ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Using FEM: Fluid Flow Analysis of Centrifugal Fan

V.Chandra Shekar Goud
Associate Professor,
Department of Mechanical Engineering,
Aurora's Scientific Technological and Research Academy, JNTUH, Telangana, India.

R.Sai Krishna
Student Scholar,
Department of Mechanical Engineering,
Aurora's Scientific Technological and Research Academy, JNTUH, Telangana, India.

G.Vikas
Student Scholar,
Department of Mechanical Engineering,
Aurora's Scientific Technological and Research Academy, JNTUH, Telangana, India.

R.Vinay Kumar
Student Scholar,
Department of Mechanical Engineering,
Aurora's Scientific Technological and Research Academy, JNTUH, Telangana, India.

ABSTRACT:

The forward backward curved (mixed) Centrifugal fan (HRE 630) has been analyzed aerodynamically for compare experimental results with simulation results by using ANSYS FLUENT (Finite Element Analysis Software). The material of the fan impeller was specified as ALUMINIUM. Boundary conditions on the HRE 630 are taken from the reference. The flow distribution across the fan is obtained. The maximum static pressure at the inlet is known and the pressure distributions across the blade are obtained accordingly. The obtained results are compared with Experimental results is discussed. In final recommended the design of the centrifugal fan and results are tabulated. It's observed that simulation results are nearer to the experimental results.

Keywords: Ansys Fluent, HRE-630.

I.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 experimental procedure and the CFD analysis and comparison between those results. 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 experimental analysis which gives the output parameters such as static pressure, static efficiency, total efficiency, velocities at entry and exit of the impeller. In the simulation the CFD analysis carried on exact model of final product and which gives result same as in experimental analysis. In comparison there is 9-10% of deviation in the static pressure, 23-24% of deviation in the shaft power and 12-13% of deviation

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

in the efficiency. From comparison 10-15% increase in outlet area thickness of volute casing gives the improvement in results.

II.WORKING MODEL:

Computational fluid dynamics is part of fluid mechanics that uses numerical method and algorithm to solve and analyze problem that related to fluid flow. A-Geometry Creation using CAD software (SOLIDWORKS) SOLIDWORKS is a parametric, integrated 3D CAD/CAM/CAE solution. The application runs on Microsoft Windows platform, and provides solid modeling, assembly modeling and drafting, finite element analysis, direct and parametric modeling, sub-divisional and nurbs surfacing, and NC and tooling functionality for mechanical engineers.

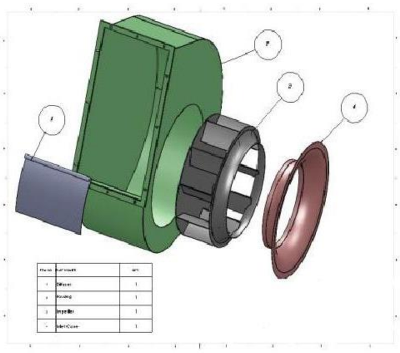

Figure: 1 CAD Model

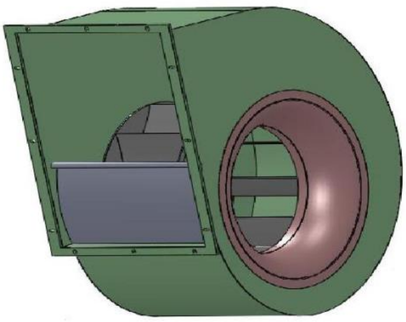

Figure: 2 Assembly Model

B- Meshing:

Meshing is done in ANSYS workbench itself. For volume meshing, a tetrahedral mesh generally provides a more automatic solution with the ability to add mesh controls to improve the accuracy in critical regions. Conversely, a hexahedral mesh generally provides a more accurate solution but is more difficult to generate. Path independent method was used for meshing which uses top down approach (creates volume mesh and extracts surface mesh from boundaries). The patch-independent method uses the geometry only to associate the boundary faces of the mesh to the regions of interest thereby ignoring gaps, overlaps and other issues that present other meshing tools with countless problems. The advanced size function is the default for fluids applications and is designed to accurately capture the geometry while maintaining a smooth growth rate between the regions of curvature and/or proximity. Maximum skewness observed was 0.9.

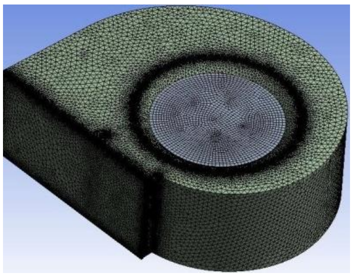

Figure: 3 Mesh Model

III.POST PROCESSING:

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. Specifications of the fan- 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.

Table-1 Final Mesh Details

Entity	Value
Total number of elements	4933605
Maximum cell skewness	0.9

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Type – Backward forward curved (mixed) centrifugal fan Outlet diameter of impeller- 634mm

Inlet diameter of fan – 480 mm

No. of blade – 10

Volume flow rate - 98 m3/ minute

Blade angle at the outlet – 63.75°

Boundary Conditions

Discharge: 98 CMM

Speed: 933rpm

Air Density: 1.2 kg/m3

Temperature: 20 deg

Inlet Area: 0.312 m2

Outlet Area: 0.312 m2

Compressibility Factor Kp: 0.99

IV. EXPERIMENT AND RESULT:

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, 16GB RAM and three processors for parallel solving. Once the solution got converged, the results were post-processed. The converged solution, velocity, pressure contours and velocity vectors are plotted as shown in Figure 4 to 6.

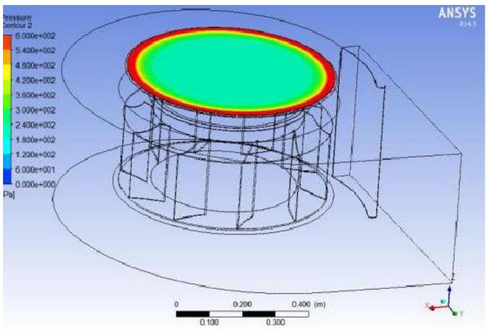

Figure: 4 Pressure Plot

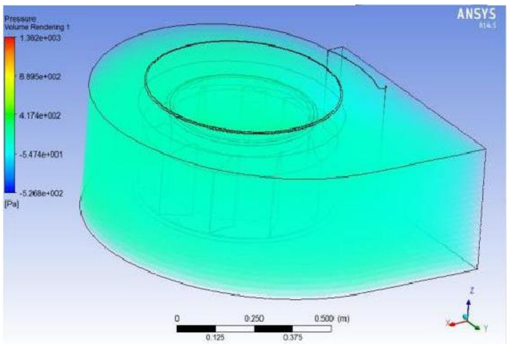

Figure: 5 Pressure Volume Plot

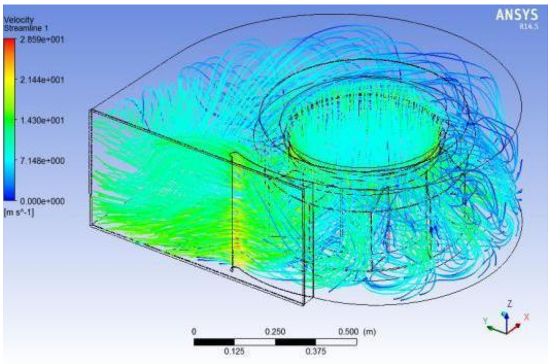

Figure: 6 Streams Line Plot

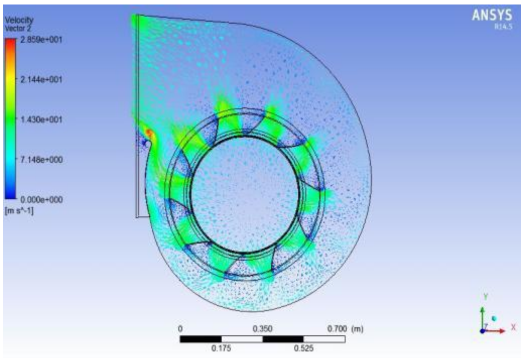

Figure: 7 Velocity Plot

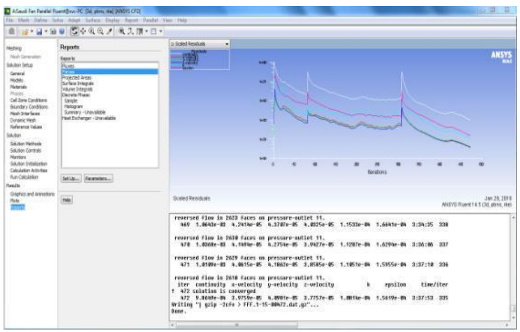

Figure: 8 Converged Solution

The results obtained from the CFD analysis of the centrifugal fan system with parametric optimization of volute casing are as shown in table-2

Table- 2 Interpretation of Results Obtained

Parameters to be Compared	Simulation Results	Experimental Results	Percentage of Deviation
Static Pressure at inlet(Pa)	480	530	9.4
Shaft Power(W)	1112.2	1457	23.6
Efficiency(%)	70.59	61.82	12.4

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 experimentally 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. 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 CFD analysis results validated successfully by experimental analysis. This completes the validation of CFD simulation for high efficiency impellers of centrifugal fans with experimental results.

V.CONCLUSION:

The three parameters static pressure (SP in inches of water column), static efficiency, power consumed by fan obtained from the experimental and CFD analysis are correlated successfully for the case undertaken. Hence it can be concluded that the CFD optimization of volute casing helps 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 experimental analysis which gives the output parameters such as static pressure, static efficiency, total efficiency, velocities at entry and exit of the impeller. In the simulation the CFD analysis carried on exact model of final product and which gives result same as in experimental analysis. In comparison there is 9-10% of deviation in the static pressure, 23-24% of deviation in the shaft power and 12-13% of deviation in the efficiency. From comparison 10-15% increase in outlet area thickness of volute casing gives the improvement in results.Finally, it can be concluded that the design methodology thus developed for high efficiency centrifugal fan impellers with airfoil blades which includes experimental 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 Mechanical Engineering (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, (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, (2012).