AERODYNAMIC GEOMETRY OPTIMIZATION OF A CENTRIFUGAL BLOWER

Bogdan GHERMAN[1], Cristina SILIVESTRU[21], Marian DRAGHICI[31]

In this paper is presented a study of a new centrifugal blower where the mass flow variation is controlled by mobile ante rotor and not by speed variation of the electrical motor. A 3-D analysis of the centrifugal blower is involved, where is studied the interaction between rotor, stator and scroll, with a special attention on losses done by each component. Also an optimization of the stator geometry is also done to maximize the efficiency of the blower. It is known that the wall functions cannot accurately predict the separation and reattachment regions. For this reason, the fine mesh of impeller has the averaged y+ nearly 1, to properly capture the separation flows near leading edge at off-design regimes.

Keywords: centrifugal compressor, scroll, CFD, optimization

1. Introduction

Centrifugal compressors due to their compact size, large capacity, high performance and the ability to run on a wider operating range than an axial compressor makes them a solution for residual water treatment plants. A centrifugal blower is often used in residual water treatment facilities. Their role is to deliver oxygen in the water tanks for the development of biological flock which substantially removes the organic material from fluid.

A centrifugal compressor can be divided in four major parts, the inlet guide vanes, the impeller, the diffuser and the volute. And each of these components can improve performances of the centrifugal compressor. For example to extent the operating range of a centrifugal compressor one way is to use variable inlet guide vanes [1]. Also improving the design of the diffuser will reduce the losses and improve compressor performances significantly [2-4]. But when we talk about the scroll (volute), in spite of its importance, there are very few detailed studies that deal with the scroll aerodynamics due to highly three-dimensional flow and its complex geometry, which makes the obtainment of a high quality mesh vital for a good aerodynamic analysis a difficult task. The CFD

[1] Eng., COMOTI - National Research & Development Institute for Gas Turbines, Bucharest, Romania bogdan.gherman@comoti.ro
[2] Eng., COMOTI - National Research & Development Institute for Gas Turbines, Bucharest, Romania
[3] Eng., COMOTI - National Research & Development Institute for Gas Turbines, Bucharest, Romania

analysis helped the understanding the flow inside a volute, which lead to the development of high performance centrifugal blowers. The studies carried out so far in this area can be split in two major categories: volute flow analysis and analysis of the impeller – volute interaction [5].

In this paper, in order to obtain realistic data, the impeller aerodynamics was investigated carefully. The interaction between rotor-stator and scroll was performed in two stages. First rotor-stator interaction to be able to see the influence they have on each other without the interference of the scroll, and second was to perform a analysis concerning the influence of the scroll on the rotor and stator.

2. Numerical Methods

For this study, the flow was assumed compressible, the equations that govern the flow, written in Reynolds averaged form, time and mass averaged, being, in the repeated indices summation convention:

1. The Continuity Equation:

$$\frac{\partial \overline{\rho}}{\partial t}+\frac{\partial \overline{\rho}\tilde{u}_j}{\partial x_j}=0 \tag{1}$$

2. The Momentum Equations:

$$\frac{\partial \overline{\rho}\tilde{u}_i}{\partial t}+\frac{\partial \overline{\rho}(\tilde{u}_i\tilde{u}_j)}{\partial x_j}=-\frac{\partial \overline{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\overline{\tau_{ij}}-\overline{\rho u_i' u_j'}\right) \tag{2}$$

where

$$\overline{\tau_{ij}}=\mu\left[\frac{\partial \tilde{u}_i}{\partial x_j}+\frac{\partial \tilde{u}_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\left(\frac{\partial \tilde{u}_k}{\partial x_k}\right)\right],$$

represents the stress tensor.

3. The Total Energy Equation:

$$\frac{\partial}{\partial t}\left(\overline{\rho}\tilde{h}\right)+\frac{\partial\left(\overline{\rho}\tilde{u}_j\tilde{h}\right)}{\partial x_j}=\frac{\partial \overline{p}}{\partial t}+\frac{\partial}{\partial x_j}\left(\overline{\frac{\mu}{\Pr}\frac{\partial h}{\partial x_j}}\right)+\frac{\partial}{\partial x_j}\left(-\overline{\rho h' u_j'}\right), \tag{3}$$

where h is the enthalpy.

4. Ideal Gas Equation of State:

$$\tilde{\rho}=\frac{w\left(\tilde{p}+p_{ref}\right)}{R_0\tilde{T}} \tag{4}$$

where w is the molecular weight.

The flow is assumed fully turbulent and it has been simulated in ANSYS CFX 11.0 with the shear-stress transport (SST) k-ω based model, which was developed by Menter [6] in order to combine the advantages of the robust and accurate formulation of the Wilcox k-ω model in the near-wall region with the free-stream independence of the k-ε model in the far field. To achieve this, the k-ε model is converted into a k-ω formulation. With this turbulence model, it is possible to obtain precise results and information regarding dimension of the separation area that forms under the influences of high pressure gradients, Coriolis and centrifugal forces.

The equations are discretized using a second order upwind scheme. This structured grid size allows us to have y+ of order 1 at the blade and the volute walls.

A structured grid has been used for the volute and rotor, as in figure 1.

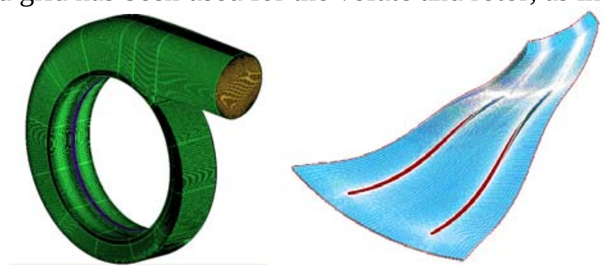
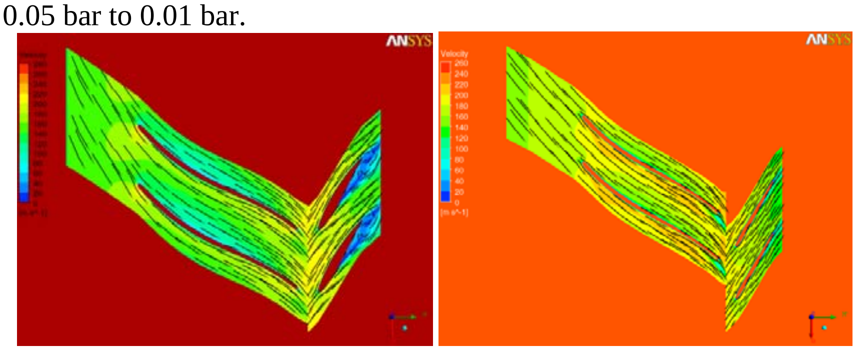


Fig. 1. Structured grid: volute (left frame) and rotor (right frame)

3. Results

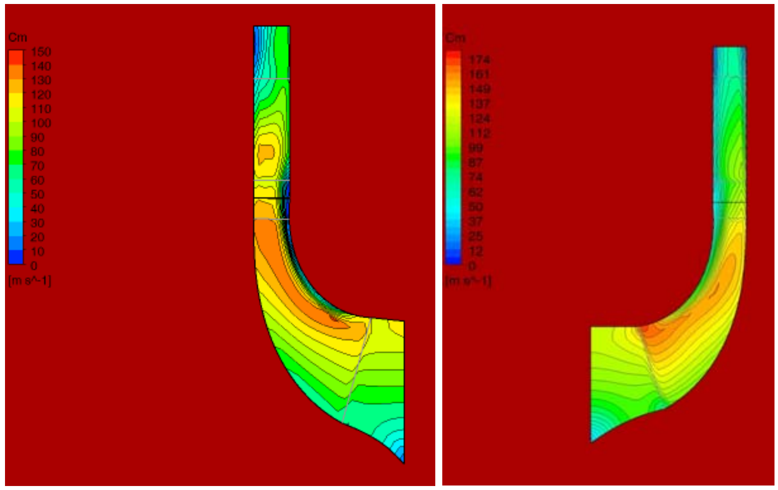
The analysis performed using ANSYS CFX clearly shows the recirculation zone present on stator part (Fig. 2a). Also after optimization is carried out, it is possible to see that the recirculation zone disappeared (Fig. 2b).

Removing the recirculation zone from the stator blade improves the performance of the machine as it can be seen in Figure 3. The pressure ratio rises with 0.04 bar in the optimized geometry case reducing the losses on the stator from 0.05 bar to 0.01 bar.



(a) (b)

Fig. 2. Velocity field at 0.5 from the blade tip: a) old geometry; b) optimized geometry



(a) (b)

Fig. 3. Velocity in meridional plane: old stator (a) and optimized stator (b)

It can be seen also the irregularities of the flow in the meridional plane, especially in the old stator (Fig. 3a) where it appears a zone, close to the leading edge where the flow accelerates. In the optimized stator the flow is more uniform (Fig. 3b). Also this affects the performance of the machine; for instance, efficiency increases from 0.86 to 0.92 in the case with the optimized geometry.

In fig. 4 it can be seen the necessity to run the calculation on the entire stator and rotor coupled with the volute. Accordingly to this figure the stator part is influenced differently depending on the position of the blade.

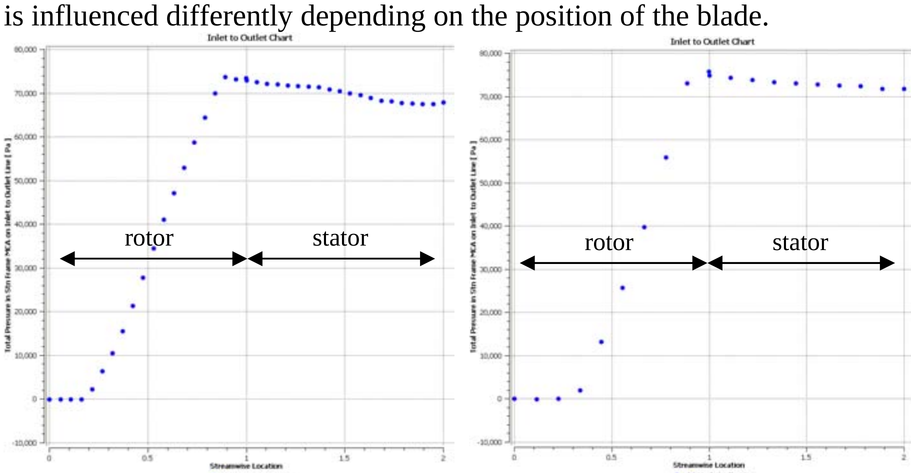


Fig. 4. Total pressure in the rotor + stator

Also inside the volute (scroll) are formed two vortices that grow as volute area increases until the exit (Fig. 5).

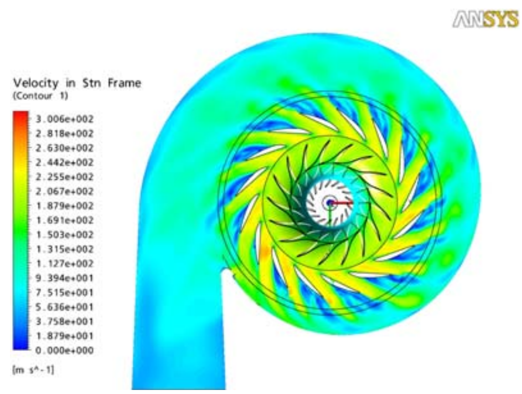
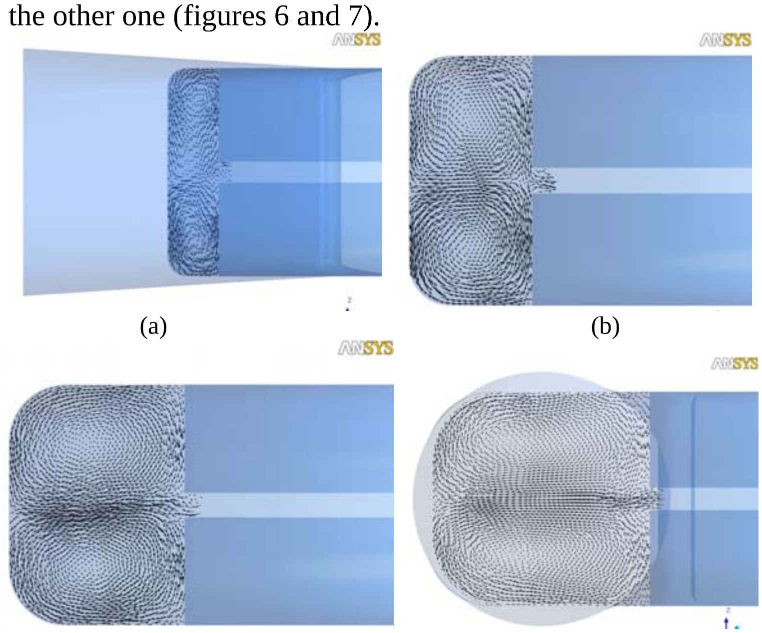


Fig. 5. Velocity field in rotor+stator+volute domain (old stator)

At the scroll exit one of the vortices becomes stronger and tends to annihilate the other one (figures 6 and 7).



(a) (b)

(c) (d)

Fig. 6. Vector field inside volute at: (a) 0^o, (b) 90^o, (c)180^o and (d) 360^o (with optimized stator)

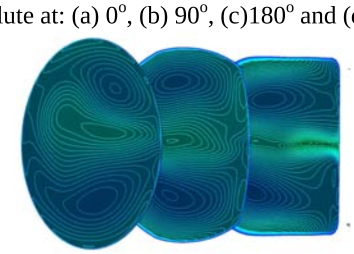


Fig. 7. Velocity field at scroll exit

In the volute the losses reach 0.05 bar of the total pressure. The power necessary to run the blower is 187 kW.

4. Conclusions

During this study were identified loses on each component of a centrifugal blower in terms of total pressure variation. Also due to a recirculation zone identified on the stator part of the blower, this part has been optimized from the aerodynamic point of view.

The performances of the centrifugal stage have been improved; the efficiency of the compressor stage changed from 0.86 to 0.92, also the total pressure loss was reduced from 0.05 bar to 0.01 bar on the stator part.

R E F E R E N C E S

[1] *H. Mohtar, P. Chesse, A. Yammine and J.F. Hetet,* "Variable inlet guide vanes in a turbocharger centrifugal compressor: local and global study", SAE paper 01-0301, 2008.

[2] *A. Engeda*, "Experimental and numerical investigation of the performance of a 240-kW centrifugal compressor with different diffusers", Exp Therm Fluid Sci, **vol. 28**, 2003, pp. 55–72.

[3] *H. Uchida,* "Transient performance prediction for turbocharging systems incorporating variable-geometry turbochargers", R&D Rev Toyota CRDL, **vol. 41**(3), 2006, pp. 22–8.

[4] *R. C. Griffith,* "Series turbocharging for the caterpillar heavy-duty on-highway truck engines with ACERT technology", SAE paper 01-1561, 2007.

[5]. *K. Jiao, H. Sun, X. Li, H. Wua, E. Krivitzky, T. Schram and L. M. Larosiliere, "*Numerical simulation of air flow through turbocharger compressors with dual volute design*",* Appl Energy, doi:10.1016/j.apenergy.2009.02.019, 2009.

[6] *F. R. Menter*, "Two-equation eddy-viscosity turbulence models for engineering", AIAA - Journal, **vol. 32**, no 8, August 1994.

[7] *B. Gherman, M. Niculescu, G. Vizitiu and V. Silivestru*, "A CFD analysis of a centrifugal stage performances**",** Computer Aided Engineering Solutions for Design, Analysis and Innovation Conference, Sinaia, Romania, 2007.

[8] *C. Xu and M. Muller***, "**Development and Design of a Centrifugal Compressor Volute", International Journal of Rotating Machinery, **vol. 3**, 2005, p. 190.

[9] *H. Pitkanen, H. Esa, A. Reunanen, P. Sallinen and J. Larjola*, "Computational and Experimental Study of an Industrial Centrifugal Compressor Volute", J. of Thermal Science, **vol. 9**, no. 1, 2000.

[10] *K. Jiao et al.*, "Numerical simulation of air flow through turbocharger compressors with dual volute design", Appl Energy, doi:10.1016/j.apenergy.2009.02.019, 2009.

[11] *C. Hirsch*, Numerical Computation of Internal and External Flow, Volume 2: Computational Methods for Inviscid and Viscous Flows, John Wiley and Sons, New York, 1990.

[12] *S. Dănăilă and C. Berbente*, Numerical Methods in Fluid Dynamics (in Romanian), Publishing House of Romanian Academy, 2003.

[13] *M. Ionescu, I. Dragason, M. Niculescu and B. Gherman*, "Verificarea si optimizarea aerodinamica a treptei de comprimare, suflanta TEOFIT si calcul de rezistenta a rotorului centrifugal" (in Romanian), Contract 21, B2/2, 5427/2, April 2008.