Proceedings of
The Second National Congress on Fluids Engineering
August 22-24, 2002, Muju, Korea

Aerodynamic Analysis and Design of Inline-Duct Fan

En-Min Guo*, Kwang-Yong Kim**, Seoung-Jin Seo***

관류익형송풍기의 공력해석 및 설계

곽은민*, 김광용**, 서성진***

Keywords : *Inline-Duct Fan(관류익형송풍기), Aerodynamic Design(공력설계), CFD(전산유체역학), Vortex(와류), Q3D Design(준삼차원설계)*

Abstract

A tubular centrifugal fan is designed by using various methods of analysis and design. A preliminary design method based on empirical optimum curves for centrifugal fan is used to determine the geometric parameters for tubular centrifugal fan. And, Quasi-3D streamline curvature duct-flow analysis is used to provide the primary position of streamlines and spanwise distribution of flow angle for generation of blade geometry based on S1 surface. Three-dimensional CFD solution then is obtained to optimize the blade design. Constriction of flow path in the region of impeller, backward swept blade, and central cone, which are introduced to improve the design, successfully remove or suppress the vortices downstream of the impeller.

1. Introduction

Tubular centrifugal fan is classified as a special type of centrifugal fan composed of full radial-flow impeller, axial annular exhaust with or without guide vane, and axial tubular exhaust with rear cone, as shown in Fig. 1. Compared to conventional centrifugal fan, the configuration of tubular centrifugal fan is more compact, but noise level is rather lower due to absence of impeller-volute interaction. Therefore, tubular centrifugal fan is widely used in the field of air-conditioner, radiator, and ventilation due to compact construction and low noise level.. Because of the relatively low flow velocity within centrifugal impeller and the employment of airfoil blade, the efficiency of the impeller alone is usually very high, up to 80%~90%. However, for centrifugal fan, most of the losses occur downstream of the impeller, and especially the flow deflection from nearly radial direction to axial direction in tubular centrifugal fan easily results in flow separation. Thus, in design of this tubular centrifugal fan, more effort should be paid on suppressing flow separation and vorticity development, and also on reducing losses within the part of tubular flow.

In this work, design method of a tubular centrifugal fan is presented to improve efficiency and to reduce noise level. Flowfield of prototype fan is analyzed first with the help of CFD simulation. Then, basic design method and some measures are proposed in the improvement design based on the investigation on loss sources. Several impellers and guide vanes are compared with the numerical results.

2. Analysis of flowfield of prototype tubular centrifugal duct fan

The steady three-dimensional flow field computation is performed

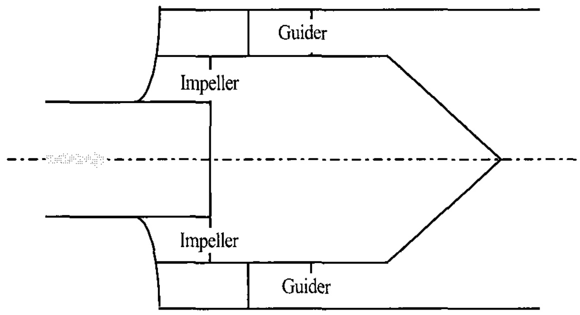


Fig.1 Configuration of tubular centrifugal fan

with a commercial Navier-Stokes code, CFX-TASCflow. Reynolds averaging is applied to the governing equations in strong conservation form for the turbulent flow. The turbulence is modeled by the k-ω based SST model[1-4]since it accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients.

The calculation is performed in one impeller blade passage (total 10 impeller blades in the prototype tubular centrifugal fan) with structured H-type mesh of 85,248 (32 nodes in spanwise direction, 37 nodes in tangential direction, and 72 nodes in streamwise direction), in the section of inflow with H-type mesh of 68,076(61336 331), and also in the section of annular exhaust without guide vane with H-type mesh of 363,096(216341341). The structured meshes are generated with CFX4-Build and an interactive code CFX-TurboGrid for blade geometry. The blade region is a block-off domain in the grid system. The periodic boundary conditions are specified on the both side of the computational domain. Detailed procedure of the calculation is presented in Ref. 5.

Based on the computational results, characteristics of the flowfield at design point can be summarized as follows:

* 인하대학교 기계공학과, kykim@inha..ac.kr
** 인하대학교 기계공학과, guoenmin@munhak.inha.ac.kr
*** 인하대학교 대학원, mrroi@netian.com

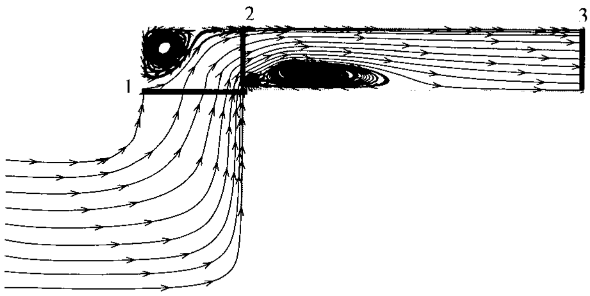


Fig. 2 Streamlines at the midpitch

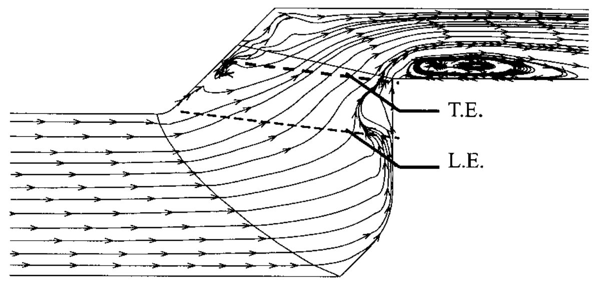


Fig. 3 Streamlines at the mid-pitch

- Due to the huge adverse pressure gradients and the deflection of the flow from nearly radial to axial direction downstream of the impeller, there occur two large scale flow recirculations, shown in Fig. 2, which obviously result in the large loss of total pressure. Computational results indicate that, the losses of total pressure from section 1 to section 2, and from section 2 to section 3, account for about 17 and 42 percent of total loss from inlet to section 3, respectively.
- Because of the employment of linear cascade (i.e., so-called straight blade) and over-dilative flowpath due to the nearly equivalent width at the inlet and exit of impeller, the distributions of blade load and incidence angle are not ideal. Computational results indicate that the hydraulic efficiency of the impeller is about 83.2%.

Based on the above analyses, it can be concluded that, in the new design of tubular centrifugal fan, the effort should be concentrated on how to remove or suppress the vortex existing in the tubular flow, and how to improve the flow within impeller.

3. Design of impeller

3.1 Basic design method

Because the conventional loss model for centrifugal fan could not accurately take into account of the effects of tubular flow in tubular centrifugal fan, the mean streamline analysis method based on loss models is not employed for this design. Instead, a preliminary design method based on empirical optimal curves for centrifugal fan[6] is used to determine the geometric parameters for tubular centrifugal fan. Quasi-3D streamline curvature duct-flow analysis is used to provide the primary position of streamlines and spanwise distribution of flow angle for generation of blade geometry based on S1 surface. Three-dimensional CFD solution then is obtained to optimize the blade design. The above steps are repeated until the design requirement is achieved.

Because the sound power of fan noise is proportional to $U^{5.6\text{–}6}$, where U is the peripheral speed of impeller, to reduce the peripheral speed by increasing the aerodynamic load of impeller blade is an effective method to reduce the noise level in the improvement of the design.

As discussed in the above section, large scale vortex is the important source of loss and aero-noise in the flowfield of the prototype tubular centrifugal fan. In order to suppress the development of vortex in the tubular flow, the following measures are proposed in this design:

- Constriction of flow path by optimizing the shape of shroud to suppress the vortices within impeller and at the shroud-exhaust corner
- Control of circulation distribution to suppress the vortices at the hub-exhaust and the shroud-exhaust corners
- Backward swept blade to improve the through-flow ability behind the impeller and to improve the flow within the impeller by enlarging the entry area of tubular centrifugal fan
- Central cone to improve the flow entering the impeller at the hub

3.2 Preliminary design of impeller

Basic geometric parameters are determined based on some empirical optimum curves and one-dimensional analysis results.

Cordier optimum curve is most widely employed to determine the outer diameter of impeller[6]. Based on the design requirement (generally volume flow rate, rotating speed, and total pressure rise), speed coefficient, σ and diameter coefficient, δ is related by Cordier curve, and then the optimum outer diameter, D_2 could be determined. In this way, the peripheral speed of newly designed impeller is reduced by 15 percent relative to the prototype impeller. The exit width of impeller could also be determined by the optimum width curve[7], which relates the ratio of exit width to diameter, b_2/D_2 to diameter characteristic, δ.

As introduced in Ref. 6, the optimum inlet diameter should satisfy the following condition.

$$\frac{D_1}{D_2} \geq 1.194\varphi^{1/3}$$

where φ is the volume coefficient. In this improvement of the design, the lower limit is taken to determine the inner diameter of impeller, D_1.

For backward-curved blade, one of the design criteria for blade passage is to limit the meridional deceleration through impeller in permissible range, which can be achieved by the following limit.

$$b_2 / b_1 \geq D_1 / D_2$$

As discussed in previous section, the shape of shroud, which depends on the width ratio (b_2/b_1), should be optimized to suppress the

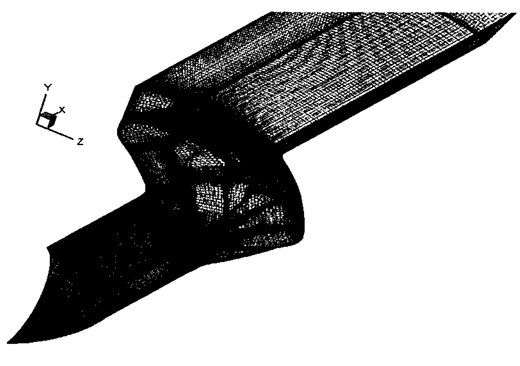
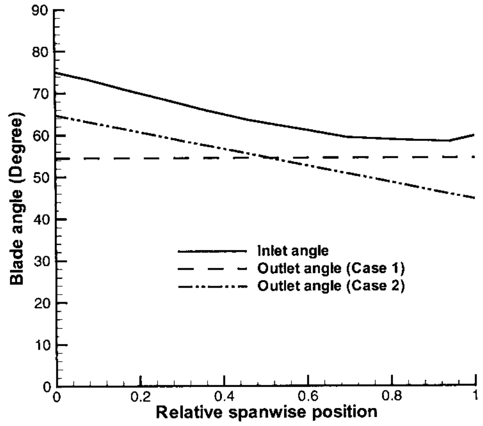
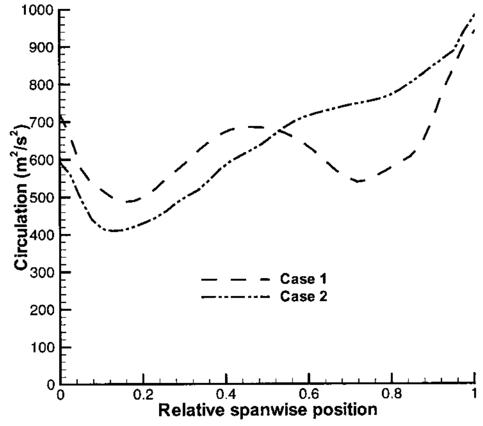


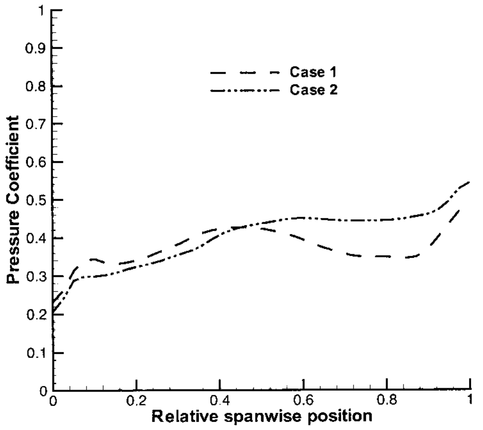
Fig. 4 Computational grids



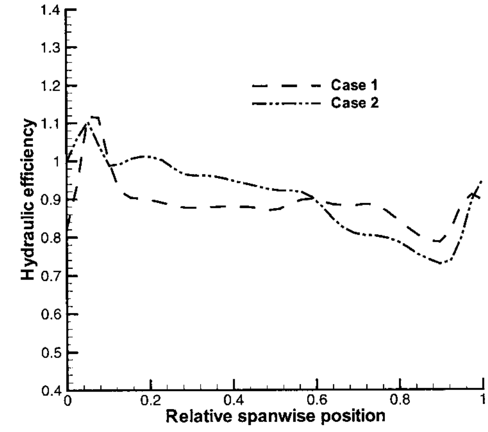
(a) Distributions of blade angle



(b) Distributions of circulation



(c) Distributions of pressure coefficient



(d) Distributions of Hydraulic efficiency

Fig. 5 Comparison between two cases of impeller

development of vortex at the shroud-exhaust corner. The lower limit for b_2/b_1 is then adopted to determine the inlet width b_1 in order to reduce the adverse pressure gradient the furthest behind the impeller, and tapered shroud is employed to simplify the process of manufacture.

In general, the blade number for backward-curved airfoil type centrifugal fan is about 8-12. The computational results indicate that too large blade number results in strong secondary flow within impeller and high friction losses. Therefore, the blade number is selected to be nine for this design.

3.3 Q3D/3D design of impeller

Fig. 3 shows the meridional view of flow path in newly designed impeller at mid-pitch position. The central cone, tapered shroud, and backward swept impeller with swept angle of about 7.5 degree in meridional view could be observed from the scheme.

The impeller blade is firstly two-dimensionally generated on S1 surfaces with single-circled camber and C-4 profile by specifying the inlet and outlet blade angle distributions, and then three-dimensionally generated by stacking 2-D blade sections on S1 surface with a straight line without lean angle specified at the position of trailing edge.

In order to evaluate the effect of distribution of circulation (UC_u) on the performance of tubular centrifugal fan, the flowfield is numerically investigated for two different impellers with nearly same total pressure rise. The computational domain consists of structured H-type mesh of 296,400 (383523150) in inflow part and impeller passage, and structured H-type mesh of 107,666(101341326) in the section of annular exhaust (without guide vane), as shown in Fig. 4..

These two impellers have the same meridional shape and inlet blade angle distribution, but have different outlet blade angle distribution. As shown in Fig. 5(a), in order to match the spanwise variable velocity triangle at inlet, the twisted blades are designed with larger inlet blade angle at hub than at shroud. The outlet blade angle in case 1 is constant along spanwise direction, but linear variation of outlet blade angle is designed in case 2, of which circulation UC_u almost linearly increases along spanwise direction as shown in Fig. 5(b).

The distributions of performance parameters of impeller alone, including pressure coefficient, $\psi(=\Delta P/\rho U_2^2)$, and hydraulic efficiency, $\eta(=\Delta P/\rho(U_2C_{u2}-U_1C_{u1}))$, are also shown in Fig. 5. It can be found that, only the distributions of circulation and pressure coefficient in case 2 are linear in most spanwise range, but in case 1, the peak values of both circulation and pressure coefficient occur at nearly 50 percent spanwise position due to the effect of secondary flow. And, the hydraulic efficiency of impeller in case 2 is higher than that in case 1 in the spanwise range less than 60% span, because of different distribution of blade load.

However, integrated performance including the loss in annular exhaust behind impeller should be evaluated. Because most of the loss occurs at the hub-exhaust corner, the whole hydraulic efficiency in the region from inlet to outlet of computational domain in case 2 is about 0.706, higher than that of 0.655 in case 1. Since higher total pressure rise through impeller in case 1 is obtained than that in case 2, the higher energy at hub enhances the intensity of vortex in the separation zone at the hub-exhaust corner.

Consequently, the impeller in case 2 is selected as the final solution for improvement of the design. From the computed blade load distribution of this impeller, it can be found that, along the whole spanwise range, the blade load of newly designed impeller is higher than that of prototype impeller. Such a blade load distribution together with lower peripheral speed can be expected to reduce the noise level in the new design of tubular centrifugal fan. Fig. 3 also shows that the effective measures employed in the new design successfully remove the vortex at the shroud-exhaust corner, and suppress the vortex at the hub-exhaust corner by reducing the spanwise extension of separated flow.

4. Conclusion

Numerical analysis indicates that most of total pressure loss occurs downstream of the impeller in tubular centrifugal fan, especially at both of hub-exhaust corner and shroud-exhaust corner, where large scale votical flow exists.

By introducing constriction of flow path in the region of impeller, backward swept blade, and central cone for the improvement of the design, the vortices behind the impeller are successfully removed or suppressed.

References

1. Menter, F.R., "Two-equation eddy-viscosity turbulence models for engineering applications," AIAA-Journal, 32(8),1994.
2. Grotjans, H., and Menter, F.R., "Wall functions for general application cfd codes," In K.D.Papailiou et al., editor, ECOMAS 98 Proceedings of the Fourth European Computational Fluid Dynamics Conference, pp. 1112-1117. John Wiley & Sons, 1998.
3. Wilcox, D.C., "Multiscale model for turbulent flows," In AIAA 24th Aerospace Sciences Meeting. American Institute of Aeronautics and Astronautics, 1986.
4. Menter, F.R., In 24th Fluid Dynamics Conference. American Institute of Aeronautics and Astronautics, 1993.
5. Moon, J.J., "Numerical analysis of flow in airfoil type tubular centrifugal fan", Master degree Dissertation, Inha University, 2002.
6. Ech, B., "Fans", Pergamon Press, Oxford, 1973.
7. Pan, D.L., "Optimal curves and its application to centrifugal ventilation fan,"Fluid Machinery, Vol.28, No. 10, 2000, pp.21-13.