The Design, Performance and CFD Analyses of Regenerative Blower used for Fuel Cell System

Chan Lee and Hyun Gwon Kil
Department of Mechanical Engineering, University of Suwon, Hwaseong, Korea

Keywords: Regenerative Blower, Design, Performance, CFD.

Abstract: For efficient design of regenerative blower used for fuel cell system, the design and the performance analysis methods of regenerative blower are developed, and CFD modelling and simulation are carried out on the designed blower. The design process of regenerative blower is conducted to determine the geometries of rotating impellers and stationary side channel with several design variables. The performance analysis on the designed blower is made by incorporating momentum exchange theory between impellers and side channel with mean line analysis method, and its pressure loss and leakage flow models are constructed from related fluid mechanics data and correlations which can be expressed in terms of blower design variables. The internal flow field of blower is analysed by using the CFX code, a CFD code specialized for fluid machinery. The present performance analysis method is applied to four existing models for verifying its prediction accuracy, and the comparison between the prediction and the test results are well-agreed with a few percentage of relative error. Furthermore, the present design and performance analysis methods are also applied in developing a new blower used for fuel cell application, and the newly designed blower is manufactured and tested through chamber-type test facility. The performance prediction by the present method is well-agreed with the test and the CFD simulation results. Therefore, from the comparison results, the prediction design and performance analysis methods are shown to be suitable for the actual design practice of regenerative blower.

1 INTRODUCTION

Regenerative blowers are usually operated with high pressure rise at low flow capacity, so widely used for air/ hydrogen supply in fuel cell applications. However, because regenerative blowers are operating with low efficiency or a lot of pressure loss (Badami and Mura, 2012), there are growing industrial needs for high-efficiency regenerative blower development. Since the pressure loss is strongly dependent on the internal flow phenomena of regenerative blower, for developing high-efficiency blower, reliable design method with accurate performance analysis model considering the flow effects should be developed and applied to actual design practice of blower industries.

The early theoretical researches on regenerative blower and pump have been conducted to investigate the flow pattern and the energy transfer mechanism of fluid inside the machines, and showed that the energy transfer to fluid is achieved by the momentum exchange of the helical-torodal fluid motion between rotating impeller and fixed side channel of regenerative machine (Wilson et al., 1955: Hollenberg and Porter, 1979). Recent researches by Badami and Mura have been devoted to improving the performance analysis method of regenerative blower by using momentum exchange theory, and the prediction results have been compared and well-agreed with test and 3-D CFD results. However, since their analysis model requires model constants which user should specify, it needs the generalization of the constants in terms of blower design and operation parameters (Badami and Mura, 2011: Badami and Mura, 2012). Lee et al. proposed an analysis method for regenerative blower performance by using momentum exchange theory, and tried to generalize their model constants from relevant regenerative blower and fluid mechanics experimental results (Lee et al., 2013).

In the present study, a simple but reliable design-analysis method of regenerative blower is developed as in-house program called as the FANDAS-Regen code. Regenerative blower performance is predicted

Lee C. and Gwon Kil H..
The Design, Performance and CFD Analyses of Regenerative Blower used for Fuel Cell System.
DOI: 10.5220/0005104007510755
In *Proceedings of the 4th International Conference on Simulation and Modeling Methodologies, Technologies and Applications* (SIMULTECH-2014), pages 751-755
ISBN: 978-989-758-038-3

by incorporating mean line analysis method with the momentum exchange theory between rotating impeller blades and fixed side channel of blower. The performance prediction accuracy of the present method is verified by comparing the prediction with the measurement results of several actual regenerative blowers, and the comparison results show the present method is capable of predicting blower pressure, efficiency and power very accurately.

Furthermore, with the use of the present design-analysis method, a new regenerative blower is designed, manufactured and tested by using chamber-type test facility, and its internal flow field is analyzed by the CFX code, a CFD code specialized for fluid machinery. The comparison between the present performance prediction, the CFD simulation and the test results are well-agreed within a few percentage of relative error, and they also show that the present design-analysis method is very suitable for the actual design practice of regenerative blower.

2 DESIGN AND ANALYSIS

2.1 Blower Design Method

In general, regenerative blower is composed of rotating impellers and fixed side channel and its geometry is shown (Badami and Mura, 2011) in Fig.1.

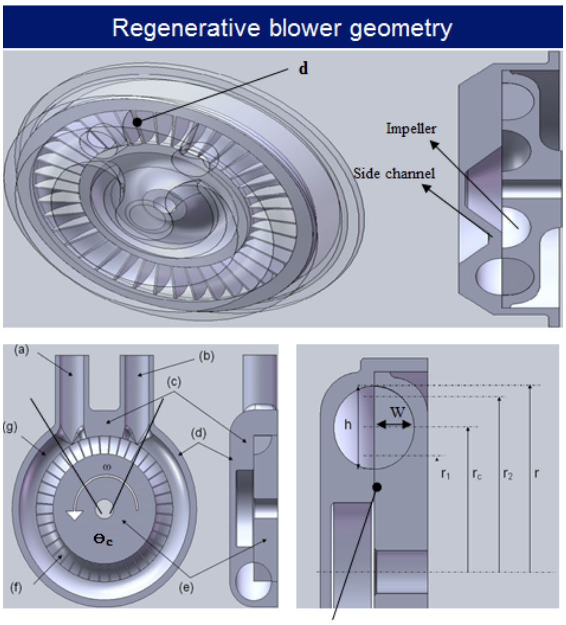


Figure 1: Geometry and design variables of regenerative blower.

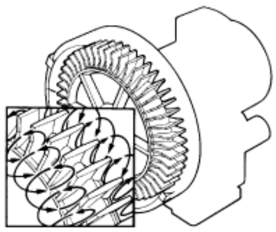
The main design variables of rotating impellers and fixed side channel are defined as follows:

- Rotation speed (N)
- Tip diameter (D_2=2r)
- Channel height (h)
- Channel width (W)
- Impeller blade inlet angle (β_1)
- Impeller blade outlet angle (β_2)
- No. of impeller blades (Z)
- Impeller blade thickness (d)
- Axial clearance(c)
- Extension angle ($\boldsymbol{\theta_c}$)

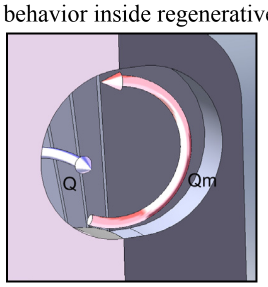
Once the blower design variables are defined, 3-D blower geometry design can be achieved and then used for the input data of performance analysis and CFD simulation.

2.2 Blower Performance Analysis Models

In the present study, the performance of blower is analysed by combining the mean line analysis method for fluid flow and the momentum exchange theory between impellers and side channel. As shown in Fig. 2, the gas flow inside regenerative blower shows typically three dimensional and helical-toroidal motion where fluid rotates in and passes along the space between rotating impeller blades and fixed side channel. The present study assumes mean streamline as the representative one of the three dimensional fluid flow phenomena.



(a) Flow behavior inside regenerative blower



(b) Cross section view on helical-toroidal flow

Figure 2: Flow pattern of regenerative blower.

Through the momentum exchange between fluid and impeller due to this flow motion, gas pressure is gradually raised along tangential flow path and its overall pressure rise(Δp_s) is calculated by

$$\frac{\Delta p_s}{\frac{1}{2}\rho u^2} = 2\frac{Q_m}{A_c u}\frac{r_1}{r_c}\left(\frac{r_2}{r_1}\frac{C_{u2}}{u} - \frac{C_{u1}}{u}\right) - (K_p + K_f/4)\phi^2 \tag{1}$$

where

$$\frac{C_{u2}}{u} = \frac{r_2}{r}\left(1 - \frac{\Delta u_2}{u_2}\right) + \frac{A_c}{A_2}\left(\frac{Q_m}{A_c u}\right)\cot\beta_2 \qquad \frac{\Delta u_2}{u_2} = \frac{1.5 + 1.1(2 - 2\beta_2/\pi)}{Z[1-(r_1/r_2)^2] + 1.5 + 1.1(2 - 2\beta_2/\pi)}$$

$$\frac{C_{u1}}{u} = \frac{r_1}{r}\phi \qquad \phi = \frac{Q}{uA_c}$$

$$\frac{1}{2}\left[K_m + \frac{1}{\sin^2\beta_2} + \left(\frac{A_2}{A_1}\right)^2\cot^2\beta_1\right]\left(\frac{A_c}{A_2}\right)^2\left(\frac{Q_m}{A_c u}\right)^2 + \left(\frac{r_1}{r} - \frac{u_{c1}}{u}\right)\cot\beta_1\frac{A_c}{A_1}\left(\frac{Q_m}{A_c u}\right)_2$$

$$+\frac{1}{2}\left[\frac{r_1^2 - r_2^2}{r^2} + \left(\frac{u_{c2}}{u_1}\right)^2 - \left(\frac{u_{c1}}{u}\right)^2\right] = 0$$

Here ρ, u, Q_m , φ are fluid density, impeller rotation speed, flow capacity and flow coefficient. More detailed description and variable definition about momentum-exchange theory are referred to Badami and Mura (Badami and Mura, 2012).

Since the fluid flow inside blower results in both the pressure losses due to fluid friction, turbulence and mixing and the leakage flow through the axial clearances between impeller disc and side channel, the present method needs the pressure loss and the leakage flow models.

So, in the present study, the pressure loss and the leakage flow models are constructed by using well-known fluid mechanics correlations corresponding to pressure loss and leakage flow sources as shown in Table 1. It is noted that all the present pressure losses and the leakage models are expressed as the functions of blower design variables (Lee et al., 2013).

Table 1: Model constants for pressure losses and leakage flows.

Related flow phenomena	Present model	Reference
Circulating flow loss(K_m)	$K_m = f(A_c/A_{cir}) = f(h,W)$	Choi et al.(2003)
Tangential flow loss(K_p)	$K_p = 0.0175 f_p(\mathrm{Re}_{D_h})\theta_c \frac{r_c}{D_h}\left(0.7 + 0.35\frac{\theta_c}{90}\right)\left(0.21\frac{1}{(r_c/D_h)^{0.25}}\right)$	Handbook of hydraulic resistance
Entry and discharge flow losses(K_L)	$K_L = 2\left(0.95 + \frac{33.5}{180 - \theta_c/2}\right)$	Handbook of hydraulic resistance
Casing-impeller leakage flow(K_{f1})	$K_{f1} = \frac{1.4(1 + 0.01Rv^2)}{\mathrm{Re}_{D_h}^{0.1}(r/c)^{0.25}}$	Kang & Lim(2005)
Stripper leakage flow(K_{f2})	$K_{f2} = f_p(\mathrm{Re}_{D_h})\frac{L_c}{D_h}$	Kang & Shim(2003)

Table 2 summarizes the main design variables of four actual regenerative blowers used to verify the present performance prediction accuracy, and all the blowers are applied in fuel cell applications (Hwang Hae Elelc., 2012: Badami and Mura, 2011: Gardner Denver, 2006).

Figs. 3-6 show the performance prediction results of Mini H-200, Mini H-100, Badami and Gardner Denver models by the present method, which are well-agreed with the measurement over entire flow capacity range.

Table 2: Design variables of four regenerative blowers.

Design variables	Hwanghae Mini-H200	Hwanghae Mini-H100	Badami-H2 (Italy, 2010)	Siemens(Gardner Denver) 2BH1002-AA53*
Rotation speed, N [rpm]	6,673	7,052	10,000	15,000
Impeller tip diameter, 2r [m]	0.111	0.104	0.138	0.110
Side channel height, h [m]	0.016	0.015	0.024	0.014
Side channel width, W[m]	0.008	0.0075	0.012	0.007
Blade inlet angle, β_1[deg]	90.0	90.0	90.0	90.0
Blade outlet angle, β_2[deg]	90.0	90.0	90.0	73.5
Extension angle, ΘC [deg]	290	290	325	290
Axial clearance, c [m]	0.0002	0.0002	0.0002	0.0002
No. of impeller blades, Z	39	39	41	52
Working fluid	Air	Air	Hydrogen	Air

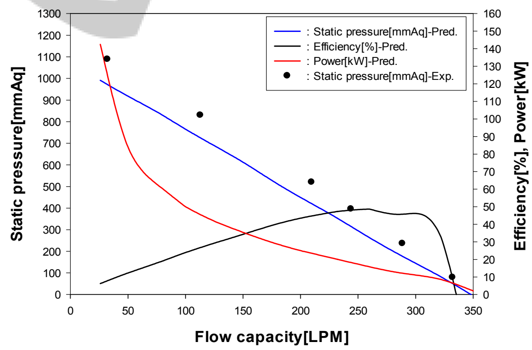


Figure 3: Performance predictions of Mini-H200 model.

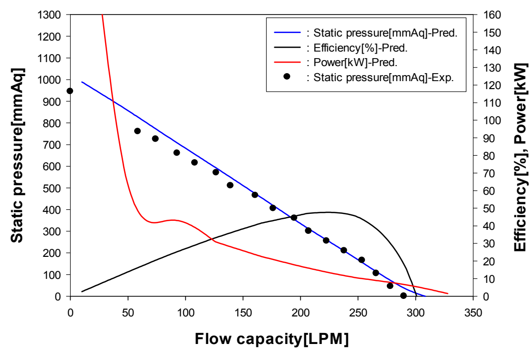


Figure 4: Performance predictions of Mini-H100 model.

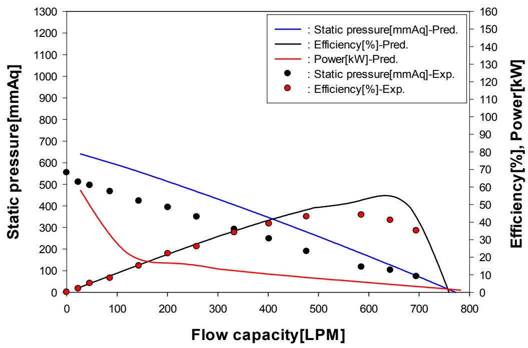


Figure 5: Performance predictions of Badami model.

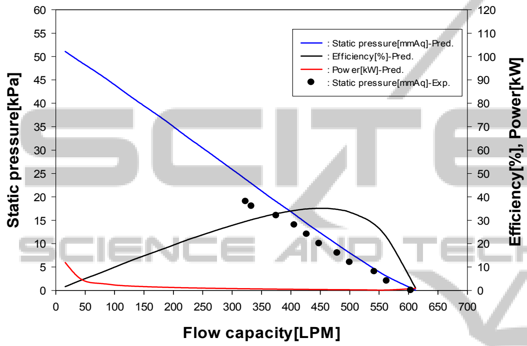


Figure 6: Performance predictions of Gardner Denver model.

2.3 Blower CFD Analysis Method

Based on the blower design of section 2.1, CFD modelling and simulation are conducted to investigate the internal flow field of blower. The present study employs the CFX code, a CFD program specialized for fluid machinery analysis, where 3-D RANS (Reynolds-stress Averaged Navier Stokes equations) solver is used with SST (Shear Stress Transport) turbulence model. The mesh generation on rotating impellers and stationary side channel is made, and the interface between rotating and stationary flow surfaces is treated by using frozen rotor scheme (CFX, 2013).

3 APPLICATION RESULTS

The present design and analysis methods are applied to develop a new regenerative blower used for fuel cell application. The design requirements and variables of new blower are summarized as follows:

- Rotation speed = 8000 rpm
- Tip diameter(2r) = 122 mm
- Side channel height(h) = 23 mm
- Side channel width(W) = 9 mm
- No. of impellers(Z) = 39

Based on the design variables, the new regenerative blower is manufactured as shown in Fig. 7.

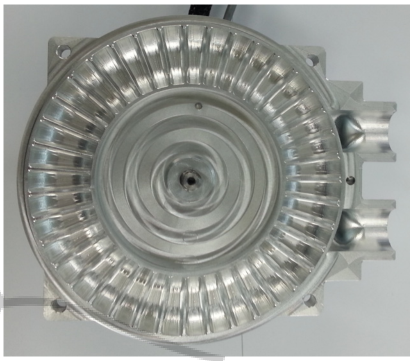


Figure 7: Manufactured model of newly designed blower.

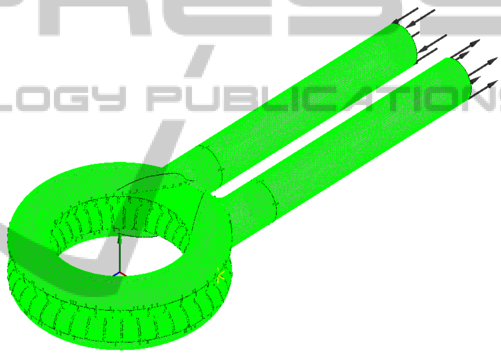


Figure 8: Mesh system of newly designed blower.

Fig. 8 shows the mesh system for the CFD analysis on the internal flow between rotating impellers and fixed side channel of newly designed blower. The CFD computation results on the fluid flow and the pressure rise through rotating impellers and side channel are depicted in Fig. 9. As shown in Fig. 9, the predicted streamline shows the fluid flow between impellers and side channel is helical-toroidal motion, and the pressure rise of fluid passing through tangential flow path is linearly increased.

Fig. 10 shows the turbulent kinetic energy inside blower, which is produced due the helical-toroidal fluid motion between impellers and side channel and is also linearly increased along tangential flow path. The performance of newly designed blower predicted by the present method is compared with the test results obtained from chamber-type test facility. As shown in Fig. 11, the predicted pressure curve is well-agreed with the test except at very low flow capacity.

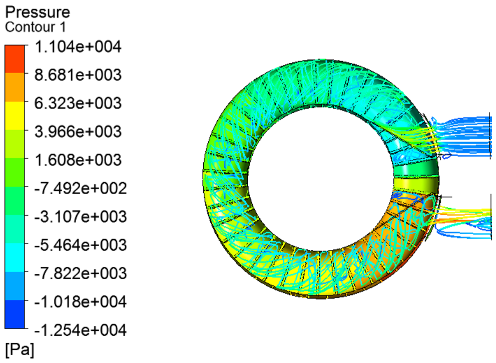


Figure 9: Streamline and static pressure of newly designed blower.

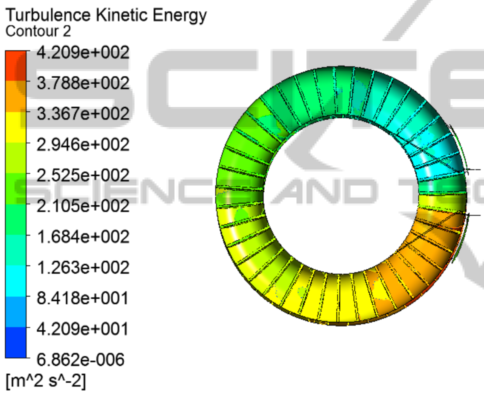


Figure 10: Turbulent kinetic energy of newly designed blower.

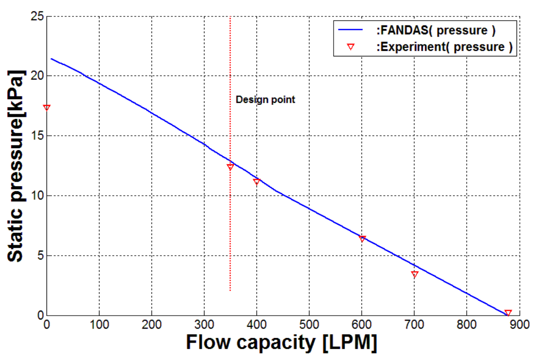


Figure 11: Pressure curve of newly designed blower.

4 CONCLUSIONS

The present study develops the design-analysis method which can be applied to the development process of regenerative blower. The present method is applied to the performance prediction of four existing blowers, and is also coupled with CFD simulation in developing a new regenerative blower used in fuel cell system. The prediction results by the present method are well-agreed with the test results within a few percentage of relative error. Therefore the present method is expected to be the reliable design tool suitable in developing regenerative blower.

ACKNOWLEDGEMENTS

This work was supported by the Development of the Regenerative Blower for fuel cell application of the Korea Institute of Energy Technology Evaluation and Planning (KETEP) grant funded by the Korea government Ministry of Trade, Industry & Energy, Republic of Korea.

REFERENCES

W.A. Wilson, M.A. Santalo and J.A. Oelrich, 1955, A Theory of the Fluid Dynamic Mechanism of Regenerative Pumps, *Trans. ASME*, 77, 1303-1316.

J.W. Hollenberg and J.H. Potter, 1979, An Investigation of Regenerative Blowers and Pumps, *Trans. ASME J. of Eng. For Industry*, 110, 147-152.

M. Badami, and M. Mura, 2012, Comparison between 3D and 1D Simulations of Regenerative Blower for Fuel Cell Applications, *Energy Conversion and Management*, 55, 93-100.

C. Lee, H.G. Kil, G.C. Kim, J.G. Kim, J.H. Ma, and K.H. Chung, 2013, Aero-acoustic Performance Analysis Method of Regenerative Blower, *J. of Fluid Machinery(in Korean)*, 16(2).

M. Badami and M. Mura, 2011, Setup and Validation of a Regenerative Compressor Model Applied to Different Devices, *Energy Conversion and Management*, 52, 2157-2164.

S.E. Wright, 1976, The Acoustic Spectrum of Axial Flow Machines, *J. of Sound & Vibration* 45(2), 165-223.

B.D. Mugridge, 1976, Noise Characteristics of Axial and Centrifugal Fans as Used in Industry, *Shock and Vibration Digest*, 45(3).

M.E. Goldstein, 1976, Aeroacoustics, McGraw-Hill.

Regenerative blower performance and noise test report, 2012, Hwang Hae Electric Co.

Catalogue module1, *A Gardner Denver Product*, 2006.

ANSYS-CFX user manual, *ANSYS Inc.,* 2013.