Simulation Studies on Impeller of Centrifugal Fan

Dr. T. Ch. Siva Reddy (Author)
Professor and Head
Dept. of Mechanical Engineering
SNIST, Hyderabad, India

Dr. MVSSSM Prasad
Professor
Dept. of Mechanical Engineering
SNIST, Hyderabad, India.

***Abstract*— The present investigations are concerning the computational studies of flow through a typical industrial centrifugal fan impeller of medium specific speed with a shape number of 0.12 used for ventilation applications. The present design of the impeller has backward curved constant thickness plate blades (8 Nos.) of circular arc camber. The drawing has been made in AutoCad and the 3-D model and the meshing were carried out using Ansys CFX Turbo Grid. The physical parameters and the Boundary conditions were stipulated using Ansys CFX- Pre and the Solution was carried out using the Ansys CFX Flow Solver. The flow cases were analysed for a set of flows ranging from 80 to 130% of the design flow rate for a nominal speed of 1450 rpm. The fan impeller performance was computed and the performance graphs of Total and Static Pressure rise, Total Pressure Ratio, the Input Power and the Polytropic Efficiency were obtained and plotted with respect to the fan flow rate for the range of flow rates investigated. Blade Loading pattern as well as the flow charts of vector and streak line patterns are also presented at 80%, 100% and 120% design flow rates for a better understanding of the flow through the centrifugal fan impeller.**

I. INTRODUCTION

Centrifugal fans find extensive use in many of the industrial and general ventilation applications because of their manufacture through simplified sheet metal fabrication. The present day energy crunch however demands a careful relook into the various designs for possible improvements and increased efficiency. The energy is transferred by the dynamic action of the impeller on the working fluid which is thrown outwards continuously and fresh fluid is inducted because of the reduced pressure. Another characteristic feature of the centrifugal impeller is that the angular momentum of the fluid flowing through the impeller is increased by virtue of the impeller outer diameter being significantly larger than the inner diameter. These can be optimized by identifying various zones of flow and pressure losses and suitable alternatives for the minimization of those losses. The shape number is another important parameter which has a bearing on the efficiency that can be realized from the fan. Impellers with shape numbers in the range 0.033 to 0.066 fall in the category of low specific speed or slow running impellers while those with shape numbers in the range 0.066 to 0.12 are of medium specific speed category. These are characterized by a medium D_1/D_2 ratio and a relatively larger b_2/D_2 ratio as compared to the low specific speed impellers.

1.1 Description Of The Fan Geometry:

The present design of the fan consists of a radial impeller with backward curved blades and housed in a volute casing. The impeller is provided with 8 Nos. of constant thickness plate blades with circular arc camber. The impeller is of medium specific speed with a shape number N_{sh} = 0.12. A bell-mouth seal connects the inlet duct with the eye of the impeller. The other specifications are as below.

Nominal Diameter at blade tip : 882mm.
Operating speed : 1450 rpm.
Nominal Air Flow rate : 3.06 m^3/ s.
Nominal Static Pressure rise : 2940 Pa (300 mmwc)
Nominal Total Pressure Rise : 3090 Pa (315 mmwc)

The present paper concerns the modeling of the impeller and the computational flow studies after suitable meshing of the rotating flow passages of the impeller.

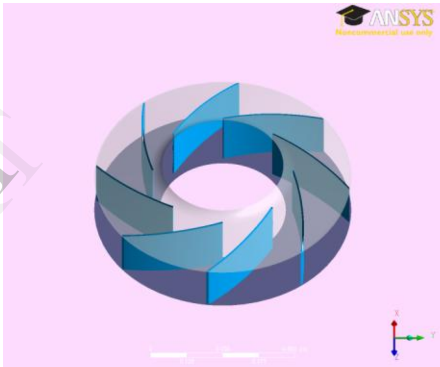


Fig.1a. View of Impeller with 8 Plate Blades

Fig.1b. View of Impeller Passage with Mesh Details

Fig.1a shows the isometric view of the impeller with 8 Plate blades with circular arc camber while Fig.1b shows the mesh details of a typical passage of the 8-bladed impeller configuration.

2. CFD MODELING AND ANALYSIS:

The elements used are of hexa-hedral structured type having typically 20538 nodes and 17680 elements for an impeller passage. The impeller flow passages were modeled and the 3-D Meshes were generated using CFX TurboGrid maintaining rotational periodicity. k – ε Turbulence model was used for the analysis with scalable turbulent wall functions. Other specifications include adiabatic type heat transfer and smooth wall, with mass and momentum corresponding to no-slip wall. The Boundary conditions were specified with standard dry air at a pressure of 1 atm or 1.0132 bar and 20^0 C as the working medium for the analysis.

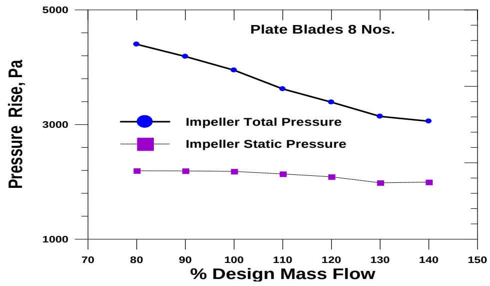


Fig. 3.1 Variation of Total Pressure Ratio across the impeller with flow rate

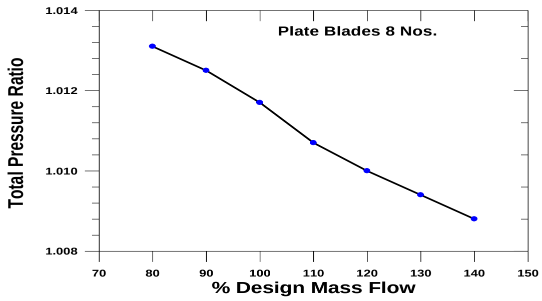


Fig. 3.2 Variation of Pressure Rise with Flow Rate

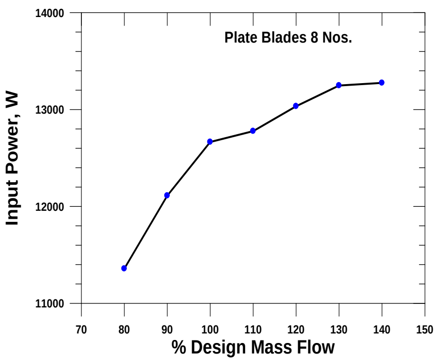


Fig. 3.3 Variation of Input Power with flow

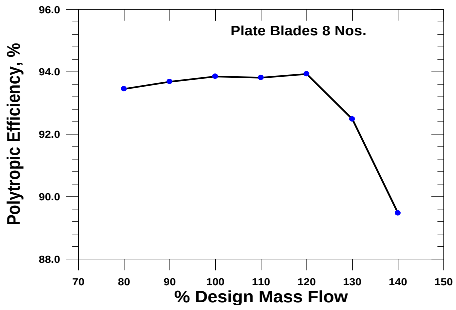
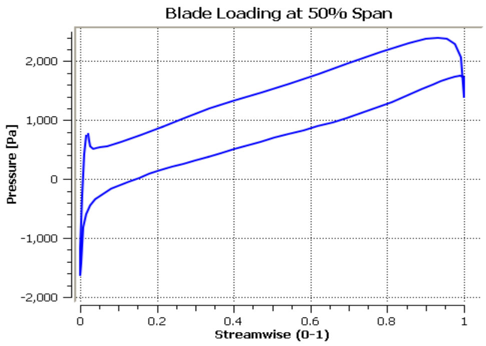
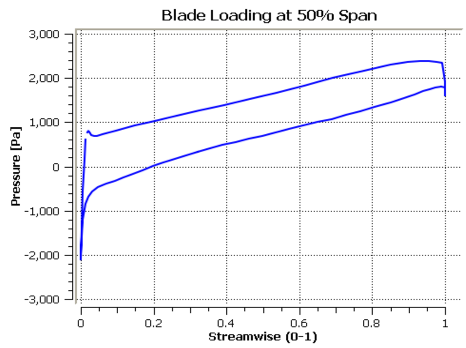


Fig. 3.4 Variation of Polytropic Efficiency of the impeller with flow rate

a. 80% Design Flow



b. 100% Design Flow



c. 120% Design Flow

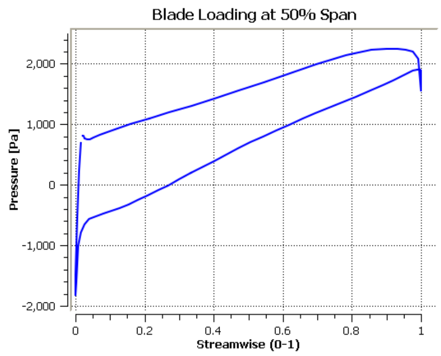
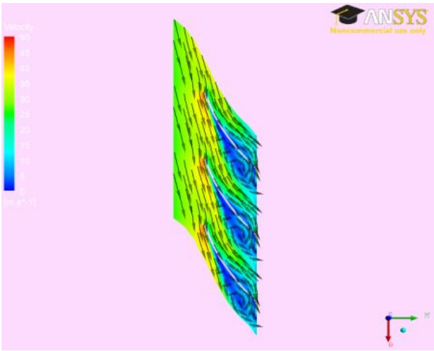
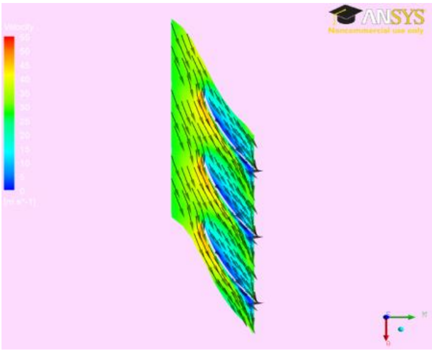


Fig. 3.5 Impeller Blade Loading

a. 80% Design Flow Rate



b.100% Design Flow Rate



c. 120% Design Flow Rate

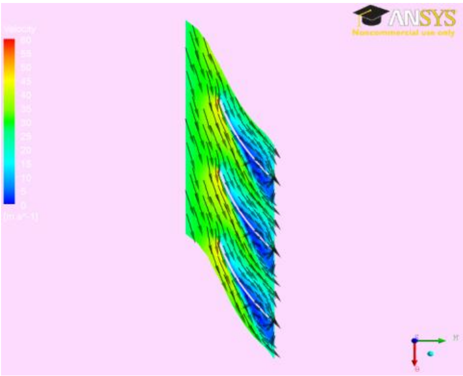
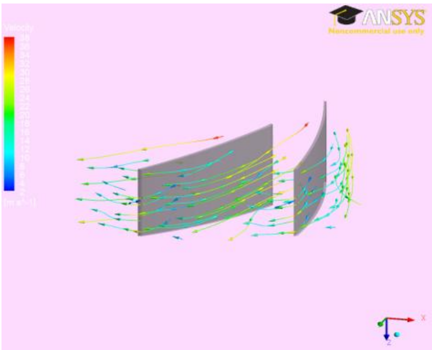
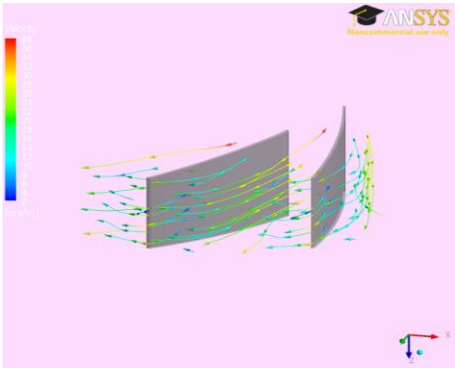


Fig. 3.6 Velocity Vectors at 50% Span

a. 80% Design Flow



b. 100% Design Flow



c. 120% Design Flow

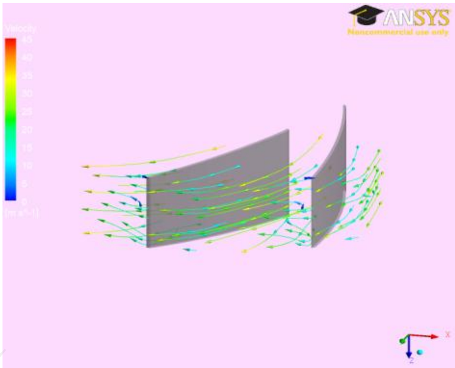


Fig. 3.7 Velocity Stream lines at Blade Trailing Edge

3. RESULTS AND DISCUSSION:

The impeller was analyzed for different sets of air flow rates covering 80% to 140% of Design flow rate, in steps of 10%. The design mass flow rate corresponded to 3.672 kg/s. The CFX solver runs were continued till a convergence limit of 10^{-4} was achieved for the RMS Residuals. The CFX Post data was analyzed to obtain the impeller performance.

Fig. 3.1shows the performance curves of Total Pressure Ratio and Fig. 3.2 shows the variation of Total and Static pressures at the impeller exit with reference to the flow rate. The curves of Total Pressure ratio and that of Total Pressure showed a stable trend with a negative slope, typical of a backward curved bladed impeller. The variation of static pressure was marginal within 200 Pa for flow increase from 80% to 140% of the design flow. The impeller total pressure was typically 3950 Pa (400 mmwc).

Fig. 3.5 a, b and c shows the Impeller blade loading distribution along the blade chord for 80%, 100% and 120% Design flow rates obtained at the mid-span. The blade loading showed a steady increase of static pressure along the blade chord from leading edge to the trailing edge.

Figs. 3.6 a, b and c shows the vector plots of velocity at 50% span for the above flow rates of 80%, 100% and 120%

respectively of the design flow rate. The plots showed a considerable flow detachment from the suction or the trailing surface from almost mid chord to the trailing edge. This was noticed for both 80% and 100% design flow rates. This could probably be due to limited guidance to the flow in this region. The flow at 120% design however, showed no such trend. The velocity stream line plots in the trailing edge region are shown in Figs. 3.7 a, b and c for the above flow rates of 80%,100% and 120% design and these are in close agreement of the trend of the vector plots discussed above.

4. CONCLUSIONS:

The impeller of a given industrial fan having a set of 8 Nos. of circular arc plate blades was modeled and the flow through the impeller was analyzed using Ansys CFX CFD software. The studies indicated a stable operating zone for the impeller in the range 80% to 140% of the design flow rate. The total pressure variation was as expected for a typical backward curved bladed fan impeller. The static pressure showed less variation of about 200 Pa or 20 mmwc in the flow range investigated.

The impeller blade loading indicated a uniform loading pattern from leading edge to the trailing edge with no negative loading.

The velocity vector plots showed considerable flow separation on the blade trailing surface from mid span down towards the trailing edge at 80% and 100% design flow rates.
The velocity stream line plots also confirmed the above trend.

5. REFERENCES:

- Bleier, F.P. 1998. Fan Handbook: Selection, Application, and Design. New York: McGraw-Hill. Calvert, W.J., and R.B. Ginder. 1999. Transonic fan and compressor design. Proceedings of the Institution of Mechanical Engineers, Part C—Journal of Mechanical Engineering Science 213(5):419–36.
- Cengel, Y.A., and J.M. Cimbala. 2006. Fluid Mechanics: Fundamentals and Applications. Boston: McGraw-Hill Higher Education. Dilin, P., T. Sakai, M. Wilson, and A. Whitfield. 1998. A Computational and Experimental Evaluation of the Performance of a Centrifugal Fan Volute. Proceedings of the Institution of Mechanical Engineers, Part A—Power & Energy 212(4):235–46.
- Fox, R.W., and A.T. McDonald. 1998. Introduction to Fluid Mechanics, 5th ed. New York: J. Wiley. Fluent. 2005a. FLUENT 6.2 User's Guide. Lebanon, NH: Fluent, Inc.
- Fluent. 2005b. GAMBIT 2.2 Manuals. Lebanon, NH: Fluent, Inc.ISO. 2002. ISO 10303-21, Industrial automation systems and integration: Product data representation and exchange—Part 21: Implementation methods: Clear text encoding of the exchange structure. Geneva: International Organization for Sandardization.
- Chen-Kang Huang, Mu-En Hsieh Performance Analysis and Optimized Design of Backward-Curved Airfoil Centrifugal Blowers VOLUME 15, NUMBER 3 HVAC&R RESEARCH MAY 200