International Journal of Fluid Machinery and Systems
Vol. 5, No. 4, October-December 2012

DOI: http://dx.doi.org/ 10.5293/IJFMS.2012.5.4.168
ISSN (Online): 1882-9554

Original Paper

Numerical Investigation on Aerodynamic Performance of a Centrifugal Fan with Splitter Blades

Jin-Hyuk Kim[1], Kyung-Hun Cha[1], Kwang-Yong Kim[1] and Choon-Man Jang[2]

[1]Department of Mechanical Engineering, Inha University
253 Yonghyun-Dong, Incheon, 402-751, Republic of Korea
jinhyuk@inha.edu, kyunghun@inha.edu, kykim@inha.ac.kr
[2]Environmental Engineering Research Division, Korea Institute of Construction Technology
283 Goyangdae-Ro, Ilsanseo-Gu, Goyang, Gyeonggi-Do, 411-712, Republic of Korea
jangcm@kict.re.kr

Abstract

This paper presents a numerical investigation on the aerodynamic performance according to the application of splitter blades in an impeller of a centrifugal fan used for a refuse collection system. Numerical analysis of a centrifugal fan was carried out by solving three-dimensional Reynolds-averaged Navier-Stokes equations with the shear stress transport turbulence model. A validation of numerical results was conducted by comparison with experimental data for the pressure and efficiency. From analyses of the internal flow field of the reference fan, the losses by the reverse-flows were observed in the region of the blade passage. In order to reduce these losses and enhance fan performance, two splitter blades were applied evenly between the main blades, and centrifugal impellers having the different numbers of the main blades were tested with their application. Throughout the numerical analyses of the centrifugal fan with splitter blades, it was found that the reverse-flow regions in the blade passage can be reduced by controlling the main blade numbers with splitter blades. The application of splitter blades in a centrifugal fan leads to significant improvement in the overall fan performance.

Keywords: Centrifugal fan, impeller, splitter, pressure, efficiency, Reynolds-averaged Navier-Stokes equations.

1. Introduction

A centrifugal fan used in a refuse collecting system is mainly connected by a circular duct with a waste inlet. In the refuse collection station of the system, three or four fans are installed serially to increase a suction pressure. The level of the suction pressure is determined by the distance between a refuse collection station and a waste inlet. Hence, centrifugal fans used in a refuse collecting system require the high pressure characteristics to cope the required suction pressure.

In general, splitter blades attached in the blade passage are known to increase turbomachinery performances such as pressure and efficiency. A splitter blade can guide flow direction along the blade passage, and especially is more effective for enhancing the pressure rise in centrifugal impeller types. Kergourlay et al. [1] investigated the influence of adding splitter blades on the hydraulic performance of a centrifugal pump through experimental and computational methods. Miyamoto et al. [2] studied experimentally the effects of splitter blades on the flow behaviours and characteristics in unshrouded and shrouded centrifugal impellers. Madhwesh et al. [3] examined the effects of splitter blades corresponding to various geometrical locations on the impeller in a centrifugal fan, and they found that the splitter blades located at the impeller leading edge improves the static pressure recovery of the fan. Golcu [4] conducted a design optimization based on an artificial neural network surrogate model to improve the hydraulic performance of a deep well pump with splitter blade. Golcu et al. [5] also tested experimentally to investigate the effects of the number of blades and lengths of the splitter blades on a deep well pump performance. Jeon [6] performed a numerical study to investigate the acoustic characteristics of centrifugal impellers with and without a splitter blade.

In this work, a numerical investigation on the aerodynamic performance of a centrifugal fan with splitter blades has been carried out based on three-dimensional Reynolds-averaged Navier-Stokes (RANS) equations. To investigate the effects of splitter blades on the aerodynamic performance of a centrifugal fan, centrifugal impellers having the different numbers of the main blades with their application were tested in comparison with the reference impeller. The objectives of this work are to investigate the effects of the different numbers of the main blades and splitter blades on the aerodynamic performance of a centrifugal fan, and to provide

Received August 9 2012; accepted for publication October 8 2012: Review conducted by Prof. Yang-jun Zhang (Paper number O12022C)
Corresponding author: Choon-Man Jang, Research Fellow, jangcm@kict.re.kr

guidelines for the design of high performance centrifugal fan with splitter blades.

2. Reference Centrifugal Fan

A centrifugal fan used in a refuse collecting system has been investigated in this work. The perspective view of the centrifugal fan is shown in Fig. 1. It consists of an impeller with eleven blades and a volute casing. The backward typed blade is selected to get larger flow rate. According to experimental test [7], the flow and total pressure rise coefficients at the design flow condition of the reference model are 0.175 and 1.06, respectively, with an efficiency of 76.1%, which are defined as,

$$\Phi = \frac{Q}{AU_t} \tag{1}$$

$$\Psi = \frac{2\Delta P_t}{\rho U_t^{\,2}} \tag{2}$$

where, Q, ΔP_t, U_t, and ρ indicate the volume flow rate, total pressure rise, rotor tip speed, and density, respectively, and A is the outlet area of a centrifugal fan. The inlet and outlet diameters of the impeller used in the present work are 465 and 885 mm, respectively. The three-dimensional geometry of the centrifugal fan is shown in Fig. 2, and additional design specifications are listed in Table 1.

3. Numerical Analysis

The Flow field in the computational domain was analyzed using the commercial code ANSYS CFX 11.0 [8]. The three-dimensional steady compressible RANS equations were discretized by using the finite volume method (FVM). Discretizations of convection and diffusion terms of the equations were adopted by modified upwind scheme and central difference scheme, respectively. The shear stress transport (SST) turbulence model [9] with scalable wall function was employed to estimate the eddy viscosity [10]. Basically, SST model combines the advantage of the k-ε and k-ω models with a blending function. The k-ω model is activated in near-wall region and the rest region uses the k-ε model. It was demonstrated that the SST model captures flow separation under adverse pressure gradient well compared to other eddy viscous models and predicts well the near wall turbulence which is vital for the accurate prediction of flow separation.

As shown in Fig. 2, the whole inside domain of the centrifugal fan was considered as the computational domain for the numerical analysis. This computational domain includes a rotating impeller domain and three stationary domains (volute casing, inlet, and outlet duct domains). The working fluid was considered to be an ideal gas. The pressure of minus 1500 mmAq and designed mass flow rate were set at the inlet and outlet of the computational domain, respectively. The inlet pressure was determined by considering the value used in the practical site. The solid surfaces in the computational domain were considered to be hydraulically smooth with adiabatic and no-slip conditions. The boundary plane between the impeller and casing regions was imposed by Frozen-Rotor interference [8].

Table 1 Design specifications for the centrifugal fan

Flow coefficient	0.175	Outlet diameter of impeller, mm	885
Pressure coefficient	1.06	Inlet diameter of impeller, mm	465
Rotational frequency of impeller, r/min	3550	Blade thickness, mm	3.2
Efficiency, %	76.1	Number of main blades	11

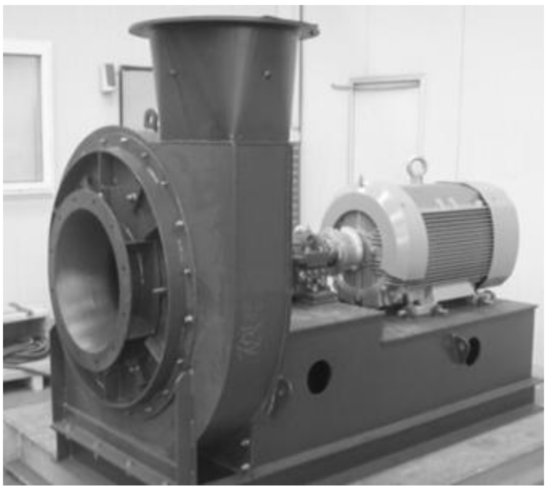


Fig. 1 Perspective view of a centrifugal fan

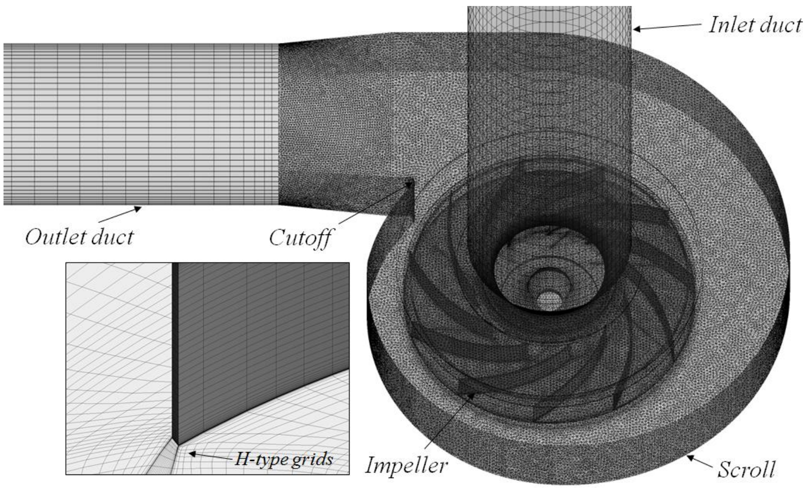


Fig. 2 Computational domain and grids of the centrifugal fan

In computational grids, tetrahedral grids were mainly imposed in a volute casing domain, whereas hexahedral grids were imposed in an impeller, inlet and outlet domains, as shown in Fig. 2. The volute casing domain was constructed using approximately 700,000 grid points, while the rotating impeller, inlet, and outlet domains were constructed using approximately 1,800,000, 100,000, and 100,000 grid points, respectively. Thus, the whole grid system in the present simulation for the centrifugal fan has about 2,700,000 nodes.

Root-mean-square (RMS) values of the equation residuals for convergence criteria were specified to be at least 10^{-5} for all equations. The computations have been performed by an Intel Core I7 CPU having clock speed of 2.94 GHz. The converged solutions were obtained after 500 iterations and the computational time was about approximately 12 hrs.

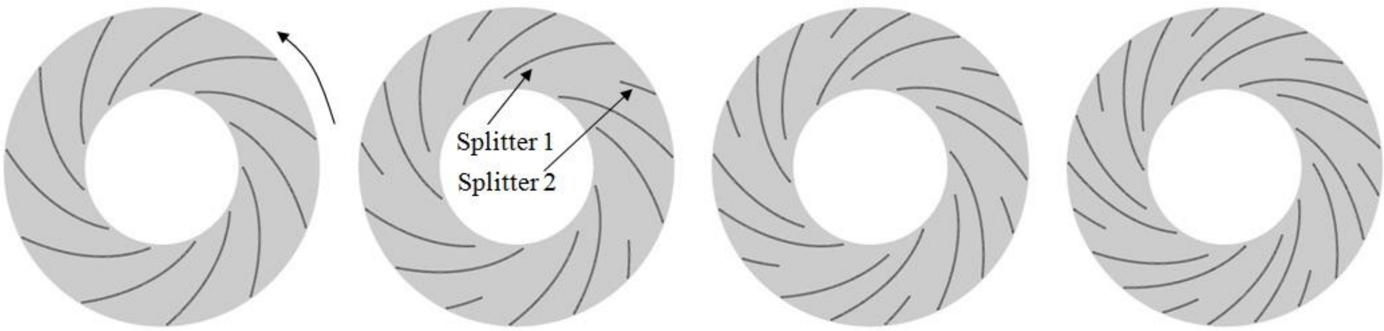
4. Centrifugal Fan Model with Splitter Blades

In this work, to investigate the effects of splitter blades on the aerodynamic performance of a centrifugal fan, two splitter blades were evenly applied between each main blade in centrifugal impellers having five, six, and seven main blades by considering a manufacturing cost, respectively. The front views of each centrifugal impeller with splitter blades are shown in Fig. 3, with the reference impeller. It is noted that the inlet and outlet angles keep constant for the reference and splitter blades. As shown in Fig. 3, the chord lengths of splitter blade 1 and 2 have 90% and 30% chord of the main blade, respectively. The positions and chord lengths of two splitter blades used in this work were determined by the preliminary study. Therefore, three types of centrifugal impellers having the different numbers of the main blades and splitter blades shown in Fig. 3 were tested to investigate their effects on the fan performance, with the reference impeller.

5. Results and Discussion

For the validation of the present numerical solutions, the characteristics of pressure and efficiency of the reference fan were compared to the experimental results according to flow rates, as shown in Fig. 4 [7]. The centrifugal fan model shown in Fig. 2 was used for this validation, and in this work this fan was regarded as the reference model. As shown in Fig. 4, the characteristics of the local pressure and efficiency obtained by numerical simulation match well with the experimental results. The computed pressure and efficiency, has a maximum of 5% error compared to the experimental data. The comparisons between the numerical and experimental results near the design flow condition show that the pressure and efficiency of the test fan are simulated correctly by the present numerical calculation. Thus, it is assured that the numerical analysis is valid and reliable.

The pressure coefficient and efficiency for each type of centrifugal impellers having the different numbers of the main blades according to the application of splitter blades are compared in Fig. 5, with the reference impeller. As shown in Fig. 5(a), all the types of the centrifugal impellers with the splitter blades show a beneficial effect on the pressure rise compared to the reference impeller. The pressure coefficient of the reference impeller is 1.060, and it is increased by the different numbers of the main blades according to the application of the splitter blades. Case 3 shows the most pressure rise of the pressure coefficient of 1.231, which is increased by 0.171 in comparison with the reference impeller. The pressure coefficients of cases 1 and 2 are also increased by



(a) Reference (b) Case 1 (b) Case 2 (c) Case 3

Fig. 3 Front views of centrifugal impellers having different numbers of the main blades and splitter blades, with the reference one

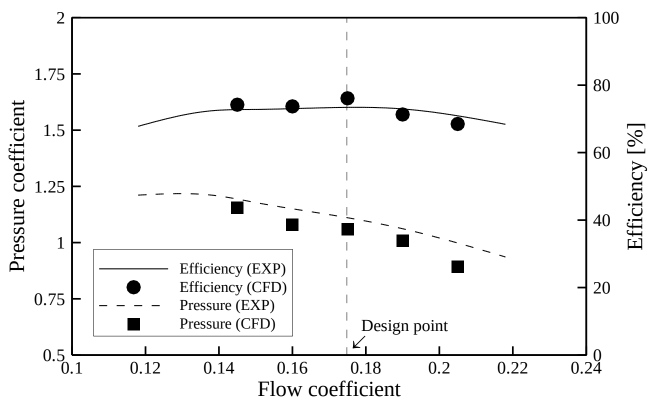
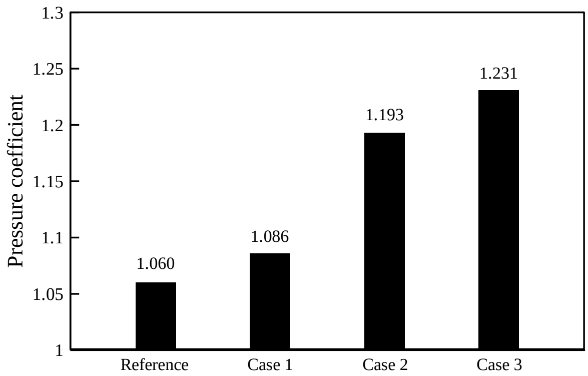
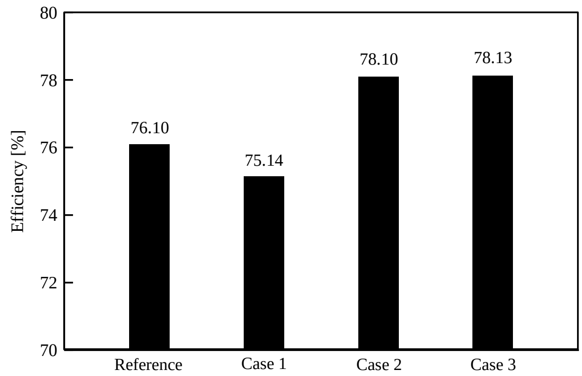


Fig. 4 Validation of the numerical analysis [7]

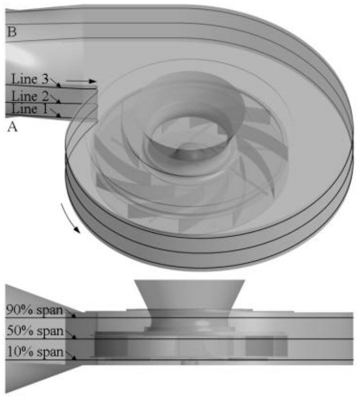


(a) Pressure coefficient

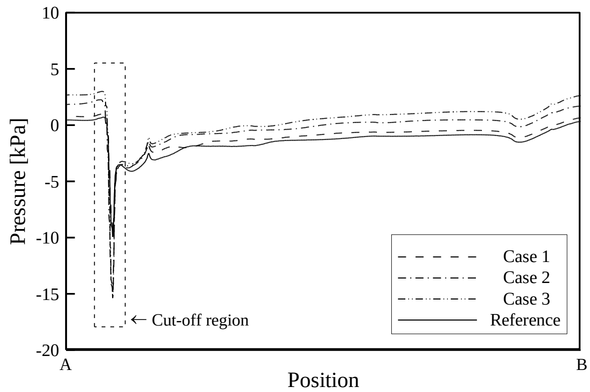


(b) Efficiency

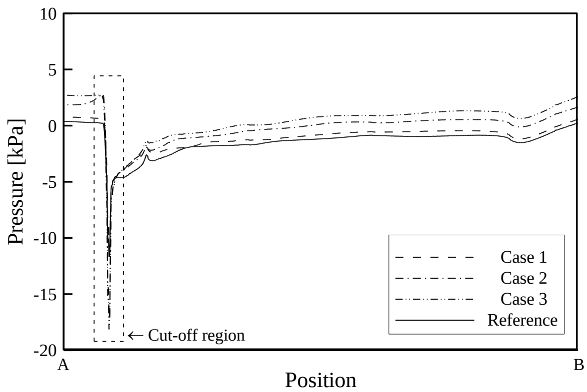
Fig. 5 Comparison of the performance parameters



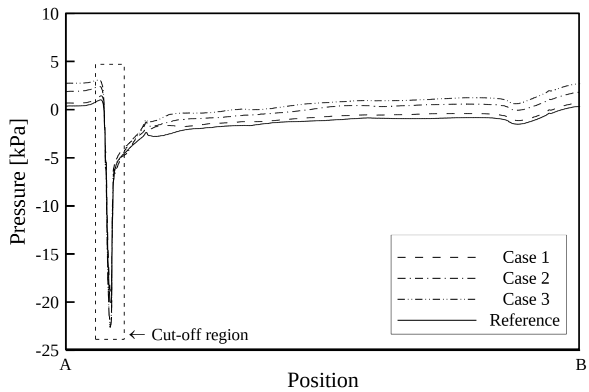
(a) Locations of observation lines



(b) Line 1 at 10% span



(c) Line 2 at 50% span



(d) Line 3 at 90% span

Fig. 6 Pressure distributions along observation lines on casing wall

0.026 and 0.133, respectively. In Fig. 5(b), it seems that case 1 having five main blades with splitter blades is less beneficial for the efficiency than the other types. The reference impeller shows the efficiency as 76.10% in this work. The efficiencies for cases 2 and 3 are increased by 2.00% and 2.13%, respectively, while the efficiency of case 1 is slightly decreased by 0.96%, in comparison with the reference impeller.

Figure 6 shows the pressure distributions along observation lines on casing surface for the reference design and cases 1~3. The observation lines, 1, 2 and 3, were located at 10, 50, and 90% spans of casing volute, respectively, as shown in Fig. 6(a). On all observation lines, a rapid pressure drop was observed at the cut-off region for the all types. It is noted that the loss occurs mainly at the volute casing cut-off region. In Fig. 6, the pressure distributions for the reference design and case 1 are similar in most of observation lines. Meanwhile, cases 2 and 3 generally maintain higher pressure distributions along the all observation lines, and especially at the case 3, and show considerably higher pressure values on most of the observation lines, compared to the reference design.

Figure 7 shows the pressure contours on the mid-span of a volute casing for the reference design and cases 1~3. As shown in Fig. 7, the gradual pressure increase was observed along the volute casing as the number of main blades and splitter blades

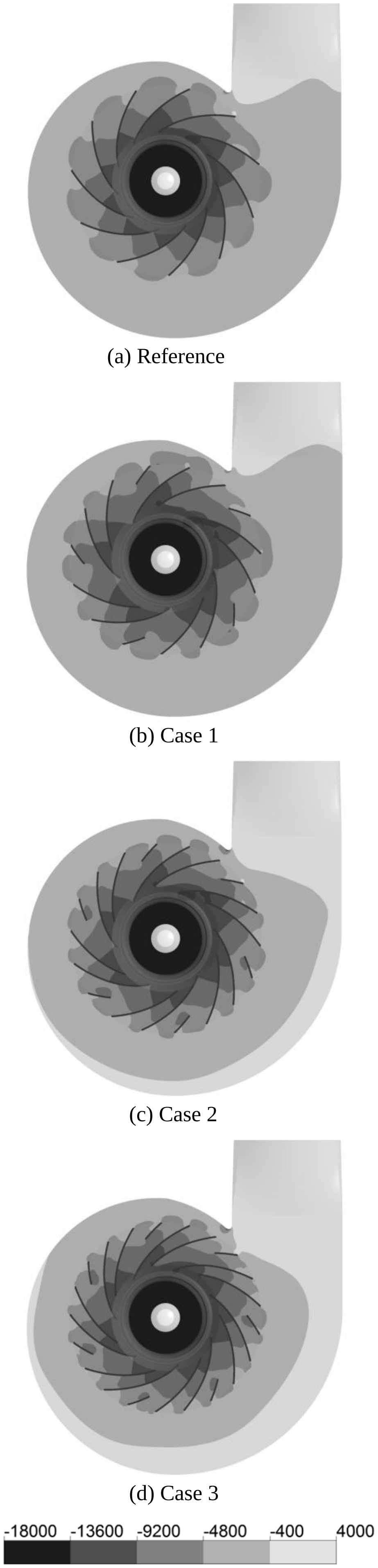


Fig. 7 Pressure contours on the mid-span of a volute casing (unit: Pa)

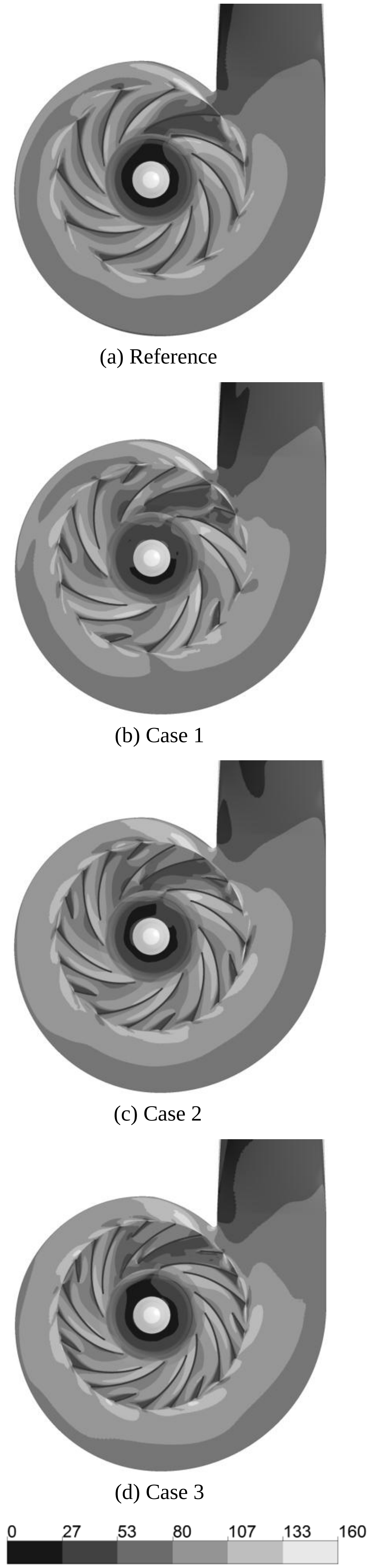


Fig. 8 Velocity contours on the mid-span of a casing volute (unit: m/s)

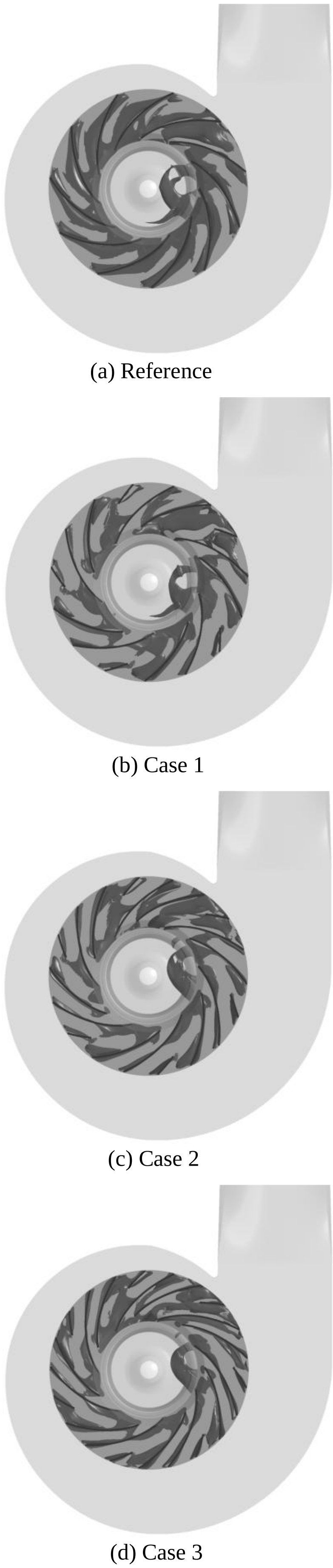


Fig. 9 Isosurfaces with velocity of 30 m/s

increase. In particular, this phenomenon was occurred remarkably in case 3, which features the highest pressure rise and efficiency values shown in Fig. 5.

Figure 8 shows the velocity contours on the mid-span of a volute casing for the reference design and cases 1~3. In the reference design, the reverse-flow phenomenon was observed in the region of the blade passage near the volute casing cut-off, as shown in Fig. 8(a). However, for cases 2 and 3, sizes of the reverse-flows in the blade passage near the volute casing cut-off were noticeably reduced in comparison with the reference design. Especially for case 3 which features the highest performance, this phenomenon is remarkable as shown in Fig. 8(d). Reducing these reverse-flows contributes to the enhancements of the pressure rise and efficiency of the centrifugal fan.

Figure 9 shows the isosurfaces having low velocity of 30 m/s in centrifugal impellers for the reference design and cases 1~3. The low velocity region observed in the blade passage near the volute casing cut-off represents the loss by the reverse-flows. As shown in Figs. 9(c) and (d), the low velocity regions in cases 2 and 3 were remarkably reduced in all blade passages compared to the reference design. These results illustrate clearly the enhancement of the centrifugal fan’s performance as a result of the application of splitter blades.

6. Conclusion

A numerical investigation on the aerodynamic performance according to the application of splitter blades in an impeller of a centrifugal fan used for a refuse collection system has been performed in this work. To investigate the effects of splitter blades on the aerodynamic performance of a centrifugal fan, two splitter blades were applied evenly between the main blades, and centrifugal impellers having the different numbers of the main blades were tested with their application. Throughout the numerical analyses of the centrifugal fan with splitter blades, the highest pressure rise and efficiency were obtained in the centrifugal impeller having seven main blades with splitter blades. This means that the centrifugal impeller having seven main blades with splitter blades produced mostly stable flows in the impeller region. These results elucidate how the centrifugal fan’s performance was improved as a result of the application of splitter blades.

Acknowledgments

This research was supported by the Korea Institute of Industrial Technology Evaluation and Planning (ITEP) grant funded by the Ministry of Knowledge Economy (No. 10032063 and 10031771).

Nomenclature

A	Outlet area	SST	Shear stress transport
CFD	Computational fluid dynamics	U_t	Rotor tip speed
EXP	Experiment	ΔP_t	Total pressure rise
FVM	Finite volume method	ρ	Density
Q	Volume flow rate	Φ	Flow coefficient
RANS	Reynolds-averaged Navier-Stokes	Ψ	Total pressure rise coefficient

References

[1] Kergourlay, G., Younsi, M., Bakir, F. and Rey, R., 2007, “Influence of Splitter Blades on the Flow Field of a Centrifugal Pump: Test-Analysis Comparison,” International Journal of Rotating Machinery, Vol. 2007, Article ID 85024, p. 13.

[2] Miyamoto, H., Nakashima, Y. and Ohba, H., 1992, “Effects of Splitter Blades on the Flows and Characteristics in Centrifugal Impellers,” JSME International Journal, Vol. 35, No. 2, pp. 238-246.

[3] Madhwesh, N., Karanth, K. V. and Sharma, N. Y., 2011, “Impeller Treatment for a Centrifugal Fan Using Splitter Vanes – A CFD Approach,” Lecture Notes in Engineering and Computer Science, Vol. 2192, No. 1, pp. 1981-1986.

[4] Golcu, M., 2006, “Artificial Neural Network Based on Modeling of Performance Characteristics of Deep Well Pumps with Splitter Blade,” Energy Conversion and Management, Vol. 47, No. 18-19, pp. 3333-3343.

[5] Golcu, M., Pancar, Y. and Sekmen, Y., 2006, “Energy Saving in a Deep Well Pump with Splitter Blade,” Energy Conversion and Management, Vol. 47, No. 5, pp. 638-651.

[6] Jeon, W. H., 2005, “A Numerical Study on the Acoustic Characteristics of a Centrifugal Impeller with a Splitter,” GESTS International Transactions on Computer Science and Engineering, Vol. 20, No. 1, pp. 17-28.

[7] Jang, C. M., Lim, S. J. and Yang, S. H., 2011, “Performance Analysis for Turbo Blower According to Inlet-Vane Angle,” Journal of the KSME B, Vol. 35, No. 3, pp. 301-307.

[8] ANSYS CFX-11.0, 2006, ANSYS CFX-Solver Theory Guide, ANSYS Inc.

[9] Menter, F. R., 1994, “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Application,” AIAA Journal, Vol. 32, No. 8, pp. 1598-1605.

[10] Menter, F. and Esch, T., 2001, “Elements of Industrial Heat Transfer Predictions,” Proceedings of 16th Bazilian Congress of Mechanical Engineering (CPBEM), Uberlandia, Brazil, CD-ROM.