Optimization of Backward Curved Aerofoil Radial Fan Impeller using Finite Element Modelling

M. Mohammed Mohaideen

Assistant Professor, Department of aeronautical engineering,
Email: mohdmohaideen@yahoo.co.in

Abstract

This paper deals with optimizing the design of radial fan impeller using finite element analysis. Most manufacturing concerns spend more than 60% of their money for materials. The design of turbo machinery has been practiced in the last half of the previous century with increasing degree of sophistication. This trend of development is not complete because design of any turbo machine is an interdisciplinary process involving aerodynamics, thermodynamics, fluid dynamics, stress analysis, vibration analysis, the selection of materials, and the requirements for manufacturing. So, products have not reached a level of full maturity. Today there is considerable room for further development of most turbo machinery products when looked at with the perspective of the broadest possible design world and when it comes to development at the design stage then various types of analysis of every component are performed on computer before the first prototype is built. Among these the major one end the most frequently used in the manufacturing of any mechanical part is stress analysis. As the mass of the fan is directly related to the thickness of the impeller parts hence our objective in this project is to minimize the thickness, keeping in consideration the operating restrictions and design parameters.

This work concentrates on to generate the finite element model of backward curved aerofoil radial fan impeller which behaves mathematically like structure model & carry out stress analysis using ANSYS software. The analysis was carried out using the default thickness of the impeller of the radial fan. This results in massive weight of the fan, leading to large vibrations and one of the reasons to failure. Ways were found out to reduce all these things, so the analysis was carried using various reduced thicknesses of the impeller and the optimum thickness of the impeller parts found for the safe stress and strain limits.

This value analysis/Engineering results in material reduction, cost reduction with reduced vibrations for the desired design and operating conditions. The basic procedures and issues involved in the Finite Element Analysis of a radial fan are also outlined. The design configuration of the radial fan has been provided by Boiler Auxiliaries Plant.

© 2012 Published by Elsevier Ltd. Selection and/or peer-review under responsibility of Noorul Islam Centre for Higher Education

Open access under CC BY-NC-ND license.

Keywords: backward curved radial tipped centrifugal fan; unified design; finite element Analysis;
1. Introduction

Finite element method provides the solution domain into simply shaped regions or elements. An approximate solution for partial difference equation can be developed for each of these elements. The total solution is then generated by linking together or assembly the individual solutions taking care to ensure continuity at the inter – element boundaries. Thus, the partial difference equation is satisfied in piecewise fashion.

The finite element method is endowed with three basic features that account for its superiority over other competing methods:

Geometrically complex domain of the problem is represented as a collection of geometrically simple subdomains called ‘finite elements’. Over each finite element the approximation, functions are derived using the basic idea that any continuous function can be represented by a linear combination of algebraic polynomials.

Algebraic relations among the undetermined co-efficient (i.e. nodal values) are obtained by satisfying the governing equation often in a weighted integral sense over each element.

1.1 Selection of Software for Modeling and Stress Analysis

The accurate result of the software analysis depends upon the effectiveness and simplicity of analysis software together with its speed and simple user interface. Therefore among many options, Ansys 5.4 is selected for modeling and stress analysis of backward curved radial tipped centrifugal fan which is having variety of modules for different tasks.

1.2 Finite Element Model

For finite element modeling the simulation module of program is used. Finite element programme divides the element into a grid of ‘element’, which form a model of the real structure.

A finite model is complete idealization of the entire structural problem, including the node locations, the elements, physical and materials properties, loads and boundary condition. The model is supposed to be defined differently for different types of analysis: static structural loads, dynamics or thermal analysis. The goal of finite element model is not to make a model look like the structure. The purpose of finite element modeling is to make a model that behaves mathematically like the structure modeled, not necessarily one looks like the real structure.

Finite Element Modelling and Stress Analysis

It consists of three steps. Which are?
1. Pre-processing,
2. Solution
3. Post processing.
2.1 Pre processing
This includes the entire process of developing the geometry of a finite element model, entering physical and material properties, describing the boundary conditions and loads and checking the model. For variable thickness of the impeller, first made model the ‘surface model’ along with ‘split surface’.

2.1.1 Entering the material properties of forward curved radial tipped impeller
- Modulus of elasticity $E = 20,680$ Kg/mm$^2$
- Poisson’s ratio $\nu = 0.3$
- Density $\rho = 7,850$ Kg/mm$^2$
- Shear modulus $G = 7953.84$Kg/mm$^2$
- Power=1500 KW
- Fan size= NDZV 20 BAB2
- Speed= 1000 rpm
- Pressure ~ 9850 Nm/Kg
- Volume= 50 m$^3$/s
- Material Used = Naxtra 70

2.1.2 Meshing
Nodes and elements are generated by one of the two methods, mapped Mesh or free mesh. Here the free mesh is used for meshing because of more flexibility in defining mesh areas, it will automatically create by an algorithm, which tries to minimize element distortion & can easily have internal holes which is shown in figure 1.

![Fig 1 Mapped mesh of the impeller model](image)

2.1.3 Boundary Condition
The boundary condition is applied to build analysis cases containing loads and restraint of the model .in finite element analysis, the model is considered to be in equilibrium. So the loads and moments should be such that the condition of $\Sigma F =0$ and $\Sigma M =0$ should satisfied.
- Structural Loads:
- Structural loads can be nodal forces or pressure on face or edge of an element. A nodal force has six values, for three forces and three moments.
2.2 Solution
The solution phase can be performed in the Model solution task of the simulation application, or an external finite element analysis program. Model solution can solve linear and non-linear static, dynamics, buckling, conduction heat transfer and potential flow analysis problems.

The model solution task in the simulation applications is where the finite element model is solved. In this case, the solution set is created for obtaining displacements and stress as the output data. Then model is solved using the 'solve command'. After the solution displacements will be stored in one dataset and stresses will be another.

Fig 2. Original fan impeller deflection plot

Fig 3. Original fan impeller stress plot

2.3 Post Processing
From the type of material, the yield stress was known.

Now by von mises theory the induced stress should be less than 0.66

Permissible yield stress $\sigma_x = 0.577$, (yield stress) = $248.26 \text{ N/mm}^2$

These involves plotting deflections and stresses and comparing these results with failure criteria on the design such as maximum deflection allowed, the material static and fatigue strengths etc. The post processing task of the simulation application provides tools to display and interpret the results after the solution is finished. Result of variable thickness impeller with different angular velocity can also be brought in from external finite element analysis for post processing which is shown in figure 4.
From this analysis, the maximum value of stresses obtained forward curved radial tipped impeller with different angular velocities for various thickness of material like 2 mm, 1.5 mm, 1.0 mm explained in result and discussion and values were compared with above limiting value of stresses.

3. Result and Discussion

3.1. Static analysis

The stress distribution and the deflection of the impeller are found. The stress distribution and the deflection plot for the various components of the impeller are plotted in the figures.

3.2. Optimization

For the original fan impeller the stress value is 4.274kgf/mm² and deflection is .0678mm. The blade in the impeller has the maximum stress of 4.274kgf/mm².

Conclusion

In this work, an attempt has been made to increase the Fan efficiency by optimizing the thickness of the various components in the fan impeller, and analyzing the stress distributions in them. Optimization of the thickness of the parts of impeller leads to decrease in weight of the Fan Impeller, and in turn the power required for driving the fan decreases. Pre-stress conditions are applied to this model, therefore the strengthening and weakening of the impeller is predicted.
After optimizing the thickness of the Fan impeller, the 18.5% weight of the Fan Impeller is reduced.

| components  | original fan |  | optimized fan |  |
|-------------|--------------|---|---------------|---|
|              | stress (kgf/mm²) | deflection (mm) | stress (kgf/mm²) | deflection (mm) |
| fan impeller | 4.274 | 0.0078 | 3.822 | 0.09945 |
| backplate (bottom) | 1.284 | 0.0126 | 1.291 | 0.016396 |
| backplate (top) | 2.318 | 0.03426 | 2.393 | 0.041143 |
| blade | 4.274 | 0.0678 | 5.822 | 0.099455 |
| cover plate | 2.832 | 0.0526 | 3.091 | 0.064376 |
| ring | 2.927 | 0.0845 | 3.159 | 0.057353 |
| flange | 38.433 | 0.0198 | 581.387 | 0.02394 |
References

[1] John F. Abel, Chandra Kant S Desai (1987), ‘Introduction to the Finite Element Method’- CBS Publishers and distributors, New Delhi.
[2] Kalyanmoy Deb (2000), ‘Optimization for engineering Design’- Prentice hall of India (p) Ltd, New Delhi.
[3] Krishnamoorthy C.S. (1994), ‘Finite Element Analysis’ -Tata McGraw-Hill publishing company, New Delhi.