CFD SIMULATION OF BOILER FEED PUMP BY VARYING NUMBER OF BLADES

R.Surendra Rao [1]| N.Srikanth [2] | K.Chandrakanth [3]| B.Sai Kiran[4]

[1,2&3] Assistant Professor, Mechanical department, Kasireddy Narayanreddy College of Engineering And Research, Hyderabad, TS.

[4] UG SCHOLARS, Mechanical department, Kasireddy Narayanreddy College Of Engineering And Research, Hyderabad, TS.

Abstract: Feed pumps are an essential subsystem of boilers used in industrial process plants and called as boiler feed pump (BFP). Normally, BFP is high pressure unit that takes suction from condensate return system and can be of the centrifugal type pump. In centrifugal pump, water enters axially through the impeller eyes and exits radially. Generally, electric motor is used as prime mover to run the feed pump. To force water into boiler, the pump must generate sufficient pressure to overcome steam pressure developed by boiler. In this paper, design and analysis of boiler feed pump having a flow of 70kg/s, rotational velocity of 1800 rpm and operating at 60 ^{0} C has been taken up. The various pump parameters are obtained from design and pump model is developed using modeling software CATIA to evaluate the results at given operating conditions, CFD analysis is carried out using ANSYS fluent. Blade number has great influence on the pump performance. Therefore, CFD analyses are carried out for the pump with 6, 8 and 10 blades. Based on performance of every pump model, the best feed pump design is selected. With suitable graphs and tables.

1. INTRODUCTION

Pumps are energy-consuming devices being used in many industrial applications and in particular in dewatering, handling water & other fluids and in agricultural, the effectiveness improvement of siphons has huge importance to vitality protection. The whole task work center is manage the exhibition and effectiveness of siphon which straightforwardly spare force. In this task we plan a multi-stage outward siphon which orders to different viewpoints require to US heater feed water siphon advertise. Feed siphons are a basic subsystem of boilers utilized in mechanical procedure plants and called as kettle feed siphon (BFP). Regularly, BFP is high weight unit that takes pull from condensate return framework and can be of the outward kind siphon. In divergent siphon, water enters pivotally through the impeller eyes and exits radially. By and large, electric engine is utilized as central player to run the feed siphon. To constrain water into evaporator, the siphon must create adequate strain to conquer steam pressure created by kettle. There is no thorough technique to be followed in structuring a siphon. Part of approaches has been created and, albeit each has a marginally unique technique for figuring, the expansive basic standards of all are comparable. The speed constraints and extents are likewise there to which it requires to follow; yet these might be surpassed in specific occurrences to meet rivalry with respect to cost or execution. The standard plan depends on a specific wanted head and limit at which the siphon is worked a large portion of time. In structure of diffusive siphon, the parts to be planned are: shaft, impeller, vane, packaging, and choice of bearing. To structure these parts various approachs can be gotten through writing overview. From the given conditions, the particular speed is acquired .According to required head, the stream rate and from explicit speed, siphon of twofold volute, pairs attractions and single stage type is chosen. The base shaft width can be gotten by utilizing most extreme shear pressure hypothesis. Impeller and vane are planned by procedure gave by Church. To plan the vane observational relations are utilized. Programming interface standard is utilized to structure the volute and for bearing choice.

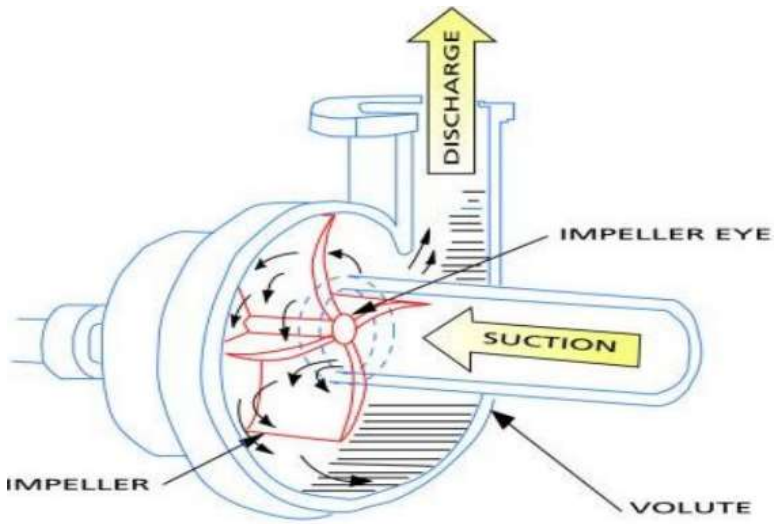


Fig 1: Boiler feed pump working

2. LITERATURE REVIEW

In the current investigation we plan and examination of kettle feed siphon having a progression of 138 m3/hr under a head of 632 m at 3550 RPM and working temperature go is 200 15 degree Celsius has been taken up. In these structure critical assignment is to set up a high head with in four phases. The different siphon boundary acquired from configuration is created utilizing 3D displaying programming Pro-E, and investigation is done by utilizing Ansys , CFD programming module.[1]design and examination of evaporator feed siphon having a progression of 2000 m3/hr, head of 470 m and working at 130±10o C has been taken up. The different siphon boundaries are acquired from plan and siphon model is created utilizing displaying programming Creo Parametric. To assess the outcomes at given working conditions, CFD investigation is completed utilizing Ansys CFX module. Cutting edge number has extraordinary impact on the siphon execution. In this way, CFD investigations are done for the siphon with 5, 6 and 7 cutting edges. In light of execution of each siphon model, the best feed siphon configuration is chosen. A consistent state CFD examination is done utilizing the K-ε disturbance model to tackle for the Navier-Stroke's condition. [2]

3. METHODOLOGY

- Study the literature review.
- Create the 3D model of BFP with help of CATIA parametric software.
- Selection of Blade geometry such as 6,8 and 10 blades.
- Perform CFD and thermal analysis on the BFP assembly for thermal loads.

PROBLEM DESCRIPTION:

The objective of this project is to make a 3D model of the evaporator and study the CFD behavior of the BFP by performing the finite element analysis.3D modeling software CATIA was used for designing and analysis software (ANSYS) was used for CFD and thermal analysis.

4. MODELING AND ANALYSIS

Computer Aided Design (CAD) is the utilization of program to structure an item or an article. PC Aided Manufacturing (CAM) is the utilization of program and equipment to design, oversee and control the activities of an assembling plant. PC Aided Engineering is the utilization of program to take care of designing issues and dissect items made utilizing CAD.CATIA is an abbreviation for Computer Aided Three-dimensional Interactive Application. It is one of the main 3D programming utilized by associations in different enterprises running from aviation, vehicle to purchaser items. CATIA is a multi stage 3D programming suite created by Dassault Systems, incorporating CAD, CAM just as CAE.

ANSYS

The overseeing conditions in ANSYS Forte follow mostly the Continuity condition, Momentum condition (Navier Stokes condition) and Energy condition to take care of computational liquid elements issue.

The protection condition for species is given by :

$\partial\rho/\partial t\ k+\nabla\cdot\rho ku=\nabla\cdot\rho DT\nabla\rho k\rho+\rho kc+\rho ksk=1,\cdots,K$ (1)

Where: ρ is the thickness, addendum k is the species record, K is the all out number of species, u is the stream speed vector. The summation of Equation 1 over all species gives the progression condition for the all out liquid

$\partial\rho/\partial t+\nabla\cdot(\rho\cdot u)=\rho$ (2)

Computational Fluid Dynamics, by and large dense as CFD, are a piece of fluid mechanics that uses numerical procedures and counts to deal with and dismember issues that incorporate fluid streams. PCs are used to play out the tallies required to reenact the association of liquids and gases with surfaces described by limit conditions. With fast supercomputers, better courses of action can be practiced. Persistent examination yields programming that improves the precision and speed of complex reenactment circumstances, for instance, transonic or rough streams. Beginning test endorsement of such writing computer programs is performed using an air stream with the last endorsement coming in full-scale testing, for instance flight tests.

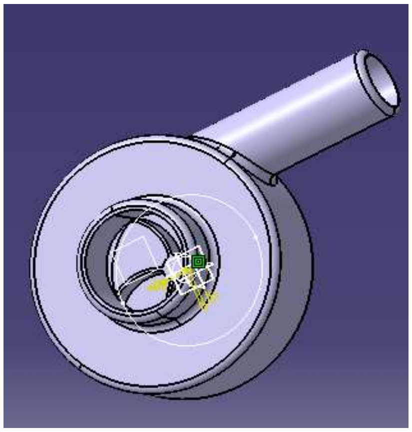


Fig 2: 3D model of feed pump

Table: 1 Design Parameters

Parameter	Value
Shaft diameter	76mm
Hub diameter	114 mm
Impeller diameter	445 mm
Inlet vane angle	11^0
Number of blades	6,8 and 10

Table: 2 fluid properties

Specification	Value
Density	998.2 kg/m^3
Specific heat	4182 j/kg k
Thermal conductivity	0.6 w/m- k
Viscosity	0.00103 kg/m-s

5. RESULTS AND DISCUSSION

CFD Analysis of boiler feed pump

Case 1-6 blades

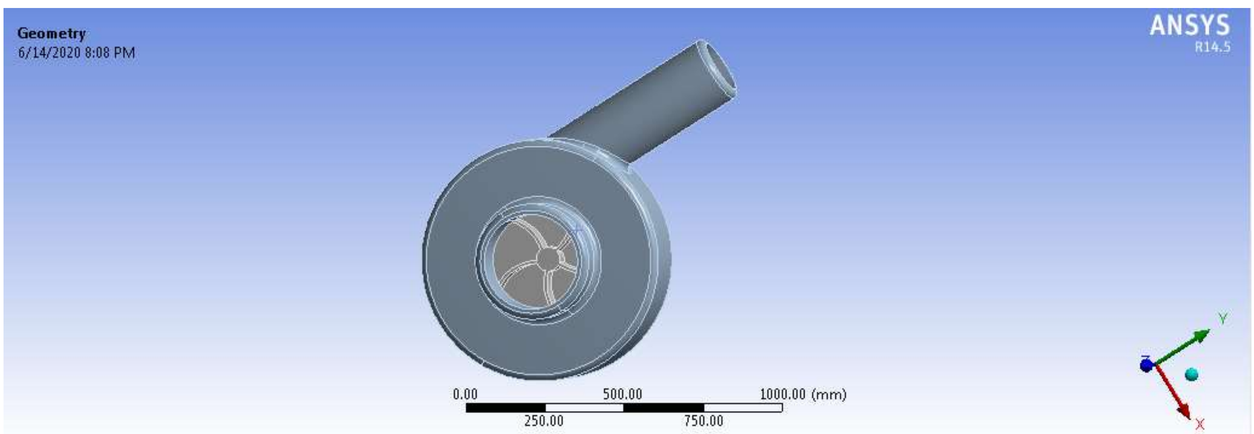


Fig 3: imported model

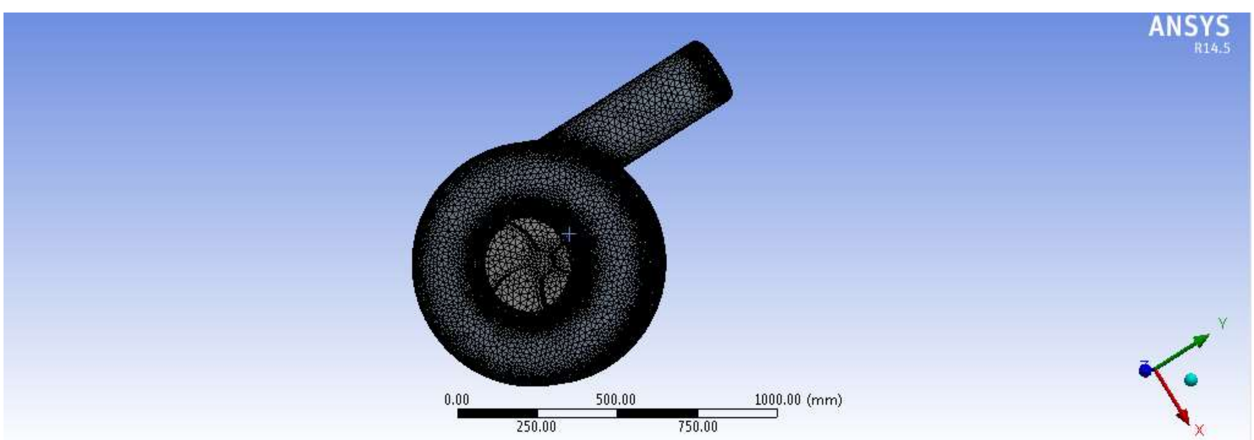


Fig 4: meshed model

The model is designed with the help of CATIA and then import on ANSYS for Meshing and analysis. The analysis by CFD approach is used in order to calculating pressure profile and temperature distribution. For meshing, the fluid ring is divided into two connected volumes. Then all thickness edges are meshed with 360 intervals. A tetrahedral structure mesh is used. So the total number of nodes and elements is 58635 and 262194.

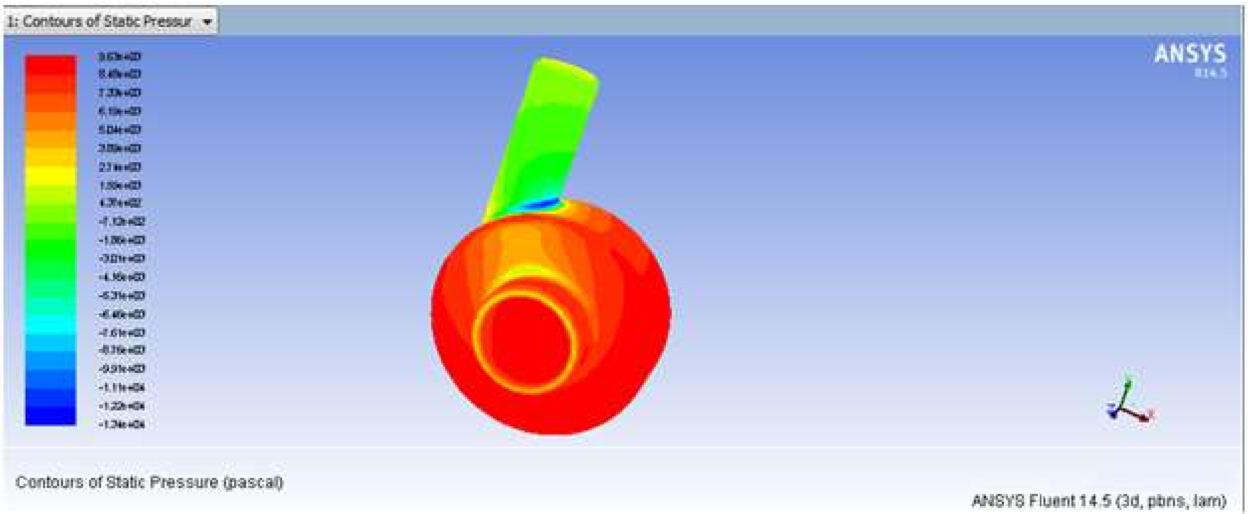


Fig 5: pressure counters at 6 blades

According to the above plot the maximum pressure distribution at inside of the casing and minimum pressure at outlet of the pump. The maximum pressure is 9.63e+03 Pa and minimum pressure is 4.32e+03 Pa

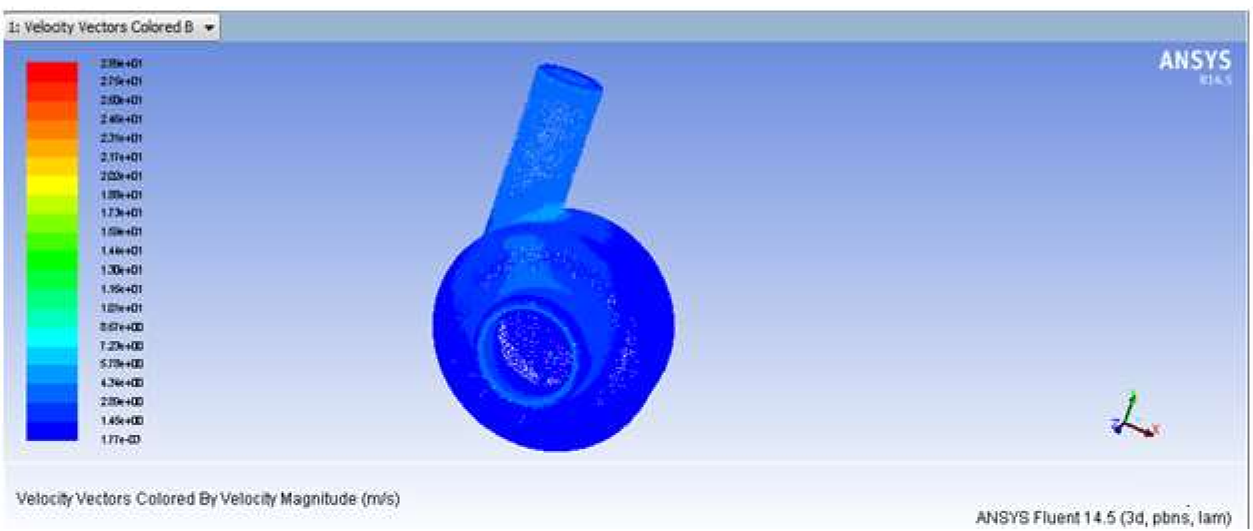


Fig: velocity counters at 6blades

According to the above plot the maximum velocity at outside of the pump and minimum velocity at inlet of the pump. The maximum velocity is 28.9 m/s and minimum velocity is 1.71e+00 m/s

Case 2: 8 blades

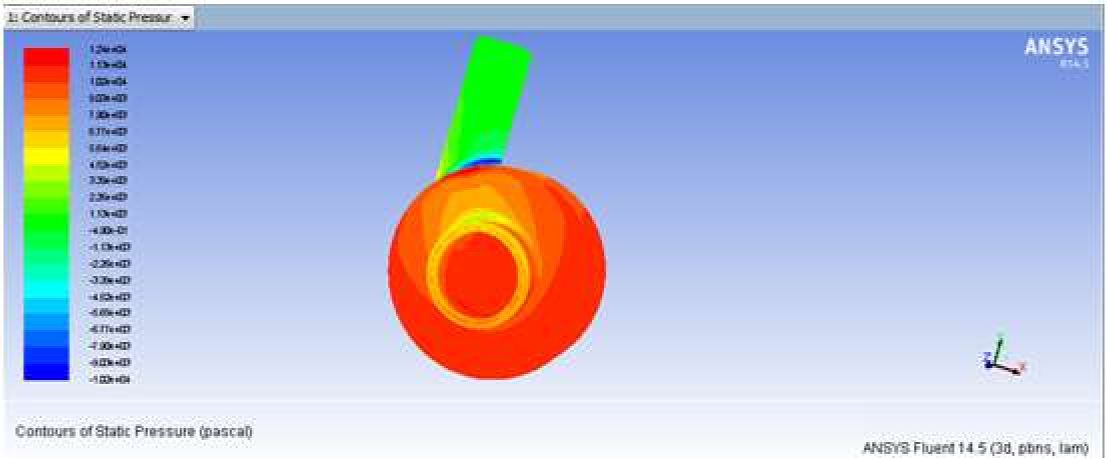


Fig: pressure counters at 8 blades

According to the above plot the maximum pressure distribution at inside of the casing and minimum pressure at outlet of the pump. The maximum pressure is 1.24e+04 Pa and minimum pressure is 5.13e+03 Pa.

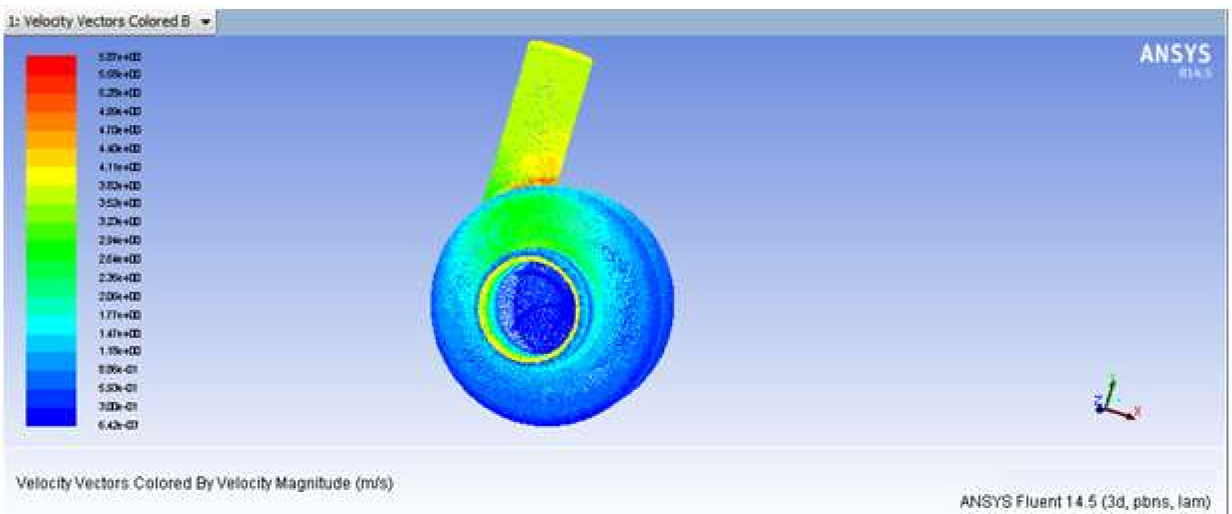


Fig: velocity counters at 8 blades

According to the above plot the maximum velocity at outside of the pump and minimum velocity at inlet of the pump. The maximum velocity is 5.87 m/s and minimum velocity is 6.41e-01 m/s.

Case 3:10 blades

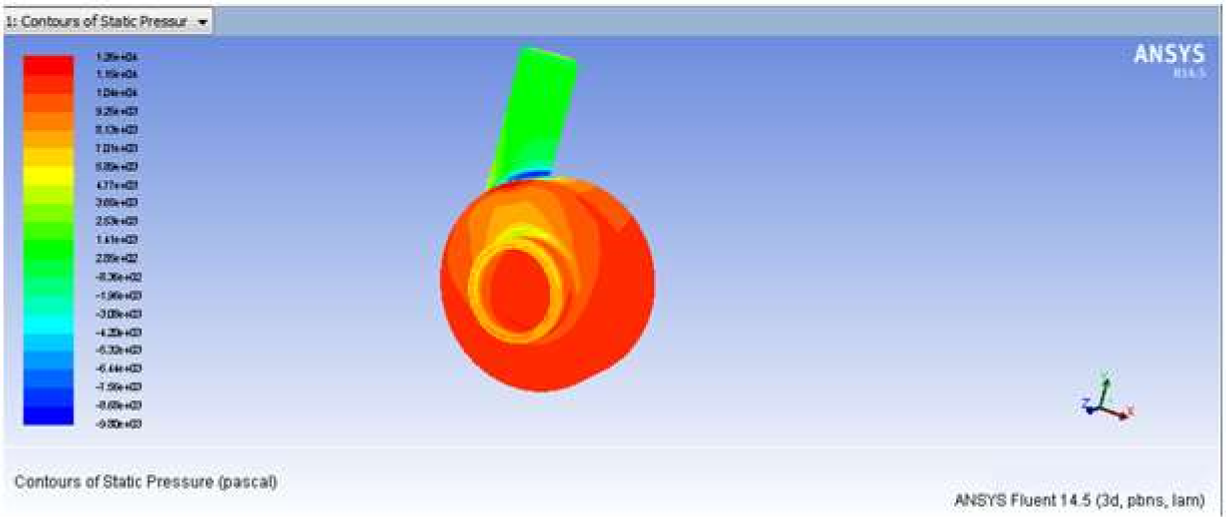


Fig: pressure counters at 10 blades

According to the above plot the maximum pressure distribution at inside of the casing and minimum pressure at outlet of the pump. The maximum pressure is 1.26e+04 Pa and minimum pressure is 2.18e+03 Pa.

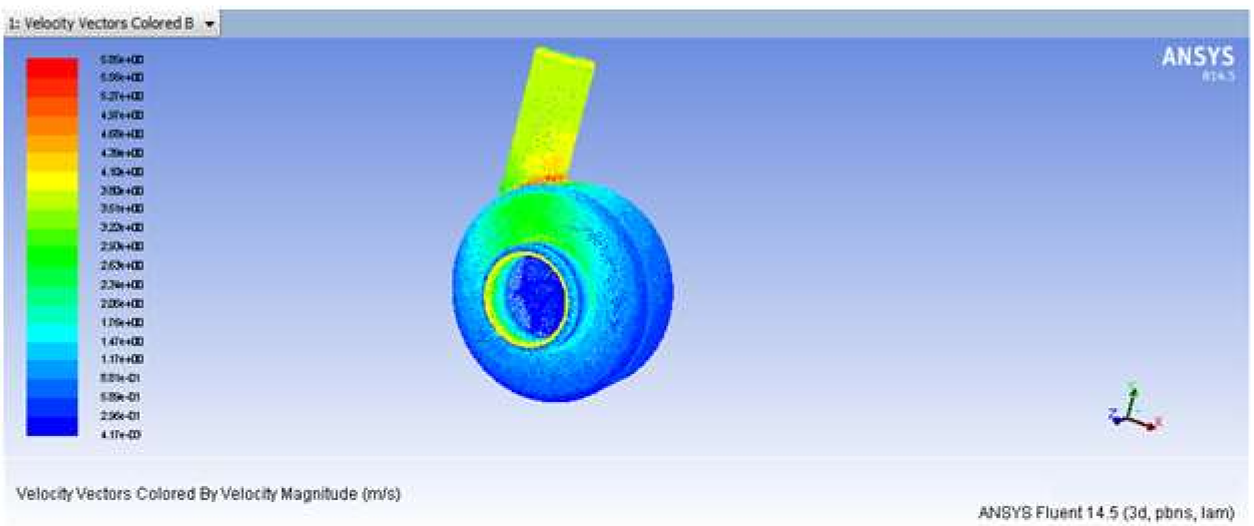


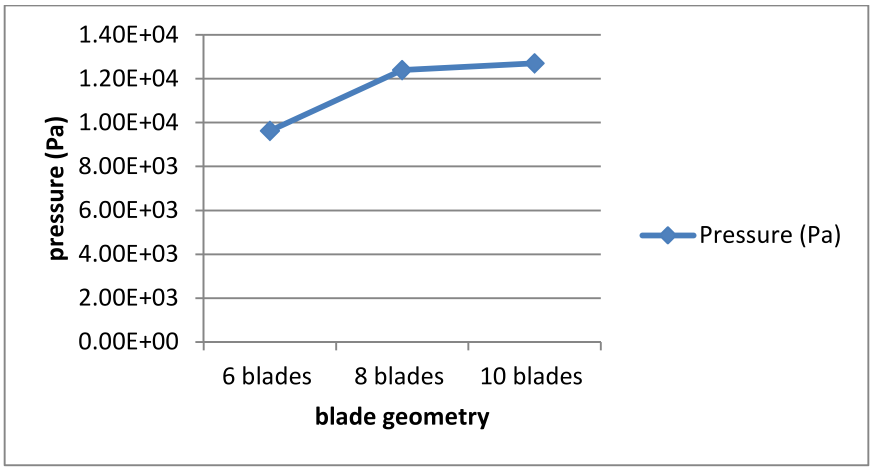
Fig: velocity counters at 10 blades

According to the above plot the maximum velocity at outside of the pump and minimum velocity at inlet of the pump. The maximum velocity is 5.56 m/s and minimum velocity is 5.41e-01 m/s.

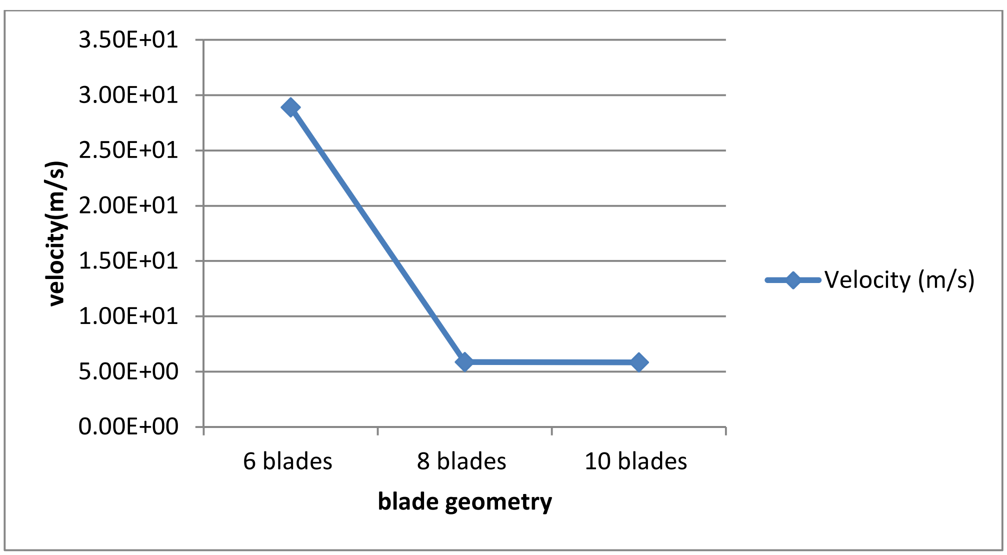
Table: 3 CFD analysis Results tables

Cases	Pressure (Pa)	Velocity (m/s)
6 blades	9.63e+03	2.89e+01
8 blades	1.24e+04	5.87e+00
10 blades	1.27e+04	5.84e+00

Graphs



Above graph shows blade geometry versus pressure, the maximum pressure at 10 blades geometry and minimum pressure at 6 blades.



Above graph shows blade geometry versus velocity, the maximum velocity at 6 blades geometry and minimum pressure at 10 blades.

CONCLUSION

- Design and CFD analysis is done on boiler feed pump.
- Brief study about boiler feed pump its application, classification is studies in this project.
- Three different models of boiler feed pump with 6,8,10 no's of impeller blades is modeled in CATIA Parametric software.
- Computational fluid dynamic analysis is performed in ANSYS Flow simulation module on three different geometry with same boundary conditions
- Flow characteristics like pressure and velocity are noted and tabulated.
- Flow characteristic pressure and velocity are shown in counters and trajectories.
- From result table we can conclude that as the numbers of blades of impller are increasing pressure inside the pump is decreasing.
- And as the numbers of blades of impeller are increasing velocity is slightly increasing as a result pressure decreases.

REFERENCES

[1] John Anagnostopoulos, "CFD Analysis and Design Effects in a Radial Pump Impeller", Wseas Transactions on Fluid Mechanics, Volume-1, Issue-7, July-2006, PP. 763-770.

[2] H Sedaghati Nasab, M Massoumian, H Karimi, "CM Experiences with Boiler Feed Pump", Journal of Worlds Pump, January-2011.

[3] Igor Karassik, J Messina, P cooper, C Heald, "Siphon Handbook", third Edition, McGraw Hill Publication, New York, 2001.

[4] S R Shah, S V Jain, V J Lakhera, "CFD for Centrifugal Pumps: A Review of the State-of-the-Art", Chemical, Civil and Mechanical Engineering Tracks of third Nirma University International Conference, Ahmedabad, India, 2012, PP. 715-720.

[5] M Mohammed Mohaideen, "Streamlining of Backward Curved Aerofoil Radial Fan Impeller utilizing Finite Element Modeling", 2012, PP. 1592-1598.

[6] S R Shah, S V Jain, V J Lakhera, "CFD Based Flow Analysis of Centrifugal Pump", fourth International Conference on Fluid Mechanics and Fluid Power, Chennai, December-2010, PP. 435-441.