Design and Analysis of Engine Cooling Fan

S. Visshnu, S. Madhu*
Department of Automobile Engineering, Saveetha School of Engineering, Chennai – 602 105

*Corresponding author: E-Mail: mathumarine@gmail.com

ABSTRACT

Car motor cooling framework deal with overabundance heat delivered in motor. This warmth delivered amid burning of powers in motor to do mechanical work. For that warmth, cooling fan are fundamental part of motor cooling framework. Working of cooling fan is to scatter abundance heat created by burning of fills. This paper is about advancement configuration of radiator fan (cooling fan) and breaking down the quality of sharp edges, static weight and different speeds created in it. Elements utilizing as a part of this paper is Finite Volume Method (FVM) accessible in agreeing examining programming.

CATIA procedures can be to give knowledge to fan outline process. Computational Fluid Dynamics (CFD) strategies used to breaking down procedure. The configuration of fan directed, presentation with computing measurements and make into a model then took after by examination in ANSYS procedures. CFD innovation has been given as effective commitment of improvement of motor cooling framework. Consequently, this innovation is broadly utilized as a part of early periods of the vehicle improvement. The outcome got from configuration and systematic investigations of fruitful outline.

KEY WORDS: radiator fan, design-CATIA, fluent, structural analysis, ANSYS APDL, ANSYS WORKBENCH.

1. INTRODUCTION

An internal combustion engine produces power by burning fuel within the cylinders; therefore, it is often referred to as a heat engine. Engines that produce their energy by heat and combustion have a problem of maintaining safe operating temperatures. Thirty to thirty five percent of the heat produced in the combustion chambers by the burning fuel is dissipated by the cooling system along with the lubrication and fuel systems. Forty to forty-five percent of the heat produced passes out with the exhaust gases. If this heat were not removed quickly, valves would burn and warp, lubricating oil would break down, pistons and bearing would overheat and seize, and the engine would soon stop. The necessity for cooling may be emphasized by considering the total heat developed by an ordinary six cylinder engine. It is estimated that such an engine operating at ordinary speeds generates enough heat to warm a six-room house in freezing weather. Also, peak combustion temperatures in a gasoline engine may reach as high as 4500°F, while that of a diesel engine may approach 6000°F. The valves, pistons, cylinder walls, and cylinder head, all of which must be provided some means of cooling to avoid excessive temperatures.

The axial flow fan is extensively used in many engineering applications. This type of fan is used in a wide variety of applications, ranging from small cooling fans for electronics to the giant fans used in wind tunnels. Axial flow fans are applied for air conditioning and industrial process applications. Its adaptability has resulted in implementation into large scale systems, from industrial dryers to automotive engine cooling and in-cabin air recirculation systems. The extended use of axial flow fans for fluid movement and heat transfer has resulted in detailed research into the performance attributes of many designs. Numerical investigations have been performed to quantify the performance of axial fans and their flow characteristics. Axial fans blow air along the axis of the fan, linearly, hence their name. The axial-flow fans have blades that force air to move parallel to the shaft about which the blades rotate. With the expressive computer capability and extensive development in the simulation field, CFD have drawn attention in recent years. With the help of CFD, the complex 3-D geometries of equipment can now be modeled with only minor simplifications. CFD models, if created correctly, can account for the complex flows in equipment. CFD models for axial fans have been used to evaluate the flow behaviour and characteristics. The models provide sufficiently accurate predictions over a range of operating conditions, which are not possible using other methods. In this paper, CFD was used to analyse the fan. The material selection also studied for metals and alloys.

Figure 1 shows the CAD 3D model of the fan. Design of fan at rated engine idle condition, Fan speed = 1.3 * 300 = 390 rpm.

![Figure.1. CAD model of the fan](image-url)
2. MATERIALS AND METHODS

Material: The first step is to identify a typical axial flow fan that can be reproduced as a 3-D CAD CATIA V5R17 software engineering drawing package (Fig.1). The 3-D models are then imported into the CFD software, remodelled into different sections, and refined to generate a finite volume meshing. This is a crucial step, where details of the geometrical shape need to be defined precisely. The flow domain is also created (Fig.2), and the final meshing of all components needs to be accurate. Any errors in the drawings and flow area need to be corrected before continuing. For the CFD analysis canvas materials was selected as the material.

Method:

Pre-Processor – ANSYS ICEM CFD: The generic working process involves the following:
- Open/Create a project
- Create/Manipulate the geometry
- Create the mesh
- Check/Edit the mesh
- Generate the input for the solver
- Post process the results

Post process the outcomes:

Tetra: The ANSYS ICEM CFD Tetra meshers takes full preferred standpoint of item arranged unstructured cross section innovation. With no dull in advance triangular surface cross section required to give all around adjusted introductory lattices, ANSYS ICEM CFD Tetra works straightforwardly from the CAD surfaces and fills the volume with tetrahedral components utilizing the Octree approach. An effective smoothing calculation gives the component quality. Choices are accessible to naturally refine and coarsen the cross section both on geometry and inside the volume. A Delaunay calculation is additionally included to make tetras from a current surface lattice furthermore to give a smoother move in the volume component size.

Post-Processor and Solver: In the isolated calculation the overseeing conditions are solved successively, isolated from each other, while in the coupled calculation the force conditions and the weight based congruity condition are illuminated in a coupled way. When all is said in done, the coupled calculation altogether enhances the meeting speed over the isolated calculation; be that as it may, the memory necessity for the coupled calculation is more than the isolated calculation. The coupled solver tackles the overseeing conditions of coherence, force, and (where suitable) vitality and species transport at the same time (i.e., coupled together). Representing conditions for extra scalars will be settled consecutively (i.e., isolated from each other and from the coupled set) utilizing the system depicted for the isolated solver. Since the representing conditions are non-straight (and coupled), a few emphasess of the arrangement circle must be performed before a merged arrangement is acquired. In both the isolated and coupled arrangement strategies the discrete, non-straight administering conditions are linearized to deliver an arrangement of conditions for the needy variables in each computational cell. The resultant direct framework is then comprehended to yield an overhauled displayed arrangement. The way in which the administering conditions are linearized may take a certain or express frame regarding the needy variable (or set of variables) of interest. By verifiable or unequivocal we mean the accompanying

Implicit: For a given variable, the obscure quality in every phone is processed utilizing a connection that incorporates both existing and obscure qualities from neighboring cells. In this manner every obscure will show up in more than one condition in the framework, and these conditions must be understood at the same time to give the obscure amounts.

Explicit: For a given variable, the obscure worth in every cell is registered utilizing a connection that incorporates just the current qualities. Subsequently every obscure will show up in one and only condition in the framework and the conditions for the obscure worth in every cell can be unraveled each one in turn to give the obscure amounts.

Solver: Customarily, these conditions have been unraveled in an isolated manner utilizing some variety of the SIMPLE calculation to couple the mutual weight with the force equation. This is achieved by successfully changing the aggregate progression into a common weight. The ANSYS FLUENT Phase Coupled SIMPLE calculation has been effectively executed and illuminates an extensive variety of multiphase streams. Be that as it may, coupling the linearized arrangement of conditions in a certain way would offer a more vigorous other option to the isolated methodology.

The feasible k-ε mode is a moderately late improvement. A prompt advantage of the feasible k-ε model is that it all the more precisely predicts the spreading rate of both planar and round planes. It is likewise prone to give better execution than streams including revolution, limit layers under solid unfavorable weight inclinations, partition, and distribution.
3. RESULTS

On post-processing the numerical CFD results, the observations are presented as velocity and pressure streamlines, static, dynamic and total pressure were analysed for the canvas material. Optimized design results were then compared with initial design for temperature contour and velocity streamline and presented in the form of contour plots.

From the figure.2 it can be noticed that the static pressure obtained from the analysis was $690899.81$ Pascal. And from the figure.3 it can be noticed that maximum dynamic pressure was $1668749.1$ pascal. Also the total pressure obtained was $1702889.8$ Pascal. From the figure.5 the maximum of $2.5$ m/sec velocity obtained from the analysis. Figure.6 explains the scaled residuals.

4. CONCLUSION

The results from the numerical simulations provided an insightful understanding of the behavior of flow around an axial fan with different material. CFD analysis was performed for both initially designed and optimized designed axial fan. The numerical CFD results for optimized design were then compared with initially designed axial fan. The key and important outcomes of this study are as follows:

a) The CFD modelling shown in this study proved to be very helpful in initiating further and more comprehensive numerical study of the axial fan.

b) CFD results were presented in the form of velocity streamlines, which provided actual flow characteristic air around the fan for different number of fan blades.

c) The different parameters like temperature, pressure, fan noise, and turbulence, were also considered while performing CFD analysis.

The study revealed that a fan with an optimum material compared to the fan with less number of fan blades.

REFERENCES


