IJREAT International Journal of Research in Engineering & Advanced Technology, Volume 2, Issue 4, Aug-Sept, 2014
ISSN: 2320 – 8791 (Impact Factor: 1.479)
www.ijreat.org

Analysis for Alternatives of Geometry for the Intake Side of the Sump to Enhance Performance of Centrifugal Pump

Ms.JagrutiRane[1], Prof. Milind Wasekar[2]

[1]*M E- Mechanical Design, appearing,*
Sinhgad Academy of Engineering, Kondhwa,
Pune University, Pune, INDIA

[2]*Department of Mechanical Engineering,*
Sinhgad Academy of Engineering, Kondhwa,
Pune University, Pune INDIA

Abstract

The marginal increment in the discharge for the Centrifugal Pump tends to depreciate with each marginal rise in capacity of the pump; especially for the higher order pumps (25HP and above). The prominence of vortices along with turbulent flow at the regions in the suction pipe affects the flow of water and consequently the discharge. The discharge could further drop if the `sump' is not favourably designed for aiding the intake through the suction pipe. This work would focus on Design alternatives for minimizing the vortices within the suction pipe and enhancing the discharge through possible use of a manifold at the suction end. Alternatively, efforts would be pursued for addressing the Design of the Sump (Tank) for facilitating the flow of water at the suction end while smoothing out the in-rush of water at the extreme end of the suction pipe.

Keywords:alternative geometry, pressure drop, suction side, performance of pump, flow analysis

1. Introduction

The centrifugal pump with high suction performance are widely used in petrochemical, pharmaceutical and natural gas industries to deliver various media with low suction head such as easily vaporised liquid, cryogenic liquid etc. To improve suction performance remains an important topic in the development and application of centrifugal pump.It is an accepted fact that faulty design of pump sump or intake is one of the major causes of unsatisfactory operation of pumps in any pumping plant. The adverse flow conditions at a pump intake lead to occurrence of swirl and vortices, which in turn reduce the pump efficiency, induce vibrations and excessive bearing loads and lead to other operating difficulties.

The flow conditions at entry to a pump depend upon flow conditions in approach channel, sump geometry, location of pump intake with respect to the walls, velocity changes and obstructions such as piers, screens etc., and rotational tendencies in flow produced upstream of the pump bays. Analytical determination of the flow conditions in a sump is not an easy task due to the complex nature of the flow. Moreover the analytical solution may not completely predict the actual conditions in the sump due to the assumptions made for simplifying the analysis. Thus at present model studies are the only tool for developing a satisfactory design Of a pump sump, yet numerical simulation is a very good facility for reducing the time and cost involved .in the design process.

The main aim of sump is to provide water with uniform velocity during the pump operation, abnormal flow phenomena such as cavitation, flow separation, pressure loss, vibration and noise occur often by flow unsteadiness and instability. Especially, free and subsurface vortices containing air occurred in sump pumps seriously damage to pump system.

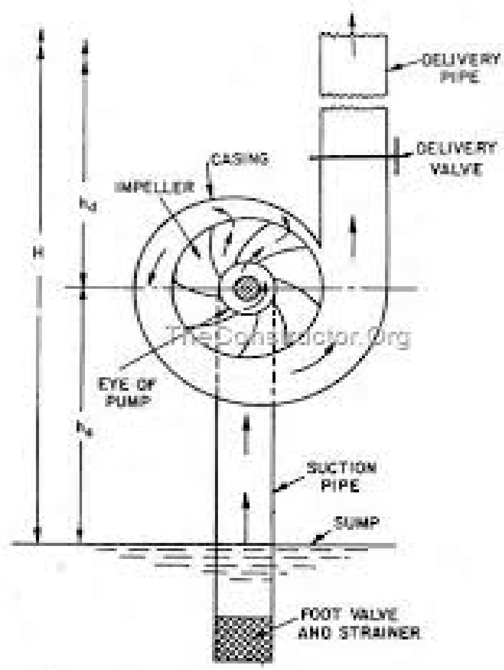

Figure 1: Centrifugal pump with suction system

According to the HI standard of Hydraulic Institute or JSME criteria for a pump sump design, these vortices should be prevented and their disappearance must be verified by model test in the construction of pump station. To reduce these vortices and for the advanced pump sump design with high performance, it is very important to know the detailed flow information in sump system.

For this purpose, many researchers have made experimental and numerical studies on the flow in pump-sump.

2. Objectives

- Identify the problem areas by studying the existing system
- Document the challenges to be addressed for enhancing the effectiveness of the pump
- Consider feasibility for redesign of the suction side of the Pump
- Analyse the multi-intake manifold design using CAE software, especially in the CFD domain
- Recommend the best alternative design for the suction side of the pumping system. And geometry were decides are:

1) Straight Pipe;
2) 2-intake suction manifold
3) 3-intake suction manifold
4) 4-intake suction manifold

3. Computational Fluid Dynamics

Computational fluid dynamics or CFD is the analysis of system involving fluid flow, heat transfer and associated phenomenon such as chemical reaction by means of computer based simulation. 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.

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. Hence all codes contain three main elements 1) pre-processor 2) solver 3) post-processor.

A. Pre-Processor

Pre-processing consists of the input of a flow problem to a CFD program by means of an operator friendly interface and subsequent transformation of this input into a form suitable for use by the solver. The solution to a flow problem (velocity, pressure) is defined as nodes inside each cell. The accuracy of a CFD solution is governed by the number of cells in the grid. In general larger the number of cells then betters the solution accuracy. The grid for the three dimensional model was created in GAMBIT. Due to the size and complexity of the pump care was taken while distribution of grid elements in the model. Considering the complexity of geometry, unstructured grid consists of triangular and tetrahedral element with TGrid scheme was used. And the element size is 5.The user activity in the pre-processing stage involves:

- Definition of the geometry of the region of interest: the computational domain.
- Grid generation-the sub-division of the domain into smaller, non-overlapping sub-domains: a grid or mesh of cells.
- Selection of the physical and chemical phenomena that needs to be modelled.
- Definition of fluid properties.Specification of appropriate boundary conditions at cells whichcoincides with or touch the domain geometry.

B. Solver

There are three different distinct streams of numerical solution technique: Finite difference, finite element and spectral methods. In outline the numerical methods that form the basis of the solver perform the following steps:

- Approximation of the unknown flow variables by means of simple functions.
- Discretisation by substitution of the approximation into the governing flow equations and subsequent mathematical manipulation.
- Solution of the algebraic equations.

C. Post-Processor

As in pre-processing a huge amount of development work has recently taken place in the post-processing field. Owing to the increased popularity of engineering workstations, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data visualisation tools. For post processor ANSYS FLUENT is used. These include:

- Domain geometry and grid display
- Vector plot
- Line and shaded contours plots
- 2D and 3D surface plot
- Particle tracking
- View manipulation
- Colour postscript output
-

4. Results ofCFD Analysis

A number of computations by the CFD analysis werecarried out for various conditions using a desktoppersonal computer. One case of computation took eighthours to get animation for flow visualization. In thefollowing analysis, water is considered as working fluid.The CFD software provided animation videofiles for visualunderstanding of flow pattern.

A. Pressure Distribution

Pressure distribution calculated by the CFD is shown infigures. Red colour indicates pressure level is maximum and blue is indicates pressure level is minimum. In the figure 2, for single pipe pressure is minimum at inlet and maximum at outlet. For two intake pipe, pressure is a maximum at the outlet as shown in figure 4. For three intake pipes and four intake pipe pressure is maximum at the outlet but for four intake pipe, pressure is minimum at the inlet side as compared with the two, three and four intake pipe.

B. Velocity Distribution

Stream lines calculated by the CFD Analysis are shown in figure. Velocity distributions are shown in Figure. Velocity of the flow is classified by colours. For single pipe, velocity is maximum at centre of pipe and minimum at inner surface of the pipe as shown in figure 3. For two intake pipe, a velocity is a maximum in single pipe connected to the two intake manifold while it's minimum in two intake manifold as shown in figure 5. For 3 intake and four intake pipe, velocity is near about constant as shown in figure 7 and figure 9.

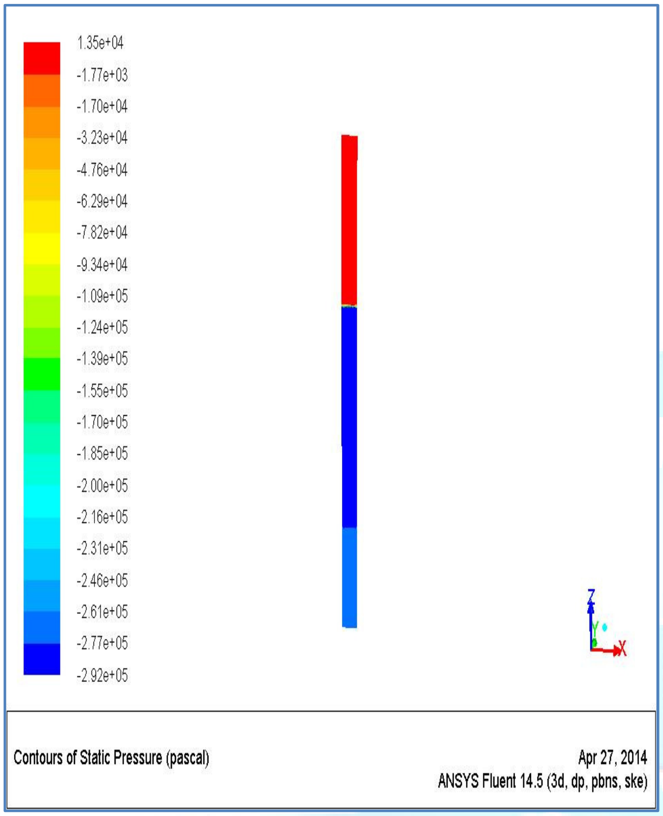

Figure 2: Pressure distribution in single pipe

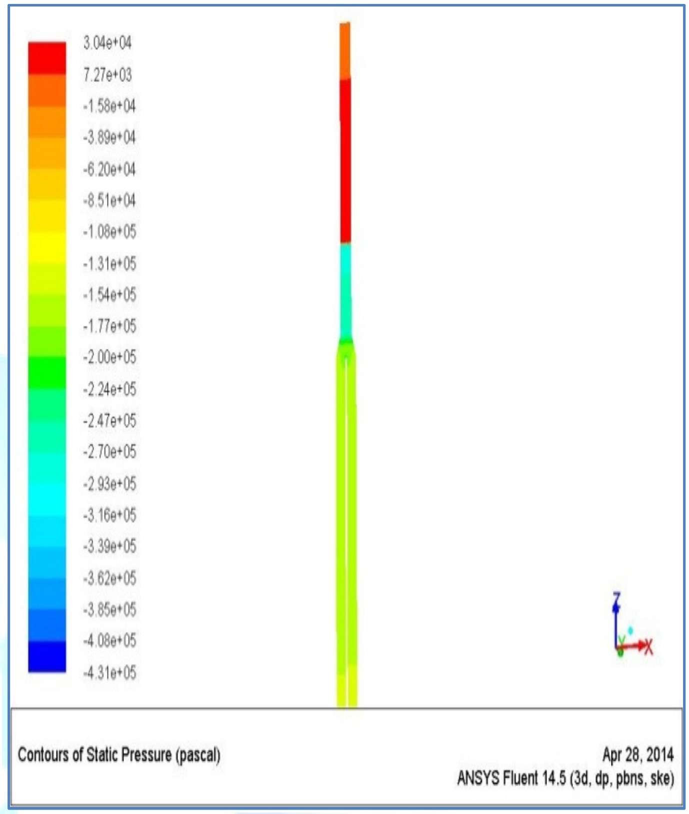

Figure 4: Pressure distribution in 2-intake pipe

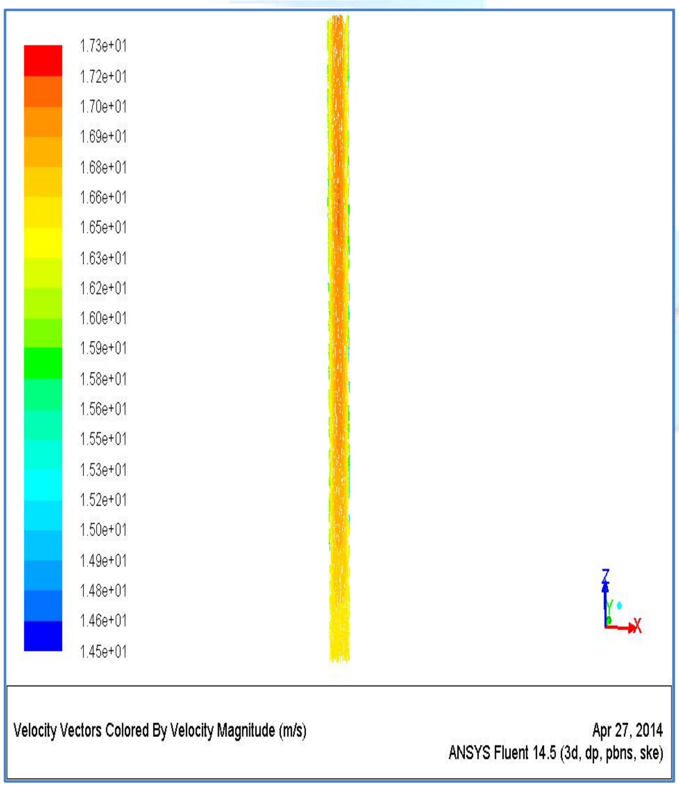

Figure 3: velocity distribution in single pipe

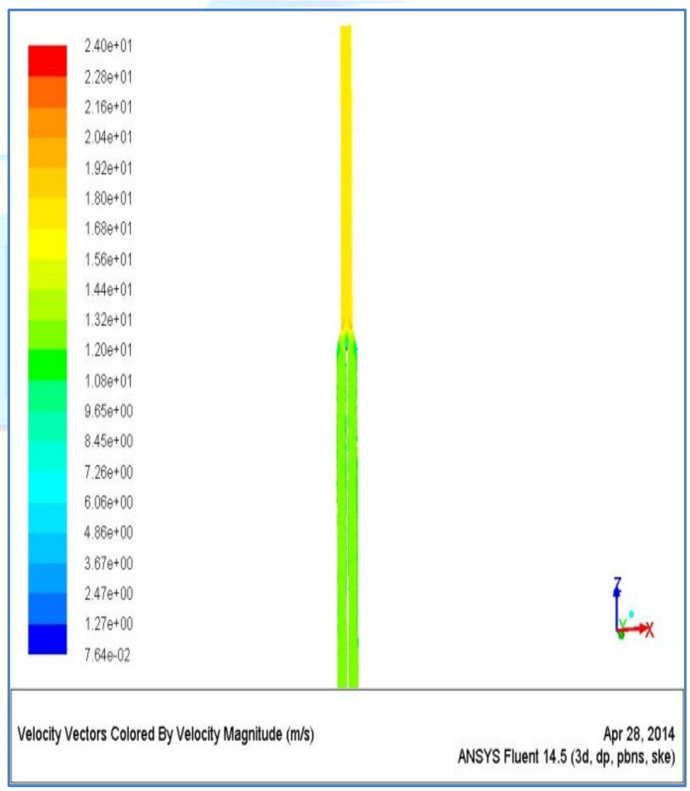

Figure 5: velocity distribution in 2-intake pipe

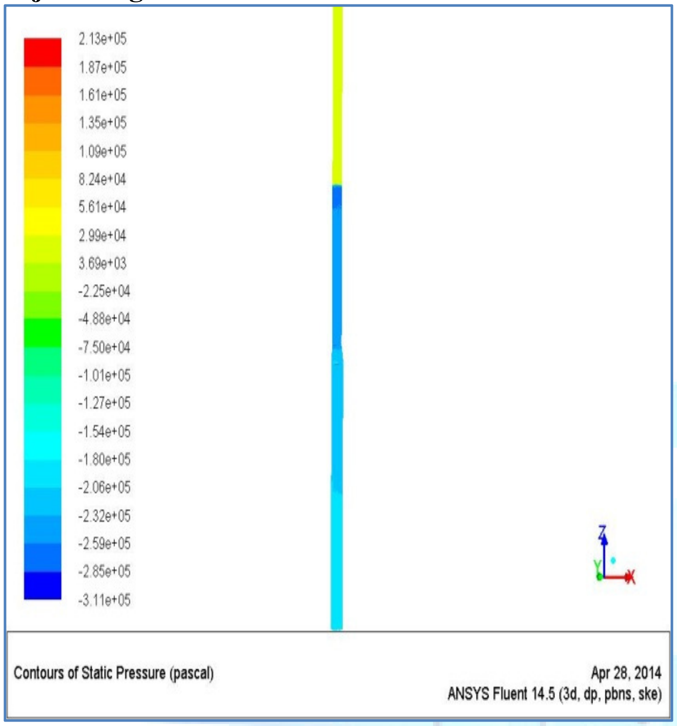

Figure 6: Pressure distribution in 3-intake pipe

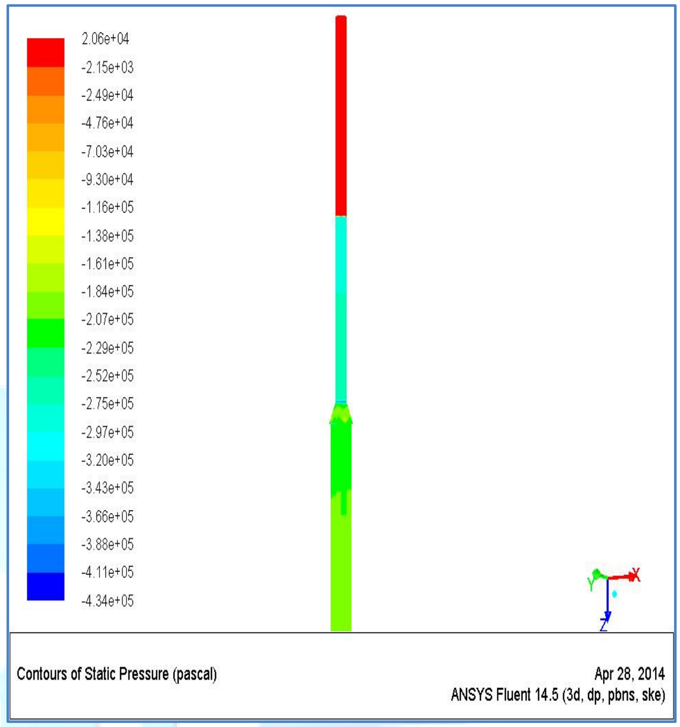

Figure 8: Pressure distribution in 4-intake pipe

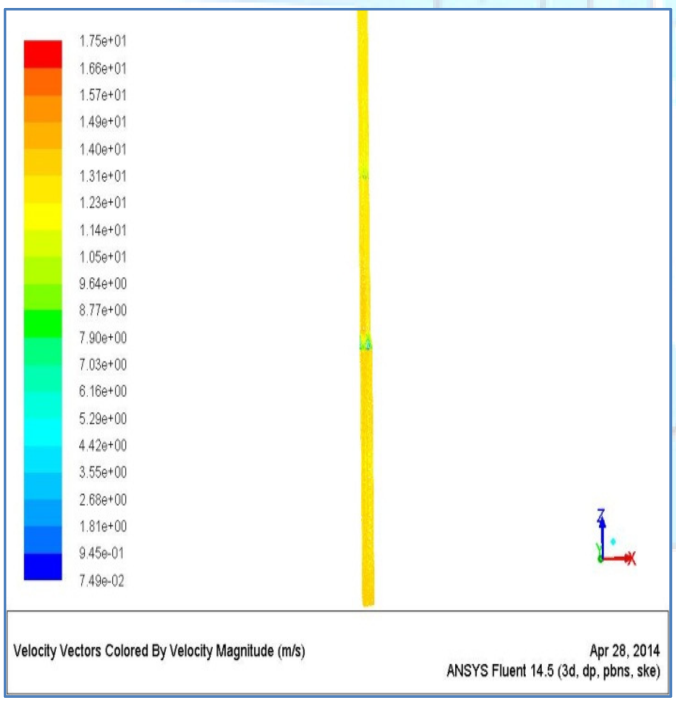

Figure 7: velocity distribution in 3-intake pipe

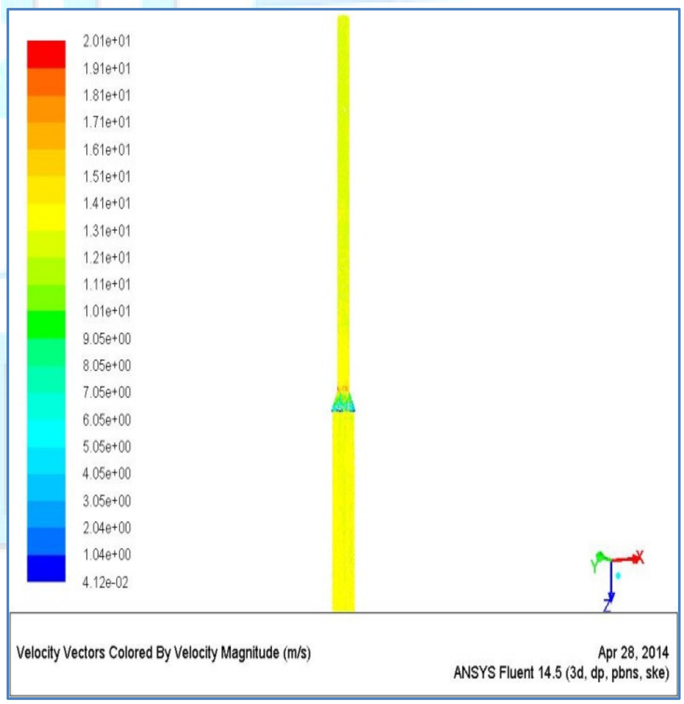

Figure 9: velocity distribution in 4-intake pipe

5. Conclusion

- The pressure drop is change for geometry depending upon the vortices and cavitation generate.
- Power required for two intake system is less (as shown in figure 10) by analyticallySo, cost is less.
- Pressure drop is less for two intake system(as shown in figure 11) by analytically and software analysis.
- For two intake geometry weight is less.
- The best geometry is 2 intake pipe manifold.

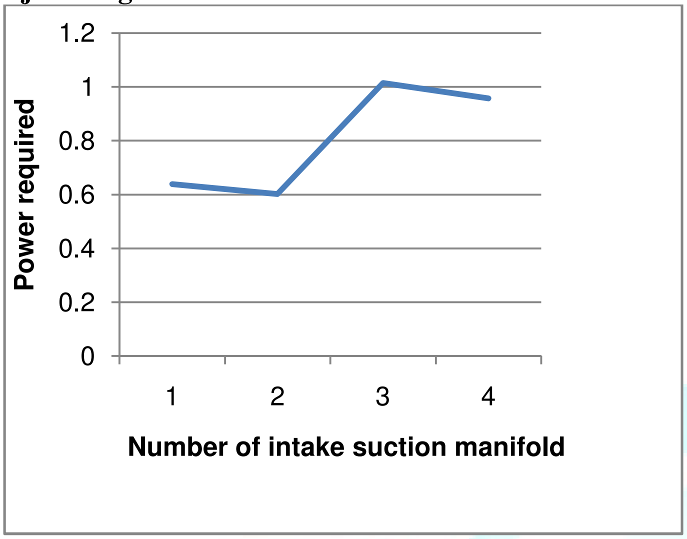

Figure 10: No. of intakesuction manifold vs. power required

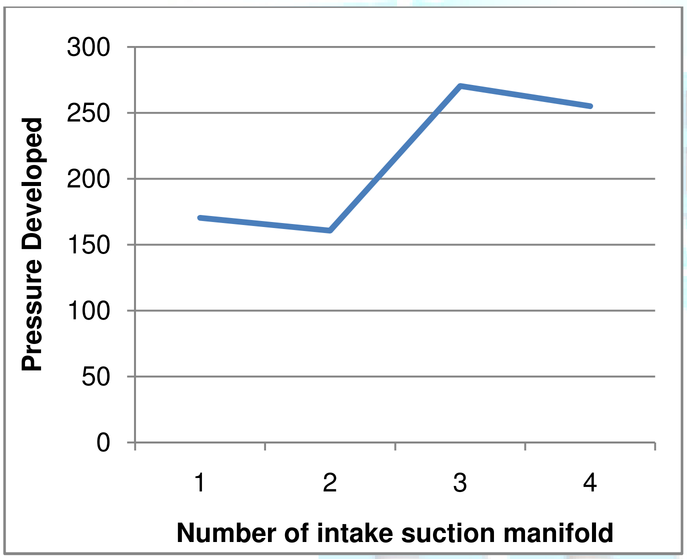

Figure 11: No. of intakesuction manifold pressure developed

Acknowledgements

Taking advantage on the occasion of submission of the paper, I take the opportunity to thank all those who are directly or indirectly involved in making of the paper and without whose guidance, co-operation and support this would not have been possible. I am highly obliged to Prof.M.K.Wasekar (Project guide), PG Co-ordinator Dr. Dinesh N. Kamble, HODProf. S. C. Shilawant, Principal Dr.Avinash G. Kharatand all the staff members for providing the spark in the dark and providing the opportunity to work under them and also for giving unfailing support and encouragement and not only for guiding me with their vast knowledge and experience, but also valuable advice while preparing the paper.

Last but not the least, I express my hearty acknowledgement to my Parents, Friends and Colleagues for their help and encouragement for making the paper complete.

References

[1] Kimberly Fernandez, Bernadette Pyzdrowski, Drew W. Schiller and Michael B. Smith, "Understand the Basics of Centrifugal Pump Operation", May 2002.

[2] Joe evans, Ph.D, "A brief introduction to centrifugal pump"

[3] Khin Cho Thin, MyaMyaKhaing, and KhinMaung Aye, "Design and Performance Analysis of Centrifugal Pump", World Academy of Science, Engineering and Technology 22, 2008.

[4] Mohamed F. Khalil, Sadek Z. Kassab, Ahmed A. Abdel Naby, A. Azouz, "Performance Characteristics of Centrifugal Pump Conveying Soft Slurry", American Journal of Mechanical Engineering,Vol. 1, No. 5, 103-112,2013.

[5] XianwuLuo, Yao Zhang, JunqiPeng, HongyuanXu, and Weiping Yu, "Impeller inlet geometry effect on performance improvement for centrifugal pumps", Journal of Mechanical Science and Technology 22 (2008) 1971~1976.