

Design and Analysis of Engine Cooling Fan

Tushar .C. Ambdekar^{A*}, Shivprakash B. Barve^A, B. S. Kothavale^A and Nilesh T. Dhokane^A

^AMechanical Engineering Department, MIT College of Engineering, Kothrud, Pune

Accepted 10 March 2014, Available online 01 April 2014, **Special Issue-3, (April 2014)**

Abstract

Engine cooling fans are an essential component of the engine cooling system which is used to dissipate the excess heat generated by the combustion of fuels inside the engine. This project consists of designing the fan and analyzing it for its strength in structure using the Finite Element Method (FEM) approach and the flow of air around it using Computational Fluid Dynamics (CFD) approach. An important factor for successful fan design is to fully understand the complex flow field within the fan. This understanding enables optimization. Computational Fluid Dynamics (CFD) techniques can be beneficial to provide insight to the fan design process. Not only can velocity vectors and pressure contours be computed, CFD can also provide critical information such as efficiency, flow uniformity at the exit, circulation or separation zones, locations of potential cavitations and noise generation. The design of the fan was conducted in phases, starting with calculating all the required dimensions followed by analytical models to prove the concept. Accordingly, finite element analysis and computational fluid dynamics were performed on simulation models. The overall shape of the first iteration was conceptualized with the aid of the above guidelines. The results obtained from the analytical studies indicated a potential for a successful design that met most of the above outlined parameters.

Keywords: Axialfan, ICEM CFD, Fluent, Structural Analysis, ANSYS APDL, ANSYS WORKBENCH

1. Introduction

Engine Cooling Systems Evolution & Its Necessity

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, absorb some

of this heat. Even though heated gases may reach high temperatures, the cylinder wall temperatures must not be allowed to rise above 400°F to 500°F. Temperatures above this result in serious damage as already indicated. The cooling system has four primary functions. These functions are as follows:

1. Remove excess heat from the engine.
2. Maintain a constant engine operating temperature.
3. Increase the temperature of a cold engine as quickly as possible.
4. Provide a means for heater operation.

2. Design

2.1 Overall Design

Select operational speed.

Designing the fan at rated engine speed: [engine rpm = 1650]

$$\begin{aligned}\text{Fan Speed} &= 1.3 * \text{engine rpm} \dots (\text{assumed}) \\ &= 1.3 * 1650 \\ &= 2145 \text{ rpm}\end{aligned}$$

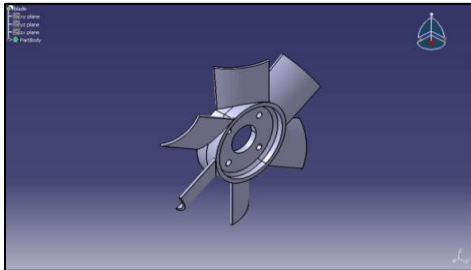
By calculating all the designing parameters such as Number of Blades, Blade Angle, Noise, Solidity of Blade, Blade Width for design of blade by taking assumptions we have got the following Blade angles.

*Corresponding author: Tushar .C. Ambdekar

Table 1 Blade angles

	At hub	At mid	At tip
$\alpha = \Phi$	1.5 ⁰	2.2 ⁰	3 ⁰
$\beta = \theta$	53 ⁰	54.6 ⁰	56 ⁰

2.2 Fan Model

**Fig.1** Fan Model created in CATIA V5R19

3. Pre-Processor – ANSYS ICEM CFD

3.1 Overall Process

The generic working process involves the following:

1. Open/Create a project.
2. Create/Manipulate the geometry.
3. Create the mesh.
4. Check/Edit the mesh.
5. Generate the input for the solver.
6. Postprocess the results

Meshing

The meshing modules available include the following:

Tetra

The ANSYS ICEM CFD Tetra mesher takes full advantage of object-oriented unstructured meshing technology. With no tedious up-front triangular surface meshing required to provide well-balanced initial meshes, ANSYS ICEM CFD Tetra works directly from the CAD surfaces and fills the volume with tetrahedral elements using the Octree approach. A powerful smoothing algorithm provides the element quality. Options are available to automatically refine and coarsen the mesh both on geometry and within the volume. A Delaunay algorithm is also included to create tetras from an existing surface mesh and also to give a smoother transition in the volume element size.

4. Post-Processor and Solver

4.1 Boundary Conditions Used In Ansys Fluent

4.1.1 Coupled solver and explicit formulation

In the segregated algorithm the governing equations are

solved sequentially, segregated from one another, while in the coupled algorithm the momentum equations and the pressure-based continuity equation are solved in a coupled manner. In general, the coupled algorithm significantly improves the convergence speed over the segregated algorithm; however, the memory requirement for the coupled algorithm is more than the segregated algorithm. The coupled solver solves the governing equations of continuity, momentum, and (where appropriate) energy and species transport simultaneously (i.e., coupled together). Governing equations for additional scalars will be solved sequentially (i.e., segregated from one another and from the coupled set) using the procedure described for the segregated solver. Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. In both the segregated and coupled solution methods the discrete, non linear governing equations are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated modelled solution. The manner in which the governing equations are linearized may take an implicit or explicit form with respect to the dependent variable (or set of variables) of interest. By implicit or explicit we mean the following:

Implicit: For a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighbouring cells. Therefore each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities.

Explicit: For a given variable, the unknown value in each cell is computed using a relation that includes only the existing values. Therefore each unknown will appear in only one equation in the system and the equations for the unknown value in each cell can be solved one at a time to give the unknown quantities.

4.1.2 Realizable k - ϵ viscous turbulence model

The realizable k - ϵ model is a relatively recent development. An immediate benefit of the realizable k - ϵ model is that it more accurately predicts the spreading rate of both planar and round jets. It is also likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation. The realizable model provides the best performance of all the - model versions for several validations of separated flows and flows with complex secondary flow features. This model has been extensively validated for a wide range of flows including rotating homogeneous shear flows, free flows including jets and mixing layers, channel and boundary layer flows, and separated flows. For the solution of the turbulence, RNG k - ϵ model is selected. This model is applicable to complex shear flows involving rapid strains, moderate swirl, vortices and locally translational flows (e.g.,

boundary layer separation, massive separation, room ventilation etc.). After the boundary conditions are set, the solution and the turbulence model are specified. The solution is obtained by segregated solver with absolute velocity formulation, three dimensions in space and steady in time. The turbulence model is selected as RNG k- ϵ model and the swirl dominated flow feature is activated to enhance the accuracy for this application. Standard wall functions are used for near wall treatment. For the numerical solution of momentum, turbulence kinetic energy and turbulence dissipation rate equations, first order upwind discretization scheme is selected. Because the flow across the fan has high rates of swirl and turbulence, and unstructured mesh is constructed in the solution domain, the flow is not aligned with the grid, thus second order discretization is preferred for higher accuracy. Linear option is selected for the pressure interpolation scheme that simply averages the pressures in adjacent cells to obtain face pressure values. To obtain the pressure field, SIMPLE algorithm is used under pressure-velocity coupling drop-down list. This algorithm uses a relationship between pressure and velocity corrections to enforce mass conservation and obtain pressure field. The details of the algorithm can be found in literature. Another thing to be controlled for the solution is the under-relaxation factors. Because of the non-linearity of the equation set being solved by the segregated solver, it is necessary to control the change of solution variables at each iteration, which is typically done by under-relaxation. For most flows, the default under-relaxation factors do not usually require modification. If unstable or divergent behavior is observed, however, the under-relaxation factors for pressure, momentum, k and ϵ from their default values may need to be reduced. During the computations for the axial fan performance, default values were kept unless a divergent and unstable trend had been observed.

5. 3D Analysis

CEM Models

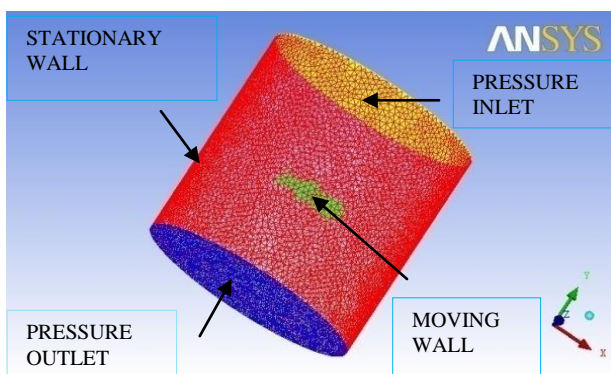


Fig.2 Boundary Conditions given to the fan geometry

The 3D mesh was built with ICEM CFD, using boundary conditions of 'pressure' at the inlet, 'pressure' at the outlet

and 'wall' around the fan. The inlet is coloured in 'Yellow', outlet in 'Blue' and wall is in 'Red'. The fan is in 'Green'. The meshed model of a fan is given below.

Numerical Simulations using FLUENT

In the CFD simulation, the following assumptions^[15] have been made:

- Steady state air flow
- Coupled solver and explicit formulation
- Realizable k-epsilon RNG swirl dominated flow viscous model
- Standard wall functions
- Moving reference frame at a constant velocity of 225 rad/s
- Momentum-First Order Upwind Scheme
- Turbulent Kinetic Energy-First Order Upwind Scheme
- Turbulent Dissipation Rate-First Order Upwind Scheme

Solver

Traditionally, these equations have been solved in a segregated fashion using some variation of the SIMPLE algorithm to couple the shared pressure with the momentum equations. This is attained by effectively transforming the total continuity into a shared pressure.

The ANSYS FLUENTPhase Coupled SIMPLE algorithm has been successfully implemented and solves a wide range of multiphase flows. However, coupling the linearized system of equations in an implicit manner would offer a more robust alternative to the segregated approach.

The realizable k- ϵ mode is a relatively recent development. An immediate benefit of the realizable k- ϵ model is that it more accurately predicts the spreading rate of both planar and round jets. It is also likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation.

6. Results

Analytically Static Pressure = 1260 Pa

Computed Static Pressure = 1162 Pa

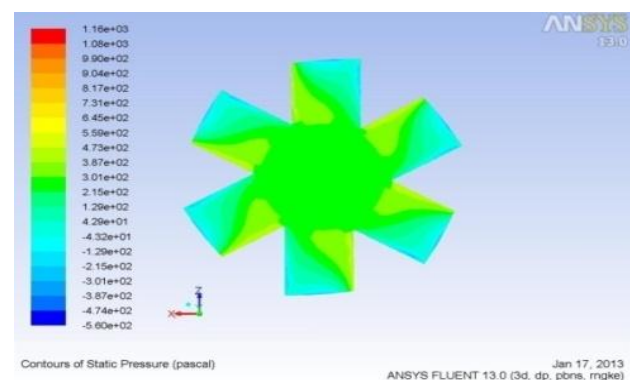


Fig.3 Contours of Static Pressure at Inlet (Pa)

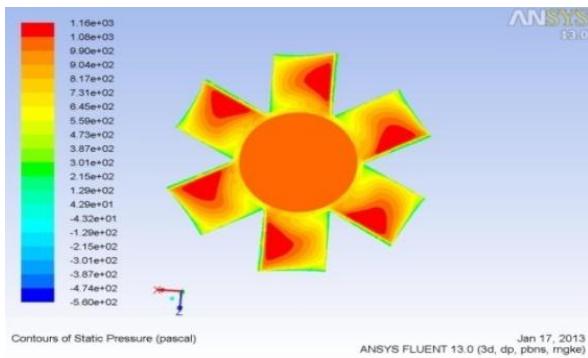


Fig.4 Contours of Static Pressure at Outlet (Pa)

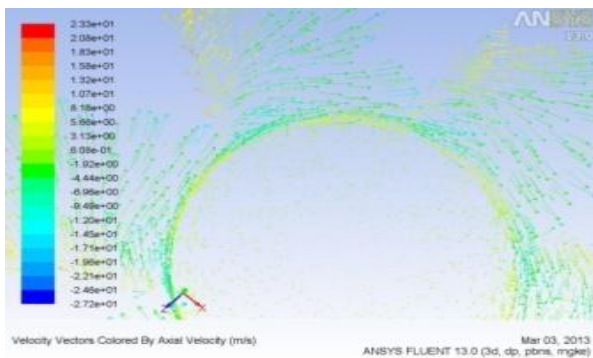


Fig.5 Vectors of Axial Velocity (m/s)

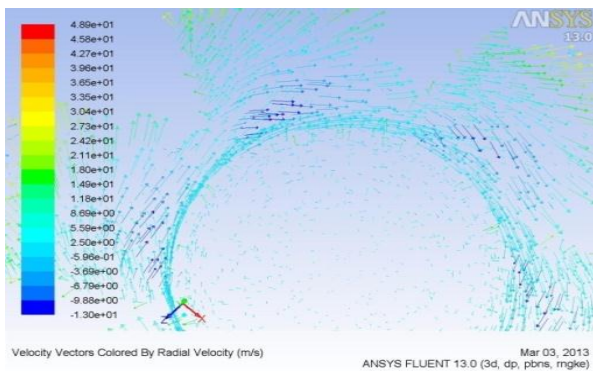


Fig.6 Vectors of Radial Velocity (m/s)

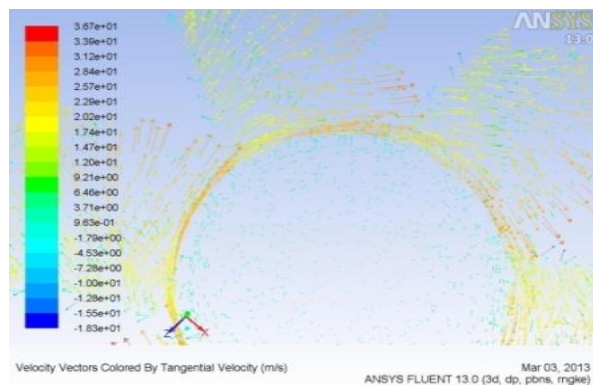


Fig.7 Vectors of Tangential Velocity (m/s)

The given mass flow rate is 2.83 kg/s. But considering 10% losses in practical conditions, the following result was achieved.

Table 2 Mass flow rate

Mass Flow Rate	(kg/s)
outlet	3.0636445
Net	3.0636445

Definition of Structural Analysis

Structural analysis is probably the most common application of the finite element method. The term structural (or structure) implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

Performing a Static analysis

The procedure for a static analysis in Workbench consists of following tasks:

- Section Build the Model
- Define material properties
- Mesh the model
- Apply the Loads and boundary condition's
- Solve the Analysis
- Review and interpretation of Results

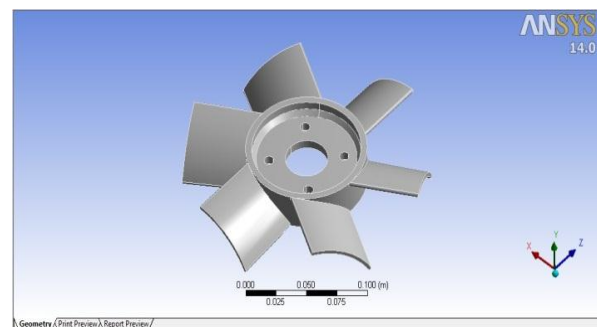


Fig.8 Fan model after importing into Workbench

The boundary conditions for the model are:

1. Load

Analytical calculation of Force: The maximum velocity of air is at the tip of the blade (by looking at contour plot in CFD results)

Max force on blade = mass flow rate * maximum axial velocity = 3.06 * 23.3

$$F_{max} = 71.298 \text{ N}$$

This is the maximum force acting on the blade.

The actual load on blade is UVL. If the maximum value of the UVL be considered and if that force be assumed as UDL, then the design would be safe in other areas as maximum value of force was considered.

The values of axial velocity are dominantly in the range of 3.13 m/s to 5.66 m/s. Hence, the net force is considered as 15.166 N.

2. Rotation: The model is rotating with rotational velocity of 225 rad/sec

3. Constraint: As the fan is pulley driven and is fixed on the shaft using nut and bolts, the model is constrained in axial direction by fixed support option.

4. Results:

Maximum principal stress = 4.3909MPa

Maximum shear stress = 1.1655MPa

Maximum Von-Mises stress = 3.2746MPa

Total directional deformation = 0.18865mm

As all stresses are below the allowable stress limit so this design is safe in structural analysis.

Below are contour plots of results.

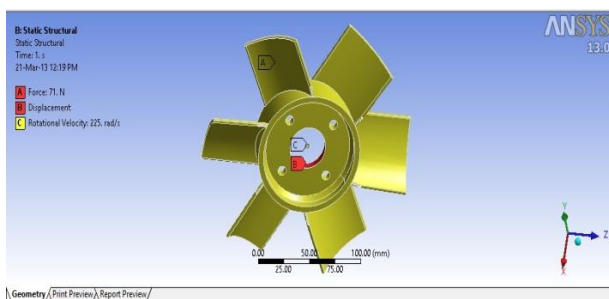


Fig.9 Picture showing the boundary conditions of model

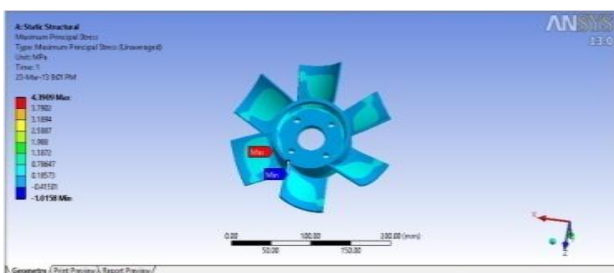


Fig.10 Figure showing maximum principal stress

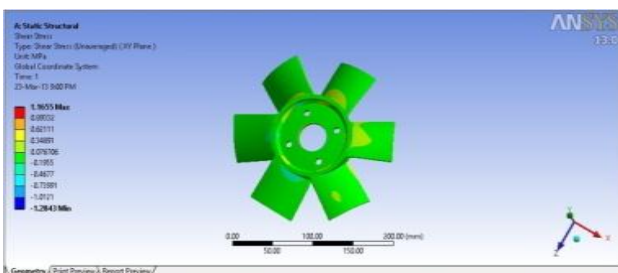


Fig.11 Figure showing maximum shear stress

5. Optimization

The maximum force which can be applied on fan is 465 N, so max torque comes out to be,

$$\tau_{max} = Force * radius$$

$$= 425 * 0.1 = 42.5 Nm$$

Also,

$$Power = \frac{2\pi NT}{60}$$

$$= 2 * 3.14 * 2145 * 42.5 / 60$$

$$= 9547.7525W$$

$$= 9.547 KW$$

Conclusions

Flow Rate: The 3D analysis of fan shows that the blade angles designed analytically are capable of producing required flow rate.

Static Pressure: The static pressure developed across the fan surface is in accordance with analytical solutions.

Velocity Vectors: The velocity vectors are in conformance with the direction and the magnitude of velocity of rotation of the fan. They also represent flow of air around the blades of the fan.

Safety in Structural: The structural analysis of the fan represents its strength structurally. The shear stress, Von-Mises stresses confirm the safety of the design in structural. Torque Optimization: The maximum torque is optimized for the fan. Its value is 42.5 Nm.

References

Robert Jorgensen, Fan Engineering, Buffalo Forge Company, Eighth Edition.

S.M.Yayha, Turbines, Compressors & Fans, McGraw Hill, Fourth Edition

Fatih Çevik, Design Of An Axial Flow Fan For A Vertical Wind Tunnel For Paratroopers, Middle East Technical University, 201

Max A. Sardou, Patricia Djomseu, 2000-01-3519 Low Energy Consumption and Low Noise Fans for Cooling Systems of Trucks, Buses, Cars and Trains, Warrendale, PA / SAE / 2000

Robert C. Mellin, Noise & Performance Of Automotive Cooling Fans, SAE No.-800031

Kenneth Campbell, Engine Cooling Fan Theory And Practice, Vol. No.:52, SAE / 194

Catharina R. Biber, Fan-plus-Heat Sink Optimization – Mechanical and Thermal Design with Reality, ISPS / 199

Computational Fluid Dynamics, the Basics with Applications, J.D. Anderson, Tata McGraw Hil

Fluid Mechanics, Cengel and Cimbala, Tata McGraw Hill.

ICEM CFD Documentation, ANSYS.

BASF Corporation, Basic Guidelines For Plastic Conversion Of Metal Axial Flow Fans