2ème conférence Internationale des énergies renouvelables CIER-2014
Proceedings of Engineering and Technology - PET

ANALYSIS OF OPERATION OF A MIXED FLOW TURBINE TO 10 BLADES FOR DIFFERENT ROTATIONAL REGIMES

O.Z.Khelloufi [#1], M. Hamel [*2], M.Abidat [#3]

Laboratoire de mécanique appliquée, faculté de génie mécanique,
université des sciences et de la technologie d'Oran – Mohamed-Boudiaf,
BP 1505, El M'Naouer,31000, Oran, Algeria.
[1]omar_zine_gm@yahoo.fr
[2] hamel_moh78@yahoo.fr

Abstract **—Turbochargers are widely used in Diesel engines as a means of increasing the output power. Most of them are fitted with radial or mixed flow turbines. This study is to determine the performance of a semi-axial turbine to 10 blades, this work was performed with ANSYS CFX-software that permits to obtain the performance of the semi-axial turbine for a wide range of speed of rotation and pressure ratios. The results obtained are in good agreement with experimental data.**

Keywords— **Mixed flow turbine, turbocharger, performance.**

I. INTRODUCTION

Radial turbines or mixed flow turbine are rotary machines for the production of mechanical energy in the small and medium sized area that offer very interesting characteristic; by their high performance and small footprint; they have been used for a long time in a variety of applications such as energy production of small and medium sized installations, in the processes of recovery of solar energy, or as a regulator in the natural gas transmission networks where they allow to recover the energy released during the trigger; However, overeating internal combustion engines is the area where the radial turbines or semi generate the most interest. This study presents a numerical performance prediction of the mixed flow turbine to 10 blades, performed with the ICEM and CFD softwares of ANSYS-11 and compare the numerical results obtained with experimental results. The mixed flow turbine already exists and forms part of a range of supercharging turbines, its study and design have summers made by Abidat (1991) [2] at Imperial College, it has been tested by several researchers, among which may be mentioned, Chen and al (1992) [3], Abidat and al (1992 and 1998) [4] et [5], Arcoumanis and al (1995) [6], Chen and al. (1996) [7], Martinez and al (1996) [1], Karamanis and al (2001) [8] and Abidat and al (2005) [9]. The study done on this subject is in comprehension and numerical prediction of the mixed flow turbine performance for a wide range of speeds and pressure ratios. The overall geometric characteristics of the turbine is given by Abidat (1991)[2] and Abidat and al. (1992 and 1998)[4].

The turbine was designed to perform the following design requirements:

Rotational speed : 98000 tr/mn
mass flow rate: 0.414 kg/s
The total temperature the entry: 923 k
The total pressure the entry: 2.91bar

II. TURBULENT FLOW MODEL

The flow is three-dimensional, compressible, viscous and turbulent in the mixed flow turbine to 10 blades for different operating regimes, is determined using the resolution of Reynolds averaged equations, the Turbulence is modeled by the two-equation model k-ε which is based on the concept of the eddy viscosity. In this model the turbulent viscosity μ_t is calculated from the turbulent kinetic energy k and its dissipation rate ε by the following equation:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \qquad (1)$$

with C_μ =0.09 and k and ε are calculated using their transport equations.

III. MESH GENERATION

The turbine geometry obtained with the ICEM CFD software, first of all, a data file that contains the Cartesian coordinates of the points required to create the geometry is imported, These points are used to create lines and then the surface. As part from our work we have used an unstructured hexahedral mesh shown in Figure 1

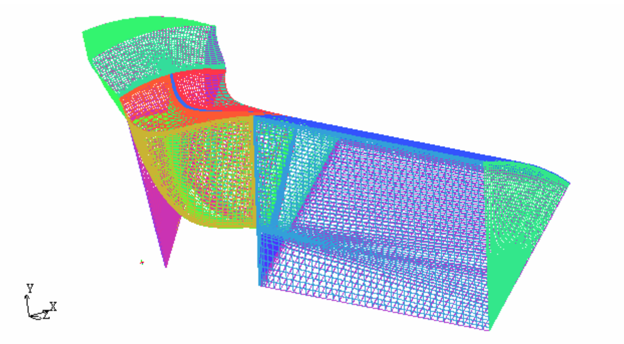

Fig. 1 Computational grid of the Mixed Flow passage

IV. NUMERICAL METHOD

The computational domain is defined using the Module CFX-Pre of the calculation code CFX-11; first of all, one defines the fluid which is an ideal gas; the regime is stationary, the turbulence model is k-ε, it remains to determine the boundary conditions at each section surrounding the computational domain, thus, for all the simulations, the boundary conditions are the following :

entry : total pressure, total temperature (333.82 k) for the two rotational speeds 29500 and 41300 tr/mn and (341.79 k) for 59700 tr/mn, the turbulence intensity (5%) and the regime is subsonic.

outlet : the regime is subsonic and relative pressure is (1 atm)

walls : adiabatic

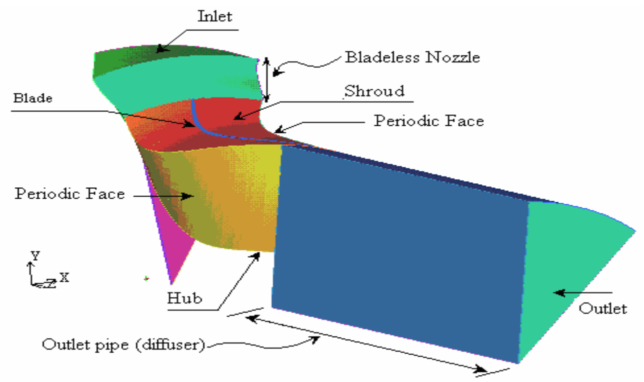

Fig.2 computation domain

V. CALCULATION PROCEDURES

In order to study the performance of a mixed flow turbine at 10 blades a numerical simulation was conducted of the steady flow for different rotational speeds and a large pressure ratios.

100% : corresponds to the maximum rotational speed (59700 tr/mn) ;

70% : corresponds to the maximum rotational speed (41300 tr/mn);

50% : corresponds to the maximum rotational speed (29500 tr/mn) ;

VI. RESULTS AND DISCUSSION

The figures show the influence of the rotational speed on the performance characteristics of the mixed flow turbine at 10 blades.

The figures (3,4) shows a comparison of the numerical results with experimental data. It is observed that the curves of the pressure ratio, the torque versus of the reduced mass flow rate calculated using the following equation are in agreement with the measured results.

$$m_r = 10^5 * \dot{m} * \sqrt{T_{in^*}}/P_{in^*} \qquad (2)$$

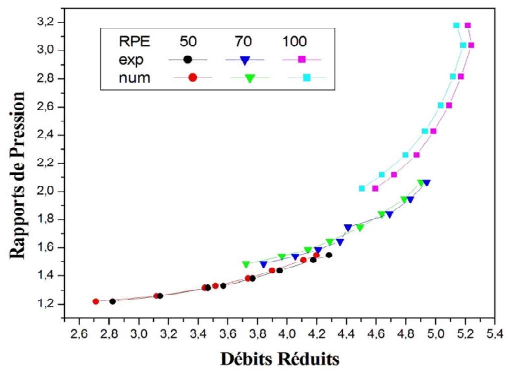

Fig.3 : Pressure ratio vs. reduced mass flow

rate

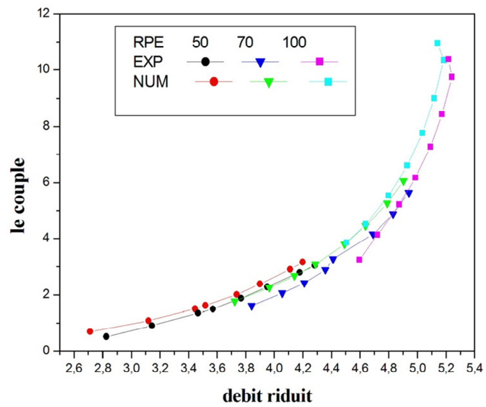

Fig. 4 : Torque vs. reduced mass flow rate

Figure (5) shows variation of the performance versus the reduced mass flow rate $\mathbf{m_r}$, one realizes that the performance grows intensely and in proportion to the reduced mass flow rate, until a maximum value. This value is considered as an optimal operating point for all flow parameters, It was observed that the total to static efficiency has reached a value 0.823 at 100% of the rotational speed to a reduced mass flow rate of 5,039.

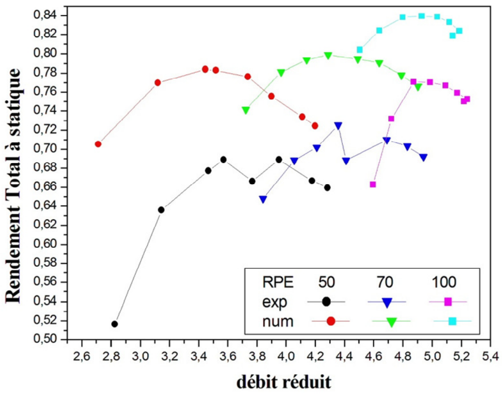

Fig. 5 : Measured total-to-static efficiency as a function of mass flow rate

The characteristics of the total to static efficiency calculated using the following equations are shown in the figure (6).

$$\eta_{ts} = (h_{0s} - h_4)/cp\, T_{0s}[1 - (P_4/P_0)]^{(\gamma-1)/\gamma} \qquad (3)$$

The calculated efficiency is slightly underrated compared to experimental. Although a reasonable agreement is obtained at each speed for high pressure ratios.

It is clear that the maximum efficiency depends on the rotational speed and it appears clearly in Figure (5).

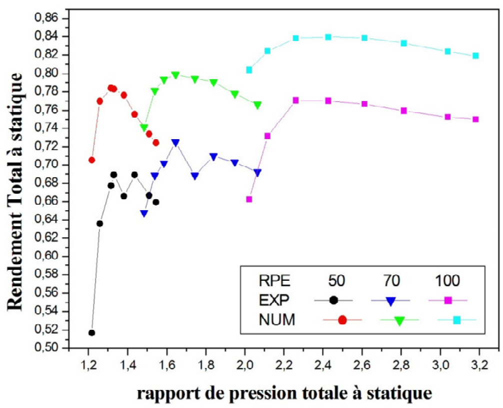

Fig. 6 : Rendement Total à statique en fonction du rapport de pression totale à statique

Figure (7) represents the flow angle of relative movement at the entry of the rotor and the total to static efficiency depending pressure ratio for know the optimal angle for each speed.

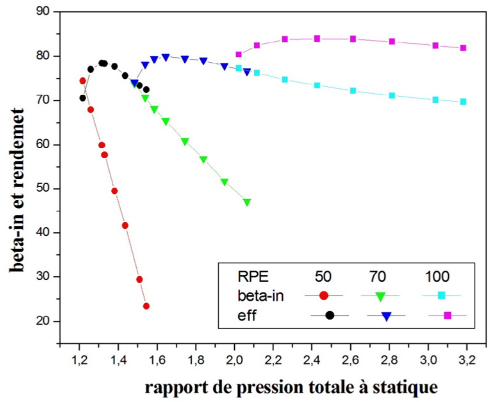

Fig. 7 the movement relative flow angle to the entry of rotor and the efficiency as a function of Pressure ratio

VII. Conclusion

This work aims to study the influence of the performance of a mixed flow turbine using the code ANSYS CFX.11 and the approach model k-Ɛ.

From the results obtained, it was shown that the structure of the flow may be disturbed by two parameters, the rotational speed and the pressure ratio.

For high speeds and at low pressure ratios, recirculation tends to weaken and disappear completely, this is observed in 100% of the rotational speed is 59700 rpm / min.

References

[1] M. Abidat, M. Hachemi, M.K. Hamidou and N.C. Baines " Prediction of the steady and non steady flow performance of a highly loaded mixed flow turbine", I.Mech . Eng. Journal of Power and Energy, Volume 212 Part A, 1998.

[2] Abidat, M. Design and testing of a Highly Loaded Mixed Flow Turbine", PhD Thesis, Imperial College, London,1991.

[3] Chen, H., Abidat, M., Baines, N. C. & Firth, M. R " The effect of blade loading in radial and mixed flow turbines", ASME, Paper 92-GT-92, 1992.

[4] Abidat, M., Chen, H., Baines N.C & Firth, M.R. " Design of a highly loaded mixed flow turbine", Proc, IMechE, Journ. Power and Energy, Vol 206, 1992.

[5] Martinez, R.F Botas, K.R Pullen and F Shi. " Numerical calculations of a turbine volute using 3-D Navier Stokes solve", ASME 96-GT-66,1996.

[6] Arcoumanis, C., Hakeem, I., Martinez-Botas, R.F., Khezzar, L. & Baines, N.C. " Performance of a mixed flow turbocharger

turbine under pulsating flow conditions", ASME, Paper 95-GT-210, 1995.

[7] Chen, H., Hakeem, I. & Martinez-Botas, R. F. " Modeling of a turbocharger turbine under pulsating inlet conditions", Proc, IMechE, Journ. Power and Energy, Vol 210, 1996.

[8] N. Karamanis, R F. Martinez, Botas and C C.Su. " Inlet and Exit Flow Under Steady and Pulsating Conditions", ASME . Journal of Turbomachinery, 2001.
M. Abidat And M. Hachemi " Off design performance analyses of a turbocharger mixed flow turbine" 6th Conference on Turbomachinery Fluid Dynamics and Thermodynamics. 7-11 March, Lille, France, 2005.