Performance Optimization of Centrifugal Fan with Ducts through CFD Simulation of Impeller Design

L.UMAMAHESWARARAO, M.Tech
Design & Development Engineer,
Saudi fan industries
Al-Khobar, KSA
l.mahesh06me23@gmail.com

V.CHANDRASIMHA, M.Tech
Associate HSE engineer
Flour Corporation
Gurgaon, INDIA
chandrasimha@gmail.com

Abstract: - The forward curved & forward impeller of Industrial Centrifugal fan along with Inlet & Outlet Ducts has been modeled in SOLIDWORKS and analyzed aerodynamically at different speeds for compare simulation results by using ANSYS FLUENT (Finite Element Analysis Software). The material of the fan impeller was specified as ALUMINIUM. Boundary conditions on the Centrifugal fan are taken from the reference. The flow distribution across the fan is obtained. The maximum static pressure at the immediate outlet of fan is known and the pressure distributions across the complete casing along with ducts are obtained accordingly. The obtained results are compared through variable speed operation for proper selection fan. As a final output the design of the centrifugal fan is tabulated and it is observed that centrifugal fan with impeller having forward ten blades have better performance compared to other designs.

Keywords: Impeller, SOLIDWORKS, ANSYS FLUENT

I. Introduction

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. The fan is always analysed by its performance curves which are defined as the plot of developed pressure, Velocity and power required over a range of air flow rates generated by fans. Also these fan characteristic curves can be used to data like fan bhp for selection of the motor being used.

The centrifugal fans with impeller having blades of Airfoil section are considered as high efficiency impellers among the six types such as Airfoil blades, Backward Inclined single thickness blades, Backward curved blades, forward curved blades, radial tip blades and radial blades[1]. But in centrifugal fans forward single thickness blades done performs better than airfoil configuration blades. The present study gives the design methodology for these forward type blades impellers of different designs. The CFD part is used to improve the results of static pressure generated at the entry of the impeller and static efficiency. The CFD optimization will also help to improve the flow pattern through the centrifugal fan system.

In the simulation phase the CFD analysis is carried out on different impeller designs at different speeds by changing the shape and as well as the number of blades. The comparisons of all the parameters at the outlet of the fan are shown in graphical representations. From the comparisons of simulations, it observed that the forward ten blades impeller design is providing better results for static pressure and velocity parameters obtained at the outlet.

II. Working model

Computational fluid dynamics is part of fluid mechanics that uses numerical method and algorithm to solve and analyze problems 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.

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

Dimensions of the Model HRE-950[2]:

Forward curved ten blades Impeller:

1. No. of Blades: 8 &10
2. Leading edge: 272^0
3. Trailing edge: 120^0
4. Radius of Curvature of Blade: 1000 mm
5. Impeller inlet eye diameter (D_{eye}): 1840 mm.
6. Impeller Diameter (D): 2400 mm
7. Impeller cross sectional thickness: 2 mm

For forward blade impeller all the dimensions are same as forward curved impeller except curvature of blade because the consideration is just inclined as shown in figure 1.

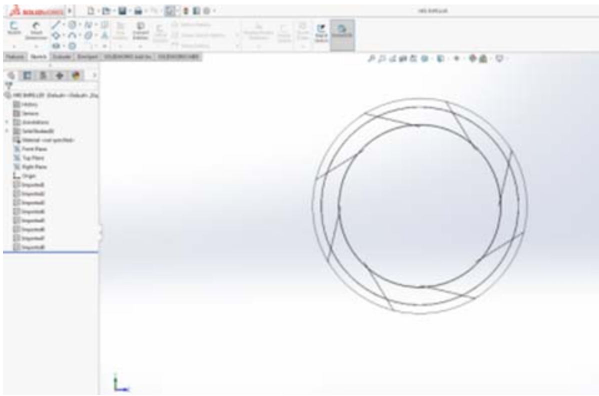

Fig 1: Forward Impeller with 8 blades front view

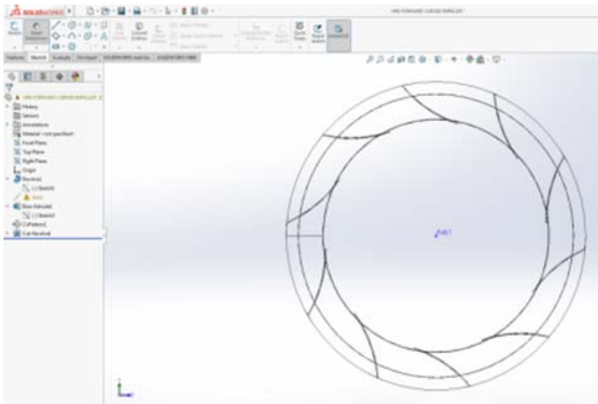

Fig 2: Forward Curved impeller with 10 blades front view

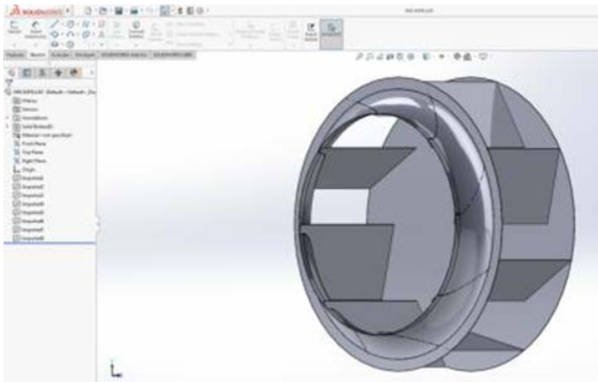

Fig 3: Forward Impeller with 8 blades ISO view

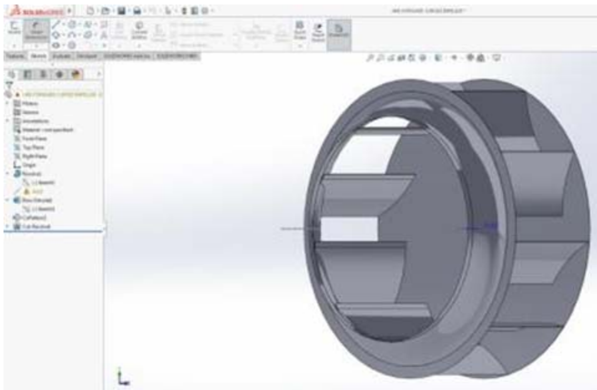

Fig 4: Forward Curved impeller with 10 blades

Dimensions of casing as shown in figure:

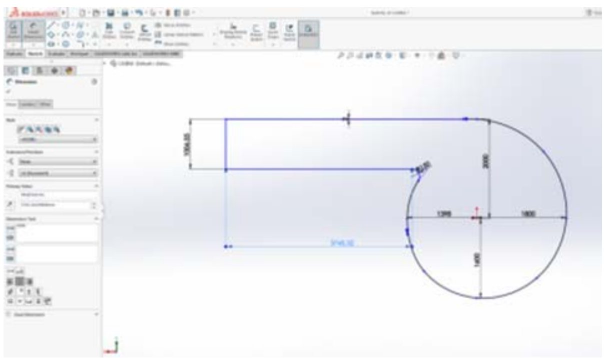

Fig 5: Outline of Casing with Ducts Dimensions

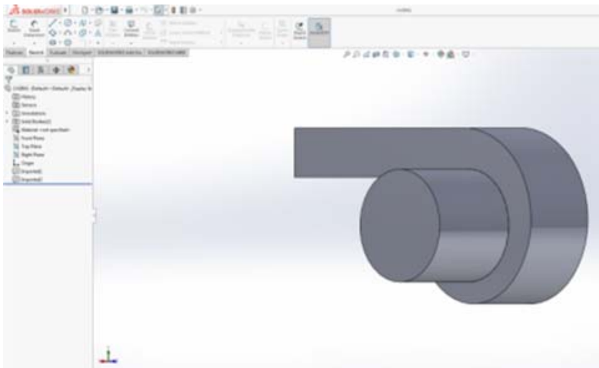

Fig 6: Isometric view of Casing with Ducts

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.

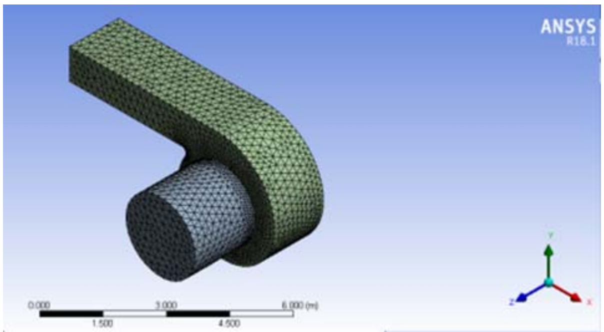

Fig 7: Mesh Model of Complete Fan with Ducts

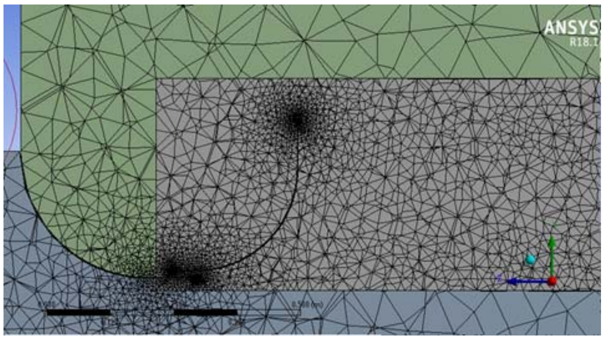

Fig 8: Meshing at the Impeller Blade

S.No	Name	Forward Curved 10 Blades	Forward Curved 8 Blades	Forward 10 Blades	Forward 8 Blades
1	Elements	9115310	13800273	15166802	14798272
2	Skewness	0.996	0.992	0.986	0.988

Table 1: No. of Elements & Skewness of Different Designs of Impellers

III. Post Processing

Post processing is the process to declare Boundary Conditions in the ANSYS FLUENT interface to perform simulation. Here flowing fluid in the fan mentioned as Air at density 1.22 kg/m^3 at temperature. Different speeds of impeller are considered in preprocessing. For better performance in first stage simulation carried at absolute pressure zero in each speed. Pressure-inlet & Pressure-outlet boundary conditions are considered as inlet & outlet of the fan ducts. Coupled solution method used for all simulations at different speeds and different modeled impellers.

Formulae [1]:

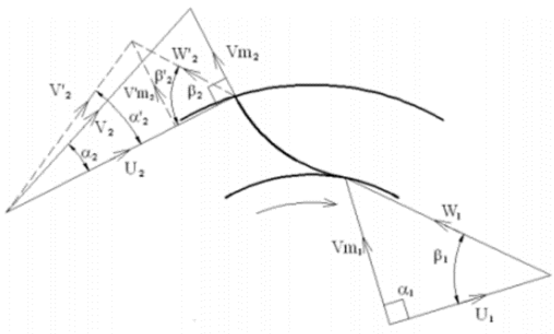

Fig 9: Theoretical and Actual Velocity Triangles as per fundamental Design

After each simulation we will get the mass flow rate, total pressure and velocity with respect to each speed

Inlet tip Velocity, $U_1 = 1.1 V_1$

V_1=Absolute Velocity

$$U_1 = \frac{\pi (D_{eye}) N}{60} = 1.1\ V_1$$

N= Speed in rpm

$$\text{Discharge, } Q = \frac{\pi}{4} \times (D_{eye}{}^2) \times V_1$$

$$\text{Efficiency, } \eta = \frac{\text{Air Horse Power}}{\text{Brake Horse Power}}$$

$$AHP = \frac{Q \times \text{Total Pressure}}{6356}\ KW$$

IV. Experiment and Results

Fluent software is used for solving the Navier-Stokes equations [3] 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 3.4GHz, 32GB 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 following Figure.

Pressure & Velocity behavior through forward curved 8 & 10 Blades impeller fans.

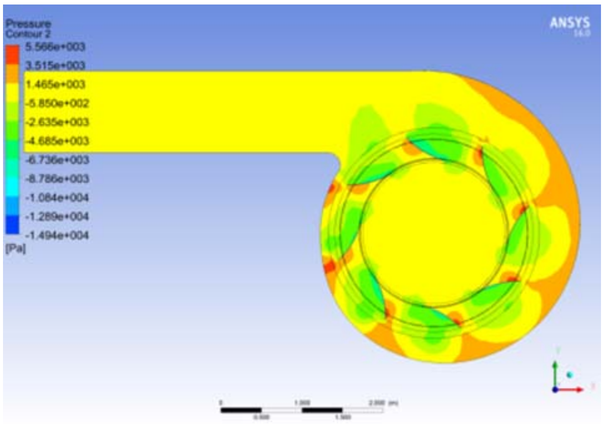

Fig 10: Pressure contour of 8 Blade forward Curved Impeller

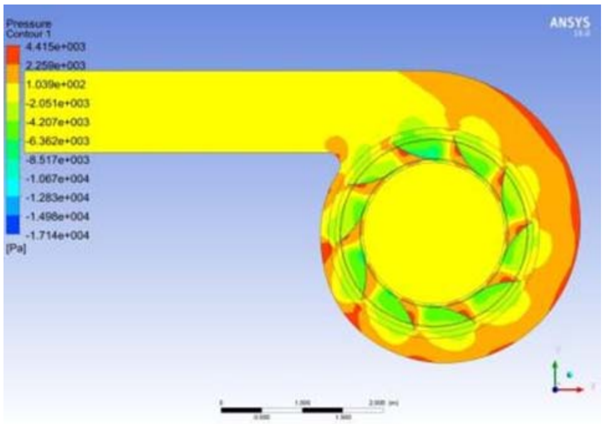

Fig 11: Pressure contour of 10 Blade forward Curved Impeller

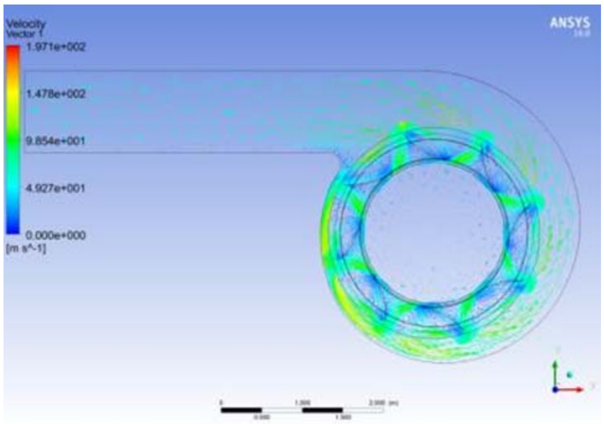

Fig 12: Velocity Vector of 8 Blade forward Curved Impeller

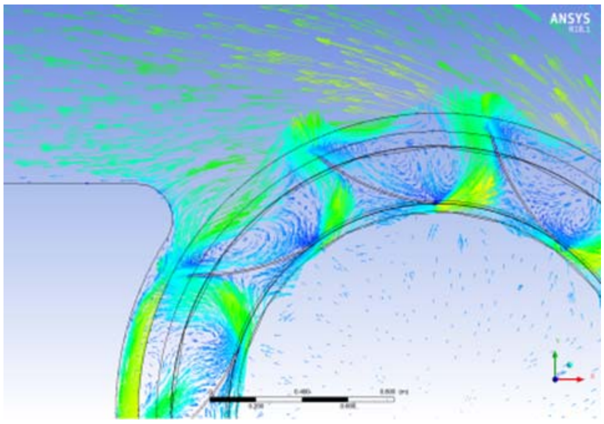

Fig 13: Velocity Vector of 10 Blade forward Curved Impeller

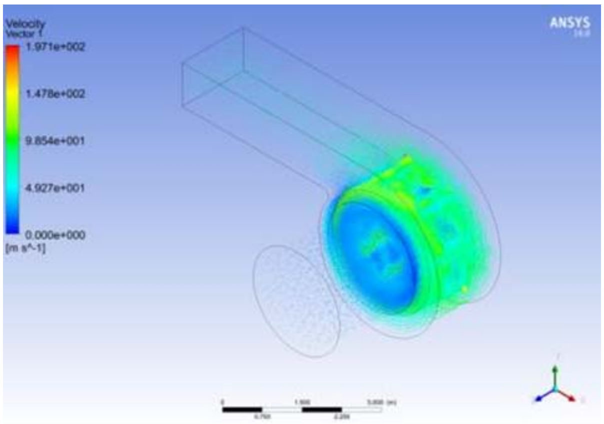

Fig 14: Velocity Vector Volume of 8 Blade forward Curved

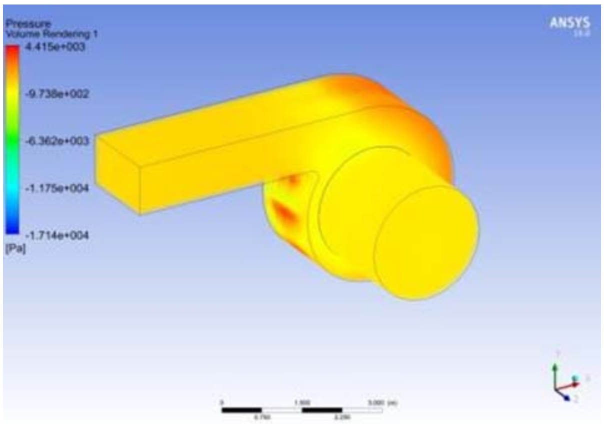

Fig 15: Pressure contour Volume of 10 Blade forward Curved

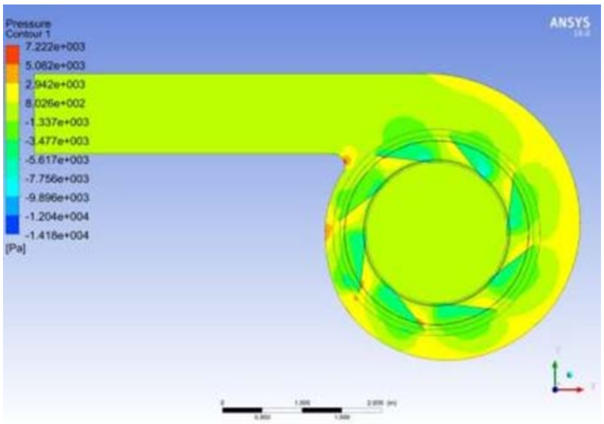

Fig 16: Pressure contour of 8 Blade forward Impeller

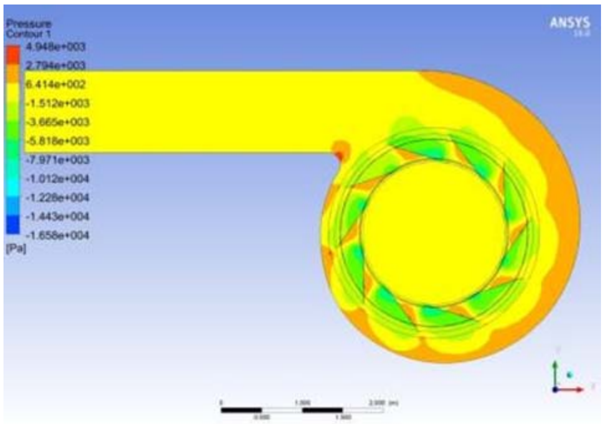

Fig 17: Pressure contour of 10 Blade forward Impeller

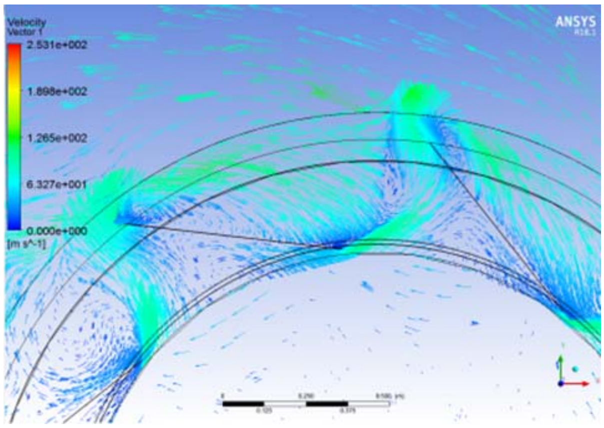

Fig 18: Velocity Vector of 8 Blade forward Impeller

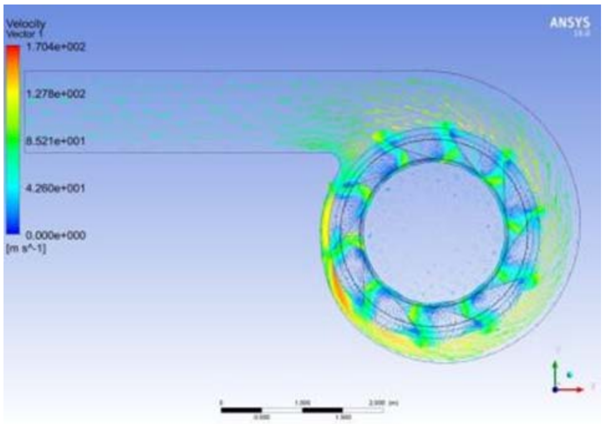

Fig 19: Velocity Vector of 10 Blade forward Impeller

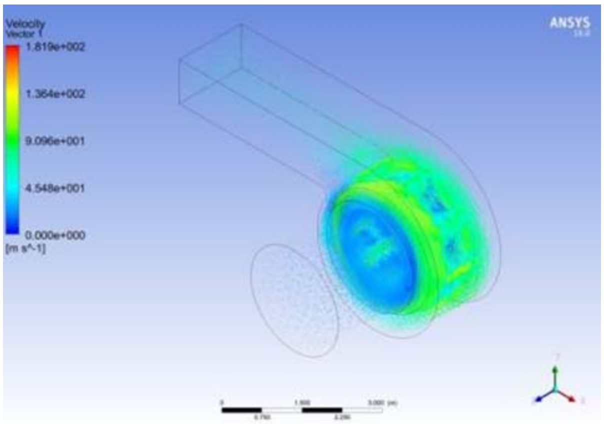

Fig 20: Velocity Vector Volume of 8 Blade forward Impeller

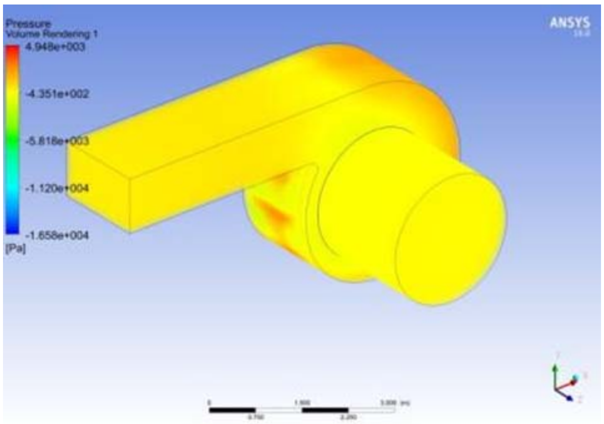

Fig 21: Pressure contour Volume of 10 Blade forward Impeller

Comparison of parameters developed from all simulations at each speed tabulated as below

S.No	Unit	Type	Speed (RPM)				
			646	**840**	**950**	**1174**	**1400**
1	Pa	[A]	137	228	272	421	572
		[B]	258	432	573	804	1130
		[C]	343	567	727	1110	1204
		[D]	346	590	744	1165	1595
2	m/s	[A]	59	77	87	109	130
		[B]	54	71	76	99	119
		[C]	55	72	78	101	118
		[D]	49	63	70	89	106
3	Kg/s	[A]	104	135	152	189	226
		[B]	92	120	134	168	200
		[C]	96	125	140	175	209
		[D]	86	113	126	158	188

Table 2: Comparison of Results

Type[A]: Forward Curved 10 Blades Impeller
Type[B]: Forward Curved 8 Blades Impeller
Type[C]: Forward 10 Blades Impeller
Type[D]: Forward 8 Blades Impeller

Graphical Representations:

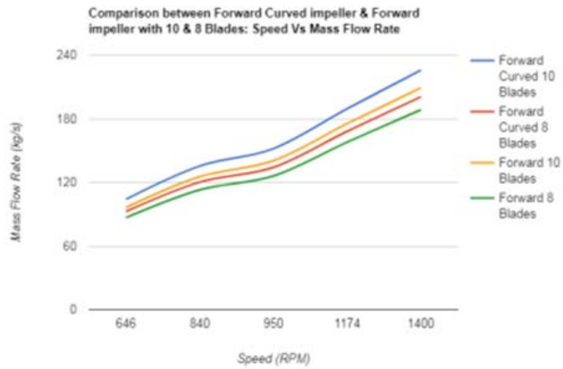

Fig 22: Speed Vs Mass Flow Rate

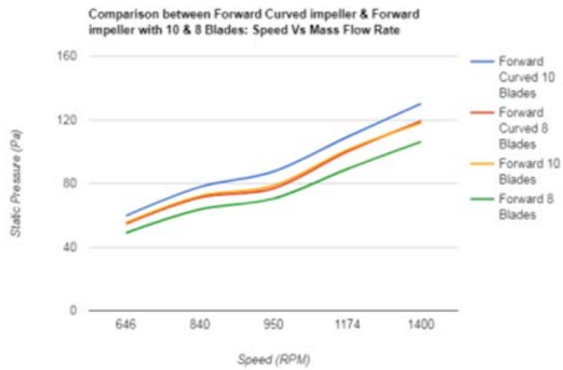

Fig 23: Speed Vs Velocity

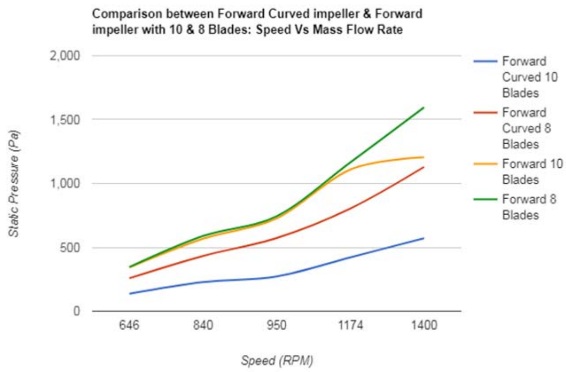

Fig 24: Speed Vs Static Pressure

From the above post-processed results of the case study obtained from CFD analysis, it is observed that the three parameters used for validation have shown good results. Hence, From the results for this case, it can be concluded that the CFD simulation/analysis has given better performance results with forward impeller having ten blades. This completes the validation of CFD Simulation provided that **forward impeller with ten b lades** has higher efficiency when compared to other centrifugal fan impellers

V. Conclusion

The three parameters Static pressure, velocity & mass flow rate obtained from CFD simulation analysis are correlated with all the types of impeller designs at variable speeds validation method. A Comprehensive simulation is performed providing a conclusive performance optimization of centrifugal fan achieved by **ten blades forward impeller** showing better static pressures with respective variable speeds and approaching nearer velocities of eight blade forward curved impeller. And it is also resulting that without curvature of forward blades showing better results than with curvature forward blades.

VI. References

1. Fan Handbook, Selection, Application and Design, Bleier
2. Catalogue V-3 2015, backward curved impeller single inlet, Saudi Fan Industries
3. computational fluid dynamics by t. j. chung.
4. Aerovent Technical Bulletin, 720, May **(2011)**
5. Bleier Frank P., Fan Handbook Selection, Application and Design, McGraw Hill publications **(1997)**
6. Eck, Bruno 'FANS'- Reference book on fan engineering, **(1975)**
7. Air and Gas Flow, Chapter Number 3, Book on Fans and ventilation
8. 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)**
9. 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)**
10. 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)**
11. Pathak Sunil, Turbocharging and oil techniques in light motor vehicle, *Research Journal of Recent Sciences,* **(2012)**
12. 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)**
13. 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), (2012)**
14. Balasubramanian P and Ramamurti V, Frequency Analysis of Centrifugal Fan Impellers, *Journal of Sound and Vibration*, **1**, 1-13 **(1987).**