RESEARCH ARTICLE OPEN ACCESS

Simulation of Flow and Structural Analysis of Industrial Blower

Rakesh Mididoddi*, A. Raja Sekhar**
*(M.E (CAD/CAM) Scholar, Department of Mechanical Engineering, Methodist College of Engineering and Technology, Hyderabad-500001 Email: rakeshmididoddi@gmail.com)
** (Professor, Department of Mechanical Engineering, Methodist College of Engineering and Technology, Hyderabad-500001 Email: arsekhar06@gmail.com)
Corresponding author: Rakesh Mididoddi

ABSTRACT
Industrial Blowers are used to transport air to different upward/downward components in power/process equipment assembly. However performance degradation during operation concerns designers for plant efficiency. To this effect performance analysis is aimed in three dimensional environments. The basic approach followed to obtain three dimensional CAD model of blower comprises suction duct, volute chamber, impeller and exit duct. The CAD model in the form of IGES format has been imported in Altair Hyper Mesh pre-processor for surface repair and fluid/solid domain extraction. The fluid domain has been further discretized in the form of unstructured and structured grids so as to generate tetrahedral and hexahedral elements respectively which in turn was imported in Ansys Fluent Software. With the prescribed boundary conditions such as inlet velocity, exit pressure and blade rotation, CFD simulation is performed to visualize the velocity and pressure distribution. Solid domain of blower impeller was imported in Ansys Mechanical to map pressure load from fluid simulation to compute deformations and principal stresses.
Keywords: Ansys, Blade rotation, Blower, CFD.

Date of Submission: 26 -07-2017 Date of acceptance: 05-08-2017

I. INTRODUCTION

Blowers are widely used in several engineering applications including process and power generation, refinery, petrochemical, pharmaceutical, heating, ventilating and air-conditioning applications. For the design and development of turbo machinery components like pumps, blowers, fans, turbines, etc., multi-disciplinary analysis is critical before development of prototype models [1]. During the operation of turbo machinery components the fluid dynamic perturbations are produced which may also alter vibration and noise levels. One of the major problems with the impeller blade design is whether the impeller blades are strong enough to withstand the hydrodynamic load. Some impeller blades have suffered failures, possibly due to the inadequate use of material and geometries or because the design load was not correctly estimated. When the internal pressure load acted on the blade reaches or exceeded the yield stress of the material, yielding and deformation occurred. Most frequently impeller blades yield and deform at the highly stressed region. This type of failure has critical importance because yield of impeller blades decreases the blade performance.

So the component develops performance degradations which are planned to be addressed in the proposed research investigation using three dimensional modeling and simulation techniques. In order to understand flow behavior and pressure rise parameters, CFD simulation is performed on the complete geometry involving blower casing as well as impeller blades. Structural simulation is also performed on the impeller blades to understand deformations and structural stresses.

The casing of the blower under consideration is of volute type and the impeller blades are of backward aero foil shape. The fluid enters at the centre of the impeller, turns through a right angle and as it moves outwards radially, is subjected to centrifugal force resulting increase in its static pressure. The airflow tends to drop drastically, as the system pressure increases is one of characteristic of centrifugal fan. Pressure Volume characteristic curve for a backward bladed centrifugal blower represents losses in the blowers [2]. Though the design and development of blowers with different configurations are discussed in the literature using advanced CFD techniques, they lack fluid-solid interaction effects [3-6]. Blowers can get failure due to vibration, lack of performance, excessive noise and premature component failure.

1.1 Experimentation and Methodology

CFD provides the fundamental improvements by simulating the performance of the turbo machinery component with the account of complete geometry. It also helps to reduce the design lead time by performing design cycles in software rather than hardware and improves the product performance by evaluation of more design alternatives in shortest time.

The first step in performing a CFD analysis is called pre-processing. This involves identifying the flow region of interest, geometrically representing the region, meshing and defining the flow physics. Once the region is defined, computer model of the geometry is created. Computational domain can be discretized with the choice of elements like tetra, hexahedral elements in "Altair Hyper Mesh". The mesh generation concepts were discussed, reviewed and resulted to several commercial softwares which are being extensively used for several industrial components [7-9]. Unstructured grid methods [10] utilize an arbitrary collection of elements to fill the domain automatically requires the volume bounded by error free surfaces. Structured grid generation of CFD simulation is explained by Bhasker [11]. The appropriate boundary conditions are applied to define the regions of inflow, outflow and moving walls. Physical models within the software are activated to simulate the flow physics especially turbulent flows can be done in "Ansys Fluent Solver". After initialization of flow field variables, the problem is submitted to solver. Solution is converged or stopped when the equation residuals are meeting the specified values and writes the result file.

This result file can be loaded in post-processing to visualize the flow characteristics in terms of vector plots, contour plots and streamlines. The post processor also provides fluxes and averaged quantities across the computational domain. Solid domain of blower impeller was imported into "Ansys Mechanical" to map pressure load from fluid simulation to compute deformations and principal stresses.

II. BLOWER MODEL FOR ANALYSIS

The blower is shown in Fig.1 comprises the casing with supports. The casing is of volute type with exit duct and inlet is on the suction plate which is placed on top of the blower casing. The impeller was designed with seven aero foil backward blades whose leading and trailing edges are filled with fillets.

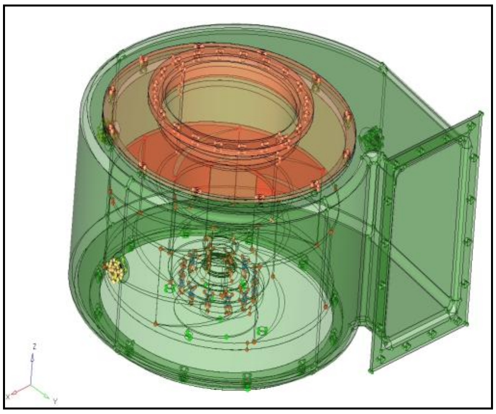

Fig 1 Blower model.

Table 1 Dimensions of blower.

1	Casing height	:	239.4 mm
2	Casing dia	:	360 mm
3	Inlet dia	:	212.0 mm
4	Inlet height	:	34.8 mm
5	Impeller height	:	98.4 mm
6	Impeller dia	:	354 mm
7	No.of aero foil blades	:	7
8	Pitch angle	:	56.429^0

The solid model used for flow simulation was comprises thickness, plugs, lining washers, casing bushes, casing lining washers, whose presence is not effected for the flow simulation. Hence, surfaces associated with these supports are deleted. As a result pin holes are present on casing at several locations need to be filled.

2.1 CFD Analysis:

Pre-processing: Pre-processing consists of the following steps. From the blower model configuration (fig.1):

Step 1: Extraction of casing from blower model.

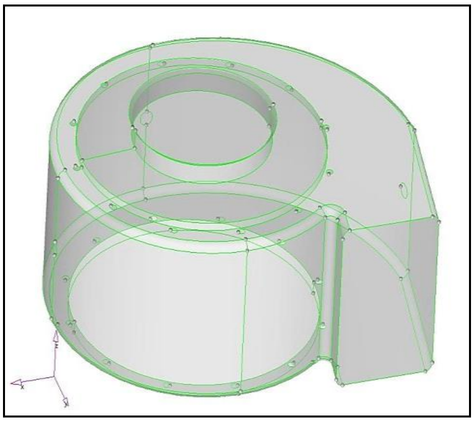

Fig.2 Topography of casing.

Step 2: Extraction of impeller from blower model.

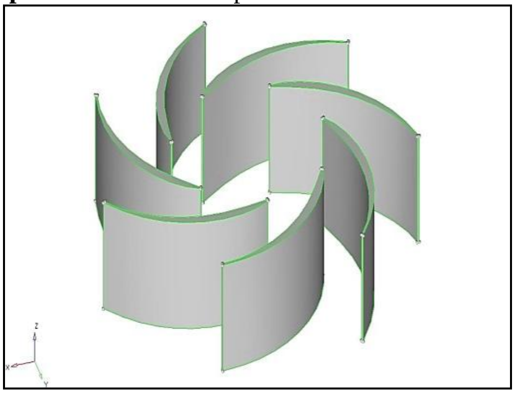

Fig.3 Topography of impeller.

Step 3: In separate group collector, using bounding surfaces, casing and blower are grouped together to form complete geometry of blower model.

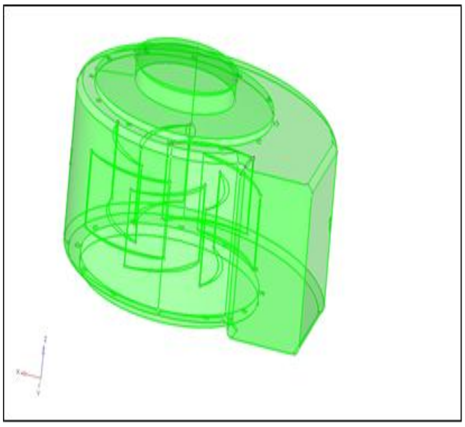

Fig.4 Grouping of casing and impeller.

Step 4: Unstructured grid mesh with boundary conditions.

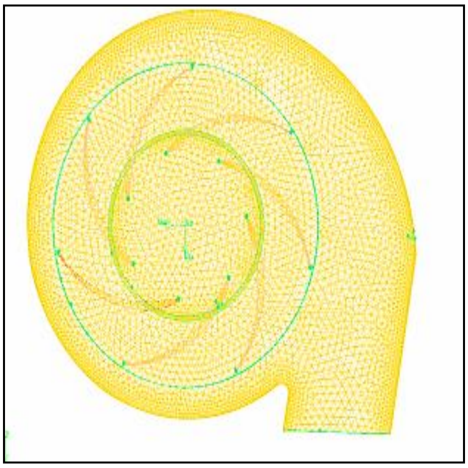

Fig.5 Unstructured grid mesh model with boundary conditions.

Step 5: Structured grid mesh of complete geometry with boundary conditions.

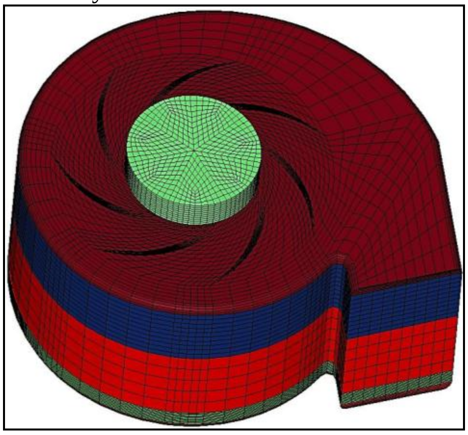

Fig.6 Structured grid mesh with Boundary Conditions.

Step 6: Simulation of blower model with unstructured grid.

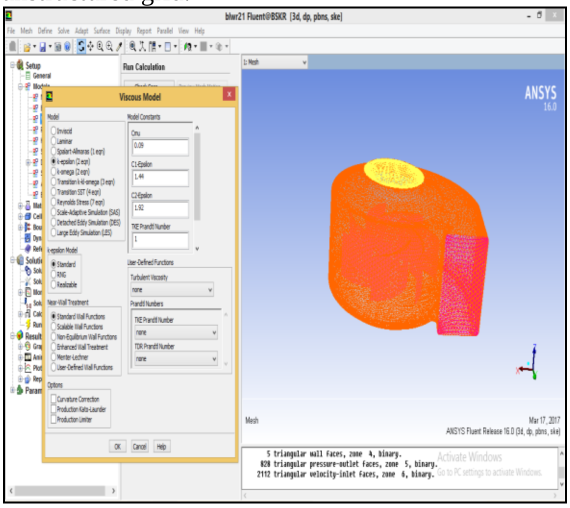

Fig.7 Critical parameters applied to the model.

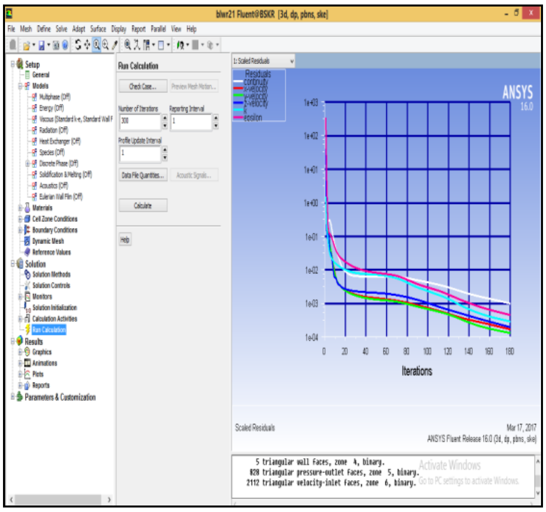

Fig.8 Convergence history of unstructured grid model.

The material considered is air. Generally operating pressures are 1 atm and its value in the "pressure out boundary condition" should be mentioned as zero. Under run calculations, mention about 300 iterations and profile update interval has made as 1. The unstructured grid with tetrahedral elements generated for blower model has taken 180 iterations to obtain numerical results for flow and pressure parameters when the blades are rotating with 2800 rpm along z direction.

Step 7: Simulation of blower model with structured grid.

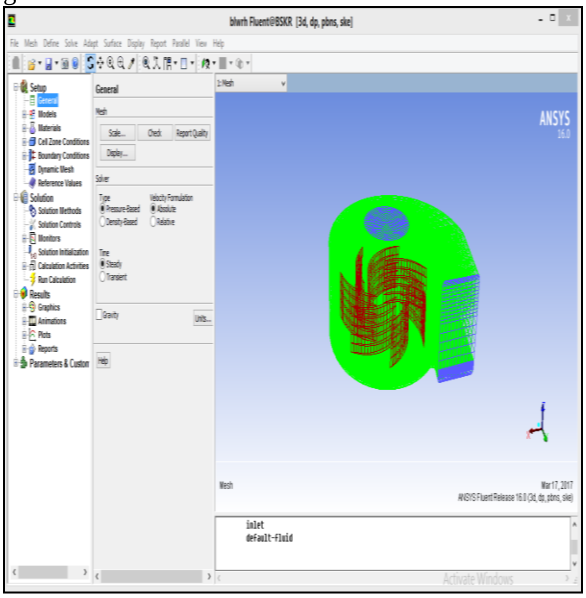

Fig.9 Structured grid model into Ansys Fluent Solver.

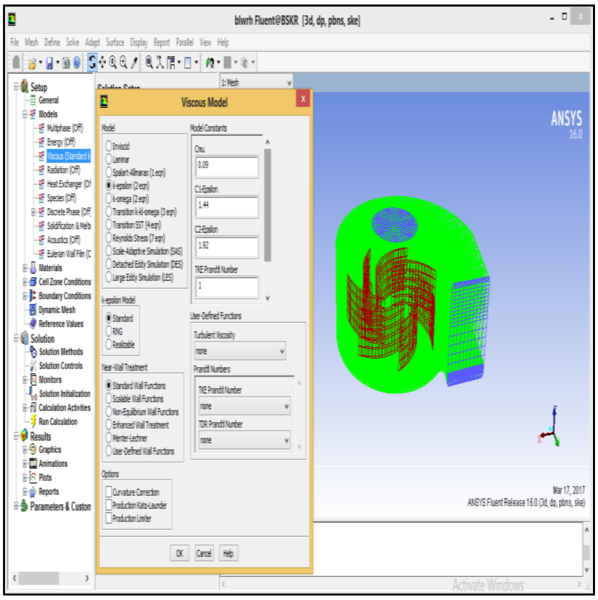

Fig.10 Critical parameters applied to the model.

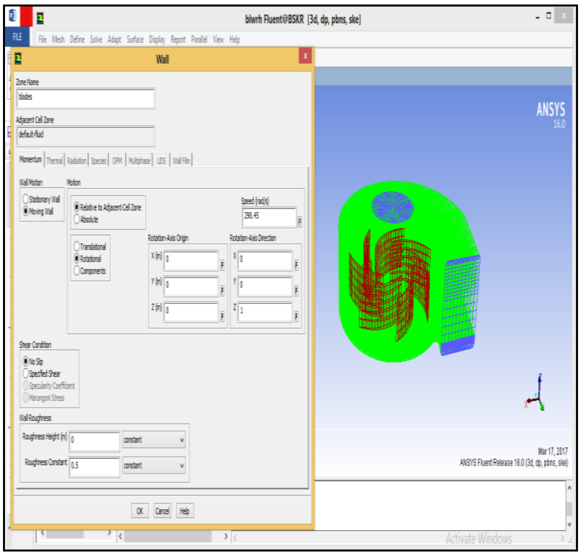

Fig.11 Boundary conditions (wall) applied to model.

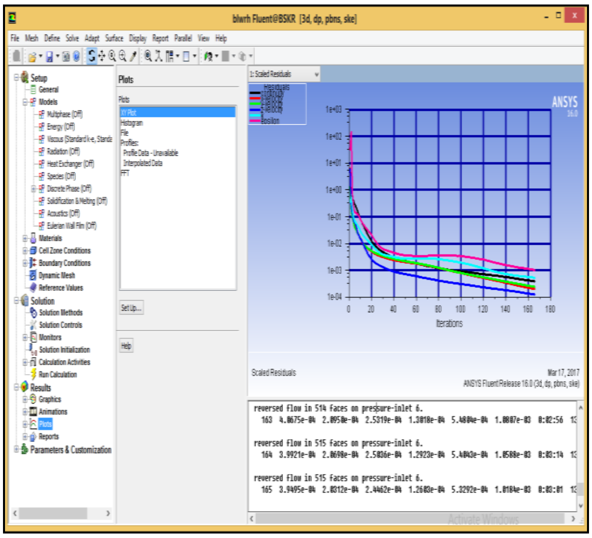

Fig.12 Convergence history of structured grid model.

The working fluid considered is air. Inlet velocity 9 m/sec, working pressure will be 1 atm and blades are rotating about z direction with 2800 rpm. When assigning the boundary conditions, the wall motion considered is as moving wall and blade motion considered is as rotatory. The convergence history of equation residuals in case of structured multi-block grid is faster than unstructured grid and took only 166 iterations.

Post-processing:

Discussion of results from CFD simulation:

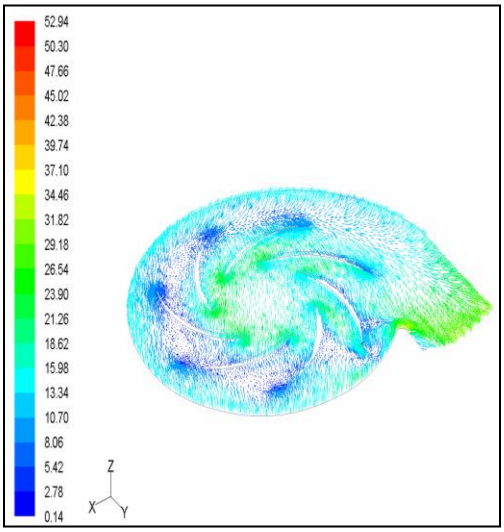

Fig.13 Describing velocity vectors in the middle plane of the impeller section.

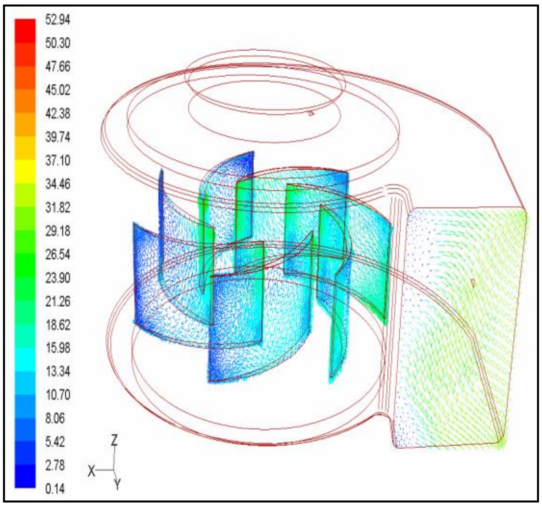

Fig.14 Velocity vector plots drawn on full blade surfaces to understand its effect on exit.

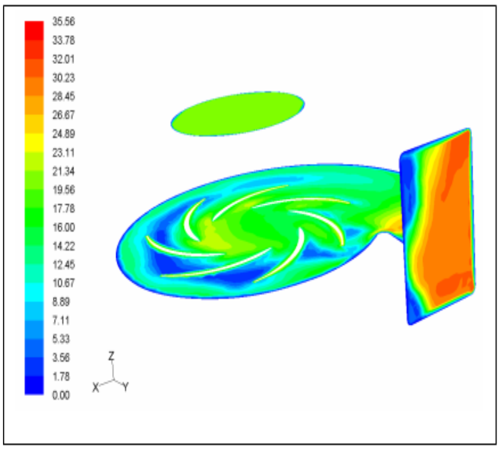

Fig.15 Contour plot indicating flow non-uniformity.

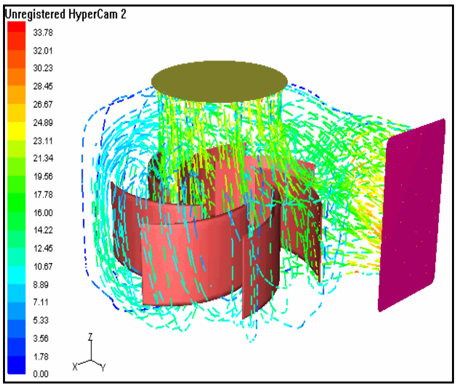

Fig.16 Flow path in the blower casing in the form of animated streams from inlet to exit.

Maximum velocity is taking place at blade leading edge and the minimum velocity is at blades trailing edges (fig.13) and is observed that low velocity of flow taking place on blades trailing edges side away from the exit location (fig.14). The flow through blade channels indicates low velocities and higher pressure drop. This kind of flow will have effect on the exit location which indicates flow non-uniformity is more clearly observed in contour plot (fig.15). Fluid flow from inlet to exit in the form of animated streams is representing the centrifugal forces due to rotation of blades changes the flow to move upwards and then leaves to exit location (fig.16). The flow also fluctuates with turbulent eddies near blade channels away from exit causing higher pressure drop and influence non-uniform flow at the exit location.

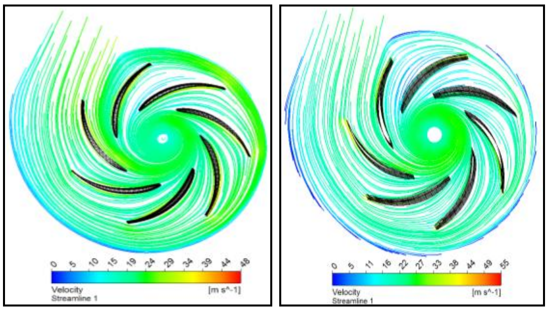

Fig.17 Surface Streamlines on the velocity scale from the suction to exit via impeller blades.

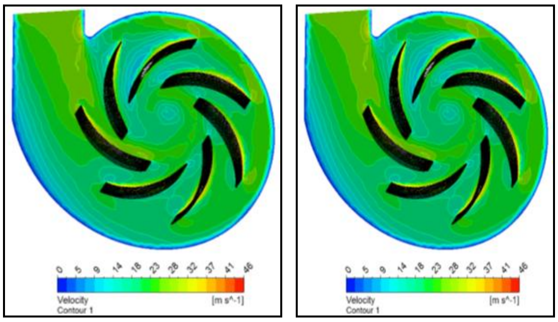

Fig.18 Flow recirculation in the blade passage away from exit.

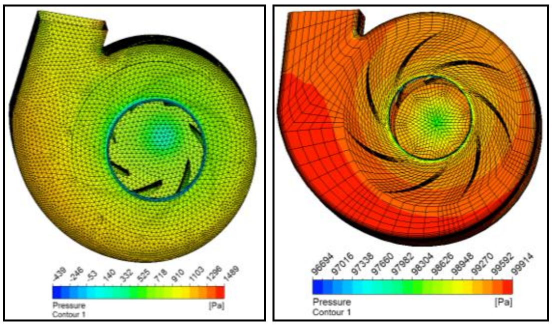

Fig.19 Contours of pressure distribution on the blade middle span and wall surfaces.

From the plots (fig.17 & fig.18), high velocity is noticed near the blade surfaces in impeller and low velocity in the volute area because the velocity is converting into pressure. The contours on velocity scale exhibit the non-uniform flow in cross section plane blade span along volute casing. The contours also reveal the high velocity regions near the impeller due to rotation of blades. As the flow moves away from the rotating wall, this high velocity however reduces. It is also observed that as the flow moves from suction to exit along the volute casing static pressure is increasing due to conversion of velocity into pressure (fig.19).

2.2 Ansys Mechanical Analysis:

Step 1: Mesh generation of impeller.

After creating the three-point spline curves for blade and tip, volumes created for blade surfaces. Then meshing has done, generally mapped mesh is preferred for impeller blade surfaces.

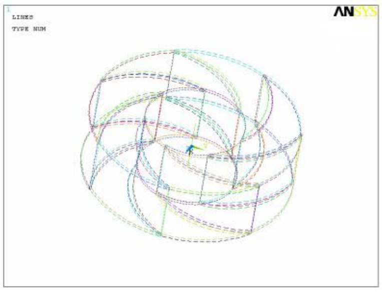

Fig.20 3-point spline curves for blade tip and hub.

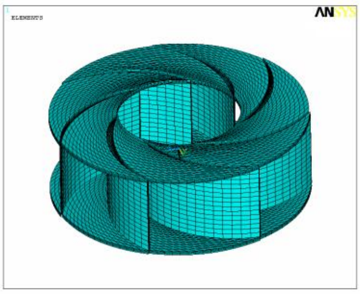

Fig.21 Mesh Generation for Impeller Blades.

Step 2: Structural simulation of blower impeller.

The geometry has to be scaled in SI units and preferences for linear structural analysis have to define with proper material properties like poisson ratio, young's modulus and density. The objective is to study the effect of internal pressure load and displacement constraints on impeller for stress analysis.

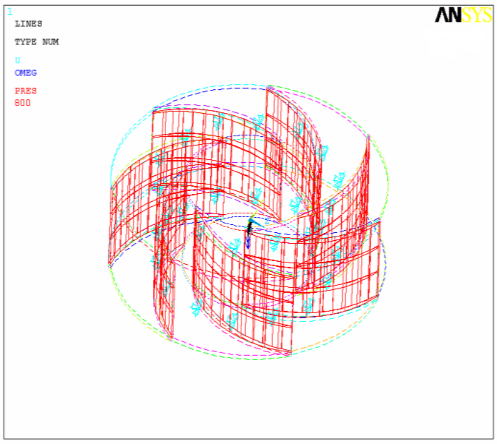

Fig.22 Critical parameters applied to impeller blades.

From the fluid flow analysis, on the blade surfaces, it is found that the average pressure load acting on the blades is about 800 Pascals.

Results from Ansys Mechanical:

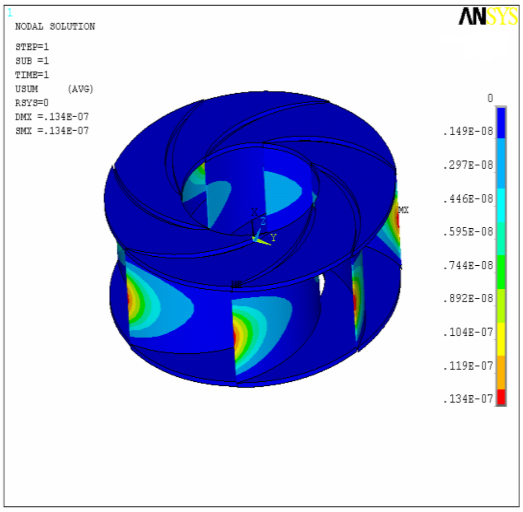

Fig.23 Deformed model with color map indicates highest displacement while blue shows the least.

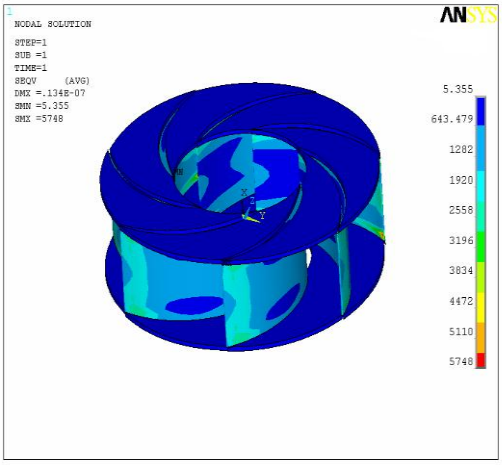

Fig.24 Equivalent stresses on blade surfaces shows maximum stress at blade trailing edges symmetrically on all blades.

It is observed that blades are deforming very small and maximum displacement is taking place at mid-section of blade towards trailing edge which decreases towards leading edge. The stress distribution on rest of the blade section and pressure surfaces is small and appears to be in limits.

III. MATHEMATICAL FORMULATION OF FLUID MECHANICS

The transport of fluid comprises gases/liquid from one component to other in power/process equipment are described through mass, moment and energy conservation principles. The Navier Stokes (transport) equations are derived from these principles are discussed and represented mathematically as [12]:

$$\frac{\partial}{\partial t}\int_V \rho\varphi dV + \oint_A \rho\varphi V.dA = \oint_A \Gamma\nabla\varphi.dA + \int_V S_\varphi dV \qquad (1)$$

Where ϕ describes the transport equation for x, y, z momentum when this variable assigns u, v, w in the above equation. If φ takes 1, equation (1) becomes continuity equation. The acceleration of fluid depends on the inertia, diffusion and sources terms. The equation (1) is further discretized into:

$$\sum_f \rho_f V_f.A_f\varphi_f - \sum_f \Gamma_f (\nabla\phi)_f.A_f = \bar{S}_f\Delta V \qquad (2)$$

Wherein diffusion term on right hand side of equation (1) is volume integral and is converted into surface integral using Green-Gauss theorem. Equation (2) after its reconstruction from cell centers forms the set of algebraic equations and can be solved for the variable φ iteratively. During the iteration process, imbalance of flow variables between left and right hand side of an equation is called Residual error (*R*) will minimizes to specified target, if the following criterion is satisfied:

$$R \equiv \sqrt{\frac{\sum_{i=1}^{N}(\varphi_i - \varphi_g)^2}{N}} \qquad (3)$$

Where φ with subscript g and i are assumed and currently calculated value; N is of Number of iterations.

3.1 Finite Element Method of Solids:

Finite element analysis is a computer based numerical technique for calculating the strength and behavior of engineering structures [13]. Structural analysis was carried using commercial solver Ansys Mechanical for which following mathematical relations are employed:

$$[K]\{u\} = \{F\} \qquad (4)$$

Where K is the system stiffness matrix; F load vector and u is displacement vector. The system stiffness matrix is obtained by assembling the element stiffness matrices consistent with the compatibility requirements between the global and local displacements. The state of stress of the idealized structure is obtained by compiling the results for the states of stresses of individual elements. The state of stress of an individual element is obtained by solving equation (8) for the system displacements. The compatibility equations between the system and element nodal displacements:

$$\{\delta\}_i = [\beta]_i\{u\} \qquad (5)$$

Where δ nodal displacement is vector and β is the compatibility matrix for the element I. The equation for the strains:

$$\{\epsilon\}_i = [B]_i\{\delta\}_i \qquad (6)$$

Where B is the strain displacement matrix and ε strain vector for the element i. The relation for computation of stresses:

$$\{\tau\}_i = [C]\{\epsilon\}_i \qquad (7)$$

Where C is the elastic constants matrix and τ is the stress vector for the element i.

IV. CONCLUSIONS

The simulation using structured grid data provides better insights (took 166 iterations) than the same observed in the simulation using unstructured grid (took 180 iterations).

The simulation results in both unstructured and structured grid method cases relatively addressed the non-uniform flow in blower volume with the flow velocity near impeller blades shows high and lower in volute section. Lower velocity in volute section converts into pressure and hence pressure rise takes place across exit and entry location.

The mechanical simulation of impeller blades with the average pressure load suggests (800 Pascals) that the component is structurally safe, as the deformations are small and equivalent stress determined on blades are within the acceptable limits.

ACKNOWLEDGEMENT

The authors would like to thank **Dr C.BHASKER**, (MD/Ex.sr Mgr (BHEL, R&D, Hyderabad) for his support in this work.

REFERENCES

[1]. Process design of Fans and Blowers, Design document, KLM Technology, April, 2001.

[2]. Takahashi, Characteristics of Centrifugal Blowers and its effective use in high static Pressure Area, Oriental Motor Co Ltd, 2002.

[3]. Tremmel M, Taulbee DB Calculation of the Time Averaged Flow in Squirrel Cage Blowers by Substituting Blades with Equivalent Forces, ASME Jr of Turbo machinery, Vol 130 PP 1-12,2008.

[4]. Huang C-K, Hsich M-E, Performance analysis and optimized Design of Backward-Curved Aerofoil Centrifugal Blowers, HVAC&R Research Magazine, Vol 15, pp 461-488,2009.

[5]. Pathak Y, Baloni D B and Channiwala SA, Numerical simulation of Centrifugal Blower using CFX, Intl.Jnl. of electronics, April, 2012.

[6]. Hsia S-Y, Chiu, S-M and Cheng J-W, Sound field analysis and simulation of fluid machines, Adv.in Engg.software, Vol.40, 2009.

[7]. Thomson, J.F et.al Numerical Grid Generation – Foundations and Applications, North Holland, Amsterdam, 1985.

[8]. Shekar Majumdar Pressure based finite volume algorithm for viscous flow computation, Lecture Notes, CFD Advances and Applications, NAL, Bangalore, India, 1994.

[9]. Bhasker C CFD applications for Industrial Flow Simualtions – Invited talk, SONIC-10 Conference, International Institute of information Technology, Pune, India, 2010(a).

[10]. Shevare, G Issues in Unstructured Mesh Generation, Lecture notes, 1994.

[11]. Bhasker, C,Simulation of Three Dimensional Flows in Industrial Components using CFD Techniques, chap-8, Computational Fluid Dynamics Technologies/Applications Editedby Igor V. Minin and Oleg V. Minin, InTech, Rijeka, Croatia.

[12]. Hoffman, K.A et al Computational Fluid Dynamics for Engineers, in two volumes, Engineering Education System, Wichita, KS, 67208-1078. 1993.

[13]. Mohd Zubair Nizami, et al Evaluation of static and dynamic analysis of centrifugal blower using FEA. Procgs of International Journal of Advanced Trends in Computer Science and Engineering, Vol.2 , No.1, Pages : 316 - 321 (2013).

International Journal of Engineering Research and Applications (IJERA) is **UGC approved** Journal with Sl. No. 4525, Journal no. 47088. Indexed in Cross Ref, Index Copernicus (ICV 80.82), NASA, Ads, Researcher Id Thomson Reuters, DOAJ.

Rakesh Mididoddi " Simulation of Flow and Structural Analysis of Industrial Blower" International Journal of Engineering Research and Applications (IJERA) 7.8 (2017):82-89