CFD ANALYSIS OF CHANGE IN SHAPE OF SUCTION MANIFOLD TO IMPROVE PERFORMANCE OF THE CENTRIFUGAL PUMP

Mr. Suraj K. Patil
PG Student,
Department of Mechanical Engineering
/BIGCE, Solapur University,
Maharashtra, India

Prof. S.M. Rajmane
Research Scholar,
WIT Research Center, Solapur University,
Solapur, Maharashtra, India

ABSTRACT

The design and optimization of turbo machine parts such as those in pumps and turbines is a highly complicated task due to the complex three-dimensional shape of the parts. Small differences in geometry can lead to significant changes in the performance of these machines. The paper uses mathematical modeling of the inlet manifold design and analysis using Computational fluid dynamics (CFD) with Geometry Parameterizations

INTRODUCTION

COMPUTATIONAL FLUID DYNAMICS

Basics of Computational Fluid Dynamics:
Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behaviour of systems involving fluid flow, heat transfer and other related physical processes. It works by solving the equations of fluid flow over a region of interest, with specified (known) boundary conditions for that region. CFD is a sophisticated computationally-based design and analysis technique. CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. This software can also build a virtual prototype of the system or device before can be apply to real world physics and chemistry to the model and this software will provide with images and data which predict the performance of that design. Computational fluid dynamics (CFD) is useful in a wide variety of applications and used in industry. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithm can be used to solve and analyze problems that involve fluid flows and also simulate the flow over a piping, vehicle or machinery. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic and turbulent flows are ongoing research. Onwards the aerospace industry has integrated CFD techniques into the design, R & D and manufacture of aircraft and jet engines. More recently the methods have been applied to the design of internal combustion engine, combustion chambers of gas turbine and furnaces also fluid flows and heat transfer in heat exchanger. Furthermore, motor vehicle manufactures now routinely predict drag forces, under bonnet air flows and surrounding car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes. This study deals with thestudy of water discharge from intake suction manifold. Computational fluid analysis is carried out to determine the velocity and pressure profile. Computational fluid analysis is carried using software CATIA V5 which is used to build the model and mesh it in GAMBIT and ANSYS FLUENT is used to carry out the velocity and pressure analysis. This total analysis is known as Computational fluid dynamics analysis. Today, well tested commercial CFD packages not only have made CFD analysis a routine design tool in industry but also helping the research engineer in focusing on the physical system more effectively. Before doing the analysis it is important to have an overview of what fluent is and how does it work.
CFD simulation process: It consist of three main steps as follows

i)   Pre-processor
ii)   Solver
iii) Post-processor

We briefly examine the function of each of these elements within the context of a CFD code. Before going for those processes we have to identify the problem; in that first includes define area means exactly for which result we looking for (i.e. pressure drop, mass flow rate, temperature drop etc.). Second things come that is identification of domain in which exact part of complete system have to identify by which expected result (i.e. pressure drop, mass flow rate, temperature drop etc.) will affect and try to examine that part only instead of complete system because it takes long time for simulation and required large memory space in computer system unnecessary.

i) Pre-Processor:
Geometry:
To trace the behavior of that fluid in terms of pressure drop, heat transfer rate, mass flow rate, temperature drop etc. we need to generate the model of that fluid not of mechanical structure. It means we need to generate modeling of fluid only not mechanical structure where it necessary.
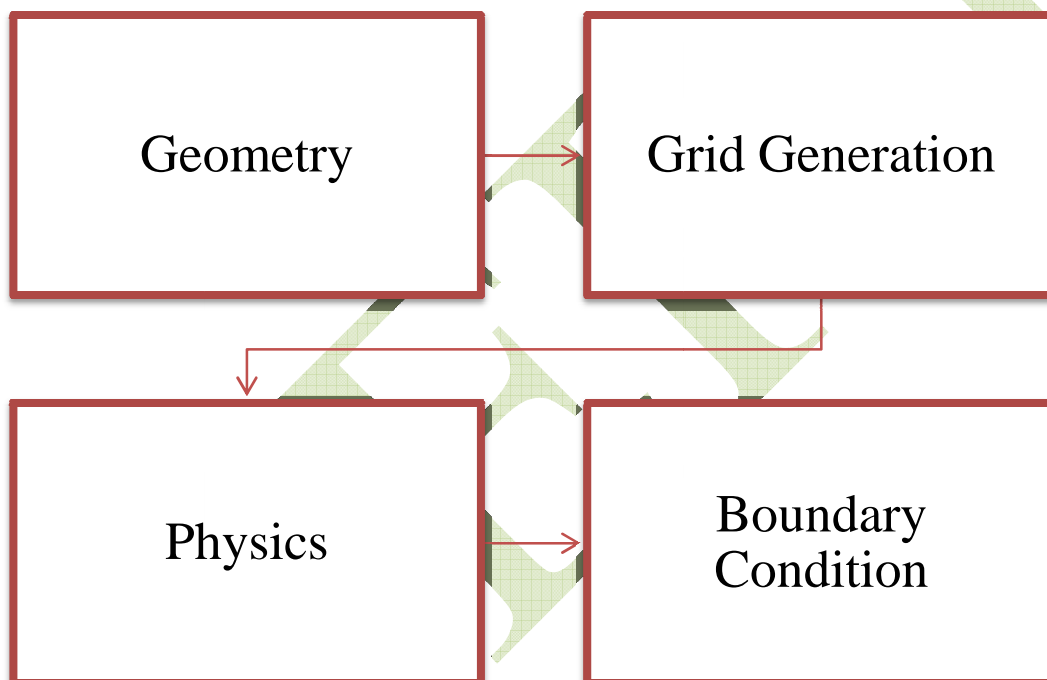
Flow chart of CFD process

Grid Generation:

Grid generation is also known as meshing which is a discretization of model into smaller regions called as grid or element. Generated grid has a significance effect on rate of convergence (or even lack of convergence), accuracy and computation time. So it is important to select appropriate elements type, meshing scheme and grid density.
For a given problem, you will need to

1. **Define material properties:**
    It is an essential step and this option is present just after the mesh scale option on ansys fluent interface. When you select appropriate solid or fluid material for your problem then all the mechanical and chemical properties belongs to that material in ansys material library get define to fluid domain model. If required material is not present in material library then there is one provision available in ansys fluent i.e.

    a)Fluid
    b)Solid
    c)Mixture

2. **Select appropriate physical models:**

   After material selection we have to define the domain of problem means weather it is related to viscous laminar flow, combustion problem or related to multiphase flow

   a)Turbulence
   b) Combustion
   c) Multiphase

3. **Prescribe boundary conditions at all boundary Zones:**

   In that we have to define exact boundary conditions to different zone of fluid domain model. Means here we defining the temperature, mass flow rate, free stream velocity wherever it is required as per the problem statement.

Solving

After assigning all boundary condition we have to select the relaxation tolerance for all equation which is used to determine the all parameter till outlet section. It is done by convergence in the solution. Convergence is reached when there is a change in solution variables from one iteration to the next which is negligible.
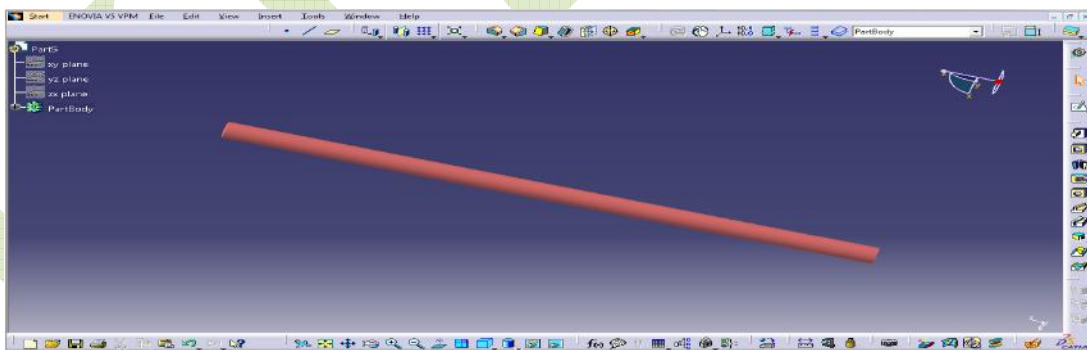
Post Processing

This is the final step in CFD analysis and it involves the organization and interpretation of the predicted flow data and production of CFD images and animations.

Result and Discussion

In this chapter we will discuss how to design the suction side. The efficiency of the centrifugal pump can be increased by number of ways such as modifying the geometry of the sump, increasing the diameter of the suction pump, having multiple pumps working in series, etc.
This results in better suction of the working fluid and as a result of it the mass flow rate of the fluid increases which directly increases the efficiency of the pump by reducing the motor HP and hence reducing the operational cost of the centrifugal pump.

The following diagram in Figure 4.1 shows a single suction pipe


In the similar way we can create all the models in CATIA for analysis. The following Figure shows a model of variants shape of manifold.
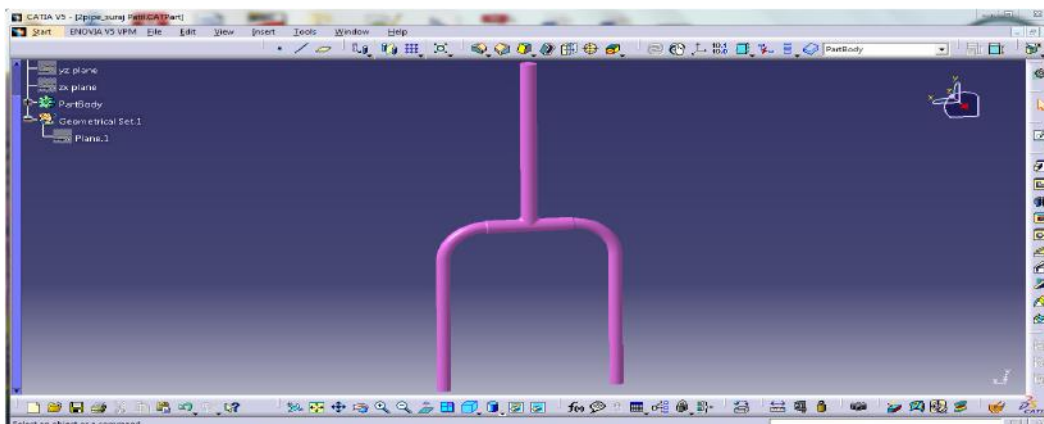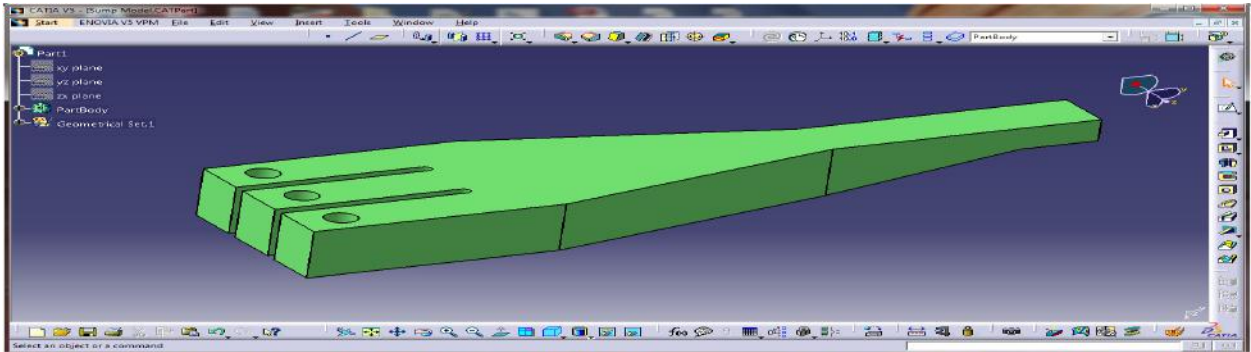
The model which is prepared in CATIA V5 is imported to new pre-processing software called GAMBIT for entering the boundary conditions and for tetrahedral meshing of the given geometry. Element Size of meshing is 5. The more fine mesh you perform the more accurate the result you will get in ANSYS FLUENT, but the fines mesh also increases the duration of the result along with the size of the file.

The following diagram in Figure 4.4 shows a single suction pipe and two suction pipe meshed geometry in Gambit Interface.
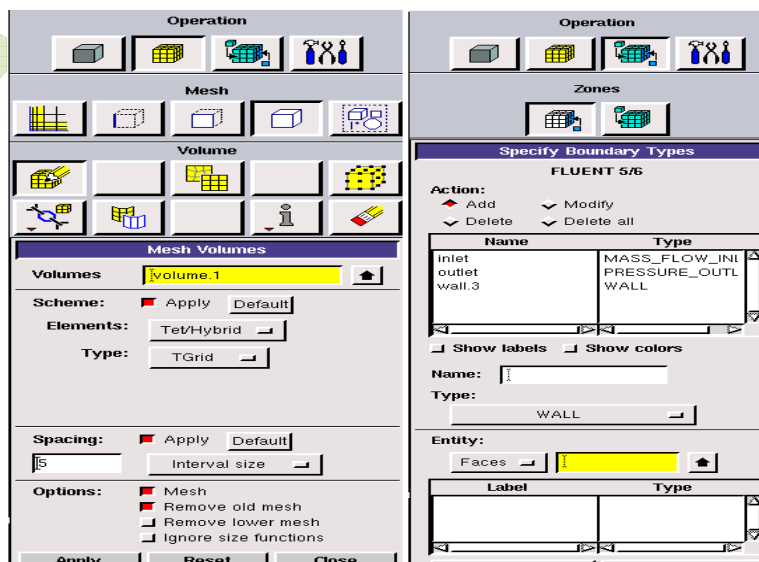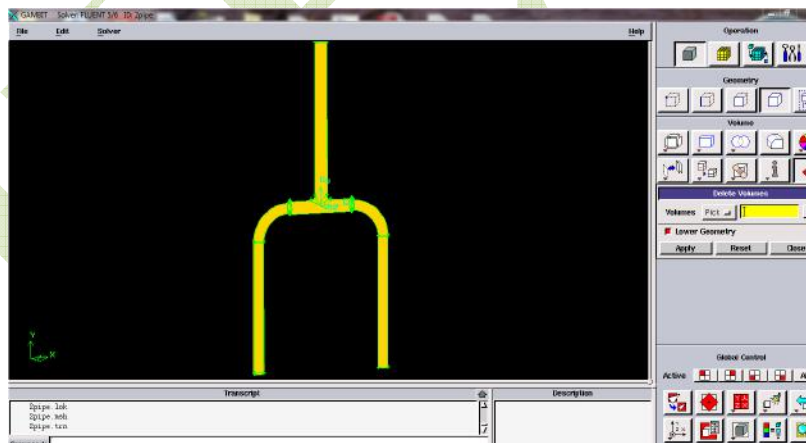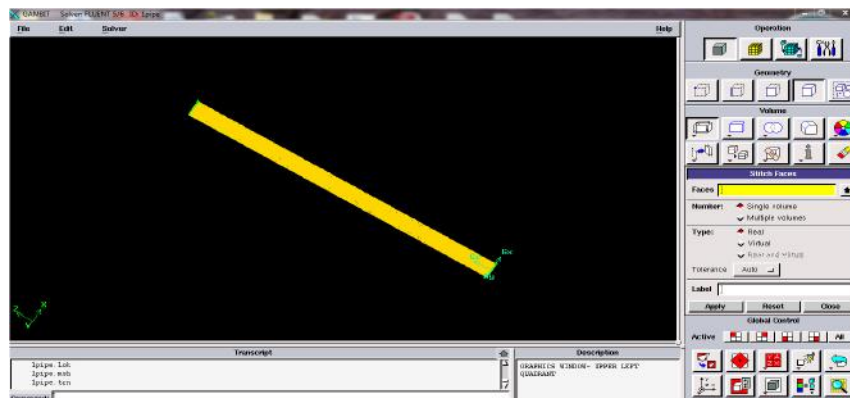
Layout of the boundary condition types in GAMBIT

Solving:
For solving we are using ANSYS Fluent Solver. In this interface following parameters used:
- System type: Pressure Based system
- Analysis type: Steady state condition
- Model: k-epsilon (2 equations) with realizable model
- Fluid Used: Water
- Mass flow rate at inlet: 2 kg/s
- Temperature: 27 deg^{0}C

Post-Processing:
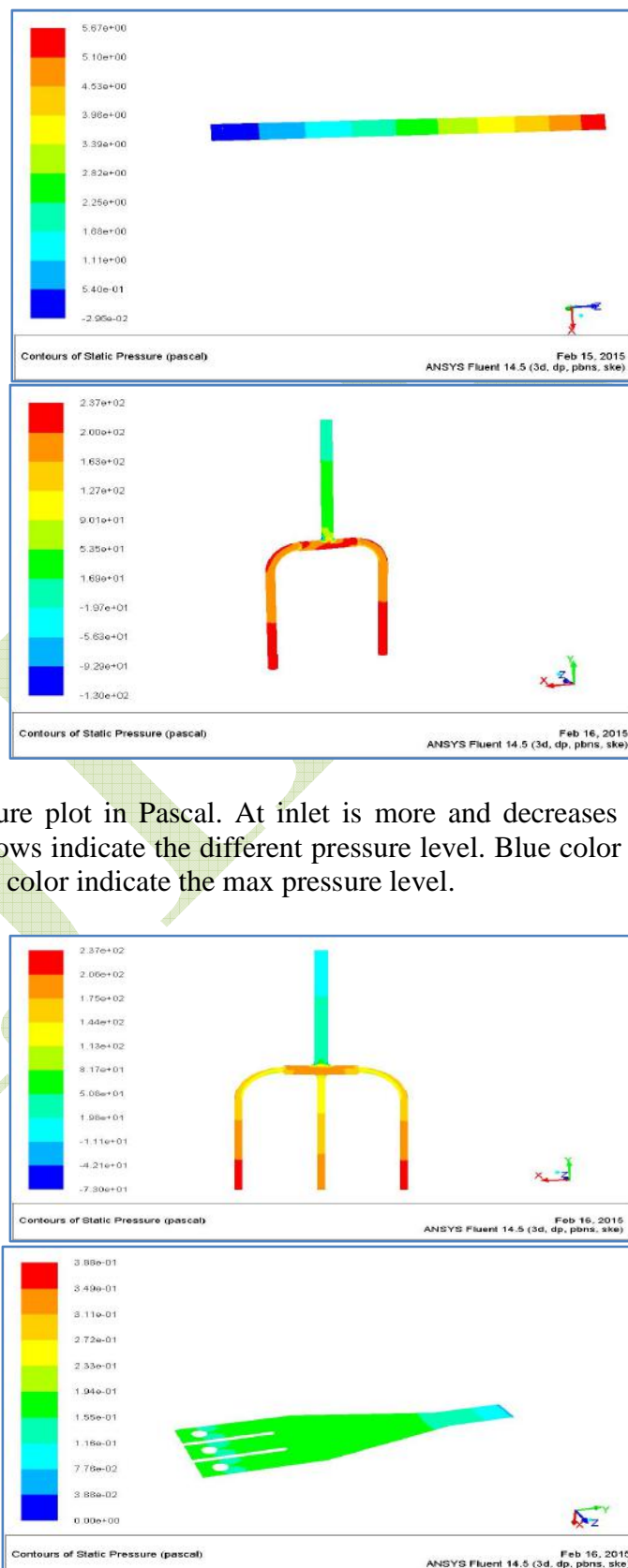ANSYS Fluent interface used for post –processing.

Fig shows that pressure plot in Pascal. At inlet is more and decreases as gradually along the length. Color strip shows indicate the different pressure level. Blue color indicates the minimum pressure level and red color indicate the max pressure level.

Velocity Plot:

Velocity plot shown in fig. color strip shows the different velocity levels in geometry. At wall o pipe velocity is minimum and at the centre of pipe velocity is max. Velocity vector shows the flow pattern of fluid. Vortices can be seen using this plot.
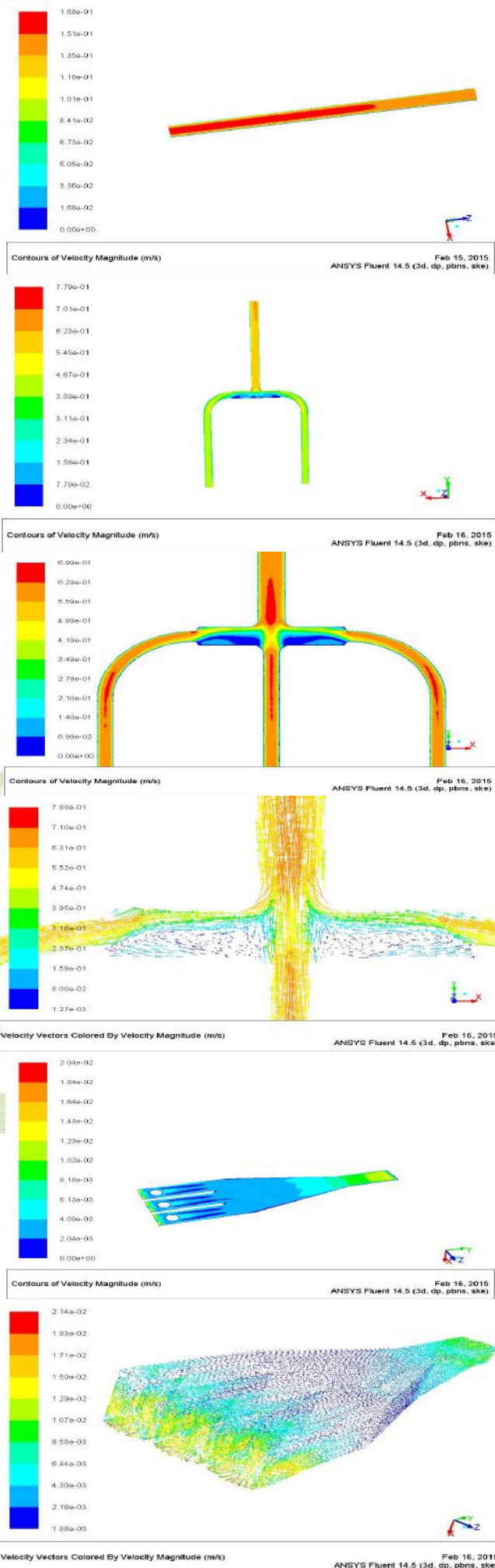
Summary

The comparison between single and modified model of suction pipe proposed in our thesis can be very well represented in a tabular format, as shown below

From below table pressure drop is minimum in 2 pipe configuration. Also minimum power required for this configuration.

Pressure developed in three pipe configuration is more as compared to other two variants. By changing the geometry of the three suction pipes, pressure drop and vortices get minimized. Efficiency of this configuration is more as compared to other variants.

Sr.No	Diameter	Length	Velocity	Pressure developed (Analytical) in Pascal	Power
1	76.2	3	0.548201	117.3169	0.36974
2	76.2	1	0.548201	77.55235	0.248601
	63.5	1	0.394705		
	63.5	1	0.394705		
3	76.2	1	0.657842	137.7397	0.430437
	44	1	0.548201		
	44	1	0.548054		
	44	1	0.548054		

CONCLUSION

CFD model used to study the effect of various parameters which reduces time as well as cost and hence could become an important tool for optimization of pump sump geometry. Redesign of the suction side of the pump facilitated the flow of water and improves the discharge and consequently the performance of the centrifugal pump.

FUTURE SCOPE

From the future point of view we can consider the following set of points:

✓ This work focused on water as a working fluid; further work could be undertaken for dealing with industrial fluids or coolants.

✓ ANSYS Fluent is used as a solver for this case; OPENFOAM or star CCM could be explored as alternative solver.

REFERENCES

1. Vibha p.pode, shylesha channapattanna, *evaluating performance of centrifugal pump through cfd while modifying the suction side for easting discharge, international journal of research in engineering and technology eissn: 2319-1163 | pissn: 2321-7308, jan-2014*

2. Sumit n.gavande, prashant d.deshmukh, swapnil s.kulkarni, *a technique to enhance the discharge of a multi intake centrifugal pump, international journal of advanced engineering research and studies e-issn2249–8974, volume: 03 issue: 01 | jan-2014*

3. S. Bin cheng, yonghai yu, *cfd simulation and optimization for lateral diversion and intake pumping stations, 2012 international conference on modern hydraulic engineering, procedia engineering 28 (2012) 122 – 127.*

4. Honggeng zhu, rentian zhang, guoqiang luo, bin zhang, *investigation of hydraulic characteristics of a volute-type discharge passage based on cfd, 2012 international conference on modern hydraulic engineering, procedia engineering 28 (2012) 27 – 32*

5. S.p. asoka, k. Sankaranarayanasamy, t. Sundararajan, g. Vaidyanathan, k. Udhaya kumar, *pressure drop and cavitation investigations on static helical-grooved square, Triangular and curved cavity liquid labyrinth seals, nuclear engineering and design 241 (2011) 843–853*

6. Chuang wei-liang, hsiao shih-chun, *three - dimensional numerical simulation of intake model with cross flow, journal of hydrodynamics 2011,23(3):314-324, doi: 10.1016/s1001-6058(10)60118-7.*

7. Zhang de-sheng, shi wei-dong, chen bin, guan xing-fan, *unsteady flow analysis and experimental investigation of axial-flow pump 2010,22(1):35-43 doi: 10.1016/s1001-6058(09)60025-1.*

8. Li yao-jun, wang fu-jun, *numerical investigation of performance of an axial-flow pump with inducer, journal of hydrodynamics, ser.b, 2007,19(6):705-711*

9. Chen hong-xun, guo jia-hong, *numerical simulation of 3-d turbulent flow in the multiintakes sump of the pump station, journal of hydrodynamics, ser.b, 2007,19(1):42-47*