Research Journal of Recent Sciences ISSN 2277-2502
Vol. **1(10),** 7-11, October **(2012)** *Res.J.Recent Sci.*

Numerical Design and Parametric Optimization of Centrifugal Fans with Airfoil Blade Impellers

Atre Pranav C. and Thundil Karuppa Raj R.
School of Mechanical and Building Sciences, VIT University, Vellore-632014, Tamilnadu, INDIA

Available online at: www.isca.in
Received 27th March 2012, revised 3rd April 2012, accepted 25th July 2012

Abstract

There are six types of centrifugal fan impellers AF, BI, BC, FC, RT, RB among which the AF i.e. impellers with airfoil blades are considered as highly efficient. The following paper presents the design methodology for the centrifugal fan system with impellers having airfoil blades. The numerical design procedure is developed for it and the CFD optimization has been carried out for volute casing to improve the results which have got from the numerical procedure only. A case is studied from technical bulletin[1] for this purpose and the results are correlated with those obtained from the numerical procedure developed. The concept of MRF (moving reference frame) is applied in the CFD analysis of the centrifugal fan as a rotating region around the impeller, keeping the components of the impeller stationary. The volute casing was optimized by decreasing the volute clearances by 10-14% and increasing the cut off height by 5% keeping it at 35% of impeller diameter. Thus the design methodology which includes the assistance of CFD optimization has been developed successfully.

Keywords: Centrifugal fans, CFD optimization, MRF.

Introduction

Fans are one of the types of turbo machinery which are used to move air continuously with in slight increase in static pressure. Fans are widely used in industrial and commercial applications from shop ventilation to material handling, boiler applications to some of the vehicle cooling systems. The performance of the fan system may range from free air to several cfm (cubic feet per min.). Selection of fan system depends on various conditions such as airflow rates, temperature of air, pressures, airstream properties, etc. Although, the fan is usually selected for nontechnical reasons like price, delivery, availability of space, packaging etc. The fan is always analysed by its performance curves which are defined as the plot of developed pressure and power required over a range of fan generated air flow. Also these fan characteristic curves can be used to data like fan bhp for selection of the motor being used.

The centrifugal fans with impellers having blades of Airfoil section are considered as the high efficiency impellers among the six types Airfoil blades, Backward Inclined single thickness blades, Backward curved blades, forward curved blades, radial tip blades and radial blades. The present study gives the design methodology for these high efficiency impellers which include the numerical design procedure and the CFD analysis of it. The CFD part is used for improvement the results of Static Pressure generated at the entry to the impeller, static efficiency. The CFD optimization also helped to improve the flow pattern through the centrifugal fan system.

The numerical procedure thus developed requires three functional input parameters volume flow rate (Q, cfm), static pressure (SP, inWC), and fan speed (rpm) which gives the output parameters such as number of blades, static efficiency, total efficiency, velocities at entry and exit of the impeller and the entry and exit angles of the blades. Also, the numerical design gave the details of volute casing. But the volute shape obtained from the numerical procedure showed the recirculation in CFD analysis of the fan system. Hence it has been concluded to decrease the volute clearances by 10-14% of the numerical design and the cut off height should be increased by 5% and kept at 35% of the impeller diameter. This brought the variation in results from 15-55% to 6-10%. The data for the case has been taken from the Aerovent Technical bulletin[1].

Methodology

Numerical Design: The numerical procedure is been developed for the high efficiency fan impellers having airfoil blades and the volute casing for it. The airfoil blades are considered to have backward inclined orientation. The design procedure gives the output parameters as below for the five functional input parameters requirements volume flow rate, Static pressure, speed of the fan rotation, mechanical efficiency and the motor bhp. The airfoil section used for the case is having NACA 2424 profile.

Table-1
Input parameters for the case

Input Parameters	Values
Volume Flow Rate (cfm)	10340
Static Pressure (inWC)	5.12
Speed of the fan rotation (RPM)	1329
Mechanical Efficiency (Assumed)	85%
Motor power (bhp)	10.61

Table-2
Output parameters for the case

Output Parameters	Values
Inner diameter (in)	19.81 (0.503 m)
Outer diameter (in)	30.64 (0.778 m)
Width of the blade at entry or at leading edge (b_1, in)	11.22 (0.285 m)
Width of the blade at exit or at trailing edge (b_2, in)	7.25 (0.184 m)
Blade entry angle (β_1, deg.)	17.18(< 35)
Blade exit angle (β_2, deg.)	51.97 (< 60)
Number of blades, Z	14
Flow coefficient (φ)	0.1894 (<0.4)
Pressure coefficient (Ψ)	2.056
Diameter coefficient (δ)	3.1681
Speed coefficient (σ)	0.166 (0.04 - 0.68)
Hydraulic efficiency (η_{hyd})	96.02 %
Volumetric efficiency (η_{vol})	95.55 %
Static efficiency (η_{stat})	78.56 %
Total efficiency (η_{total})	77.10 %

CAD Preparation according to numerical design data: The CAD modelling is divided into three parts viz.: i. Modelling of airfoil blade, ii. Modelling of fan impeller and iii. Modelling of volute casing.

The NACA Airfoil section is modelled using CAD software using surface modelling feature. The co-ordinates for the points contributing to Airfoil section curve are as mentioned in table-7 in appendix

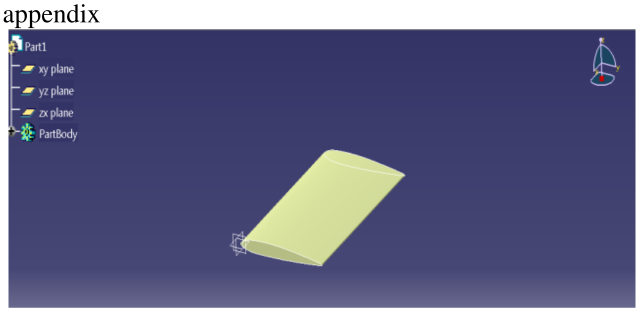

Figure- 1
CAD model for Airfoil section NACA 2424

Further, the CAD for airfoil section is imported in other CAD software and rest of the fan system is modelled in the same CAD package. The CAD for impeller is as shown in figure-2.

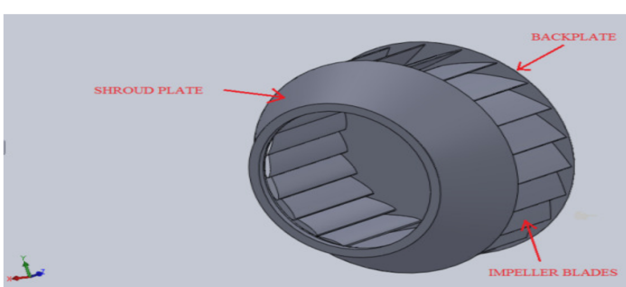

Figure-2
Impeller

The volute casing was designed according to four point method[2]. From the previous case studies, it was decided to decrease the volute clearances by 10-14% for improving the results further.

Table-3
Volute casing details

Design Parameter	Value
Smallest Radius (r_1), m	0.474
Second Radius (r_2), m	0.571
Largest Radius (r_3), m	0.679
Width of the volute, m	0.457
Corner Radius of curvature, m	0.0254
Cut-off region height, m	0.232

(**Note**: The values given are after parametric optimization)

The CAD for whole centrifugal fan system is prepared in CAD package as shown in figure-3

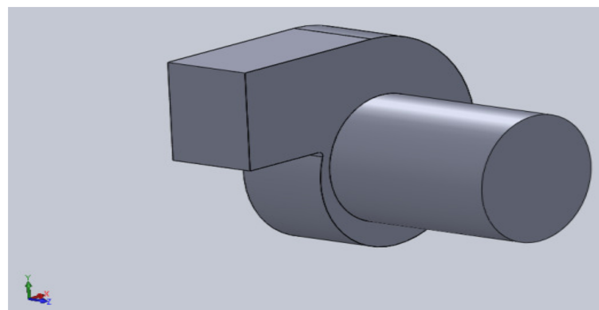

Figure- 3
CAD for Centrifugal Fan system

Grid Generation: The detailed CAD model is prepared in CAD packages and is meshed using two different softwares for surface as well as for volume meshing respectively. The mesh details are as given in table-4,

Table-4
Final mesh details

Entity	Value
Total number of elements	3216076
Maximum cell skewness	0.86

For surface meshing 2D tria element is used and for volume meshing 3D tetra element is used. The maximum cell skewness is maintained less than 0.9 to save computation time of solver. The meshed model for centrifugal fan system is shown in figure-4

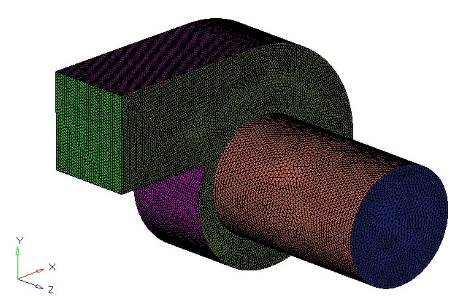

Figure-4
Meshed model for Centrifugal fan system

Boundary Conditions and case setup: The final meshed model is imported in the CFD solver fluent for pre-processing in which the case setup is carried out. The details of the case setup are given below.

Material: Air (atmospheric temperature at standard density of 1.225 kg/m^3 and viscosity of 1.789e-05 kg/m-s), Inlet: velocity, Outlet: pressure (atmospheric).

Table-5
Boundary conditions for the case setup

Boundary Condition	Value
Velocity at inlet (m/s) (calculated from volume flow rate through inlet face)	14.26
Pressure at outlet (Pa)	0
Hydraulic diameter at inlet (m)	0.66
Hydraulic diameter at outlet (m)	0.66
Turbulence model used	k-ω (2 eqn. SST)

Governing Equations: For the case analysis, the k-ω (SST) turbulence model has been used. The advantage of using k-ω (SST) turbulence model over k-ε turbulence model is that when k goes to zero the final equation appears in terms of ε for low turbulence regions. The destruction term $\frac{\varepsilon^2}{k}$ becomes problematic when k tends to zero. On contrary no such problems appear in ω. As k tends to zero the turbulence diffusion term simply becomes zero. The k-ω model contains two equations as follow,

$$(\rho U_j k)_{,j} = \left[\left(\mu + \frac{\mu_t}{\sigma_k^\omega}\right)k_j\right]_j + P_k - \beta^*\omega k \qquad (1)$$

$$(\rho U_j \omega)_{,j} = \left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)k_j\right]_j + \frac{\omega}{k}\left(C_{\omega 1} P_k - C_{\omega 2}\rho k\omega\right) \qquad (2)$$

$$\mu_t = \rho\frac{k}{\omega}, \varepsilon = \beta^*\omega k$$

The constants are

$$\beta^* = 0.09,\ C_{\omega 1} = \frac{5}{9}, C_{\omega 2} = \frac{3}{40}, \sigma_k^\omega = 2, \sigma_\omega = 2$$

When wall functions are used k and ω are prescribed as,

$$k_{wall} = (\beta^*)^{-1/2} u_*^2, \omega_{wall} = (\beta^*)^{-1/2}\frac{u_*}{ky}$$

Moving Reference Frame (MRF)[5]: The moving reference zone is nothing but the fluid domain created around the moving parts of the particular system. In case of centrifugal fan system the MRF is rotating domain. The faces of the MRF are specified as internal faces. The steady state approximation in moving reference frame (MRF) model, allows individual cell zones to rotate or translate with different speeds. MRF model is used in fans as the rotating member. This is achieved by dividing the domain into separate zones where the flow is solved in stationary or rotating coordinate systems.

To transform fluid velocities from stationary to rotating frames,

$$\vec{u_r} = \vec{u} - \vec{v_r} \qquad (3)$$

$$\vec{v_r} = \vec{\omega} \times \vec{r} \qquad (4)$$

Where, $\vec{u_r}$ is the velocity relative to the rotating frame, $\vec{u}$ is the absolute velocity and $\vec{v_r}$ is the whirl velocity due to moving frame). $\vec{\omega}$ is the angular velocity and $\vec{r}$ is the position vector to the rotating frame.

Solving the equations of motion in the rotating reference frame results in additional terms in the momentum equation,

$$\frac{\partial\rho}{\partial t} + \nabla.\rho\vec{u_r} = 0 \qquad (5)$$

$$\frac{\partial}{\partial t}\rho\vec{u} + \nabla.(\rho\vec{u_r}\vec{u}) + \rho(\vec{\omega}\times\vec{u}) = -\nabla p + \nabla\bar{\bar{\tau}} + \vec{F} \qquad (6)$$

Where, $\bar{\bar{\tau}}$ is the viscous stress

The Coriolis component and centripetal acceleration are included in the momentum equation with the term $(\vec{\omega}\times\vec{u})$. These equations are solved by commercially available Fluent software. The MRF zone created in Hypermesh is as shown in Figure-5,

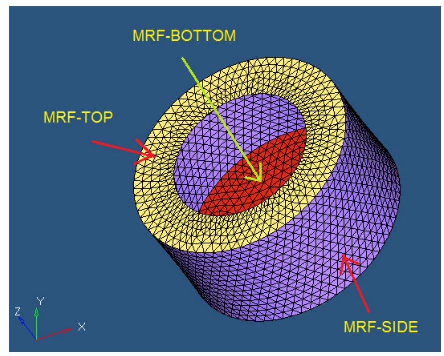

Figure- 5
Meshed model of Moving Reference Frame zone

Computational approach: Fluent software is used for solving the Navier-Stokes equations governing the physics of the flow inside the centrifugal fan system. Fluent code is based on finite volume method. The Fluent is been used for pre-processing, solving and post-processing purpose. Run is fired on workstation having processor of 2GHz, 12GB RAM and three processors for parallel solving. Once the solution got converged, the results were post-processed. The velocity, pressure contours and velocity vectors are plotted as shown in Figure-6(a), (b), (c),

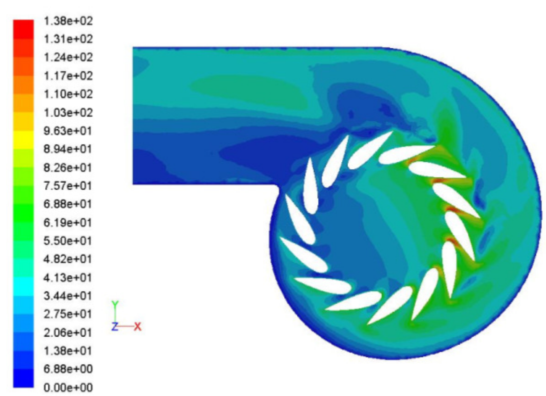

Figure-6(a)
Velocity contours plotted on plane normal to z-axis

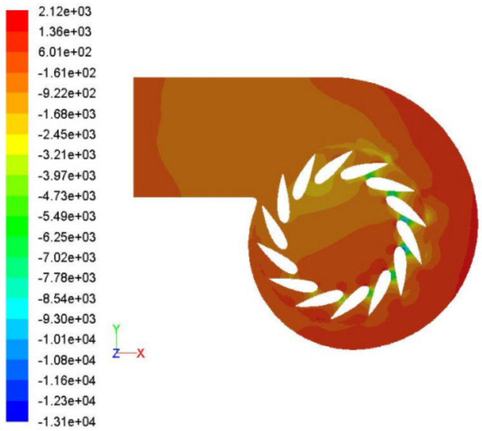

Figure-6(b)
Pressure Contours plotted on plane normal to z-axis

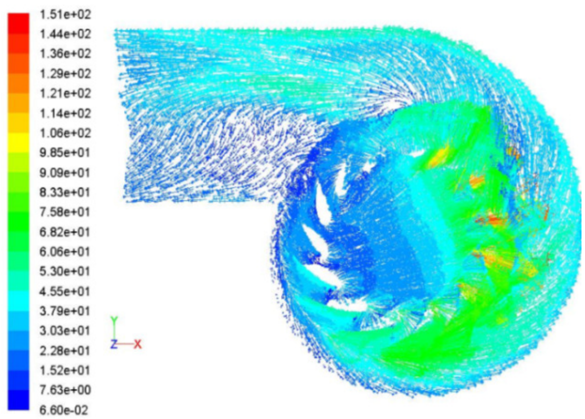

Figure-6(c)
Velocity Vectors plotted on plane normal to z-axis

Results and Discussions
The results obtained from the CFD analysis of the centrifugal fan system with parametric optimization of volute casing are as shown in table-6,

Table-6
Interpretation of results obtained

Parameters to be compared	CFD results	Design data	% Deviation
Static Pressure at the entry to the impeller (inWC)	4.63	5.12	9.5
Power consumed by fan impeller (BHP)	9.92	10.61	6.5
Static Efficiency (%)	71.11	78.63	9.56

From the above post-processed results for CFD analysis of the case study it can be observed that the three parameters used for validation of numerically obtained results are been correlated with good extent. There was no recirculation observed in any region of volute casing which can be seen from figure-6(c). This helped in improving the flow pattern in fan system and consequently the results. Hence, from the results of this case, it can be concluded that the numerical design procedure developed for centrifugal fan system with airfoil blade impellers with backward inclined orientation is validated successfully by CFD analysis. This completes the validation of numerical design for high efficiency impellers of centrifugal fans with CFD simulation.

Conclusion

The three parameters static pressure (SP in inches of water column), static efficiency, power consumed by fan obtained from the numerical design and CFD analysis are correlated successfully for the case undertaken. Hence it can be concluded that the CFD optimization of volute casing helps numerical procedure in improvement of results. However, the variation of 8-9% is observed due to the assumptions in preparing the numerical procedure and CAD model for it. The following conclusions are obtained from the study. The Numerically obtained volute casing design has drawback of recirculation phenomenon which was clearly observed in CFD analysis. The cut off height plays the major role in prevention of recirculation occurring near outlet region of the fan system. The k-ω (2 eqn. SST) turbulence model gives more accurate computational solution for the fluid physics.

Finally, it can be concluded that the design methodology thus developed for high efficiency centrifugal fan impellers with airfoil blades which includes numerical design as well as the CFD parametric optimization of Volute casing has been successfully implemented and validated.

References

1. Aerovent Technical Bulletin, 720, May **(2011)**
2. Bleier Frank P., Fan Handbook Selection, Application and Design, McGraw Hill publications **(1997)**
3. Eck, Bruno 'FANS'- Reference book on fan engineering, **(1975)**
4. Air and Gas Flow, Chapter Number 3, Book on Fans and ventilation
5. Singh O.P, Rakesh Khilwani T. Shrinivasulu M. Kannan, Parametric Study of Centrifugal Fan Performance: Experiment and simulation, *International Journal of Advances in Engineering and Technology*May **(2011)**
6. Shah K.H., Vibhakar N.N., Channiwala S.A, Dec-2003, Unified and comparative performance evaluation of forward and backward curved radial tipped centrifugal fan, *International Conference on MechanicalEngineering (ICME)* (**2003)**
7. Vibhakar N., Masutage S.D., Channiwala S.A., Three dimensional analysis of backward curved radial tipped

blade Centrifugal fan designed as per unified methodology with varying number of blades, Jan **(2012)**

8. Pathak Sunil, Turbocharging and oil techniques in light motor vehicle, *Research Journal of Recent Sciences,* **1(1)** 60-65 **(2012)**

9. Purkar T. Sanjay and Pathak Sunil, Aspect of Finite Element Analysis Methods for Prediction of Fatigue Crack Growth Rate**,** *Research Journal of Recent Sciences,* **1(2),** 85-91 **(2012)**

10. Kumar Krishnan and Aggarwal M.L., A Finite Element Approach for Analysis of a Multi Leaf Spring using CAE Tools, *Research Journal of Recent Sciences,* **1(2),** 92 -92 **(2012)**

11. Balasubramanian P and Ramamurti V, Frequency Analysis of Centrifugal Fan Impellers, *Journal of Sound and Vibration*, **1**, 1-13 **(1987)**

Table-7
Details of Airfoil section NACA 2424 from reference data sheet

UPPER SURFACE		LOWER SURFACE	
X	Y	X	Y
10.0000	10.4954	10.0000	-8.2724
20.0000	13.0514	20.0000	-9.9195
30.0000	13.8817	30.0000	-10.1319
40.0000	13.6060	40.0000	-9.6060
50.0000	12.5478	50.0000	-8.6292
60.0000	10.9398	60.0000	-7.3176
70.0000	8.8792	70.0000	-5.7842
80.0000	6.4143	80.0000	-4.0888
90.0000	3.5514	90.0000	-2.2492
100.0000	0.2574	100.0000	-0.2480
L.E. radius = 6.347 percent C (C= chord length 231.899 mm for this case)			
All co-ordinates are in percentage of chord length C			

Table-8
Nomenclature

AF	Airfoil Blades
BI	Backward Inclined blades
BC	Backward Curved blades
FC	Forward Curved Blades
RT	Radial Tipped blades
RB	Radial blades
in	inches
cfm	Cubic Feet per metre
fpm	Feet per metre
inWC	Inches of water column
RPM	Revolutions per minute
Deg.	Degrees of angle
CFD	Computational Fluid Dynamics