INTERNAL FLOW ANALYSIS OF SUBMERSIBLE PUMP IMPELLER USING CFD

Prithiviraj.N(1), Sasikumar.R(2)

[1]PG scholar,Mechanical Engineering,Gnanamani college of technology,Tamilandu, India
[2]PG scholar,Mechanical Engineering,Gnanamani college of technology,Tamilandu,India

ABSTRACT

Submersible pump has been playing an important role in industrial and house hold applications. A submersible pump (or electric submersible pump (ESP)) is a device which has a hermetically sealed motor close-coupled to the pump body. The whole assembly is sub merged in the fluid to be pumped. The submersible pumps used in ESP installations are multistage centrifugal pumps operating in a vertical position. The liquids, after being subjected to great centrifugal forces caused by the high rotational speed of the impeller, lose their kinetic energy in the diffuser where a conversion of kinetic to pressure energy takes place. And thus in this pump impeller plays an important role by which the efficiency has been calculated. In an impeller the design parameters such as number of blades, blade angles, diameter of the impeller, width of the blades are the important parameter to be considered because which affects the performance of the pump. And so here we made an analysis on the impeller by changing the outlet blade angle from the existing blade. The analysis is made by using Computational Fluid Dynamics (CFD) software by which the hydraulic efficiency has been calculated. The results are obtained from the Computational Fluid Dynamics (CFD) it has been calculated that by increasing the outlet blade angle by 5^0 the hydraulic efficiency of the impeller has been increased by 9.85% from existing impeller model which has the hydraulic efficiency of 73.5%. . It has been evident that by increasing the blade angle the hydraulic efficiency is increased. For each impeller, the flow pattern and the pressure distribution in the blade passages are calculated and finally the head-capacity curves are compared and discussed.

Keywords*:* *computational fluid dynamics, impeller, submersible pum*

---

1. CFD (COMPUTATIONAL FLUID DYNAMICS):

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions,The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations. Further simplification, by removing terms describing vortices yields the full potential equations. Finally, these equations can be linearized to yield the linearized potential equations.

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process.

The result of CFD analyses is relevant engineering data used in:

- Conceptual studies of new designs.
- Detailed product development.
- Troubleshooting.
- Redesign.

1.1. METHODOLOGY:

In all of these approaches the same basic procedure is followed.

- During preprocessing
- The geometry (physical bounds) of the problem is defined.
- The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non uniform.

- The physical modeling is defined.
- Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined.

1.1.1. MESH:

In CFD analysis meshing is the important term to be considered. In meshing at first the impeller has been split to different sub parts and the surface meshing is done. After that an volumetric is done by using different meshing software such as ANSA , T GRID etc. The geometry and the mesh of the computational pump domain were generated with Fluent's pre-processor, Gambit. Unstructured wedges are generated to define the inlet and outlet zones. An unstructured mesh with tetrahedral cells is also used for the zones of impeller. The mesh is refined in the near tongue region as well as in the regions close to the leading and trailing edge of the blades. Around the blades, structured hexahedral cells are generated. Though the size of the cells in the wall regions is not adequate to resolve the viscosity-affected region inside the boundary layer,the appropriate number of cells exists inside the boundary layer for the approach of standard wall functions. The latter provides correct values for the pump performance.

The partial differential equations that govern fluid flow and heat transfer are not usually amenable to analytical solutions, except for very simple cases. Therefore, in order to analyze fluid flows, flow domains are split into smaller subdomains (made up of geometric primitives like hexahedra and tetrahedra in 3D and quadrilaterals and triangles in 2D). The governing equations are then discretized and solved inside each of these subdomains. Typically, one of three methods is used to solve the approximate version of the system of equations: finite volumes, finite elements, or finite differences. Care must be taken to ensure proper continuity of solution across the common interfaces between two subdomains, so that the approximate solutions inside various portions can be put together to give a complete picture of fluid flow in the entire domain. The subdomains are often called elements or cells, and the collection of all elements or cells is called a mesh or grid. The origin of the term mesh (or grid) goes back to early days of CFD when most analyses were 2D in nature. For 2D analyses, a domain split into elements resembles a wire mesh, hence the name.

The most basic form of mesh classification is based upon the connectivity of the mesh: structured or unstructured.

- Structured mesh
- Unstructured mesh
- Hybrid mesh

Structured Meshes

A structured mesh is characterized by regular connectivity that can be expressed as a two or three dimensional array. This restricts the element choices to quadrilaterals in 2D or hexahedra in 3D. The above example mesh is a structured mesh, as we could store the mesh connectivity in a 40 by 12 array. The regularity of the connectivity allows us to conserve space since neighborhood relationships are defined by the storage arrangement. Additional classification can be made upon whether the mesh is conformal or not.

Unstructured Meshes

An unstructured mesh is characterized by irregular connectivity is not readily expressed as a two or three dimensional array in computer memory. This allows for any possible element that a solver might be able to use. Compared to structured meshes, the storage requirements for an unstructured mesh can be substantially larger since the neighborhood connectivity must be explicitly stored.

Hybrid Meshes

A hybrid mesh is a mesh that contains structured portions and unstructured portions. Note that this definition requires knowledge of how the mesh is stored (and used). There is disagreement as to the correct application of the terms "hybrid" and "mixed." The term "mixed" is usually applied to meshes that contain elements associated with structured meshes and elements associated with unstructured meshes (presumably stored in an unstructured fashion).

2. PROJECT DESCRIPTION

The Experimental results of an submersible pump impeller given by HARI INDUSTRIES, Coimbatore (submersible pumpset performance test report as per IS:8034-2002.The report has been given the below table 1 & 2

TABLE 1

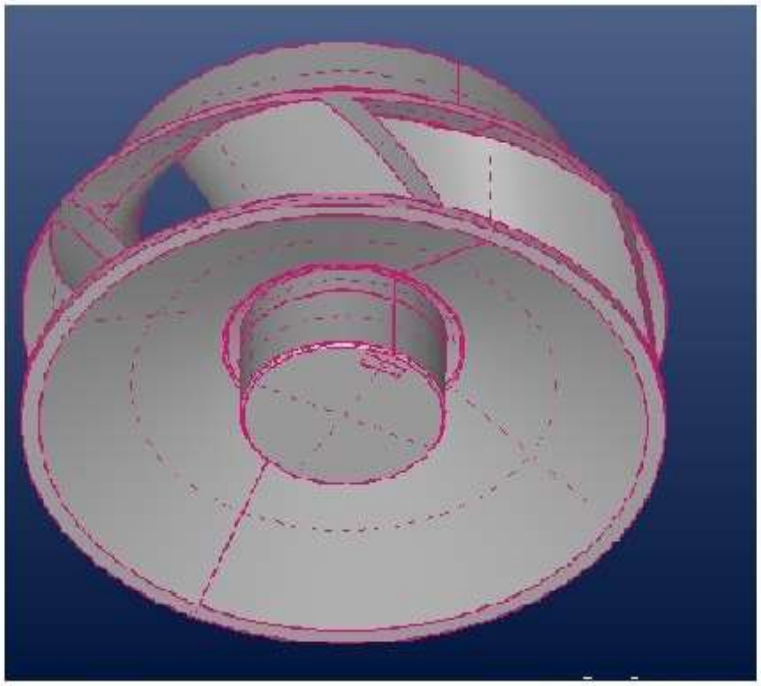
PUMP TYPE	ISM121	MOTOR RATING (Kw/ HP)	5.5/7.5	SPEED(rpm)	2880	TOTAL HEAD(m)	44
PUMP SL.NO	90309	MOTOR SL.NO	90309	FREQUENCY (Hz)	50	DISCHARGE (lps)	9
DELIVERY SIZE (mm)	65	VOLTAGE(v)	415	MOTOR TYPE	WET	EFFICIENCY(%)	44
MIN.BORE SIZE(mm)	150	PHASE	3	MOTOR CATEGORY	B	HEAD RANGE(m)	27/41.6
NO.OF STAGES	6	MAX.CURRENT(A)	14.5	MIN SUBMERGE (m)	1.5	CORRECTION HEAD(m)	2

TABLE 2

Sl.No	TOTAL HEAD			DISCHARGE(Q)		PERFORMANCE AT RATED FREQUENCY 50.0 Hz				
	DELIVERY HEAD	VOLUMETRIC HEAD	TOTAL HEAD	FLOW METER	DISCHARGE	DICHARGE (Q)	TOTAL HEAD	MOTOR INPUT	PUMP OUTPUT	OVER ALL EFFICIENCY
	M	M	M	Reading	Lps	Lps	m	Kw	Kw	%
1	61	0	63	0	0	0	65.33	5.449	0	0
2	50	0.03	52.03	2.72	2.72	2.77	53.95	6.04	1.465	24.26
3	40	0.18	42.18	6.23	6.23	8.34	43.74	6.167	2.719	44.09
4	30	0.29	32.29	7.87	7.87	8.01	33.48	5.914	2.629	44.45
5	20	0.4	22.4	9.26	9.26	9.43	22.23	5.808	2.148	36.98
6	10	0.5	12.5	10.37	10.37	10.56	12.96	5.386	1.342	24.92

2.1. IMPELLER MODEL AND SPECIFICATIONS:

The impeller model and their specification is given by the Hari Industries is given below:



2.1.1. IMPELLER PARAMETERS:

IMPELLER INLET(D_i)	75mm
IMPELLER OUTLET(D_O)	105mm
BLADE NUMBER	6
INLET BLADE ANGLE(β_i)	69^0
OUTLET BLADE ANGLE(β_O)	49^0
BLADE THICKNESS(t)	1.25mm
BLADE INLET HEIGHT(L_i)	21mm
BLADE OUTLET HEIGHT(L_O)	16mm

As for the existing impeller model, an design optimisation has been planned by changing their number of blades(n), Inlet blade angle(α) and Outlet blade angle (β) using CFD software in terms of Trial and Error method and kept the other specifications as to be constants.

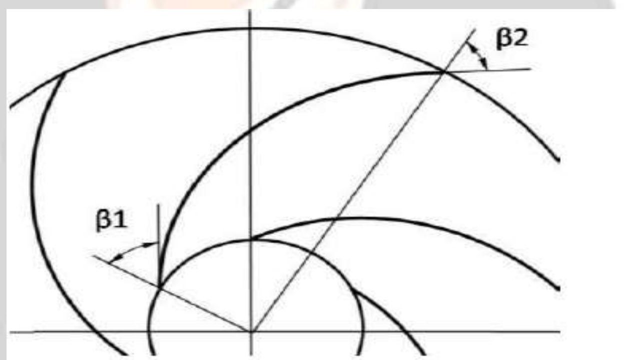
3. DESIGN OPTIMIZATION:

The design optimization has been made on the outlet blade angle as 44^0 and 49^0 to compare the hydraulic efficicency of the existing impeller having the outlet blade angle of 54^0.

Impeller design	Inlet blade angle(β_i)	Outlet blade angle(β_o)
Existing model	69^0	49^0
Impeller 1 (optimum)	69^0	44^0
Impeller 2 (optimum)	69^0	54^0

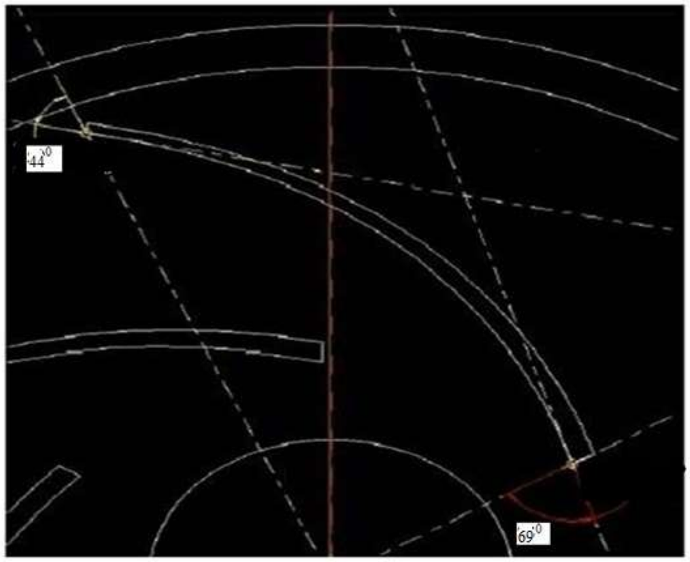
3.1. DESIGN OF IMPELLER MODELS:

Here the existing model has been obtained from the industries and so it will be easy to analysis. And also the outlet blade angle of the existing impeller is modified by using some design softwares such as AUTO CAD, PRO E, SOLID WORKS, UNIGRAPHICS & Blade Gen. Here we using Solid works software to change the outlet blade angle for the two impeller models. The below figure shows that discription of the inlet blade angle(β_1) & outlet blade angle(β_2) of the impeller.The energy usage in a pumping installation is determined by the flow required, the height lifted and the length and friction characteristics of the pipeline



3.1.1. IMPELLER MODEL 1:

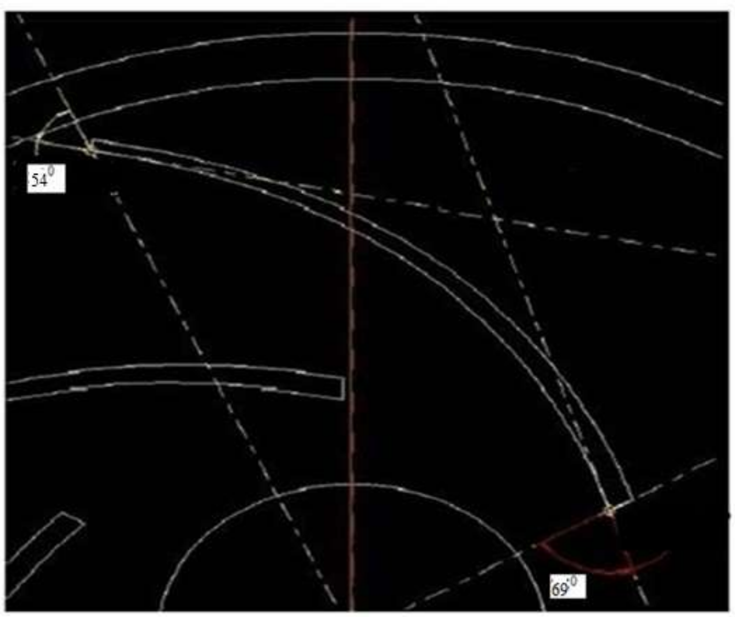
In an existing impeller, the outlet blade angle has been modified to 44^0 and kept the inlet angle of the impeller is same as that of the existing one which is 69^0. And the modified outlet blade angle for theimpeller model has been given below:



IMPELLER MODEL

3.1.2. IMPELLER MODEL 2:

In an existing impeller, the outlet blade angle has been modified to 54^0 and kept the inlet angle of the impeller is same as that of the existing one which is 69^0. And the modified outlet blade angle for theimpeller model has been given below:



IMPELLER MODEL 2

4. RESULT AND ANALYSIS:

For the existing impeller model and optmized model of the impeller an analysis is made to find the hydraulic efficiency by comparing the results the best design model has been found.Before that by using the data we have obtained from the pump industry we can find the hydraulic efficiency by using mathematical calculations which is given below:

hydraulic efficiency(η) = [Q*H / ω* T]
where

Q ⟶ discharge (m^3/s)
H ⟶ pressure head (m)
T ⟶ Torque (N-m)
ω ⟶ Angular velocity(rad/sec)

The angular velocity(ω) is calculated by using the formula,

ω = 2π N/ 60 (rad/sec)
= (2 π * 2880)/ 60

ω = 301.44 rad/sec.

Then the torque value is calculated by using the formulagiven below:

T= {Horse power(H.P)* 772}/ r.p.m
= {7.5 * 772}/2880

T=2.01 N-m

Then the hydraulic efficiency calculations be

hydraulic efficiency(η) = [Q*H / ω* T]

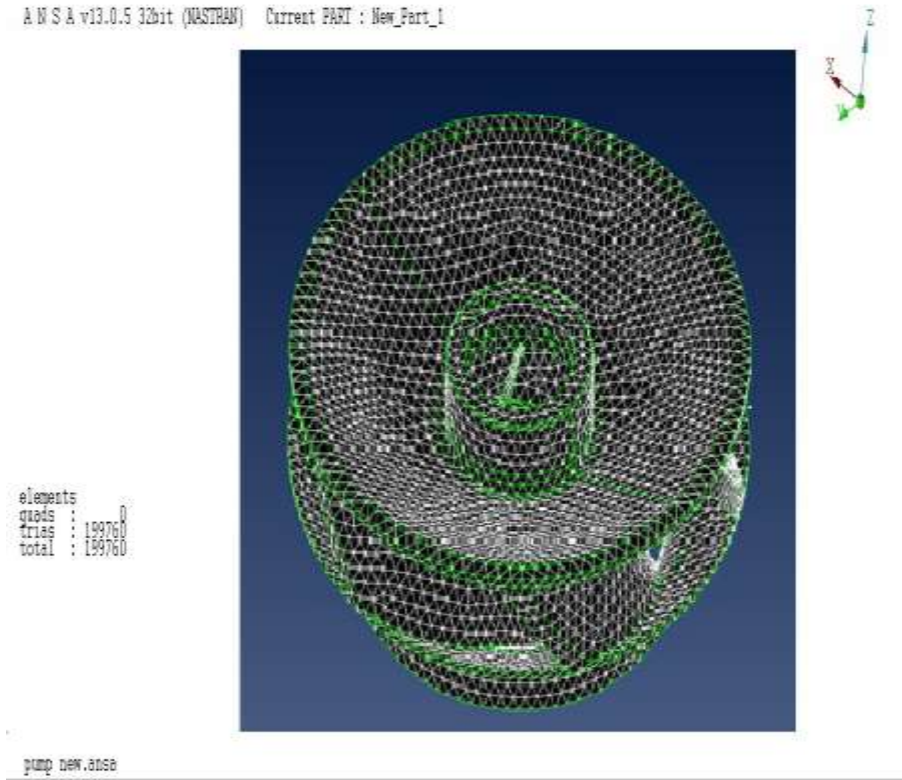
= [{9*44}/{301.2*2.01}]

=0.653 *100

hydraulic efficiency(η) = 65.3%

Meshing of an impeller:

Before the analysis, the meshing of the impeller is very important term to be considered. And the meshing is done by using ANSA(surface mesh) & T GRID software (Volume mesh). And while on surface meshing the impeller has been splited into four parts as inlet, outlet, blade outer and blade. After completing surface mesh on these parts the volume mesh on the impeller has been done. Then finally the analysis is made after complete completion of the meshing



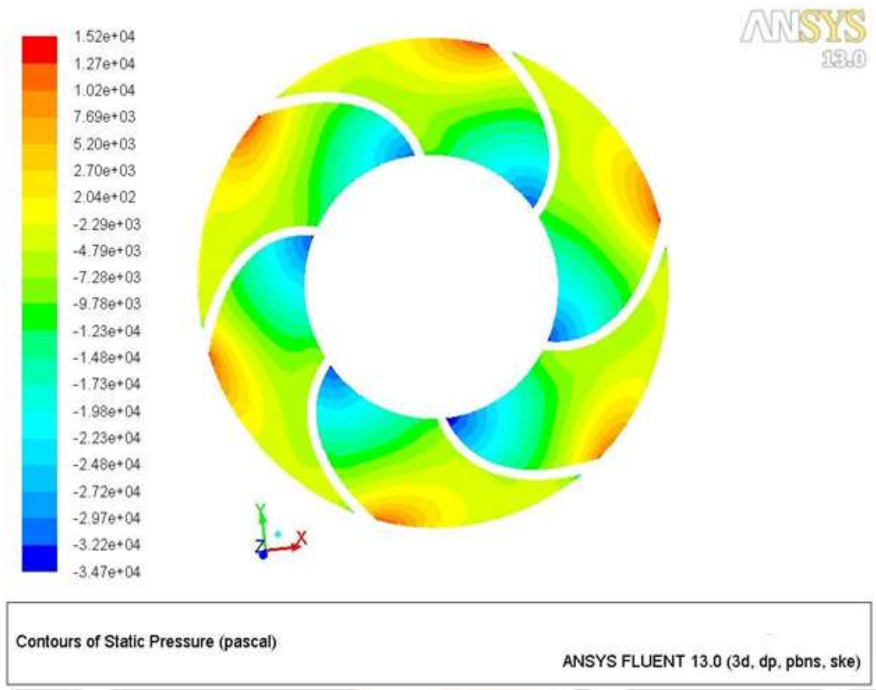
The above figure show the meshing of the existing impeller and it has a total mesh count of 199760 elements. As like this the two impeller model also have some mesh elements which is shown below:

MESH ELEMENTS:

Impeller models	Number of elements while mesh
Existing model (β_2) = 49^0	199760
Impeller model 1(β_2) =44^0	199710
Impeller model 2 (β_2) = 54^0	199770

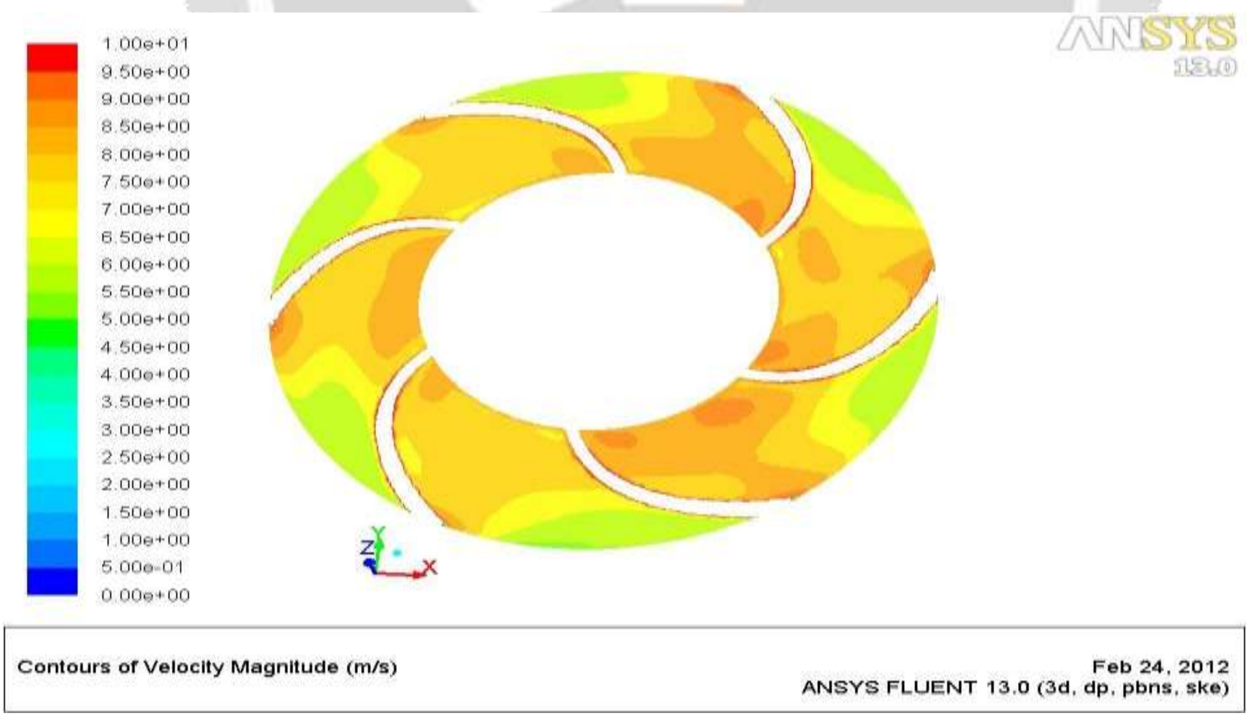
4.1. Analysis of the existing impeller:

4.1.1. Static pressure(Pascals):



The above analysis is the static pressure analysis of the existing impeller model and the results show in the above figure be in pascal. This figure shows that at the tip of the outlet bladeside the presssure be maximum when compare to the inlet blade side.

4.1.2. Relative velocity magnitude(m/s)



ANSYS FLUENT 13.0 (3d, dp, pbns, ske)

The above figure shows that the relative veloicity of the existing impeller achieved by Computational fluid dynamics(CFD) software. It shows that the maximum velocity of the fluid is 9.5m/s at the tip of the blade in an impeller.

4.1.3. Efficiency calculations:

We already know that the angular velocity(ω) of the impeller having the roatational speed (N) of 2880 r.p.m. is

$$\omega = 2\pi N/ 60 \text{ (rad/sec)}$$

$$= (2\,\pi * 2880)/ 60$$

$$\boxed{\boldsymbol{\omega = 301.44 \text{ rad/sec.}}}$$

We know that the hydraulic efficiency(η) of the impeller is

hydraulic efficiency(η) = [Q*H / ω* T]

=0.0098332679*45415.718/301.4*2880

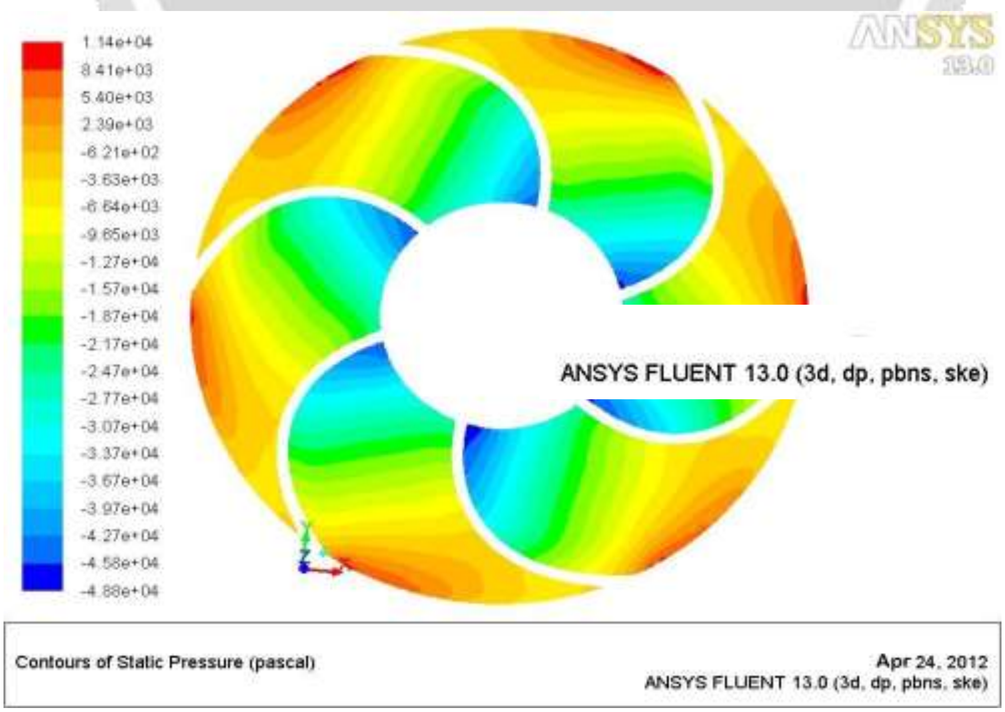
= 0.735 *100

$$\boxed{\textbf{hydraulic efficiency}(\boldsymbol{\eta}) \textbf{ = 73.5 \%.}}$$

5. ANALYSIS OF AN IMPELLER (MODEL 1):

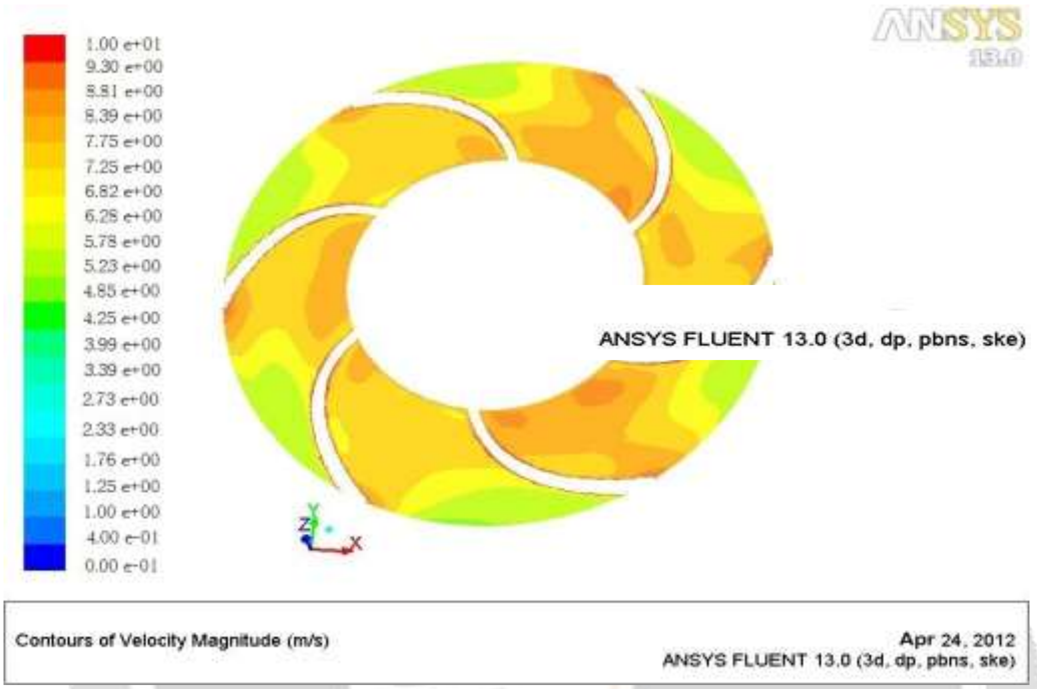
In the existing impeller model which has the outlet blade angle is changed to 44^0 using the solid works software and the analytical hydraulic efficiency is achieved by means of Computational Fluid Dynamics (CFD) analysis and it is 73.5% which is comparitively higher than the hydraulic efficiency of the same existing impeller calculated by means of mathematical model it as 65.3%.And now the first optimum model has been analytical evalued for hydraulic efficiency which is given below:

5.1. Static pressure (Pascals):

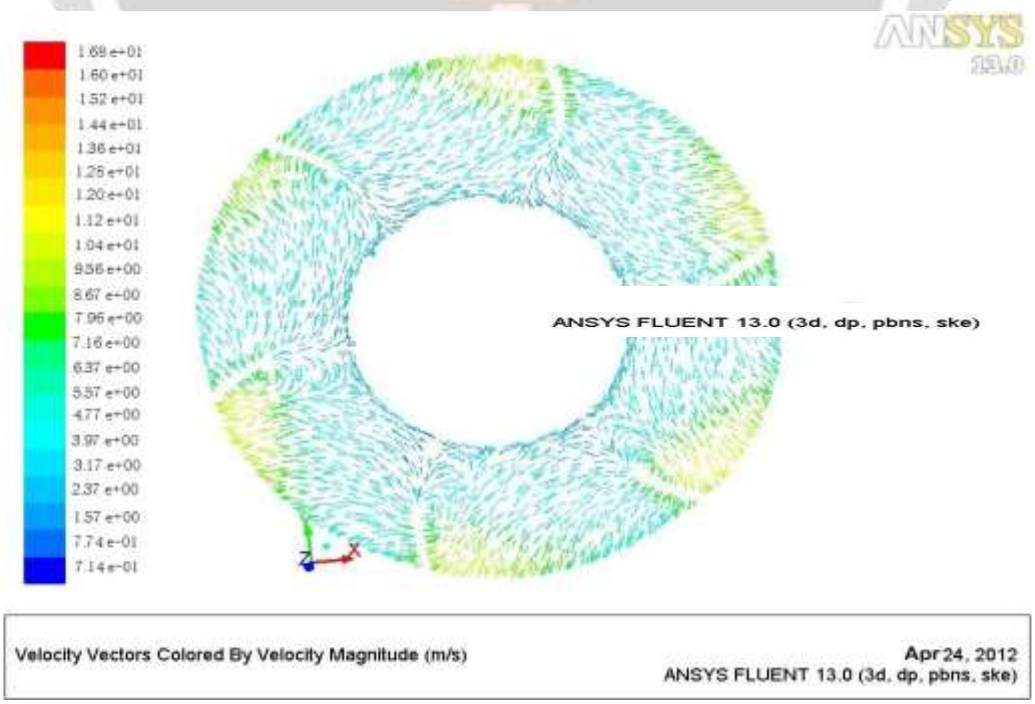


The above figure shows the static pressure results of an impeller which have the outletblade angle is changed to 44^0 from the existing model which is having the outlet blade angle of 49^0. And the maximum pressure is achieved the tip of outlet region which is shown in the figure.

5.2. Relative velocity magnitude:



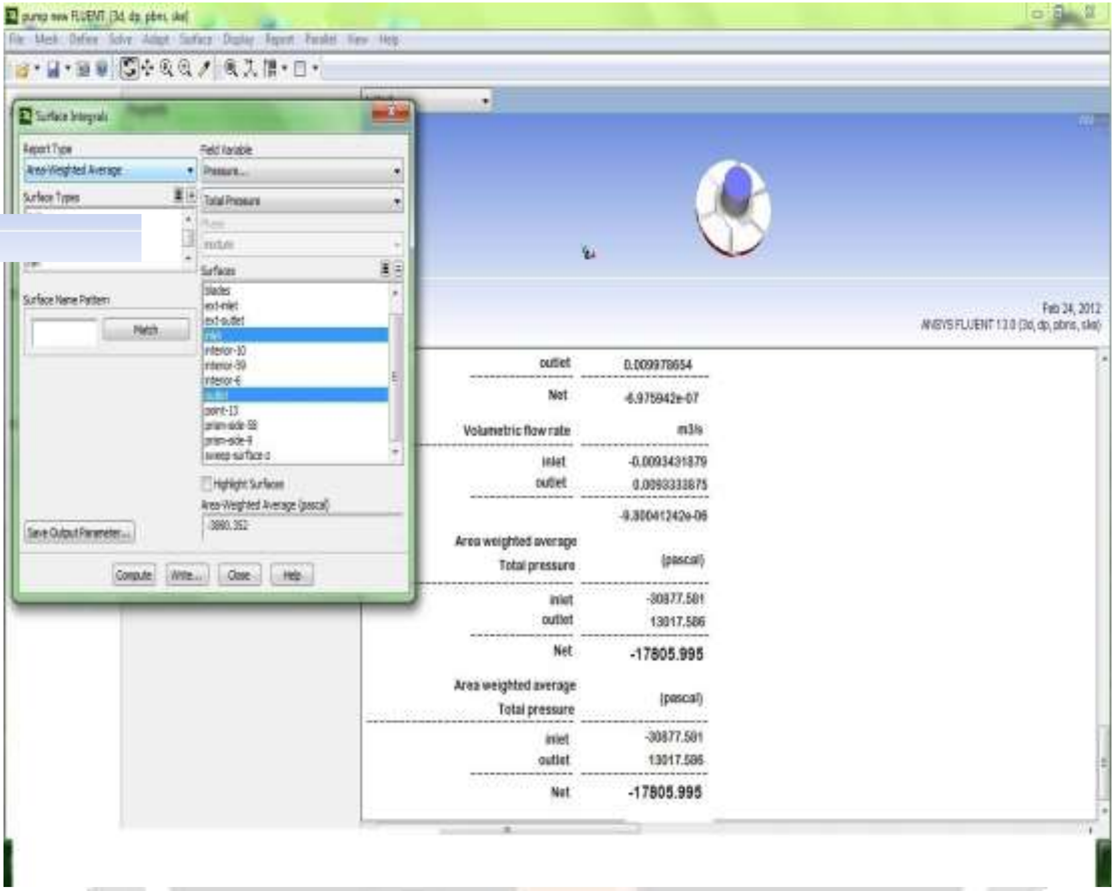
5.3. Velocity vector:



6. Post processing:

6.1. Head and discharge:

c



From the post processing results we can find the value of discharge(Q) and pressure headrange (H).

Discharge (Q) = 0.0093333875 m^3/s

Pressure head(H) = outlet pressure – inlet pressure (pascals) = 13017.586-(-30877.581)

= 43895.167 (pascals

6.2. TORQUE:

Here the torque value doesn't change it is same that of the existing impeller and also the rotational speed also doesn't change. And so the angular velocity is same as that of the existing impeller.

Torque(T) = 2.01N-m.

Angular velocity(ω) = 301.44 rad/sec.

6.3. HYDRAULIC EFFICIENCY CALCULATIONS:

hydraulic efficiency(η) = [Q*H / ω* T]

=0.0093333875 *43895.167 /301.4*2.01

= 0.6712 *100

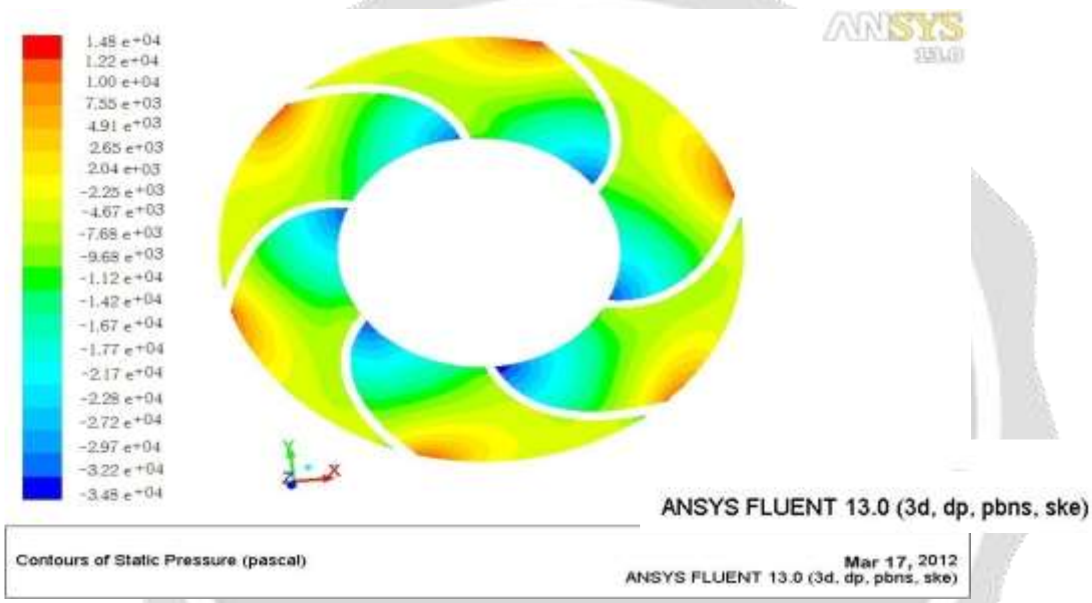
hydraulic efficiency(η_h) = 67.12 %.

This is the hydraulic efficiency of the model which has the outlet blade angle 44^0 instead of 49^0. And this hydraulic efficiency is comparitively low than the existing model which has the hydraulic efficiency of 73.5%.

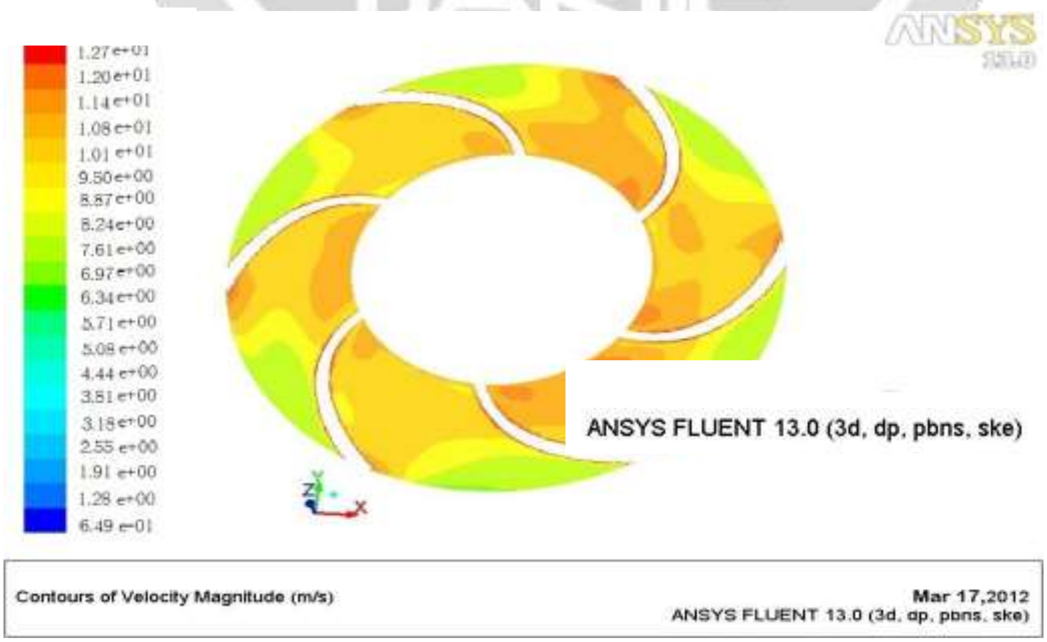
7. ANALYSIS OF THE IMPELLER(MODEL 2):

Before analysis the Model 2 it has been concluded from the above analysis while decreasing the blade angle the hydraulic efficiency of the impeller is comparatively less than the existing model in both analytically and mathematically.But for the optimized model 1 the discharge be slightly increased while compare to the existing model but head range will decreased heavily compared to existing model. After analysing the model 2 we have been concluded to the comparitive result. And the analytical results for the model 2 is given below:

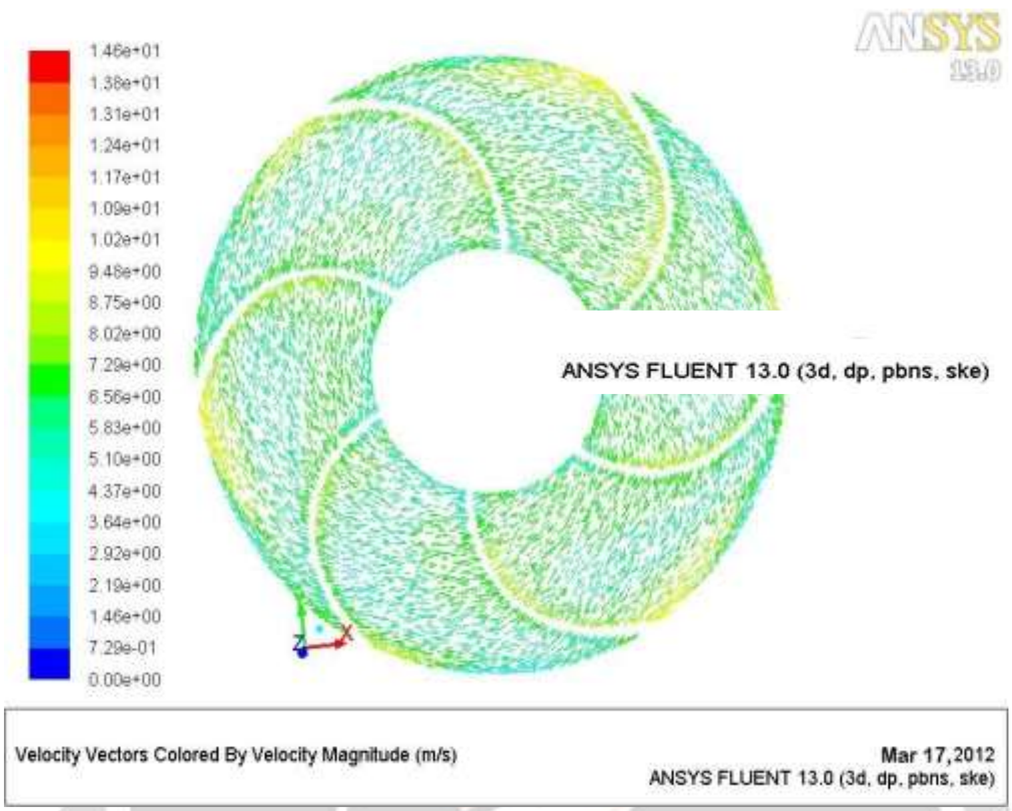
7.1. STATIC PRESSURE (Pascals):



7.2. Relative velocity magnitude(m/s):

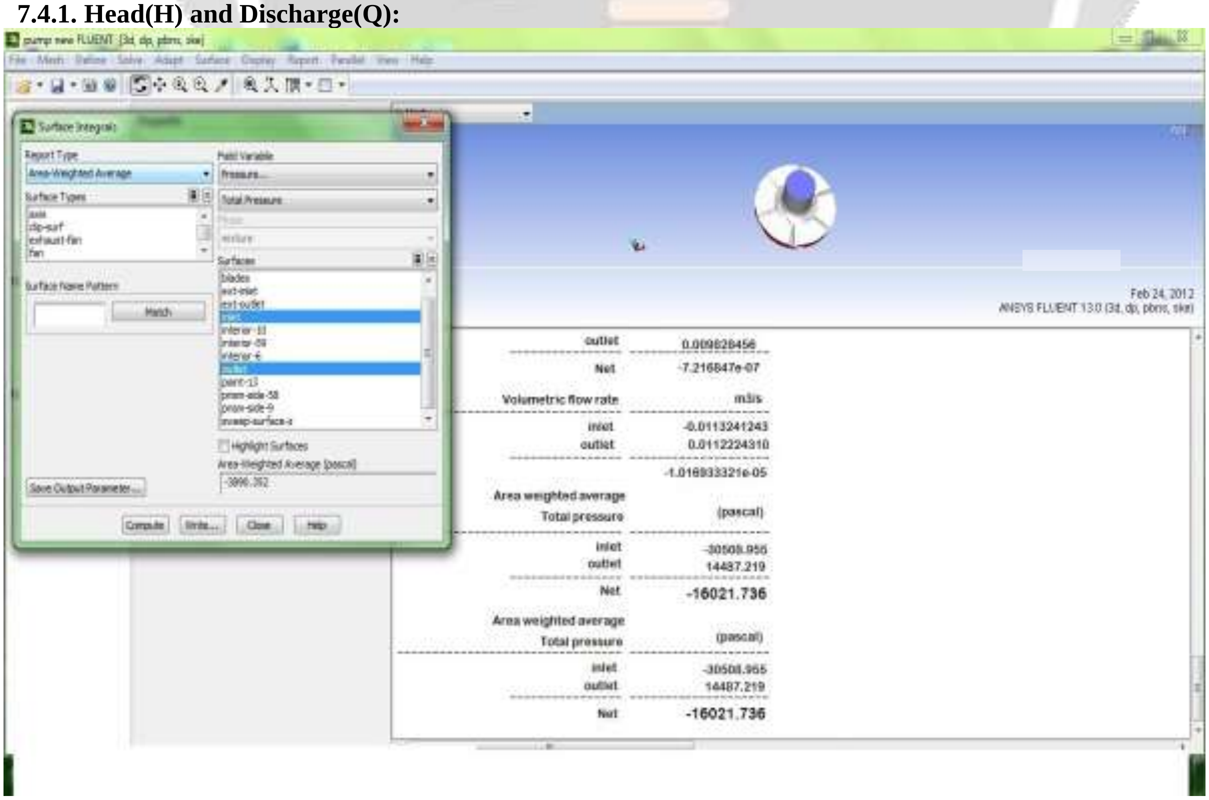


7.3. Vector velocity(m/s):



7.4. Post processing :

7.4.1. Head(H) and Discharge(Q):



From the post processing results we can find the value of discharge(Q) and pressure hearange (H).

Discharge (Q) = 0.0112224310 m^3/s

Pressure head(H) = outlet pressure – inlet pressure (pascals) = 14487.219-(-30508.955)

= 44996.174 (pascals)

7.4.3.TORQUE:

Here the torque value doesn't change it is same that of the existing impellerand also the rotational speed also doesn't change. And so the angular velocity is same as that of the existing impeller.

Torque(T) = 2.01N-m.

Angular velocity(ω) = 301.44 rad/sec.

Hydraulic efficiency calculations:

hydraulic efficiency(η) = [Q*H / ω* T]

=0.0112224310 *44996.174 /301.4*2.01

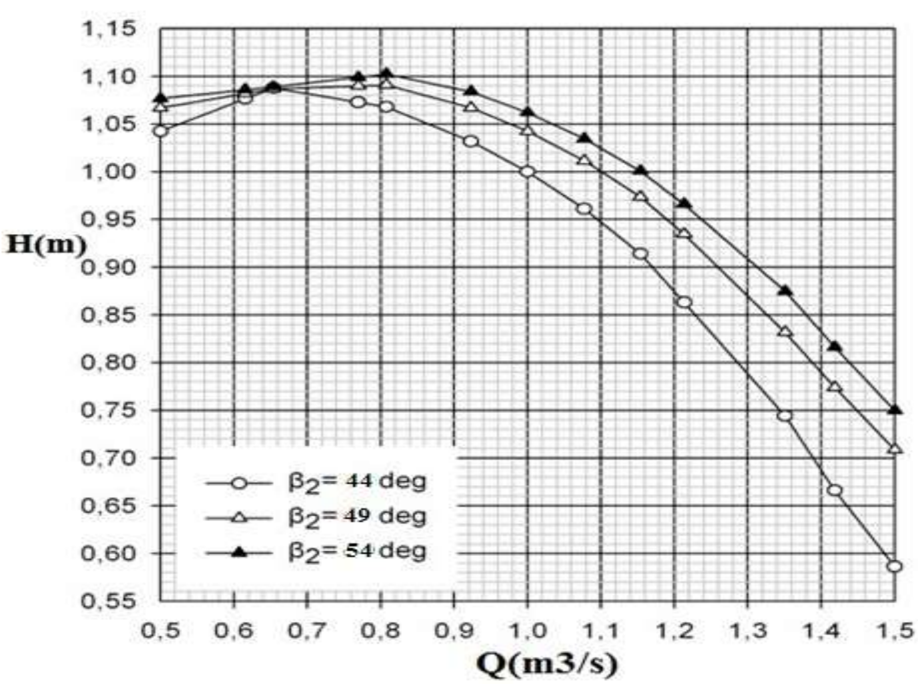
= 0.8035 *100

hydraulic efficiency(η_h) = 80.35 %.

In the impeller model 2 which is having the outlet blade angle 54^0 the hydraulicefficiency is obtained by means of Computational Fluid Dynamics (CFD) software is 83.5%. This hydraulic efficiency is higher than existing model which is having the outlet angle is 49^0 and the impeller model 1 which is having the outlet blade angle is 44^0.

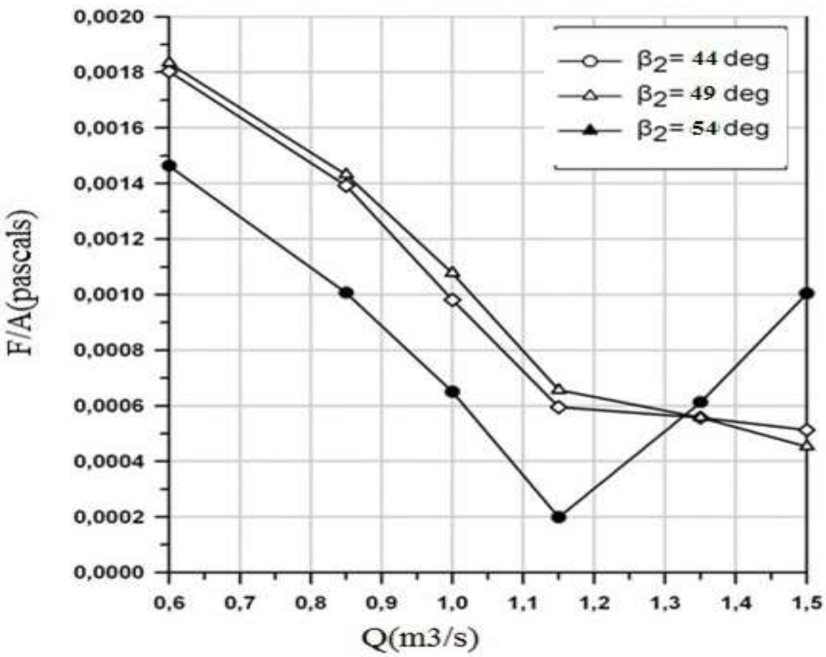
7.2. RESULT AND DISCUSSIONS:

7.2.a. HEAD AND DISCHARGE CURVE:



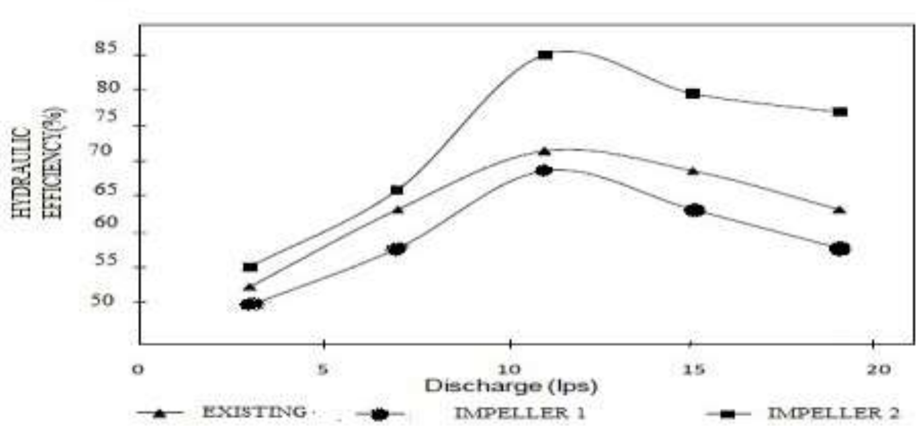
In this graph we can find that at 54^0 outlet blade angle the discharge and head will bemaximum when compared to the outlet blade angle 44^0 and 49^0. And hence it is conclude that at 54^0 angle the head and discharge will be higher than the other angles 44^0 and 49^0.

7.2.b.PRESSURE AND DISCHARGE CURVE:



The above figure represents the curve between pressure(P) and discharge(Q). It shows that at greater the discharge the pressure will be very high at 54^0 angle compared to the the 44^0 and 49^0 angles.But here our objective is to increase the discharge and so no matter the pressure should be considered.

7.2.c HYDRAULIC EFFICIENCY AND DISCHARGE:



The above figure shows that the graph between hydraulic efficiency and discharge. Themaximum efficiency point is given by the impeller model 2 which is having the outlet blade angle 54^0.

8. RESULT COMPARISON:

S.No	JOURNAL	IMPELLER OPTIMUM (MIXED FLOW TYPE)			HYDRAULIC EFFICIENCY (η_h)
2)	**Internal flow analysis of a submersible pump impeller using CFD**	**IMPELLER OPTIMUM (MIXED FLOW TYPE)**			**HYDRAULIC EFFICIENCY (η_h)**
		MODELS	**β_1**	**β_2**	**%**
		EXISTING	69	49	73.5
		MODEL 1	69	44	65.3
		MODEL 2	69	54	83.3

9. CONCLUSION

Here the impeller model of a submersible pump has been analysed mathematically and analytically. In mathematical analysis the existing impeller model have an hydraulic efficiency of 65.3% and furthur the same model has been analysed made by using Computational Fluid Dynamics (CFD) software then it has been obtained that the analytical hydraulic efficiency is 73.5%. From the above results it has been conclude that analytical results of the existing impeller is higher than the mathematical results. And further the same existing impeller has been optimum at the outlet blade angle region which is decreased to 5^0 and increased to 5^0 angle. The existing impeller has an outlet blade angle of 49^0 and the modified angle is 44^0 for model 1 and 54^0 for model 2. For these two model an analysis is done by using same Computational Fluid Dynamics (CFD) software and it shows the results of hydraulic efficiency as 67.12% for model 1 and 83.35% for model 2. It has been seen that by increasing the blade angle the hydraulic efficiency of the impeller gets incresed but the pressure head range of the impeller is decreased when compared to the existing model and optimum model 1. Thus by increasing the outlet blade angle the hydraulic efficiency.

10. REFERENCES:

1) **A. Manivannan, “Computational fluid dynamics analysis of a mixed flow pump impeller”** International journel of science and technology Vol. 2, No. 6, 2010, pp. 200-206

2) **Khalid. S. Rababa,** The Effect of Blades Number and Shape on the Operating Characteristics of Groundwater Centrifugal Pumps . European Journal of Scientific Research ISSN 1450-216X Vol.52 No.2 (2011), pp.243-251.

3) **E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris,** Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle. The Open Mechanical Engineering Journal, 2008, 2, 75-83.

4) **LIU Houlin*, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG** Kai, Effects of Blade Number on Characteristics of Centrifugal Pumps, CHINESE JOURNAL OF MECHANICAL ENGINEERING Vol. 23,aNo.*,a2011

5) Christopher Earls Brennen “**HYDRODYNAMICS OF PUMPS**” by OPEN © Concepts NREC 1994

6) Yunus A.Cengel and John M.Cimbala “ **Fluid Mechanics** ”

7) D. Eckardt, “**Detailed flow investigations within a high-speed centrifugal compressor impeller**”, ASME Journal of Fluids Engineering, vol. 98, pp. 390-402, 1976.

8) M. W Johnson, and J. Moore, **“The development of wake flow in a centrifugal impeller”,** ASME Journal of Engineering for Power, vol. 102, pp. 382-390, 1980.