International Journal of Research Studies in Science, Engineering and Technology
Volume 1, Issue 5, August 2014, PP 52-59
ISSN 2349-4751 (Print) & ISSN 2349-476X (Online)

CFD Analysis of a Radiator Pump to Investigate the Cause and to Eliminate the Unwanted Vibrations

Dr S B Prakash [1], Ajay Bharadwaj M B [2]

[1]Professor & Head, Department of TPE, Department of P.G. studies, VTU Regional Centre
Hanchya Sathagalli Layout, Mysore, Karnataka, INDIA.
sajjalprakash@gmail.com

[2]PG Student, Department of TPE, Department of P.G. studies, VTU Regional Centre
Hanchya Sathagalli Layout, Mysore, Karnataka, INDIA
ajaybharadwaj.m.b1990@gmail.com

Abstract: *A pump is a device that moves fluids (liquids or gases), or sometimes slurries, by mechanical action. Pumps operate by mechanisms (typically reciprocating or rotary) to perform mechanical work for moving the fluid. Mechanical pumps serve in a wide range of applications such as pumping water from wells, pumping coolants etc. the present work emphasizes on computational analysis of a centrifugal pump used in an automobile (car) radiator. The present design of the manufacturer is experiencing unwanted vibrations during the operation. Change in the operating conditions of the pump (such as mass flow at inlet, inlet pressure etc) is not desired. So the study involves CFD analysis a centrifugal pump to determine the cause of the unwanted vibrations and to suggest suitable design modifications for the same operating conditions and to analyze the new design, to minimize/eliminate the unwanted vibrations. The parameters which are observed during the analysis are velocity magnitude, velocity vectors and turbulent kinetic energy. The contour plots from the analysis of the old design confirm that the vibration is due to turbulence. Design modification should reduce vibrations, but it should not reduce the pumping efficiency of the pump.*

Keywords: *Centrifugal pump, Velocity vectors, Velocity magnitude, Turbulence.*

1. Introduction

Mechanical pumps serve in a wide range of applications such as pumping water from wells, aquarium filtering, pond filtering and in the car industry for engine cooling and fuel injection, in the energy industry for pumping oil and In the medical industry. Since the beginning of time, there has been a need to push, suck or lift liquid from one place to another. Pumps provide the solution for this. Thanks to the revolutionary innovation, creativity and vision of the industry’s most forward thinking people and companies, it doesn’t matter whether the liquid is water, peanut butter or oil; today there is a pump that will move it.

The type of pump used in this study is a centrifugal pump. Centrifugal pumps are used to transport fluids by the conversion of rotational kinetic energy to the hydrodynamic energy of the fluid flow. The rotational energy typically comes from an engine or electric motor. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward into a diffuser or volute chamber (casing), from where it exits. The volute converts kinetic energy into pressure by reducing speed while increasing pressure, helping to balance the hydraulic pressure on the shaft of the pump. Centrifugal pumps don’t require any valves, or many moving parts. It also allows them to move at high speeds with minimal maintenance. Their output is very steady and consistent. Most of all, they are very small compared to other types of pumps that create the same output. A centrifugal pump must be put under water, or primed, before it will move water.

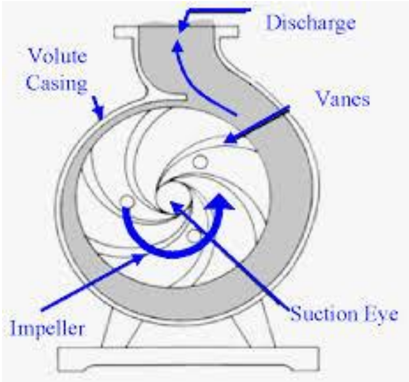

Figure 1. *Parts of centrifugal pump.*

A centrifugal pump converts the input power to kinetic energy in the liquid by accelerating the liquid by a revolving device - an impeller. Fluid enters the pump through the eye of the impeller which rotates at high speed. The fluid is accelerated radially outward from the pump casing. A vacuum is created at the impellers eye that continuously draws more fluid into the pump. Centrifugal pumps are usually the preferred choice for lower viscosity (thin) liquids than higher viscosity (thick) liquids and high flow rates. They are typically used across many residential, commercial, industrial, and municipal applications. Common uses of a centrifugal pump include water, sewage, petroleum and petrochemical pumping.

2. Methodology

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results.

During pre processing, the geometry of the existing design from the manufacturer is obtained. The geometry and the volume occupied by the fluid are divided into discrete cells (the mesh). The mesh may be uniform or non uniform. Boundary conditions are defined. This involves specifying the fluid behavior and properties at the boundaries of the problem. During post processing the meshed model is analyzed and visualization of the resulting solution is done. Then suitable design modification is suggested. After obtaining the modified model, the process of meshing, defining boundary condition and analysis is repeated.

3. Designing and Modeling

Using well modeled geometry and having an understanding of the expected simulation result helps the process of achieving a successful analysis.

Figure 2. Model of the pump

For a good analysis, the model needs to be detailed enough to reflect reality, but not so detailed that it takes an inordinate amount of time to mesh and to run analysis. Fluid flow can be very sensitive to small details and over simplifying the model can miss some of the details. However, under simplification will cause meshes that take excessively long run without necessarily increasing the accuracy of the result. Therefore, it is important to omit those details that will not affect the flow, while including those details that will affect the flow. So from geometry clean up we can reduce the excess time and extra meshes without disturbing the model and having no effect on the final result. After the geometry cleanup the model gets simplified. The hollow interior space or gap created between the wall, rotor, inlet and outlet is used as fluid domain.

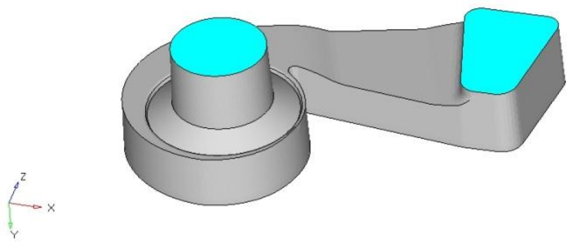

Figure 3. *Model after geometry clean up.*

The above figure shows the simplified model of the pump where the grey colour structure is the outer wall of the pump. The colour blue indicates the fluid inside the casing.

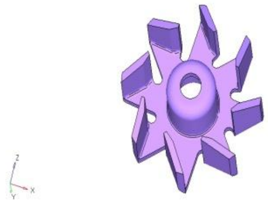

Figure 4. Impeller.

The Rotor (Rotating Element), is made up of the main component which is the impeller, In the centre of an impeller, is the 'EYE' which receives the inlet flow of liquid into the 'Vanes' of the impeller. The outer wall and the impeller is made out of steel.

The assembly of the Wall and the Rotor is as shown below.

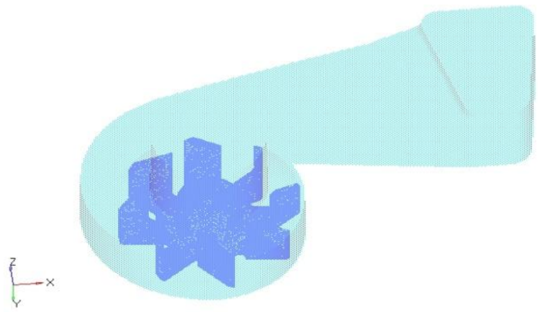

Figure 5. *Assembly of Wall, Fluid domain and rotor.*

4. CFD ANALYSIS

Grid generation for CFD is very important. Depending on purpose, the mesh can be generated in any form for 2D as well as 3D as per specific requirements. Prior to running CFD analysis, the geometry has to be broken up into small manageable pieces called element. HYPERMESH V10 software is used for the meshing purpose. The model is meshed using tetra meshes.

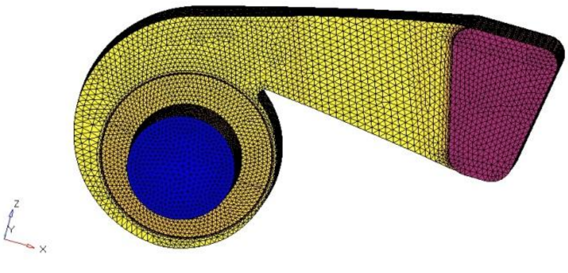

Figure 6. *Model meshed using tetra mesh.*

4.1 Inputs and Boundary Conditions

The whole system is kept at normal physical conditions. The system is operated at room temperature. To apply boundary conditions for the inlet, few parameters such as velocity at inlet, Reynolds no. and turbulent intensity are calculated using the known parameters such as mass flow rate at the inlet and radius at inlet etc.

Table 1. *Input and boundary condition*

SL. No.	Parameters	Values
1	System Operating temperature	27 °C
2	System Operating pressure	101325 Pa
3	Inlet radius	14 mm
4	fluid	Ethyl alcohol liquid
5	Fluid density	790 kg/m^3
6	Fluid viscosity	0.0012 kg/m-s
7	Velocity at inlet	4.185 m/s
8	Mass flow rate at inlet	2.036 kg/s
9	Hydraulic diameter	28 mm
10	Reynolds number	77143
11	Turbulence intensity	4%
12	Impeller and wall material	Steel
13	Impeller rotation speed	1000 Rpm

4.2 Solver Step

Setting up of the solver is very important in any of the fluid flow problem; the solver setting indicates the method and also a procedure for solving (analysis) the problem. In FLUINT the solution can be obtained by many solving methods. All the methods will be working by considering the average of the fluctuation of the flow in-spite of considering the whole path of fluctuation.

Table 2. *Solver setting applied for the analysis*

Solver type	Pressure based 3 Dimensional Formulation = implicit Steady state
Viscous model	Standard K Epsilon model
Impeller Motion type	moving frame of reference Type = Rotational (along Y axis) No slip condition
Fluid condition	Moving frame of reference
Pump casing/wall	Wall motion: Stationary No slip condition
Pressure velocity coupling	Simple

5. Results and Discussions

After applying the boundary conditions analysis is done and the respective contour plots are obtained using the display screen.

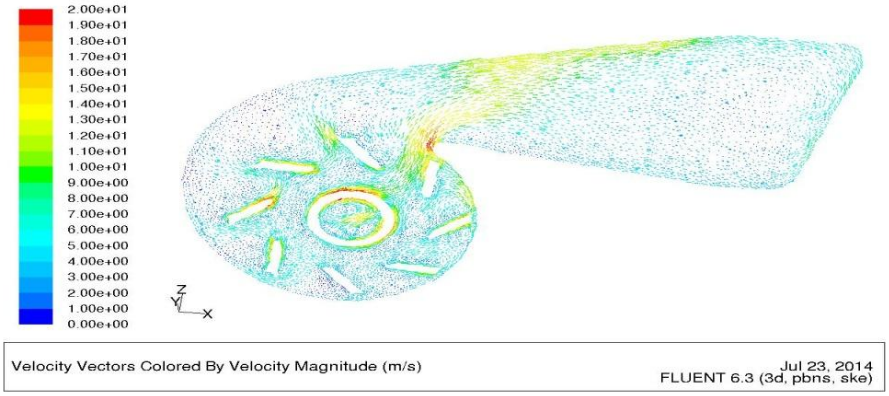

Figure 7. *Velocity vector contour plot of old design.*

In the above contour plot we can see velocity vector plot of the pump for the existing design. In this plot the region which is coloured in blue shows that the velocity at that region is normal and stable. But the region which is highlighted with yellow or red colours shows that at these regions the velocity is very high.

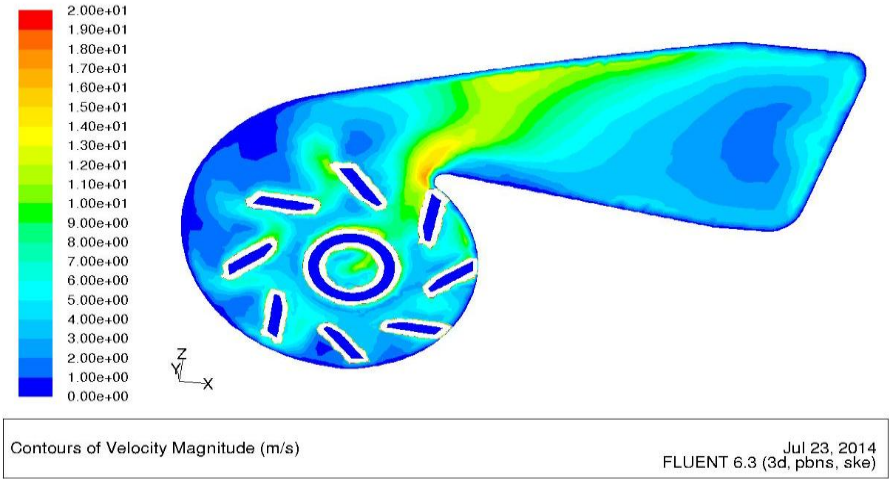

Figure 8. *Velocity magnitude Contour plot of old design.*

From both the contour plots of the velocity vectors and velocity magnitude, it is clearly seen that the velocity is very high around the suction eye and also in the gap of the blade clearance. And it continues to be slightly high for a little distance as the fluid moves away from the rotor and towards the outlet. From the above contour plots we can spot and identify the areas with high velocities in order to suggest the suitable design modification to reduce the velocity and in turn reduce the turbulence.

5.1 Modification

According to the factors observed from the previous plots, the modification to the existing design is suggested. The curvature or the slant structure that connects between the inlet and the body of the pump near the suction eye is increased by 0.5mm. And the other modification is that the clearance area between the blade and the pump casing is increased by 0.5mm. The difference between the existing design and the new design is not much. Even though the modification is very minute it is strongly presumed that the unwanted vibration due to turbulence will be reduced.

The modified design is meshed and furthers the new or the modified design is analyzed based on the previous boundary conditions only. This is because in this case only the slight design modification has been made but all the other parameters such as operating condition, mass flow rate, rotation speed etc will remain same.

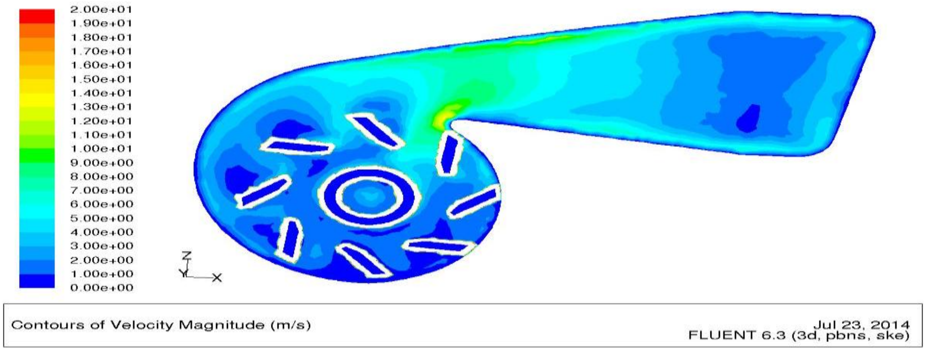

Figure 9. *Velocity contour plot of new design.*

Even though the modifications were very minute we can clearly see the difference in the velocity contour plot. The magnitude of velocity at certain regions is high in the old design, but it can be seen that the magnitude of velocity at those regions has been significantly reduced in the new design due to the modification made.

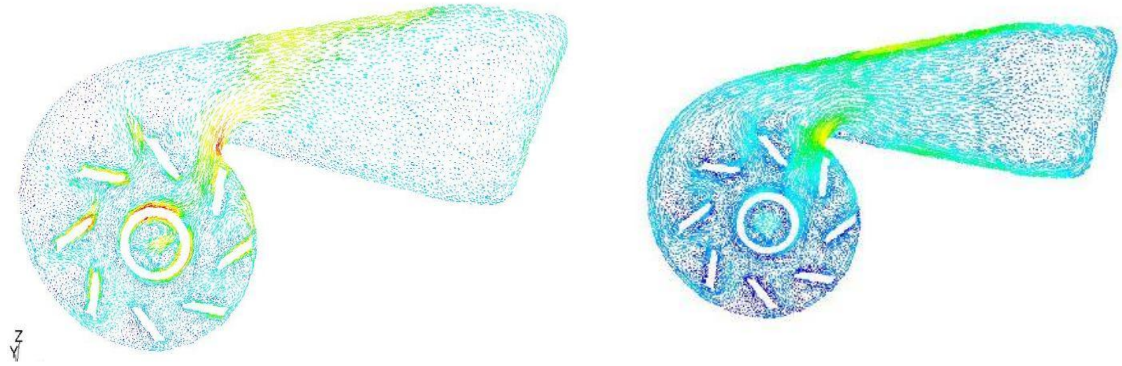

Figure 10. *Comparison of velocity vectors of old and new design.*

From the above figure we can see that the velocity which was causing the turbulence in the old design is reduced in the new design. Due to the modification done to the old design, the pump is now stable and able to with stand the velocity without creating unwanted turbulence and vibration.

It is clearly known that the turbulence in any cross section increases with the increase in the velocity. Now the effect of the high velocities on the pump can be analyzed by using the turbulence contour plots. The turbulent kinetic energy contour plot shows the regions that are affected by the turbulence inside the pump. It can be seen that the impeller blades and the area around suction eye are subjected to the effect of the turbulence.

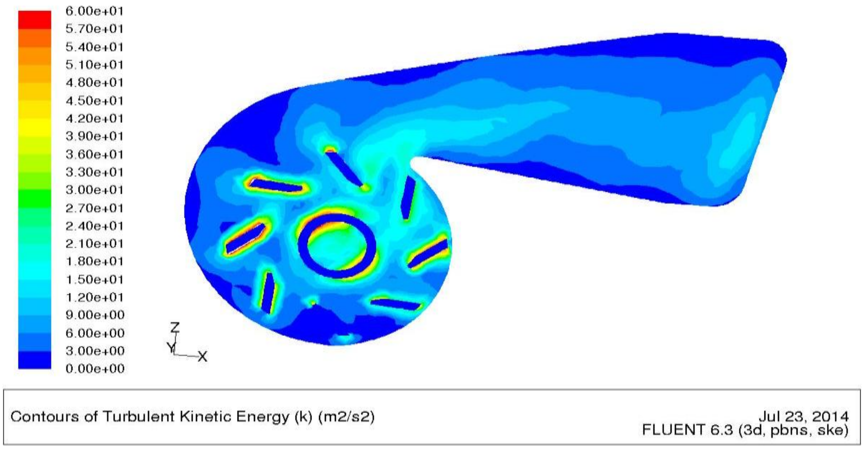

Figure 11. *Turbulent Kinetic energy plot of old design.*

The turbulence kinetic energy plot of the modified design is taken to compare with the old design.

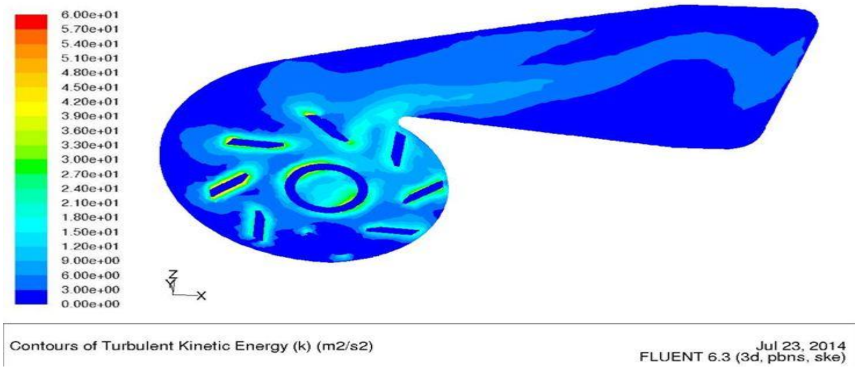

Figure 12. Turbulence Contour plot of new design

The difference between the old and new design can be clearly understood by comparing the contour plots of both the designs. It can be seen that the region affected by the high turbulence is reduced in the new design when compared to the old design. With this reduction of the turbulence, the unwanted vibrations are eliminated.

6. Conclusion

The centrifugal pump of radiator of a 4 wheeler vehicle (car) of the manufacturer was generating vibration, the cause of which was turbulence in impeller section of the water pump. The cause vibration was determined and to the regions where velocity is high is located. So during the operation the design should be made to withstand the velocity of the fluid and the turbulence will be reduced, which will reduce the unwanted vibrations. A suitable design modification has been suggested to minimize the unwanted vibration. The modified model is also meshed and analyzed with the same boundary conditions.

- From the contour plots we can clearly see that the turbulence in the pump is reduced. So the vibrations are eliminated.
- It can also be seen that there is an increase in the mass flow rate at the outlet of the new design when compared to the old design.
- The experimental values are in a good agreement with the software results.
- The experimental results shows that after the modification the mass flow rate at the outlet has increased from 1.7233 kg/s to 1.7753 kg/s.
- The experimentally determined values show an increase in the mass flow rate at outlet. This means the pumping efficiency has also been increased.

References

[1] Swapnil Urankar, Dr. H S Shivashankar, Sourabh Gupta "design and cfd analysis of single stage, end suction, radial flow centrifugal pump for mine dewatering application" IJREAS Volume 2, Issue 2 (February 2012) ISSN: 2249-3905.

[2] Maitelli, c.w.s dep, Bezerra, da Mata w " simulation of flow in a centrifugal pumpof ESP system using CFD" brazilian journal of petroleum and gas | v. 4 n. 1 | p. 001-009 | 2010 | ISSN 1982-0593.

[3] J H Kim1, K T Oh, K B Pyun, C K Kim, Y S Choi and J Y Yoon "Design optimization of a centrifugal pump impeller and volute using computational fluid dynamics" 26th IAHR Symposium on Hydraulic Machinery and Systems IOP Publishing IOP Conf. Series: 15 (2012) 032025.

[4] A. Manivannan "Computational fluid dynamics analysis of a mixed flow pump impeller" International Journal of Engineering, Science and TechnologyVol. 2, No. 6, 2010, pp. 200-206

[5] Sunsheng Yang, Fanyu Kong, and Bin Chen "Research on Pump Volute DesignMethod Using CFD" International Journal of Rotating Machinery Volume 2011, Article ID 137860

[6] Weidong Zhou, Zhimei Zhao, T. S. Lee, and S. H.Winoto "Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics" International Journal of Rotating Machinery, 9(1): 49–61, 2003.

[7] Tihomir Mihalic, Zvonimir Guzovic, Andrej Predin “Performances and Flow Analysis in the Centrifugal Vortex Pump” Journal of Fluids Engineering Copyright VC 2013 by ASME JANUARY 2013, Vol. 135 / 011002-1

[8] S. C. CHAUDHARI1, C. O. YADAV2 & A. B. DAMOR3 “a comparative study of mix flow pump impeller cfd analysis and experimental data of submersible pump” IMPACT: International Journal of Research in Engineering & Technology (IMPACT: IJRET) ISSN 2321-8843 Vol. 1, Issue 3, Aug 2013, 57-64.

[9] Bhavik M.Patel, Ashish J. Modi, Prof. (Dr.) Pravin P. Rathod “analysis of engine cooling waterpump of car & significance of its geometry” International Journal of Mechanical Engineering and Technology (IJMET), ISSN 0976 – 6340

[10] K.W Cheah, T.S. Lee, and S.H Winoto “Unsteady Fluid Flow Study in a Centrifugal Pump by CFD Method” 7th ASEAN ANSYS Conference Biopolis, Singapore 30th and 31st October 2008.

[11] Rodrigo Lima Kagami, Edson Luiz Zaparoli, Cláudia Regina de Andrade “ cfd analysis of an automotive centrifugal pump” 14th Brazilian Congress ofThermal Sciences and Engineering Copyright © 2012 by ABCM October18-22, 2012, Rio de Janeiro, RJ, Brazil.

[12] S V Jain, S R Shah, V J Lakhera “CFD based flow analysis of centrifugal pump” Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power December 16-18, 2010, IIT Madras, Chennai, India. FMFP10 - TM - 08