IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X
PP 15-32
www.iosrjournals.org

Virtual analysis and experimental investigation of slurry pump for performance enhancement

P. Gurupranesh [1], R. C.Radha [2], S. Saravanan [3],
[1] *Assistant Professor, Indra Ganesan College of Engineering, Trichy, India*
[2] *Assistant Professor, St Michael College of Engineering and Technology, Karaikudi, India*
[3] *Assistant Professor, Indra Ganesan College of Engineering, Trichy, India*

ABSTRACT: *Slurry is a mixture of liquid combined with some solid particles. There are varieties of pumps used for pumping slurries. Centrifugal slurry pump utilises the centrifugal force generated by a rotating impeller to impart energy to the slurry in the same manner as domestic centrifugal pumps. Slurry pumps are widely being used to transport corrosive/abrasive and high concentration slurries in Industries. The aim of this project is to improve the performance of the slurry pump by optimizing the impeller size.*

The impeller was modelled in Solidworks 2012 software and CFD analysis was done using fluid flow simulation package. CFD analysis was carried to predict the performance of the pump; a comparative analysis was made with experimental investigations. Four pump impeller models were developed for the critical design parameters of the pump. Inlet diameter, outlet diameter, Inlet angle, outlet angle, Vane number, Vane thickness and impeller width were the parameters which got varied to improve the efficiency of pump.CFD analysis was performed in the virtual models to predict the pump performance before experimental study.CFD results were validated by real time experiments. CFD results reveals that the efficiency of the centrifugal pump filled with optimized was impeller increased from 20.71% to 28.95% of efficiency compared to base line pump. The substantial rise in efficiency of the pump was due to change in vane numbers from 3 to 4 and width from 4.5mm to 5mm

I. INTRODUCTION

A pump is a mechanical device for moving a fluid from a lower location to a higher location, or from a low pressure area to a high pressure area. Mechanical energy is given to the pump and it is then converted into hydraulic energy of fluid. Pumps produce negative pressure at the inlet so that the atmospheric pressure pushes the fluid towards the pump. The fluid coming into the pump is pushed towards the outlet mechanically where positive pressure is generated. Pumps are classified in number of ways according to their purpose, specifications, design, environment etc

II. SLURRY PUMP

2.1. Definition of a slurry.

Slurry is a mixture of virtually any liquid combined with some solid particles. The combination of the type, size, shape and quantity of the particles together with the nature of transporting liquid determine the exact characteristics and flow properties of the slurry.

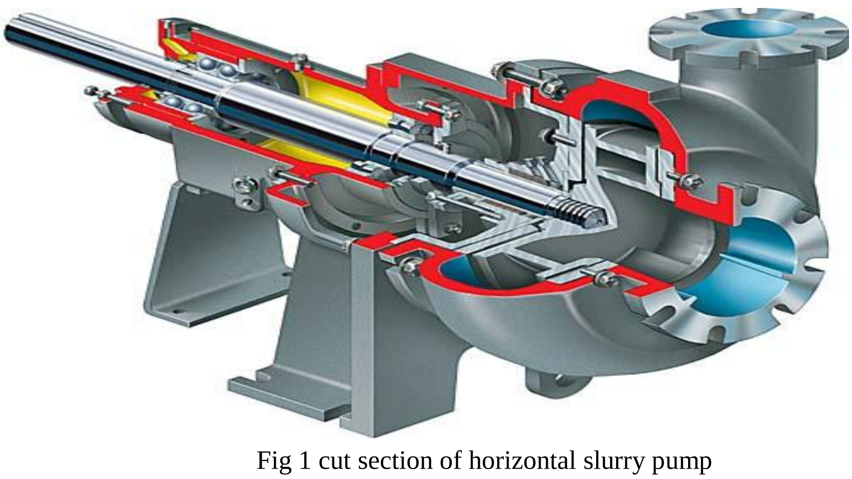


Fig 1 cut section of horizontal slurry pump

IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X
PP 15-32
www.iosrjournals.org

2.2. What is a slurry pump?

There are large numbers of pumps of various types used in the pumping of slurries. Positive displacement and special effect types such as Venturi eductors are used but by far the most common type of slurry pump is the centrifugal pump. The centrifugal slurry pump utilises the centrifugal force generated by a rotating impeller to impart energy to the slurry in the same manner as clear liquid type centrifugal pumps.

Centrifugal slurry pumps need to consider impeller size and design, its ease of maintenance, the type of shaft seal to be used and the choice of the optimum materials. This is needed to withstand wear caused by the abrasive, erosive and often corrosive attack on the materials. Many other important considerations are also required.

A slurry pump is a type of centrifugal pump, lobe pump or peristaltic hose pump based on physics principle that increases the pressure of liquid and solid particle mixture (aka slurry), through centrifugal force (a rotating impeller) and converts electrical energy into slurry potential and kinetic energy.

Slurry pumps are widely used to transport corrosive/abrasive and high concentration slurry in many industries such as Gold, Silver, Iron ore, Tin, Steel, Coal, Titanium, Copper, Mineral sands, Lead and Zinc. Various other industries include Molybdenum, Electric Utilities, Oil Shale, Water and Sewage Utilities, Building areas, Sand and Gravel, Tobacco and Agriculture (hog, poultry, and dairy manure).

Slurry pumps are grouped based on their impeller quantity, Shaft position from the horizontal, way of impeller suction and pump casing structures. Types of pump on each group are given below.

- impeller quantity: single stage and multistage slurry pump
- shaft position from the horizontal: horizontal and vertical slurry pump
- Way of impeller suction: single suction and double suction slurry pump
- pump casing structure: solid casing, horizontal split-case, and vertical split-case slurry pump. Fig 1.2 shows the cut section of the horizontal slurry pump.

The parameters determined before selecting an appropriate slurry pump include capacity, head, solids handling capacity, efficiency and power, speed and NPSH.

2.3. Characteristics of slurry

Slurries can be broadly divided into the two general groups of non-settling or settling types. Non-settling slurries entail very fine particles which can form stable homogeneous mixtures exhibiting increased apparent viscosity. These slurries usually have low wearing properties but require very careful consideration when selecting the correct pump and drive, because they often do not behave in the manner of a normal liquid. When fine solids are present in the slurry in sufficient quantity to cause this change in behaviour away from a normal liquid, they are referred to as being non- Newtonian.

Settling slurries are formed by coarser particles and tend to form an unstable mixture and therefore particular attention must be given to flow and power calculations. These coarser particles tend to have higher wearing properties and form the majority of slurry applications. This type of slurry is also referred to as being heterogeneous.

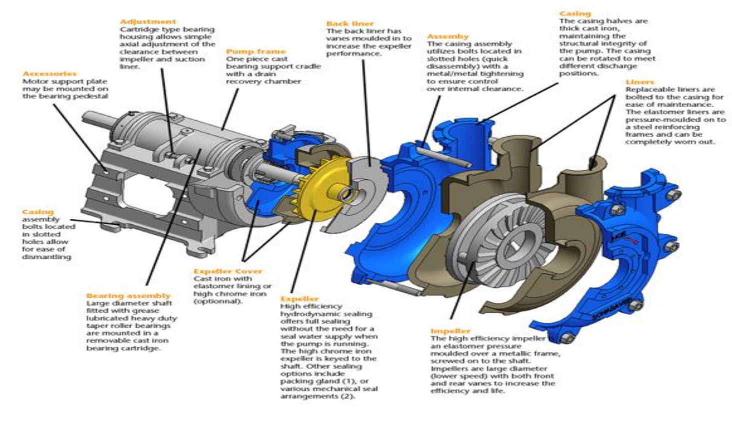


Fig 2 component of slurry pump

2.4. Components of slurry pump

2.4.1. Impeller

The impeller, either elastomer or high-chrome material, is the main rotating component which normally has vanes to impart the centrifugal force to the liquid.

2.4.2. Casing

Split outer casing halves of cast contain the wear liners and provide high operation pressure capabilities. The casing shape is generally of semi-volute or concentric, efficiencies of which are less than that of the volute type.

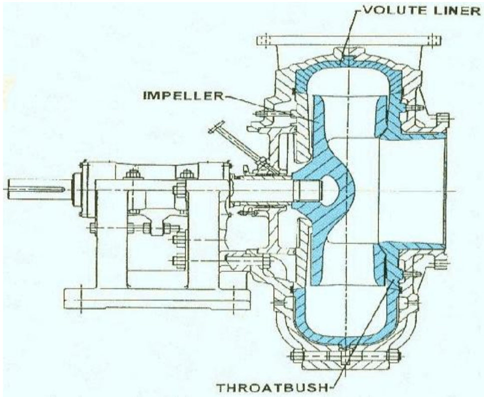


Fig 3 Cut section of slurry pump

2.4.3. Shaft and Bearing Assembly

A large diameter shaft with a short overhang minimizes deflection and vibration. Heavy-duty roller bearing are housed in a removable bearing cartridge

2.4.4. Shaft sleeve

A hardened, heavy-duty corrosion-resistant sleeve with O-ring seals at both ends protects the shaft. A split fit allows the sleeve removed or installed quickly.

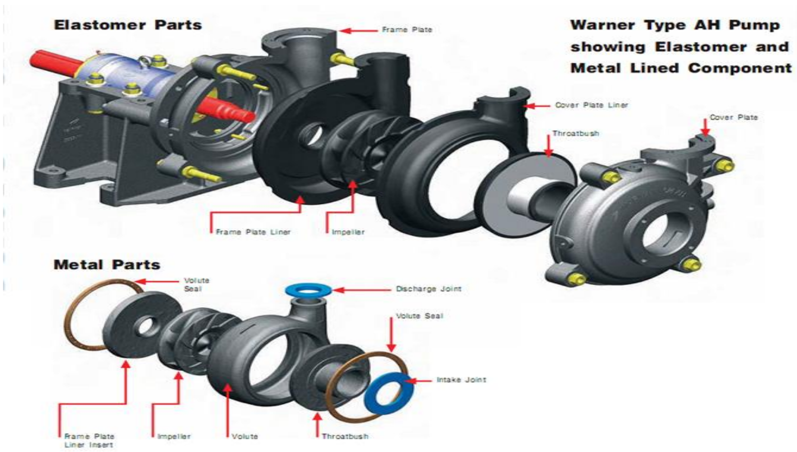


Fig. 4Explode diagram of slurry pump

2.4.5 Shaft Seal

Expeller drives seal, packing seal, and Mechanical seal.

2.4.6 Drive Type

V-belt drive, gear reducer drive, fluid coupling drive, and frequency conversion drive devices.

2.5. Concepts of material selection

Selection of the type of materials to be used for slurry pumping applications is not a precise procedure. The procedure must first account for all the factors (variable characteristics) of the particular slurry. The procedure must take into account the constraints imposed by the following:

a) Type of pump
b) Pump speed
c) Options within the range of the models available.

The basic data required to make a selection of the type of material is:

a) The particle sizing of the solids to be pumped
b) The shape and hardness of these solids
c) The corrosive properties of the "liquid" component of the slurry to be pumped.

The material selection for the pump liners and impellers is made from two basic types of materials:

a) elastomers
b) wear/erosion resistant cast alloys

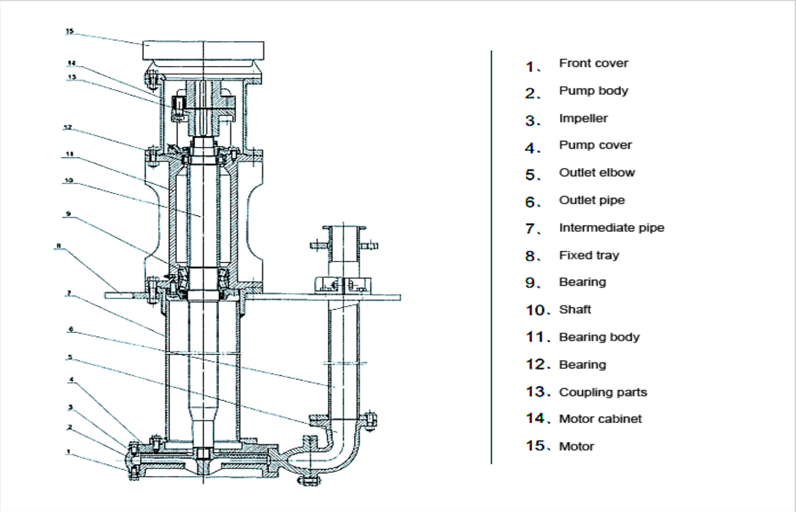


Fig 5 cut section of slurry pump

2.5.1. Elastomers

The criteria for selection of the three elastomers commonly used are:

a) Natural Rubber
i) Excellent erosion resistance for liners (against solids up to 15mm size), but limited to particles of 5mm size for impellers.
ii) May not be suitable for very sharp edged solids.
iii) May be damaged by oversized solids or trash.
iv) Impeller peripheral speed should be less than 27.5 m/s, to avoid the thermal breakdown of the liner, adjacent to the outer edge of the impeller. (Special formulations are available to allow speeds up to 32 m/s in certain cases).
v) Unsuitable for oils, solvents or strong acids.
vi) Unsuitable for temperatures in excess of 77°C.

(b) Polyurethane
i) Used for pump side liners, where the peripheral speed of the impeller is higher than 27.5 m/s, (and precluding the use of standard rubber) and used for impellers where occasional trash may damage a rubber impeller.
ii) Erosion resistance is greater where erosion is of a sliding bed type rather than one of directional impact.

IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X
PP 15-32
www.iosrjournals.org

iii) Has less erosion resistance to fine solids than natural rubber. Has greater erosion resistance to coarse sharp edged particles than natural rubber, in some circumstances.

iv) Unsuitable for temperatures exceeding 70°C and for concentrated acids and alkalies, ketone, esters, chlorinated and nitro hydrocarbons.

c) Synthetic Elastomers: Neoprene, Butyl, Hypalon, Viton A and others

These are used in special chemical applications under the following conditions:

i) Not as erosion resistant as natural rubber.

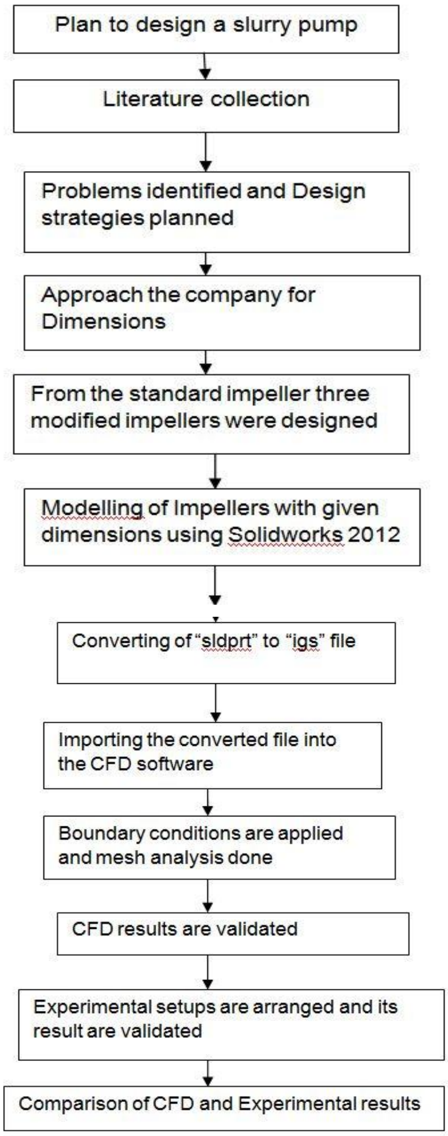
ii) Have a greater chemical resistance than natural rubber or polyurethane.

iii) Generally allows higher operating temperature than natural rubber or polyurethane.

2.5.2. Wear/erosion resistant cast alloys

Wear resistant cast alloys are used for slurry pump liners and impellers where conditions are not suited to rubber, such as with coarse or sharp edged particles, or on duties having high impeller peripheral velocities or high operating temperatures.

III. METHODOLOGY



3.1Plan to design a slurry pump

It is planned to design a slurry pump because it is widely used to transport corrosive/abrasive and high concentration slurry in many industries.

3.2 Literature Collection

Literature related to centrifugal slurry pump flow analysis, CFD application in slurry pump, Dimension analysis, idea for the formulae used for Performance enhancement are done.

3.3 Problems identified and Design strategies planned

The problem identified from the slurry pump is that the mixture of slurry water density rises means it will decrease the efficiency of the pump. So it is planned to design a slurry pump having increased in the thickness of the impeller and modifying the vane angle as well as vane number as per the reference from the journal. This design is planned to increase the performance of the slurry pump in order to enhance the efficiency. For that the Modelling of the slurry pump impeller is done using CAD (solid works 2013)software and analyzing the impeller to find the process parameters in CFD (fluid flow simulation)software.

3.4 Dimension for design from Industrial setup

This project has four type of impeller. One standard impeller which the dimension is taken from the industrial setup. The inlet diameter, outlet diameter, vane number, inlet angle, outlet angle, vane thickness, impeller width which are taken from the industrial setup are modified and three more impellers are designed to check for performance improvement.

3.5 Model construction

Three-dimensional model of an impeller was first created in Solidworks 2012 software and exported into STEP files. The STEP files were then imported into fluid flow simulation, the mesh generator. The fluid volume was split into a rotating fluid volume, a scroll volume, an inlet cone volume, and an inlet/outlet duct volume. The inlet and outlet ducts were intentionally set to simulate the actual measuring situation and to provide better boundary conditions for simulations. The flow was assumed fully developed when leaving the inlet and outlet ducts. The impeller wheel volume was defined as a rotating reference frame with constant speed and other blocks were defined in a stationary frame. This setup is referred to as a “frozen rotor” model.

3.6 Design is made by solid works software

The four impellers including the standard impeller and three modified impellers which are named as impeller A, impeller B, impeller C are designed by solid works 2012 software. Three Dimensional models of the Impellers created using solid works 2012 software are

Fig. 6 Model of standard impeller

Fig. 7 Model of modified impeller A

Fig. 7 Model of modified impeller B

Fig. 8 Model of modified impeller C

Table 1: initial conditions

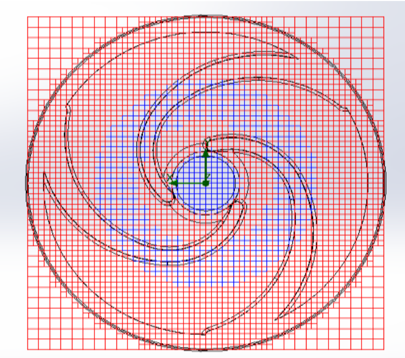
Thermodynamic parameters	Static Pressure: 101325.00 Pa Temperature: 293.20 K
Turbulence parameters	Turbulence intensity and length Intensity: 2.00 % Length: 3.891e-004 m

Meshing

In this software we have eight mode of meshing. Of them we go for mode3 which is default mesh which is having good accuracy and time factor when compare to other modes. Each mode is having their disadvantage when we take mode 1and 2 means the accuracy is rough but time consumption is low due to the size of the control volume. If we go for mode 4 and 5 means accuracy is more when compare to lower modes but time consumption is high due to small size of mesh. If we go for mode 6 and 7 means accuracy is more when compare to lower modes but time consumption is very high due to very small size of mesh. If mode 8 is selected it is a fine mesh so the accuracy is more compare to all other modes but time consumption for meshing and solver is high. It is because of considering all this factor we go for mode 3 which is the optimum mode. Meshing here is to divide one control element into 33,728 equal control volumes. The control volumes are hexahedral. The operation of meshing is performed as follows.

To start meshing in this software the requirements in system software, General information, Initial mesh settings, computational domain and its size

Meshing Diagram



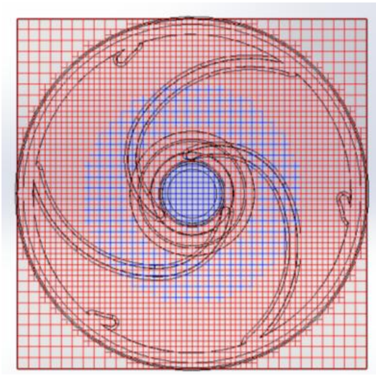
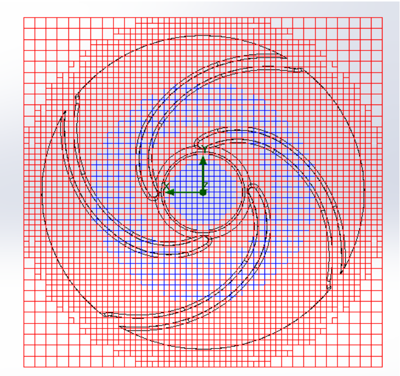


Fig 9 Standard Impeller **Fig 10 Impeller A**



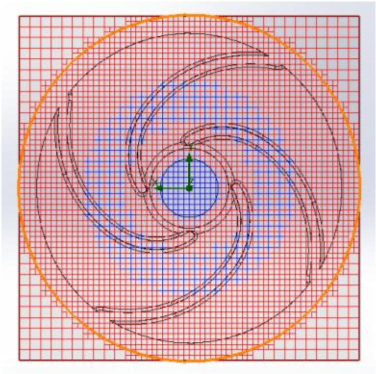


Fig 11Impeller B **Fig 12 impeller C**

Introduction to CFD

Computational fluid dynamics (CFD) uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow (the Navier- strokes and allied equations), for predefined geometries and boundary conditions. The result is a wealth of the predictions for flow velocity, temperature, density, and chemical concentrations for any region where flow occurs.

CFD analysis begins with a mathematical model of a physical problem, conservation of matter, momentum, and energy must be satisfied throughout the region of interest. Fluid properties and modeled empirically. Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, in viscid, two dimensional). Provide appropriate initial and boundary conditions for the problem. CFD applies numerical methods (called discretization) to develop approximations of the governing equations of fluid mechanical in the fluid region of interest. The solution is post processed to extract quantities of interest (e.g. lift, drag, torque, heat transfer, separation, pressure loss, etc.).

Practical advantages of employing CFD

The followings are among the many reasons why CFD is being widely used today.

- ✓ CFD predicts performance before modifying or installing the systems:

  Without modifying and/or installing actual systems or prototype, CFD can predict what design change is most crucial to enhance performance.

- ✓ CFD provides exact and detailed information about HVAC design parameters:
  The advances in HVAC/IAQ technology require broader and more detailed information about the flow within an occupied zone, and the CFD technique meets this goal better than any other methodize. Theoretical or experimental methods.

- ✓ CFD saves cost and time:
  - CFD costs much less than experiments because physical modifications are not necessary. (Note that the cost and time for physical changes/ modifications increase almost exponentially as the size of the system increases)
- ✓ C F D is reliable:
  - Most importantly, numerical schemes and methods that CFD is based on are improving rapidly so that reliability on the results produced by CFD is getting very high. Increased reliability makes CFD a dependable tool in any design and analysis purpose. CFD as an engineer's tool the concept of virtual prototyping conceptual design engineering design CFD simulation data analysis final design (prototype).

- ✓ Limitations of CFD
    - ✓ Physical models

- ✓ CFD solutions rely upon physical models of real world processes (e.g. turbulence, compressibility, chemistry, multiphase flow, etc.).
- ✓ The CFD solutions can only be as accurate as the physical models on which they are based.
    - ✓ Numerical errors

- ✓ Solving equations on a computer invariably introduces numerical errors.

- ✓ Round-off error: due to finite word size available on the computer. Round- off errors will always exist (through they can be small in most cases).
- ✓ Truncation errors: due to approximations in the numerical models.

- ✓ Truncation errors will go to zero as the grid is redefined. Mesh refinement is one way to deal with truncation error.

    - ✓ Boundary conditions
- ✓ As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model.
- ✓ Example: flow in a duct with sudden expansion. If flow is supplied to domain by a pipe, you should use a fully-developed profile for velocity rather than assume uniform conditions.

    - ✓ Discretization

    - o Domain is discretized into a finite set of control volumes or cells. The discretized domain is called the “grid” or the “mesh”. General conservation (transport) equations for mass, momentum, energy, etc., are discretized into algebraic equations. All equations are solved to render flow field.
    - o Experimental study
- ✓ The Impellers are manufactured then the test setup is assembled. The test is conducted for all the impellers with four trials. From the four trials the optimum values are selected.
    - o Comparison of CFD value and Experimental value
    - o Comparison of CFD and experimental values are done to find the impeller design which is optimum for increasing the performance of the slurry pump in order to enhance the efficiency.

IV. RESULT AND DISCUSSION

CFD simulations are carried out on the impeller model to predict its performance by giving its boundary conditions as input. Successive iterations are done by the software to obtain the characteristics such as Velocity, pressure head, flow rate, discharge and efficiency. Fluid consider for analysis is slurry.

4.1. Total Pressure
From the standard impeller three impellers were designed. They are considering as Impeller A, Impeller B, Impeller C. The following figures give the total pressure for the impellers.
4.1.1 Total Pressure for Standard Impeller
The diagram shows the total pressure for standard impeller. In this the pressure increased to 116995 Pa. In standard impeller when the impeller rotates in 2000rpm means the pressure generate from the centre passage of the impeller.

IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X
PP 15-32
www.iosrjournals.org

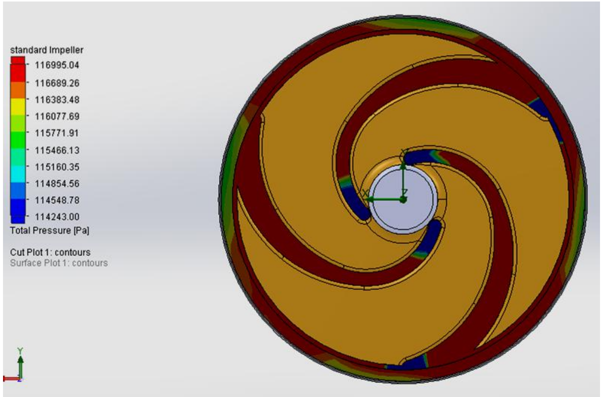


Fig 13 Total pressure for standard impeller

4.1.2 Total Pressure for Impeller A

The diagram shows the total pressure for impeller A. In this the pressure increased to 145709 Pa. In impeller A when the impeller rotates in 2000rpm means the pressure generate from the centre passage of the impeller.

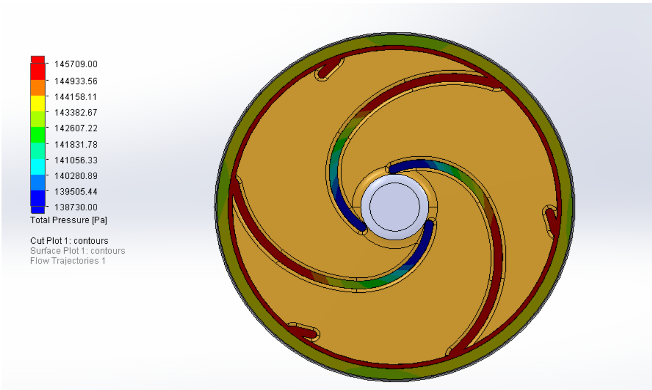


Fig 14 Total pressure for impeller A

4.1.3 Total Pressure for Impeller B

The diagram above shows the total pressure for impeller B. In this the pressure increased to 142939 Pa. In impeller B when the impeller rotates in 2000rpm means the pressure generate from the centre passage of the impeller.

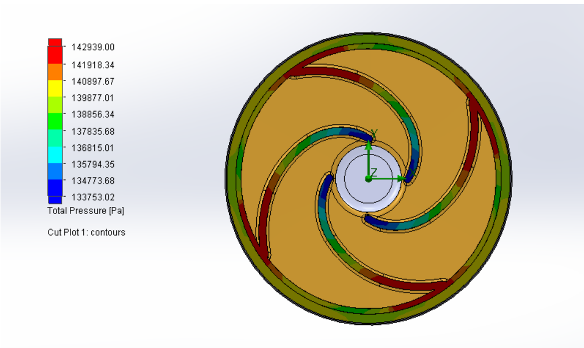


Fig 15Total pressure for impeller B

4.2 VELOCITY OF IMPELLERS

When total pressure increases for the impeller means velocity decreases. The velocity diagram for the impellers are given below.

4.2.1 VELOCITY OF STANDARD IMPELLER

The figure below gives the velocity of standard impeller and the value is 15.08m/s for the total pressure of 116995 Pa.

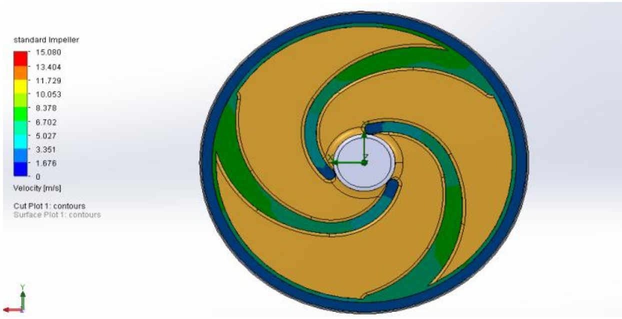


Fig 16 velocity for standard impeller

4.2.2 VELOCITY OF IMPELLER A

The figure below gives the velocity of impeller A and the value is 14.00m/s for the total pressure of 145709 Pa.

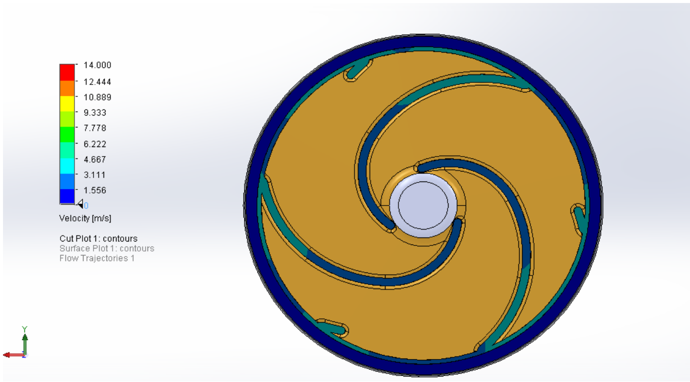


Fig 17 velocity for impeller A

VELOCITY OF IMPELLER B

The figure below gives the velocity of impeller B and the value is 13.02m/s for the total pressure of 142939 Pa.

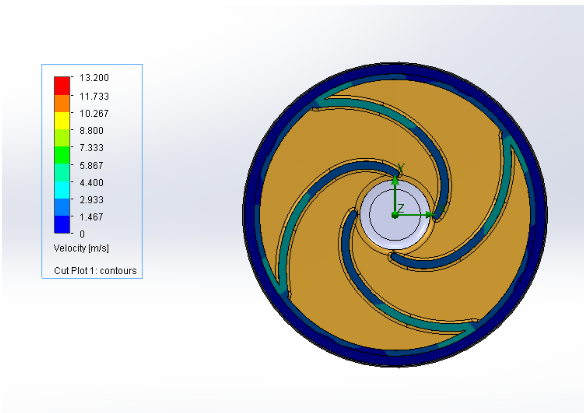


Fig 18 velocity for impeller B

EFFICIENCY OF THE IMPELLERS

Efficiency of impeller is obtain in CFD by calculating the flow rate, Total pressure and discharge of fluid

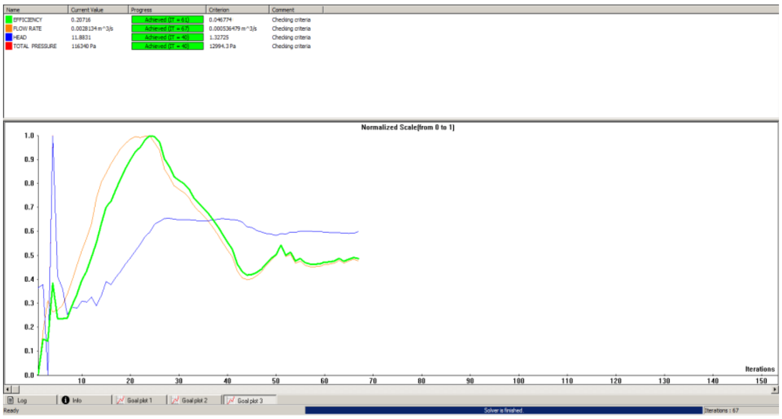


Fig 19 Efficiency graph for standard impeller

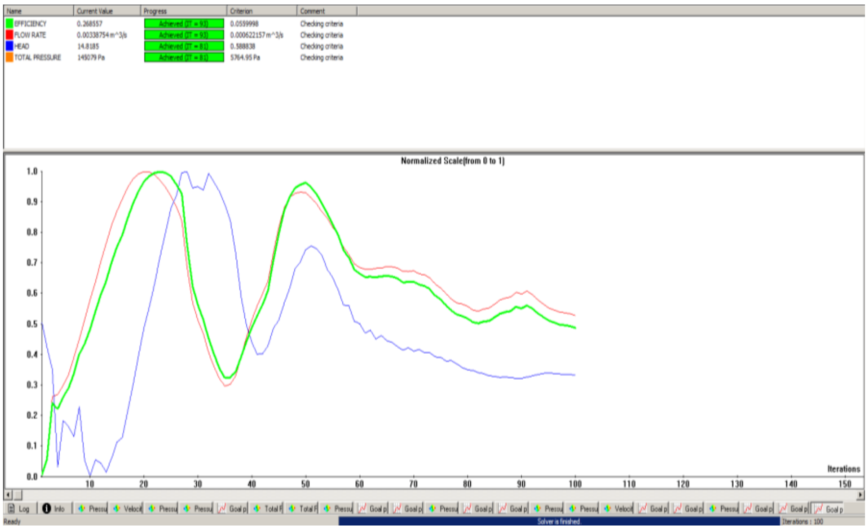


Fig 20 Efficiency graph for impeller A

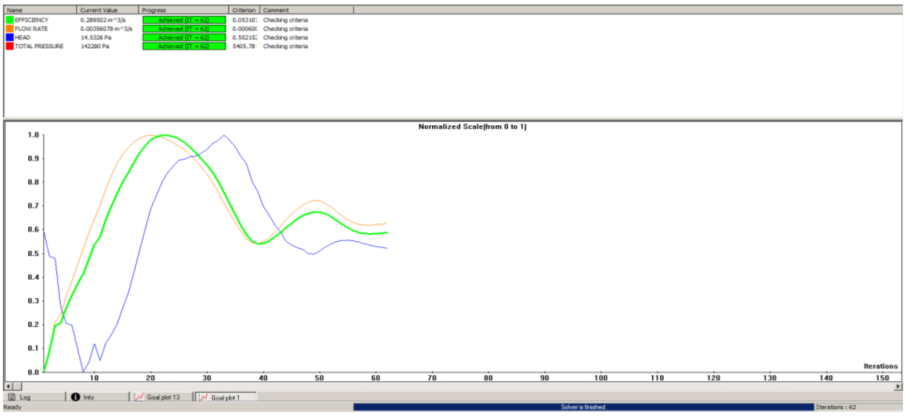


Fig 21 Efficiency graph for impeller B

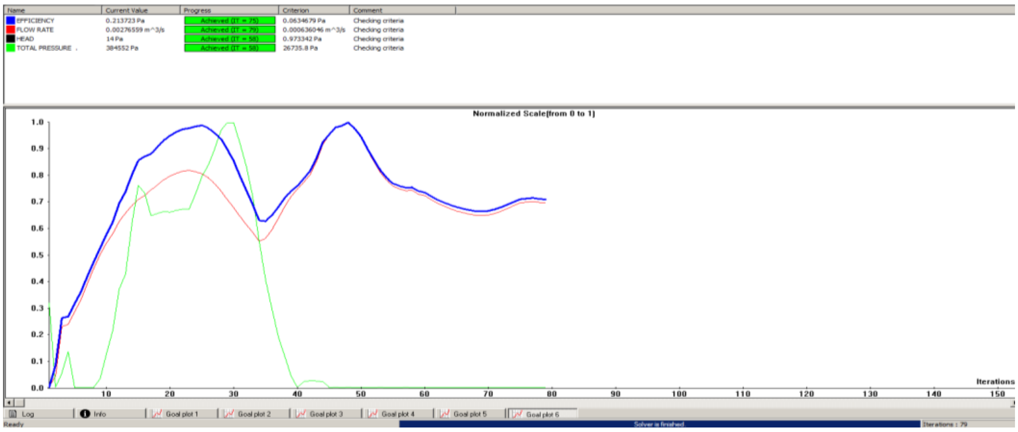


Fig 22 Efficiency graph for impeller C

4.2 Experimental study

CFD results are analysed and verified by experimental study. Four impeller models which are developed by CFD are physically designed and the impellers are tested in the experimental setup. From the literature it is found that the maximum total head possible for impellers is 25Pa. So it is consider to be the maximum limit and flow in the impeller, power consumption in terms of kW, suspension flow and efficiency are noted. Then the total head is reduced step by step and absolute change in efficiency readings are noted. For that readings, the flow rate, power consumption, suspension flow are also observed and noted.
The tabulated values for the four impeller are given below:

Table 4.1 Experimental value of standard impeller

TOTAL HEAD	Q (lpm)	Q (m^3/H)	I	kW	efficiency
11.95	169.8	611.28	7.35	1.6	20.70
13.07	163.5	588.6	7.35	1.6	21.80
14.84	150.78	542.808	7.3	1.58	23.10
16.55	133.68	481.248	7.2	1.54	23.48
17.39	123.06	433.016	7.05	1.52	23.01
18.26	113.4	408.24	7	1.52	22.26
19.97	88.92	320.112	6.75	1.44	20.14
21.69	56.52	203.472	6.4	1.36	14.75
24.50	0	0	5.9	1.2	0.00

Table 4.2 Experimental value of impeller A

TOTAL HEAD	Q (lpm)	Q (m^3/H)	I	kW	efficiency
14.90	188.76	679.536	7.4	1.72	26.71
17.22	170.76	614.736	7.2	1.64	29.29
18.86	151.86	546.696	7	1.6	29.23
20.51	130.86	471.096	6.9	1.56	28.11
22.18	107.28	386.208	6.7	1.5	25.92
24.69	57	205.2	6.2	1.36	16.91
25.50	0	0	5.9	1.24	0.00

Table 4.3 Experimental value of impeller B

TOTAL HEAD	Q (lpm)	Q (m^3/H)	I	kW	efficiency
14.60	189	680.4	7.3	1.58	28.54
15.32	175.8	632.88	7.2	1.56	28.21
16.84	151.2	544.32	7	1.52	27.38
17.60	137.04	493.344	6.9	1.48	26.64
18.40	123.48	444.528	6.75	1.44	25.78
19.99	91.56	329.616	6.5	1.4	21.37
23.00	25.2	90.72	65.9	1.2	7.39
24.50	0	0	5.75	1.16	0.00

Table 4.4 Experimental value of impeller C

TOTAL HEAD	Q (lpm)	Q (m^3/H)	I	kW	efficiency
17.54	207.96	748.656	8	1.8	33.12
18.57	187.8	676.08	7.8	1.74	33.76
19.27	173.4	624.24	7.7	1.72	31.34
20.01	160.2	576.92	7.5	1.68	31.18
20.77	147	529.2	7.4	1.64	30.42
21.52	132	475.2	7.3	1.6	29.02
22.29	115.8	416.88	7.1	1.56	27.03
23.08	98.88	355.968	6.9	1.52	24.53
23.80	71.1	255.96	6.7	1.44	19.20
24.71	0	0	6.1	1.34	0.00

From the readings graphs were drawn between total head and the efficiency for the impellers

Fig 4.1 Total head versus Efficiency graph for Standard impeller

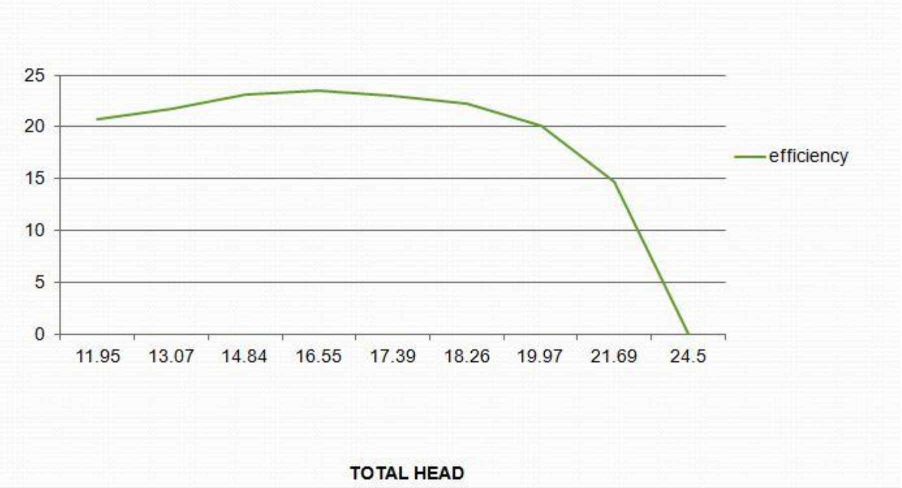


Fig 4.2 Total head versus Efficiency graph for impeller A

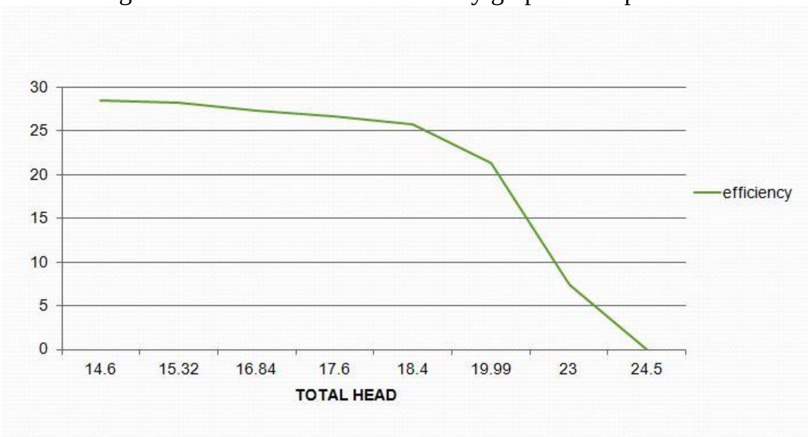
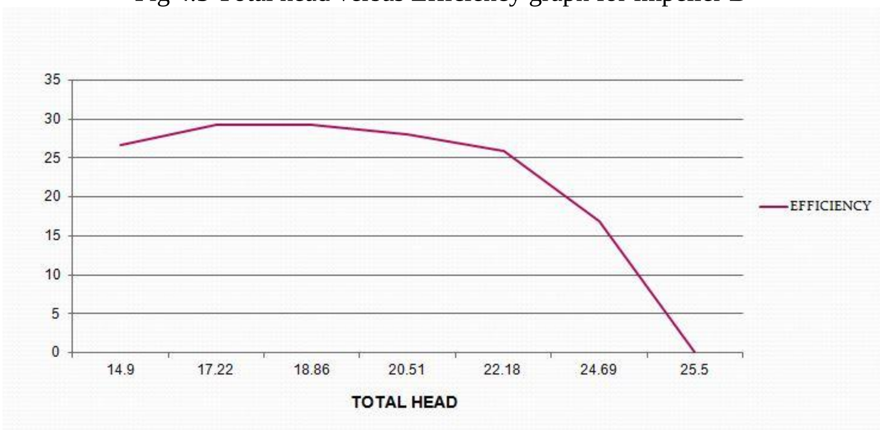


Fig 4.3 Total head versus Efficiency graph for impeller B



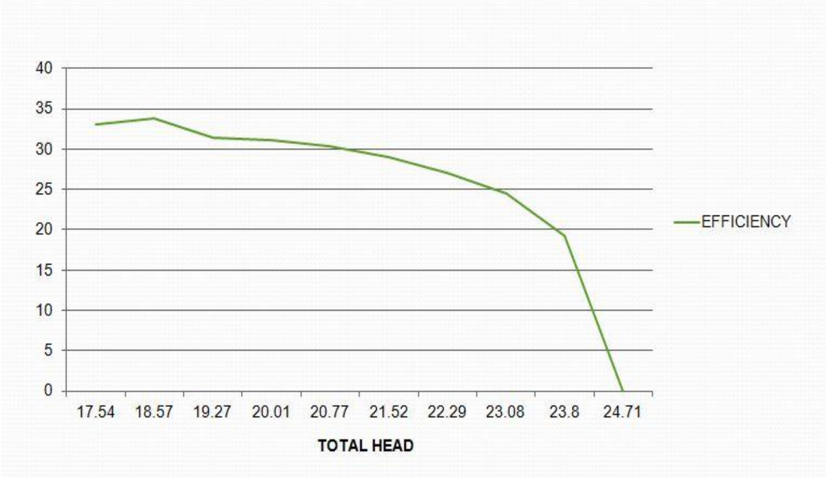


Fig 4.4 Total head versus Efficiency graph for impeller C

4.3 COMPARITIVE STUDY

Two types of comparison are done here. First comparison is with the result of impellers obtained within the CFD and experimental data. The total head and efficiency of four impellers modelled with CFD are

compared and found which is effective design. Then the total head and efficiency of four impellers physically designed for experimental study compared and found which design is effective. From both comparison the modified impeller b is having effective performance and yield maximum efficiency.
The second comparison is to compare the CFD result with experimental result. It is to validate and verify CFD results. The tables below gives the efficiency obtain in CFD and experimental analysis.

Table 4.5 Impeller Performance in CFD analysis

Sl. NO	Impellers	Total head	Efficiency
1.	Standard Impeller	11.8831	20.71
2.	Impeller A	14.8185	26.557
3.	Impeller B	14.5326	28.95
4.	Impeller C	14.000	21.37

Table 4.6 Impeller Performance in experimental analysis

Sl. NO	Impellers	Total head	Efficiency
1.	Standard Impeller	11.95	20.70
2.	Impeller A	14.90	26.71
3.	Impeller B	14.60	28.54
4.	Impeller C	17.44	33.76

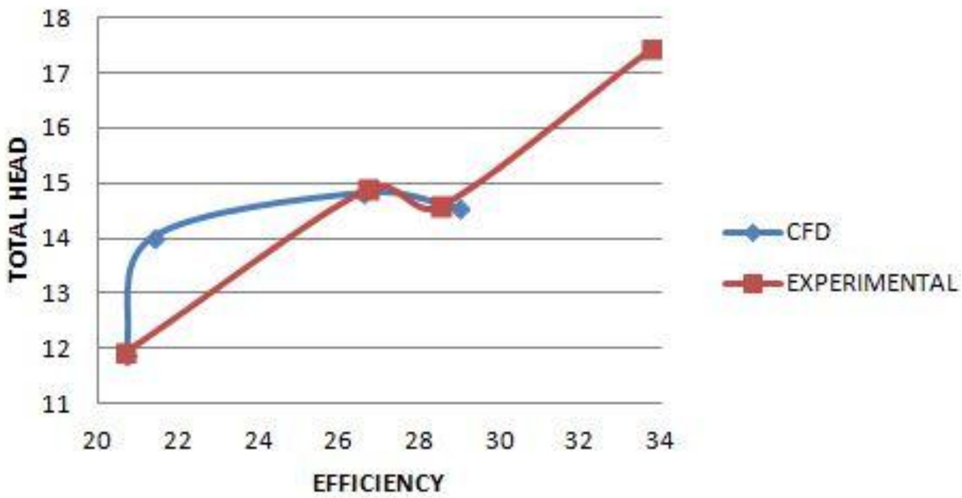


Fig. 4.5 Comparison for the performance of CFD and Experimental analysis.

From the result it was observed that Experimental values agreed with the CFD results. It was concluded from both the analysis that the efficiency of slurry pump fitted with optimised impeller B was increased from 20.71% to 28.95% compared to base line pump. This substantial rise in efficiency of the pump was due to change in vane number from 3 to 4 and impeller width from 4.5mm to 5mm.

V. SUMMARY AND CONCLUSION

It is summarised that Particle transport through pipes is an important operation in many industries including food, pharmaceutical, chemical, oil, mining, construction and power generation industries. In many of these applications the carrier fluid may be highly viscous and may have a Newtonian or non-Newtonian rheology and flow is usually turbulent. It has been a serious concern of researchers around the world to develop accurate models for increasing the performance of slurry pump over the years. The need and benefits of accurately predicting velocity profiles, concentration profiles and pressure drop of slurry pipelines during the design phase is enormous as it gives better selection of slurry pumps, optimization of power consumption and thereby helps maximize the economic benefit.

In this project, the capability of CFD was explored to model the impellers of slurry pump. It was found that the CFD is capable to successfully model the impeller and analyse the efficiency of impellers and the CFD results agreed with the experimental data.

This project is planned to improve the performance of slurry pump that is to increase its efficiency by optimising the impeller size. For that the impeller was modelled in solidworks 2012 software and CFD analysis was done using fluid flow simulation package. Four impeller models were developed for the critical design parameters of the pump. Inlet diameter, Outlet diameter, Inlet angle, Outlet angle, vane number, vane thickness and impeller width were the parameters which got varied to improve the efficiency of pump. CFD analysis was performed in the virtual models to determine the performance of pump. Then the CFD results were validated with real time experiments. CFD results agreed with experimental data. From the CFD results it was concluded that the efficiency of the slurry pump fitted with optimised impeller was increased from 20.71% to 28.95% of efficiency compared to base line pump. This substantial rise in efficiency of the pump was due to change in vane number from 3 to 4 and impeller width from 4.5mm to 5 mm. The computational model and results discussed in this work would be useful for extending the applications of CFD models for simulating large slurry pumps.

REFERENCE

[1]. Miedema, S.A., Lu, Z., Matousek, V., "Numerical Simulation of a Development of a Density Wave in a Long Slurry Pipeline“.23rd WEDA Technical Conference & 35th TAMU Dredging Seminar, Chicago, USA, june 2003.

[2]. Hofstra, C.F., & Rhee, C. van, & Miedema, S.A. & Talmon, A.M., "On The Particle Trajectories In Dredge Pump Impellers“. 14th International Conference Transport & Sedimentation Of Solid Particles. June 23-27 2008, St. Petersburg, Russia.

[3]. Miedema, S.A., & Riet, E.J. van, & Matousek, V.,"Theoretical Description And Numerical Sensitivity Analysis On Wilson Model For Hydraulic Transport Of Solids In Pipelines“. WEDA Journal of Dredging Engineering, March 2002.

[4]. Miedema, S.A., "Considerations on limits of dredging processes“.19th Annual Meeting & Technical Conference of the Western Dredging Association. Louisville Kentucky, May 16-18, 1999.

[5]. Graeme R. Addie, Robert J. Visintainer, Mihail C. Roco, Experiences with a numerical method of calculating slurry pump casing wear 1976,1980.

[6]. Hofstra, C. & Hemmen, A. van & Miedema, S.A. & Hulsteyn, J. van, "Describing the position of backhoe dredges".Texas A&M 32nd Annual Dredging Seminar. Warwick, Rhode Island, June 25-28, 2000

[7]. Asuaje M, Bakir F, Kouidri S, Rey R (2004). Inverse design method for centrifugal impellers and comparison with numerical simulation tools. International Journal of Computational Fluid Dynamics 18:101–110

[8]. Asuaje M, Bakir F, Kouidri S, Rey R (2004). Inverse design method for centrifugal impellers and comparison with numerical simulation tools. International Journal of Computational Fluid Dynamics 18:101–110

[9]. Chen SL, Wang WT (2001). Computer aided manufacturing technologies for centrifugal compressor impellers. Journal of Materials Processing Technology 115:284–293.

[10]. Dawes WN, Dhanasekaran PC, Kellar WP, Savill AM (2001). Reducing bottlenecks in the CAD-to-Mesh-to-Solution cycle time to allow CFD to participate in design. Journal of Turbo machinery 123:552–557.

[11]. B. K. Gandhi, S. N. Singh, V. Seshadri “Performance characteristics of centrifugal slurry pump”, ASME 2001

[12]. J. A. Sharpe “How to Specify the Operating conditions for a Slurry Pump” , Calgary Pump Symposium 2005.

[13]. Sandip Kumar Lahiri, “Study on slurry flow modeling in pipeline” NIT Durgapur Ph.D thesis, 2009

[14]. Lin BJ, Hung CI, Tang EJ (2002). An optimal design of axial-flow fan blades the machining method and an artificial neural network. Proceedings of the Institution of Mechanical Engineers. Part C, Journal of Mechanical Engineering Science 216:367–376.

[15]. Majidi K (2005). Numerical study of unsteady flow in a centrifugal pump.

Journal of Turbomachinery 127:363–371.
[16]. Medvitz RB, Kunz RF, Boger DA, Lindau JW, Yocum AM (2002). Performance analysis of cavitating flow in centrifugal pumps using multiphase CFD. Journal of Fluids Engineering124:377–383.
[17]. Morishige K, Takeuchi Y (1997). 5-axis control rough cutting of an impeller with efficiency and accuracy. Proceedings of the 1997 IEEE International Conference on Robotics and Automation 2:1241–1246.
[18]. Nerurkar AC, Dang TQ, Reddy ES, Reddy DR (1996). Design study of turbo-machinery blades by optimization and inverse techniques. AIAA-96-2555 (American Institute of Aeronautics and Astronautics, Lake Buena Vista, Florida).
[19]. Oh HW, Chung MK (1999). Optimum values of design variables versus specific speed for centrifugal pumps. Proceedings of the Institution of Mechanical Engineers. Part A, Journal of Power and Energy
[20]. Oyama A, Liou MS (2002). Multiobjective optimization of rocket engine pumps using evolutionary algorithm. Journal of Propulsion and Power 18:528–535.
[21]. Pak ET, Lee JC (1998). Performance and pressure distribution changes in a centrifugal pump under two-phase flow. Proceedings of the Institution of Mechanical Engineers. Part A, Journal of Power and Energy 212:165–171.
[22]. Pierret S, Van den Braembussche RA (1999). Turbomachinery blade design using a Navier- Stokes solver and artificial neural network. Journal of Turbomachinery 121:326–332.
[23]. Tanaka S, Isomura K, Togo SI, Esashi M (2004). Turbo test rig with hydroinertia air bearings for a palmtop gas turbine. Journal Micromechanics and Micro engineering 14:1449–1454.
[24]. Tsay DM, Lin BJ (1997). Design and machining of globoidal index cams. Journal of Manufacturing Science & Engineering 119:21–29.
[25]. Wang WP, Wang KK (1986). Geometry modeling of swept volume of moving solids. IEEE Computer Graphics and Applications 6:8–17.
[26]. Yedidiah S (1991). An alternate method for calculating the head developed by a centrifugal impeller. ASME/JSME Fluids Engineering Conference 107:131–138.