International Journal of Ethics in Engineering & Management Education
Website: www.ijeee.in (ISSN: 2348-4748, Volume 3, Issue 10, October 2016)

Optimization and CFD Analysis of Radiator Fan

Gandasiri Venkatesh
M.Tech Scholar - Engineering
Mechanical Engineering
Sphoorthy Engineering College
Nadergul, Hyderabad, TS, India
vrsiri6@gmail.com

M. Venkata Ramana
Assistant Professor
Mechanical Engineering
Sphoorthy Engineering College
Nadergul, Hyderabad, TS, India

P. Sudheer Rao
Associate Professor
Mechanical Engineering
Sphoorthy Engineering College
Nadergul, Hyderabad, TS, India

Abstract: **The efficiency of automotive radiator is largely dependent on the ability of the fan to force the air draught as much as possible. In order to devise an effective fan design, the primary objective is to maintain desired pressure difference between the fan inlet and outlet. The radiator fan design was first evaluated through simulations to obtain pressure difference and torque values. In order to obtain the desired pressure difference and torque. The radiator fan with 12 blades was first analyzed through CFD simulations and the pressure difference between the fan inlet and outlet were measured. To improve performance keeping the same number of blade and discharge with changing the rotational speed of the fan were suggested and flow analysis for the same was performed. Desired pressure difference was obtained through the various rotational speeds. Final results show better efficiency calculating by the numerical simulation. This solution can also be provided using FLUENT.**

Keywords: **CFD, Fan, Radiator, Fluid, Fluent, Rpm, Contours, Vectors**

1. INTRODUCTION

Axial flow fan is extensively used in many engineering applications. Its adaptability has resulted in implementation into large scale systems, from industrial dryers and air conditioning units to automotive engine cooling and in-cabin air recirculation systems. The benefit of using axial flow fans for the purpose of augmenting heat transfer is particularly evident in the automobile industry because of the need for relatively compact designs. 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 [1], [2]. Numerical investigations have been performed to quantify the performance of axial fans and their flow characteristics [3], [4]. However, the more-practical example of cooling a heated engine or heated plate using an axial flow fan has received more attention in regards to understanding flow characteristics and heat transfer [5]-[7]. 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 behavior and characteristics. The models provide sufficiently accurate predictions over a range of operating conditions, which are not possible using other methods. Without an understanding of the characteristics of air flow passing through a fan, problems related to engine cooling systems can never be fully resolved. In this PROJECT, CFD were used to model the flow passing through a radiator fan, which was then compared with actual experimental data. An APT T4 repower radiator fan and fan shroud (Fig. 1) play a crucial role in complicated engine cooling systems, such as the one shown in Fig. 2. A radiator (Figs. 1 and 2) is a type of heat exchanger designed to transfer thermal energy from the coolant to the surrounding air by means of a mechanism known as natural or forced convection. The latter case concerns the use of a radiator fan to pull the air through the radiator core.

The fan provides air flow through the radiator. The orientation of the blade also plays an important role in understanding the flow of air across the radiator and fan.

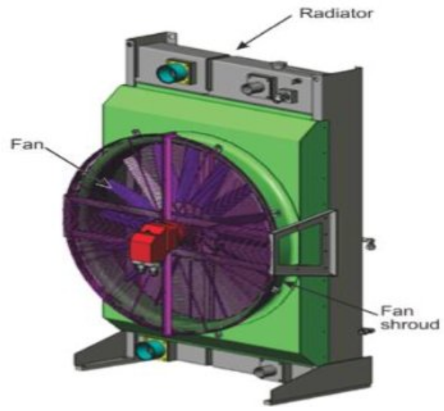


Fig. 1 Radiator and fan assembly

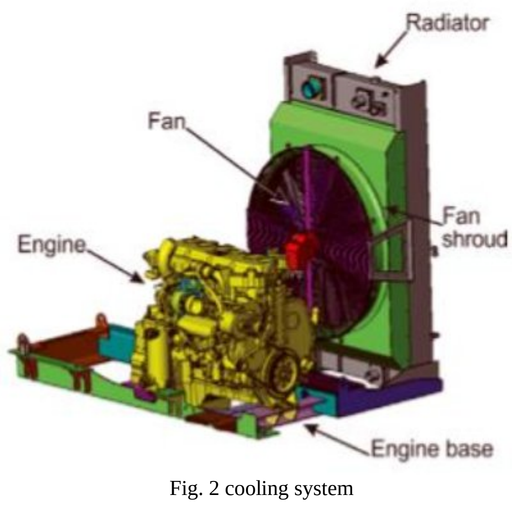


Fig. 2 cooling system

International Journal of Ethics in Engineering & Management Education
Website: www.ijeee.in (ISSN: 2348-4748, Volume 3, Issue 10, October 2016)

2. FLUID MECHANICS

There are three approaches to Fluid Mechanics – Experimental, Theoretical and Computational**.** Experimental approach is the oldest approach, perhaps also employed by Archimedes when he was to investigate a fraud. It is a very popular approach where you will make measurements using a wind tunnel or similar equipment. But this is a costly venture and is becoming costlier day by day. Then we have the theoretical approach where we employ the mathematical equations that govern the flow and try to capture the fluid behavior within a closed form solution i.e., formulas that can be readily used. This is perhaps the simplest of the approaches, but its scope is somewhat limited. Not every fluid flow renders itself to such an approach. The resulting equations may be too complicated to solve easily. Then comes the third approach- Computational. Here we try to solve the complicated governing equations by computing them using a computer. This has the advantage that a wide variety of fluid flows may be computed and that the cost of computing seems to be going down day by day. With the result the emerging discipline Computational Fluid Dynamics, CFD**,** has become a very powerful approach today in industry and research.

2.1. FLUID:

It is well known that matter is divided into solids and fluids. Fluids can be further divided into Liquids and Gases. Solids have a definite shape and a definite size, while the liquids have a definite size, but no definite shape. They assume the shape of the container they are poured into. Gases on the other hand have neither a shape nor a size. They can fill any container fully and assume its shape.

2.2. METHODS TO STUDY FLUID FLOW:

2.2.1 Lagrangian Method:

In this a fluid particle is selected, which is pursued throughout its course of motion. And changes in its parameters are studied.

2.2.2 Eulerian Method:

In this method any point in the space occupied by the fluid is selected and observation is made on changes in fluid parameters at the point.

2.3 MODELS OF THE FLOW:

In obtaining the basic equations of fluid motion, the following philosophy is always followed:

1. Choose the appropriate fundamental physical principles from the law of physics
   Such as:
   a. Mass conserved
   b. F=ma (Newton second law)
   c. energy conserved.
2. Apply these principles to a suitable model of the flow.
3. From this application, extra the mathematical equations which embody such physical Principles.

A solid body is rather easy to see &define; on the other hand, a fluid is a "squishy" substance that is hard to grab hold of. If a solid body is in translation motion, the velocity of each part of the body is the same; on the other hand, if a fluid motion, the velocity may be different at each location in the fluid. How then do we visualize a moving fluid so as to apply to it the fundamental physical principles? For a continuum fluid the answer is to construct one of the 4 models described below.

3. COMPUTATIONAL FLUID DYNAMICS

Over the past half-century, we have witnessed the rise in the new methodology for attacking complex problem in fluid mechanics, heat transfer and combustion. It has come to the state that wherever there is a flow, computer can help to understand and analyze the same. This new methodology of solving a flow problem using a computer is given the name CFD. Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based numerical approach, In this numerical approach, the equations (usually in partial differential form) that govern a process of interest are solved numerically. The technique is very powerful and spans a wide range of industrial and non-industrial application areas.

Advantages

- Low cost, high speed
- Complete information at any inaccessible point
- Ability to simulate realistic conditions and also ideal conditions
- Can handle any complex geometry

Disadvantages

- Proper mathematical model may not be available
- Validation of computer results needs experimental data

Pre-Requisites for CFD

- Fluid mechanics
- Heat transfer
- Partial differential equations
- Numerical methods
- Any programming language with graphical tools.

Table 3.1: Comparison of approaches.

Approach	*Advantages*	*Disadvantages*
Experimental	1.Capable of being most realistic	1. Equipment required 2. Scaling problems 3.Measurement difficulties 4. Operating costs
Theoretical	1.General information(which is usually in formula form)	1. Restricted to simple geometry and physics 2. Usually restricted to linear problems
Computational	1.No restriction to linearity 2.Complicated physics can be treated 3.Time evolution of flow can be obtained	1. Truncations errors 2. Boundary conditions problems 3. Computer costs

International Journal of Ethics in Engineering & Management Education
Website: www.ijeee.in (ISSN: 2348-4748, Volume 3, Issue 10, October 2016)

3.1. Mathematical Behaviour Of Governing Equations In Computational Fluid Dynamics

The development of the high speed digital computer combined with the development of a accurate numerical algorithms for solving problems on these computers has had a great impact on the way principles from the science of Fluid Mechanics are applied to problems of design in modern engineering practice

The physical aspects of any fluid flow are governed by three fundamental principles: conservation of mass, conservation of momentum, conservation of energy and these can be expressed in terms of basic mathematical equations which in their more general form are either integral equations or partial differential equation in computational approach; these equations that govern a process are solved numerically.

These partial differential equations have certain mathematical behavior. This behavior is not fixed and varies from one circumstance to another, depending on the magnitude of the dimensionless flow parameters governing, the situation, the equations governing the flow and the steady or unsteady nature of the flow.

4. DESIGN AND CFD ANALYSIS OF RADIATOR FAN ASSEMBLY

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 [1], [2]. Numerical investigations have been performed to quantify the performance of axial fans and their flow characteristics

4.1. Problem Statement:

The efficiency of automotive radiator is largely dependent on the ability of the fan to force the air draught as much as possible. In order to devise an effective fan design, the primary objective is to maintain desired pressure difference between the fan inlet and outlet.

The radiator fan design was first evaluated through simulations to obtain pressure difference and torque values. In order to obtain the desired pressure difference and torque.

The radiator fan with 12 blades was first analyzed through CFD simulations and the pressure difference between the fan inlet and outlet were measured. To improve performance keeping the same number of blade and discharge with changing the rotational speed of the fan were suggested and flow analysis for the same was performed. Desired pressure difference was obtained through the various rotational speeds. Final results show better efficiency calculating by the numerical simulation. This solution can also be provided using FLUENT.

4.2. DESIGN:

The first step is to identify a typical radiator axial flow fan that can be reproduced as a 3-D CAD Solidworks® software engineering drawing package (Fig. 5). The 3-D models are then imported into the CFD software, remodeled into different sections, and refined to generate a finite volume meshing (Fig. 6). This is a crucial step, where details of the geometrical shape need to be defined precisely. The flow domain is also created, and the final meshing of all components needs to be accurate. The total element count will be around 1.6 million, with an inflation layer on the blades. Any errors in the drawings and flow area need to be corrected before continuing.

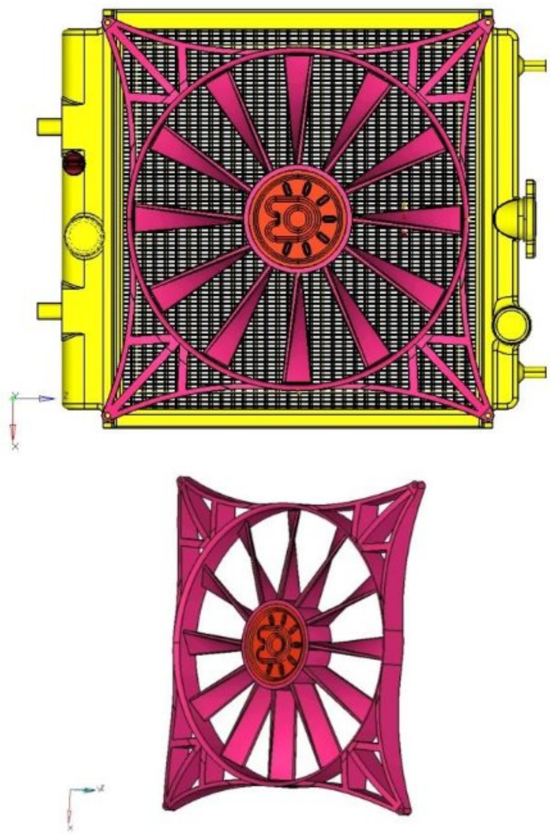


Fig. 3 Computational domain of a fan

The second step is to import the files into the CFD code preprocessor, which will solve the flow equations. Here, the flow fields boundary conditions are set. These include inlet air mass flow, outlet pressure, fluid properties, and flow domain characterization, such as moving internal zone and stationary solid walls. The next step is to set the simulation process as a 3-D steady and turbulent problem .

Table.2. FAN-SHROUD Assembly Specifications:

Specification	
Number of Blades	12
Fan Diameter	300 mm
Blade Thickness	4.25mm
Height of the blade	100mm
Hub outer diameter	100mm
Hub Thickness	37mm
Rotation	CW from Front End

International Journal of Ethics in Engineering & Management Education
Website: www.ijeee.in (ISSN: 2348-4748, Volume 3, Issue 10, October 2016)

FAN-Geometry and Domains

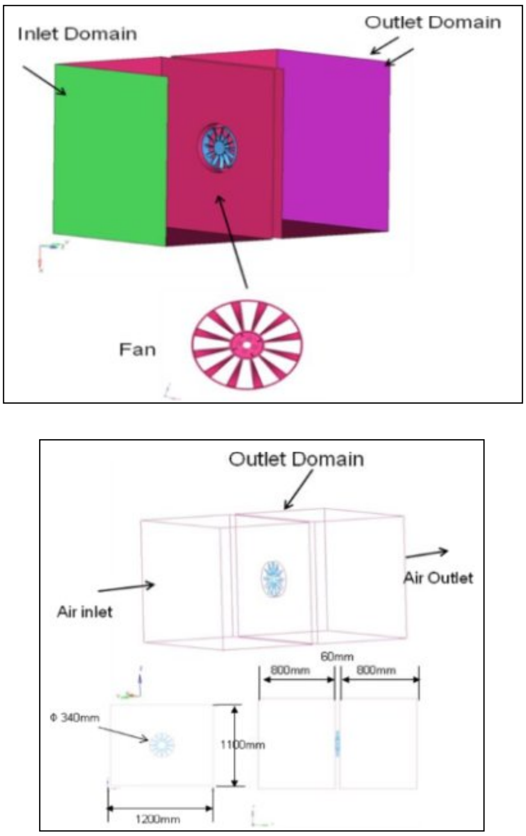


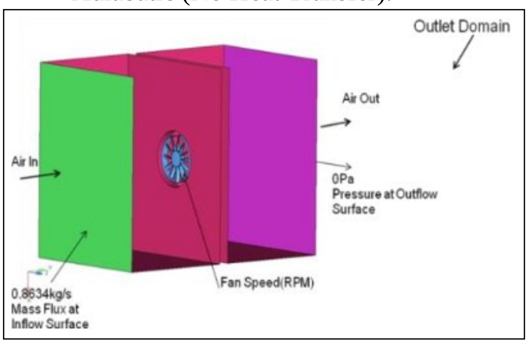
Fig. 4 FAN-Geometry and Domains

4.3. Problem setup:

For the steady RANS simulations, the single equation Spalart-Allmaras (SA) turbulence model was used. The turbulence equation is solved segregated from the flow equations using the GLS formulation. Reference Frame is used to simulate the Rotating Flow

4.4. Boundry Conditions:

Type of flow: Steady-state, K-e model, Incompressible, Adiabatic (No Heat Transfer).



Inlet – Mass Flow inlet
Outlet – Pressure Outlet
Fan –Reference Frame
All other surfaces are No-Slip wall BC

Table.3 Boundary conditions summary:

S. No	Inlet:(Mass flow rate(kg/s)	Outlet:(Pressure (Pa)	RPM
case1	0.8634	**0**	**2700**
case2	0.8634	**0**	**3000**
case3	0.8634	**0**	**3300**
case4	0.8634	**0**	**3600**

Fluid properties:
Water properties @ Ambient conditions

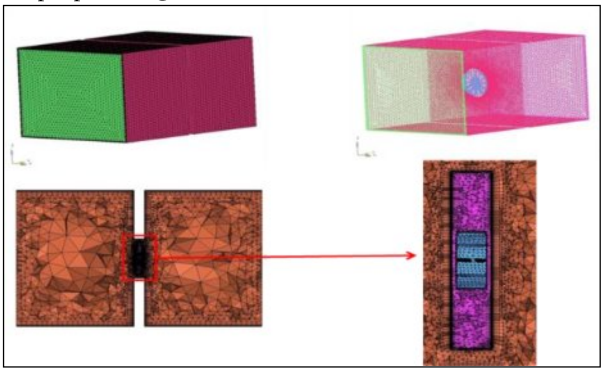


Fig.5 Meshed Model-1(Total number of cells ~954261)

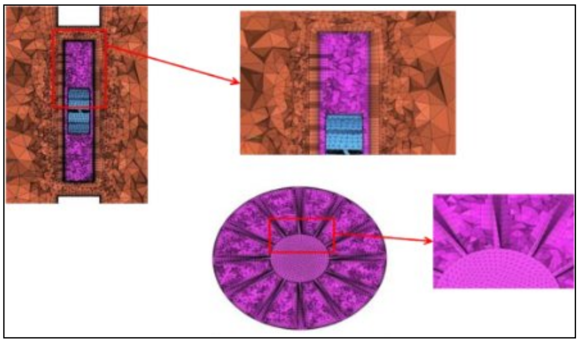
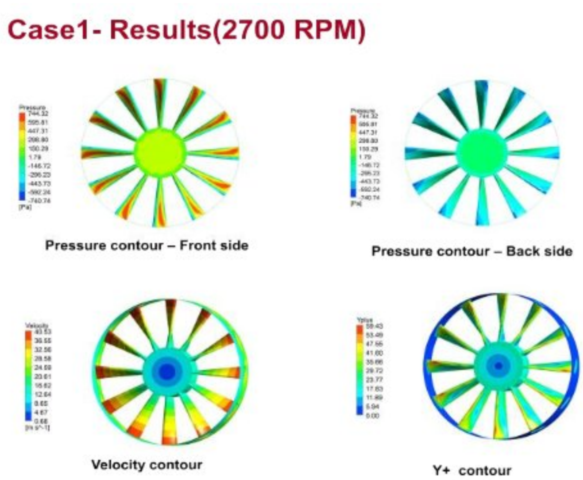


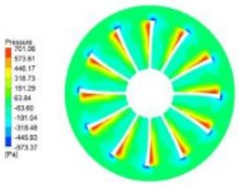
Fig. 6 Meshed Model 2 (Total Number Of Boundary Layers—10)

5. RESULTS

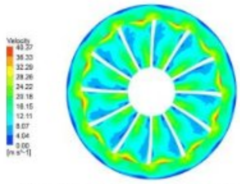


International Journal of Ethics in Engineering & Management Education
Website: www.ijeee.in (ISSN: 2348-4748, Volume 3, Issue 10, October 2016)

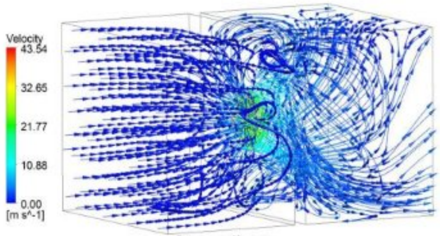
Case1- Results(2700 RPM)



Pressure contour – mid section fan

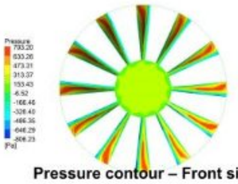


Velocity contour – mid section fan



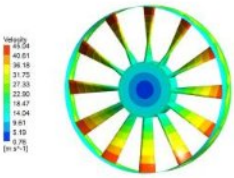
Stream lines

Case2- Results(3000 RPM)

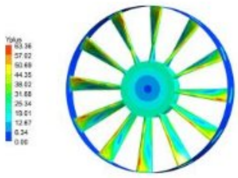


Pressure contour – Front side

Pressure contour – Back side

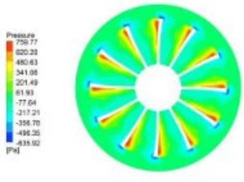


Velocity contour

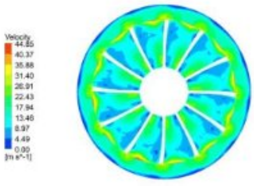


Y+ contour

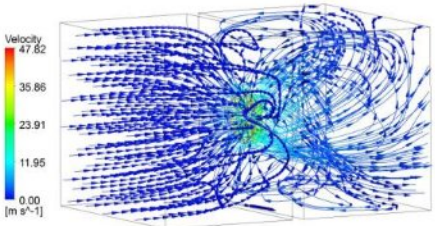
Case2- Results(3000 RPM)



Pressure contour – mid section fan

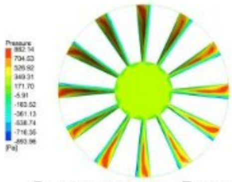


Velocity contour – mid section fan

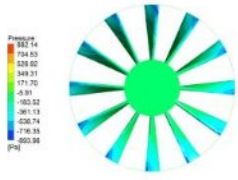


Stream lines

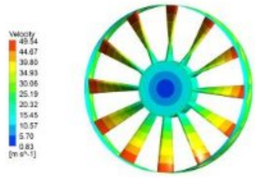
Case3- Results(3300 RPM)



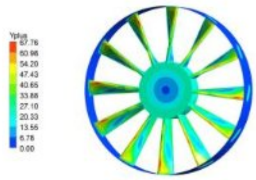
Pressure contour – Front side



Pressure contour – Back side

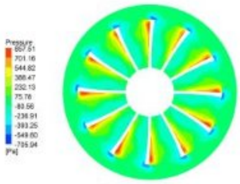


Velocity contour

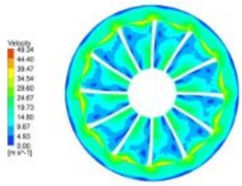


Y+ contour

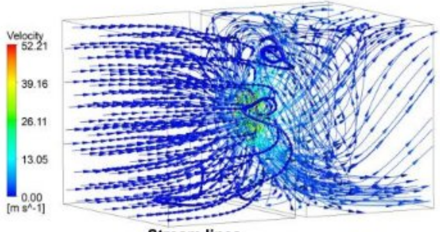
Case3- Results(3300 RPM)



Pressure contour – mid section fan

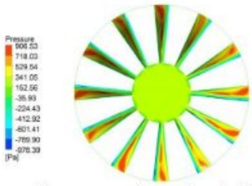


Velocity contour – mid section fan

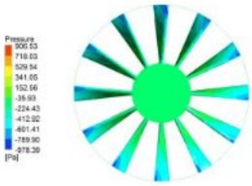


Stream lines

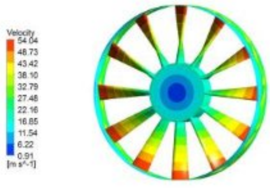
Case4- Results(3600 RPM)



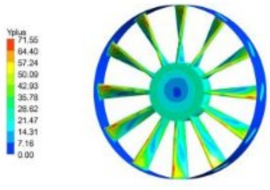
Pressure contour – Front side



Pressure contour – Back side

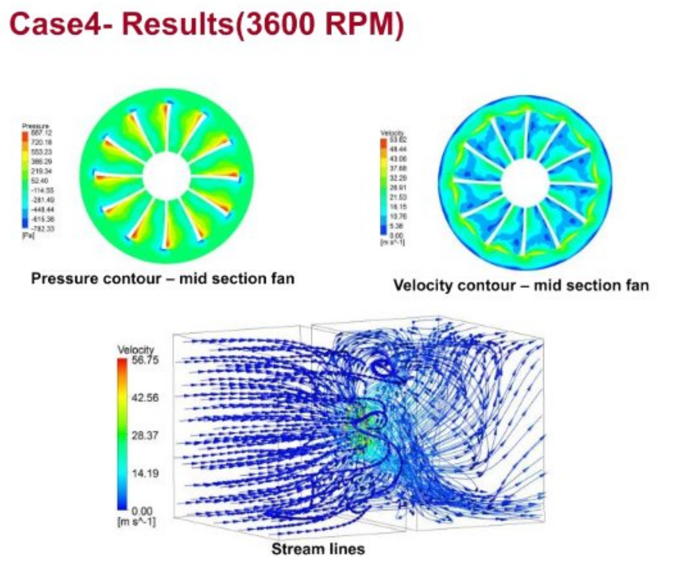


Velocity contour



Y+ contour

International Journal of Ethics in Engineering & Management Education
Website: www.ijeee.in (ISSN: 2348-4748, Volume 3, Issue 10, October 2016)



6. CONCLUSION

S. No	Mass flow rate(kg/s)	Blade speed(RPM)	Delta Pressure(Pa)
Case1	0.8634	2700	163.215
Case2	0.8634	3000	168.356
Case3	0.8634	3300	195.055
Case4	0.8634	3600	189.072

In this work numerical simulation was performed for radiator fan. CFD results to study flow distribution and back pressure estimate. Velocity magnitude, pressure contours & steam line shows flow characteristics that help to understand and verify the main vortex structures found by CFD. Based on this simulation the following conclusions were reached. Radiator fan best operating speed was 3300rpm.

REFERENCES

[1]. Gimenez J., Ramajo D., Nigro N., Particle Transport in Laminar/Turbulent Flows. MECOM 2012, Salta, Argentina, 2012.

[2]. Jaworski A.J., Meng G., On-line measurement of separation dynamics in primary gas/oil/water separators: Challenges and technical solutions—A review, Journal of Petroleum Science and Engineering, 68, 47-59, 2005.

[3]. Jeelani S. A. K. and Hartland S., Effect of Dispersion Properties on the Separation of Batch Liquid-Liquid Dispersions, Ind. Eng. Chem. Res, 37, 547-554, 1998.

[4]. Kang W, Guo L., Fan H., Meng L., Li Y., 2011, Flocculation, coalescence and migration of dispersed phase droplets and oil-water separation in heavy emulsion, Journal of Petroleum Science and Engineering, 81, 177-181.

[5]. Lee C., Frankiewicz T., The Design of Large Diameter Skim Tanks Using Computational Fluid Dynamics (CFD) For Maximum Oil Removal, 15th Annual Produced Water Seminar, Houston, USA, 2005.

[6]. Lee D. W., Bateman W.J.D, Owens N., 2007, Efficiency of Oil/Water Separation Controlled by Gas Bubble Size and Fluid Dynamics within the Separation Vessel, GLR Solutions, Calgary, Canada, 2007.

[7]. Wilkinson D., Waldie B., Mohamad M. I., Lee H. Y., 1999, Baffle plate configurations to enhance separation in horizontal primary separators. Short communication. Chemical Engineering Journal, 77, 221-226

[8]. Zhang L., Xiao H., Zhang H., Xu L., Optimal design of a novel oil-water separator for raw oil produced from ASP flooding, Journal of Petroleum Science and Engineering, 59, 213-218, 2007.

[9]. ANSYS.ANSYS CFX-Solver Theory Guide, 14th edition, ANSYS Inc., 2015.