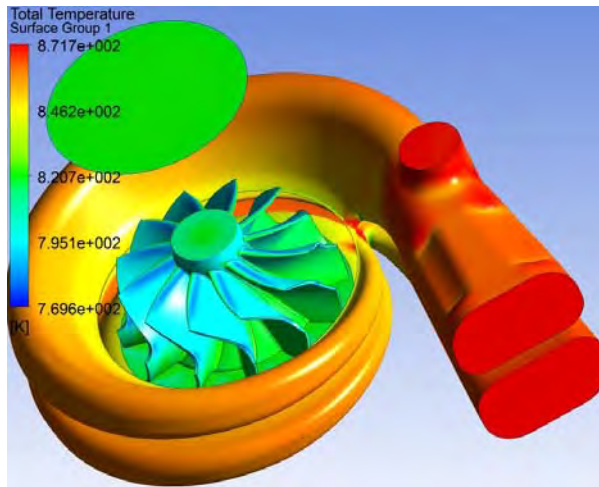

Figure 12. Total pressure contour
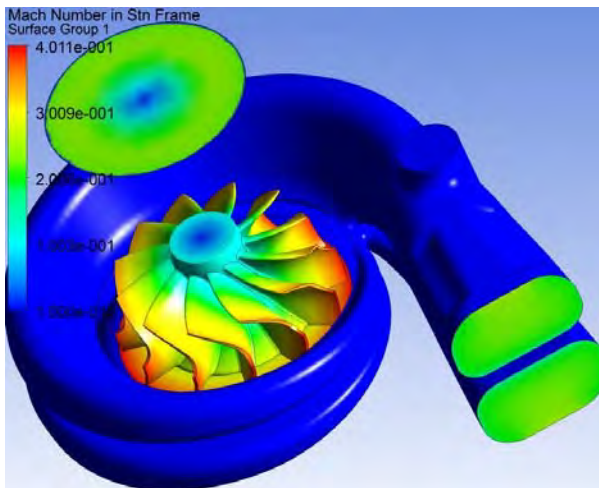
Figure 13. Total temperature contour

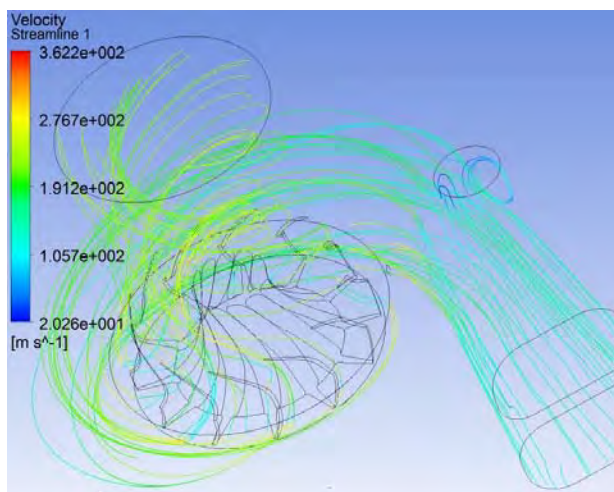Figure 14. Relative Mach number contour

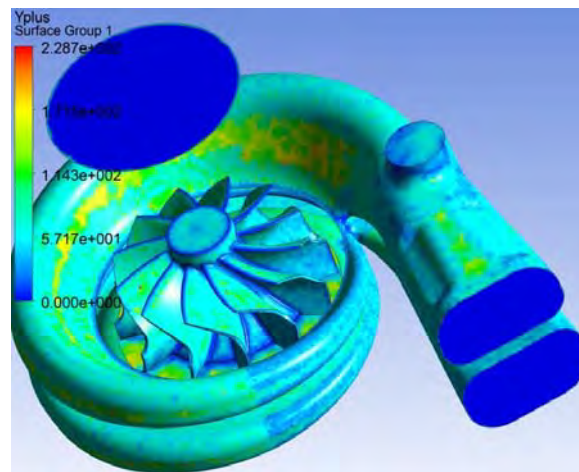Figure 15. Streamlines along the radial turbine

Figure 16. y^+ parameter field

The y^+ parameter has values within the recommended in ANSYS CFX v.13.0 manual. With the value of mass flow obtained in the simulation it is calculated the Mass Flow Parameter $\left(\text{MFP} = \frac{\dot{m}\sqrt{T_{to}}}{P_{to}}\right)$. This value and the total-to-static efficiency obtained in the simulation are compared with the corresponding results of experimental tests. The Tab. 4 below shows these values.

Table 4. Results of CFD and experimental data on for turbine design-point operation

N_corr_med	P_{t0}/P_4	MFP CFX	MFP exp	$\eta_{s\text{-}t}$ CFX	$\eta_{s\text{-}t}*\eta_m$ exp
rpm/√K	-	kg/s*√K/bar	kg/s*√K/bar	%	%
3135	2,35	3,461	3,697	71,002	68,655

It is noted a relative deviation of CFD results compared with experimental data of 6.4% below for MFP and 3.4% above for efficiency. A small portion of this shift in efficiency can be attributed to the fact that the experimental efficiency include the mechanical efficiency, that is, indeed it is $\eta_{s-t}*\eta_m$. But, as the mechanical efficiencies of gas turbines tend to be very high, the actual deviation is still considered slightly high.

6. DISCUSSION AND CONCLUSION

In this work a procedure to perform 3D turbulent flow calculations in radial turbines based on CFD techniques was described, tested, and commented. Moreover difficulties in complex geometry handling and its mesh generation were commented. The pre-processing step requires several hours, sometimes days, of work to reach a good geometry resolution.

To certify and validate the procedure used, the results were compared with experimental data at turbine design point operation. The boundary conditions, numerical schemes and turbulence model used were described in order to provide information of interest to the academic community that performs research in turbomachines. The difficulties of starting the numerical iterations were commented and an option to become the numerical procedure more robust are described and performed well in the case of radial turbine.

The results from experimental data and numerical solution are in good agreement. Deviations around 5% between the two results were detected. The radial turbine manufacturer Master Power also reported that obtained efficiency values greater than experimental values in CFD simulations. These deviations may have different causes and can be studied in more detail. An important point to note is that the position of sensors and probes installed during the test of radial turbine can be different from the positions where the boundary conditions were imposed. Other aspect is the mode in which the efficiency were calculated based on mass averaging. Maybe this average is not adequate and should be explored with more details.

The mesh used in this work is relatively well refined, but other mesh types can be studied, for example the use of a full hexahedral mesh or the inclusion of prismatic control volumes close to wall surfaces with tetrahedrons far from walls. These options may improve the boundary layer calculation. Other option is to generate a very refined mesh close to the wall to avoid the use of wall functions by CFD solver. The use of different turbulence models can be also

explored. To obtain the turbine performance map several operation points can be performed using CFD tool with the procedure above described.

This work and the adopted procedures can also be extended for radial turbines used in aerospace technology in the area of liquid propulsion based on small turbopumps. Optimization tools can be coupled with CFD solver aiming high performance machine design.

7. ACKNOWLEDGEMENTS

8. REFERENCES

BROKAW, R. S, 1968. Viscosity of Gas Mixtures, Lewis Research Center, Cleveland, Ohio, USA: NASA Technical Note.

Carbon Dioxide – Specific Heat. 05 Nov. 2012 <http://www.engineeringtoolbox.com/carbon-dioxide-d_974.html>.

Chemical Professionals, Viscosity of Gaseous Mixtures. 06 Nov. 2012 <http://profmaster.blogspot.com.br/2009/06/viscosity-of-gaseous-mixtures.html>.

Federal Register/ Vol. 70, No. 133/ Wednesday, July 13, 2005/ Rules and Regulations. Procedures for Testing Highway and Nonroad Engines and Omnibus Technical Amendments, Pages 40569-40614, USA. 05 Nov. 2012 <http://www.epa.gov/fedrgstr/EPA-AIR/2005/July/Day-13/a11534d.pdf>.

Nitrogen – Specific Heat. 05 Nov. 2012 <http://www.engineeringtoolbox.com/nitrogen-d_977.html>.

Tomita, J.T.; Silva, L.M. Silva; D.T. Comparison between unstructured and structured meshes with different turbulence models for a high pressure turbine application. ASME TurboExpo 2012, GT2012-69990, Copenhagen, Dinamarca.

Water Vapor – Specific Heat. 05 Nov. 2012 <http://www.engineeringtoolbox.com/water-vapor-d_979.html>.

Windawi, H., 1992. Controlling the Exhaust Emissions from Alternative Fuel Vehicles, New Opportunity for Platinum Metals, Johnson Matthey Catalytic Systems Division, Wayne, Pennsylvania, USA. 30 Oct. 2012 <www.platinummetalsreview.com>.

9. RESPONSIBILITY NOTICE

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