Numerical Investigation on the Role of Leading-Edge Blade Angle in Controlling Cavitation in Centrifugal Pump Impellers

C. Syamsundar[1*], G. Venkatasubabaiah[2], Lakshmipathi Yerra[1]

[1]Department of Mechanical Engineering, CMR Engineering College, Hyderabad, Telangana 501 401, India

[2]Department of Mechanical Engineering, MVSR Engineering College, Nadergul, Hyderabad, Telangana 501 510, India

[*]Corresponding author. Tel.: +91-98408 91252, syamsundariitm@gmail.com

Abstract

The operation of the centrifugal pump itself is the cause of cavitation due to the possibility of formation and collapse of bubbles. It is commonly agreed that a microjet causes cavitation erosion or a shock wave generated at the moment of bubble collapse near a solid surface, producing a pressure ranging up to 1000 MPa and causing severe pitting and removal of material, decrease of the pump performance and total head. A study of more research conducted on the same lines is needed for a better understanding of the challenges of cavitation.

In the present paper, we are using three different blade leading edge angles (β1) 9º, 15º, and 21º with a rotational speed of 1200 rpm and different flow rates (designed and off-designed) are applied in order to reduce the cavitation and enhance the pump performance by using CFD. Cavitation models are imposed by using Rayley-Plesset Equations.

For each leading-edge angle, the performance of the centrifugal pump through head drops and total efficiency curves, pressure variation in the blade passage from hub to shroud, and vapor distribution on the blades at different span locations are discussed. With the preliminary results, by increasing the blade leading edge angle on the cavitation, it is observed that both the head and total efficiency increase at different operating conditions. For the blade leading edge angle of 9º, developed cavitation occurred at the leading edge on the pressure side, and for 21º, it is observed on the suction side. For a 15º leading edge angle, there is no cavitation development, and the danger of pump damage and erosion is eliminated. These are further discussed with respect to other results in this paper.

Keywords: Centrifugal pump; Blade leading edge angle; Cavitation; CFD

1. Introduction

The main applications of centrifugal pumps are in the fields of industries, domestic applications and agriculture fields. Its main component, i.e., impeller design, depends on a complete understanding of internal flow and operations conditions [Usha, and Syamsundar, 2010; Shah et al., 2013]. The flow through the impeller is extremely complex, mainly due to turbulence secondary flow losses, and flow is mainly unsteady. In the design of a centrifugal pump, the impeller inlet and outlet blade angle, design and off-design operating condition and effect of cavitation are most important. We all know that cavitation is a multi-physics process, which involves three phases: formation, growth and collapse of vapor bubbles in any fluid due to dynamic pressure changes. It results in damage to pump performance, abnormal noise and vibration, and erosion, which reduces the efficiency and life of the pump impeller [Xavier Escaler et al., 2006]. Cavitation within pumps has been studied by some researchers such as Delgosha et al. [2003], Cudina [2003], Baldassarre et al. [1998], McNulty and Pearsall [1982]and others. When a cavity collapses from away from a solid boundary, it results in a shock wave [Fujikawa and Akamatsu, 1978;]; on the other hand, if it collapses near a wall, it results in microjet formation.

For both radial and forward-facing blades, the power is rising monotonically as the flow rate is increased. In the case of backward-facing blades, the maximum efficiency occurs in the region of maximum power. In the majority of cases, the outlet blade angle should be between 15° and 40°, and the inlet blade angle should be in the range of 16 to 20° from consideration of cavitation. The blades should be longer and smaller in number.

The experimental studies are a lot more time-consuming and also more costly. Similarly, the theoretical analysis also gives a rough estimation, but it also leads to poor understanding and performance [Patel, and Satanee, 2006; Shah et al., 2013]. Computational fluid dynamics (CFD) have successfully contributed to the prediction of the flow through pumps. Now, some researchers have studied the impeller, but the effect of inlet blade angle is not reported till now. This paper mainly focuses on the effect of inlet blade angle and its effect on performance. We have carried out CFD analysis on three different inlet blade angles at a rotational speed of 1200 rpm and a flow rate of 35 m^3/h. These results are compared with the experimental results of Kyparissis and Margaris, 2012.

2. Numerical Formulation

As discussed in the previous section, due to complex impeller geometry and flow conditions, we have to go for a three-dimensional flow field for better understanding. Here, for studying the cavitation phenomenon, we have to go for two-phase flows. Kyparissis and Margaris, 2012, have done an experimental investigation to study the effect of impeller inlet blade angle on the performance of centrifugal pump impeller, as shown in Figure 1. The line diagram with dimensions and the regions where the pressure transducers are installed are shown in Figure 2.

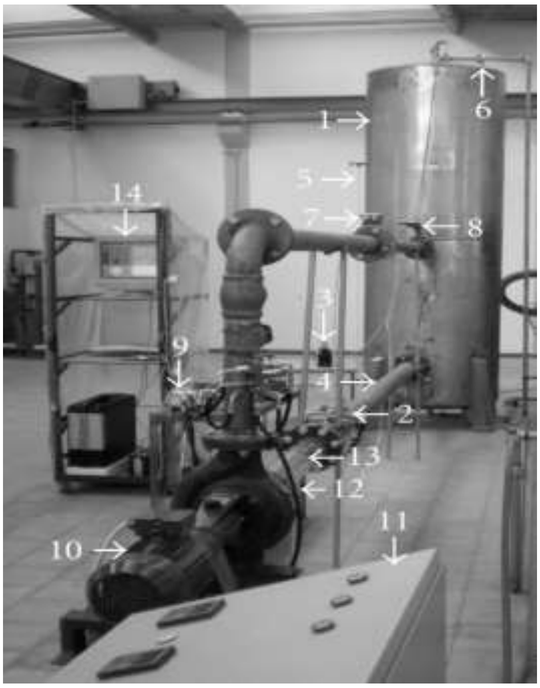

Figure 1: Experimental test facility [Kyparissis and Margaris, 2012].

The main components are: 1. Supplying tank, 2. Pressure transducer, 3. Vacuum pump, 4. Temperature sensor, 5. Fluid level meter, 6. Vacuum-pressure gauge, 7. Electromagnetic flowmeter, 8. Throttling and butterfly valve, 9. Differential pressure transducer, 10. 3-phase AC electric motor, 11. Inverter, 12. Pump suction cover, 13. Suction pipe, 14. Data acquisition system and LabVIEW]

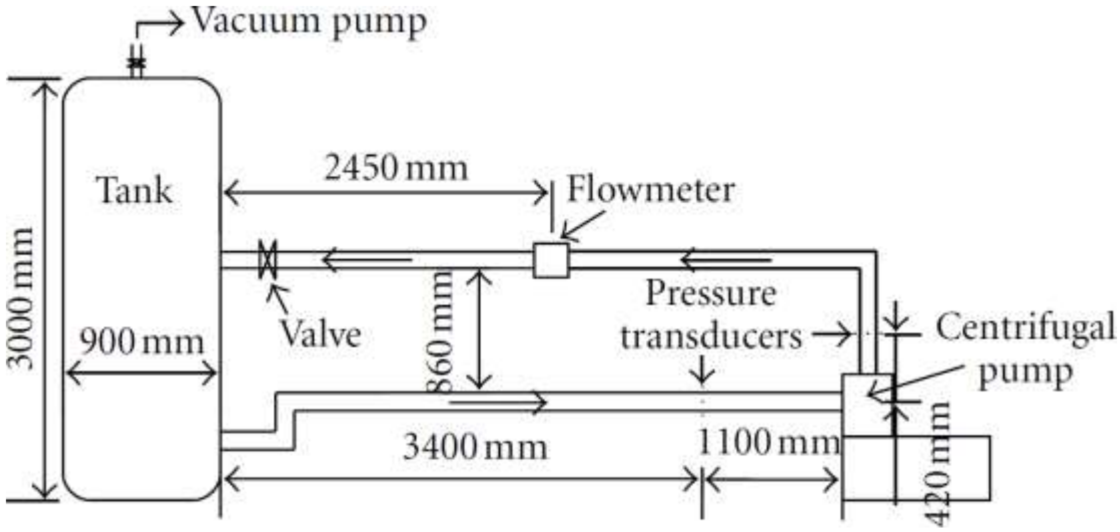

Figure 2: Line diagram of the test rig and location of pressure transducers [Kyparissis and Margaris, 2012].

The main specifications of the three tested impellers are presented in Table 1 [Kyparissis and Margaris, 2012].

Table 1: Main dimensions of the tested impellers [Kyparissis and Margaris, 2012].

Impeller dimensions	**Specifications**
Diameter of suction pipe, D_s	: 100 mm
Suction side impeller diameter, D_1	: 108 mm
Pressure side impeller diameter, D_2	: 180 mm
Width of the impeller at the suction side, b_1	: 15.6 mm
Width of impeller at pressure side, b_2	: 15.6 mm
Leading edge blade angle, β_1	: 9º, 15 º, and21º
Trailing edge blade angle, β_2	: 20 º
Blade width, S	: 7.5 mm
Number of blades, Z	: 5

All three testing impellers are made with Plexiglas for flow visualization for cavitation studies, as shown in Figures 3 (a)-(c). We have conducted the CFD simulations on three blades leading edge angles of 9º, 15º and 21º. The mass flow rates considered are 2.28 kg/s, 6.16 kg/s, 8.42 kg/s, 9.8 kg/s, 10.12 kg/s, 10.36 kg/s and 10.5 kg/s. CFD simulations are carried out on a 3-D flow impeller by using incompressible N-S equations over an unstructured grid by using Ansys-CFX. The turbulence model considered in this work is k-ε, with a turbulence intensity of 2%. The model pump impeller with the above specifications is shown in Figure 4. Here, we have considered unstructured meshes, with the main aim of reducing the computational time. The mesh geometry of a five-bladed pump impeller with tetrahedral cells is shown in Figure. 5. The mesh is fully refined near the blades and also in the regions of the leading and trailing edges of the blades. Around the blades, structured prismatic cells are generated to obtain better boundary layer details. Figure 6 shows the mesh refinement around the blade surface and inflated layers around the blades. The mesh statistics for all three impellers are shown in Table 2.

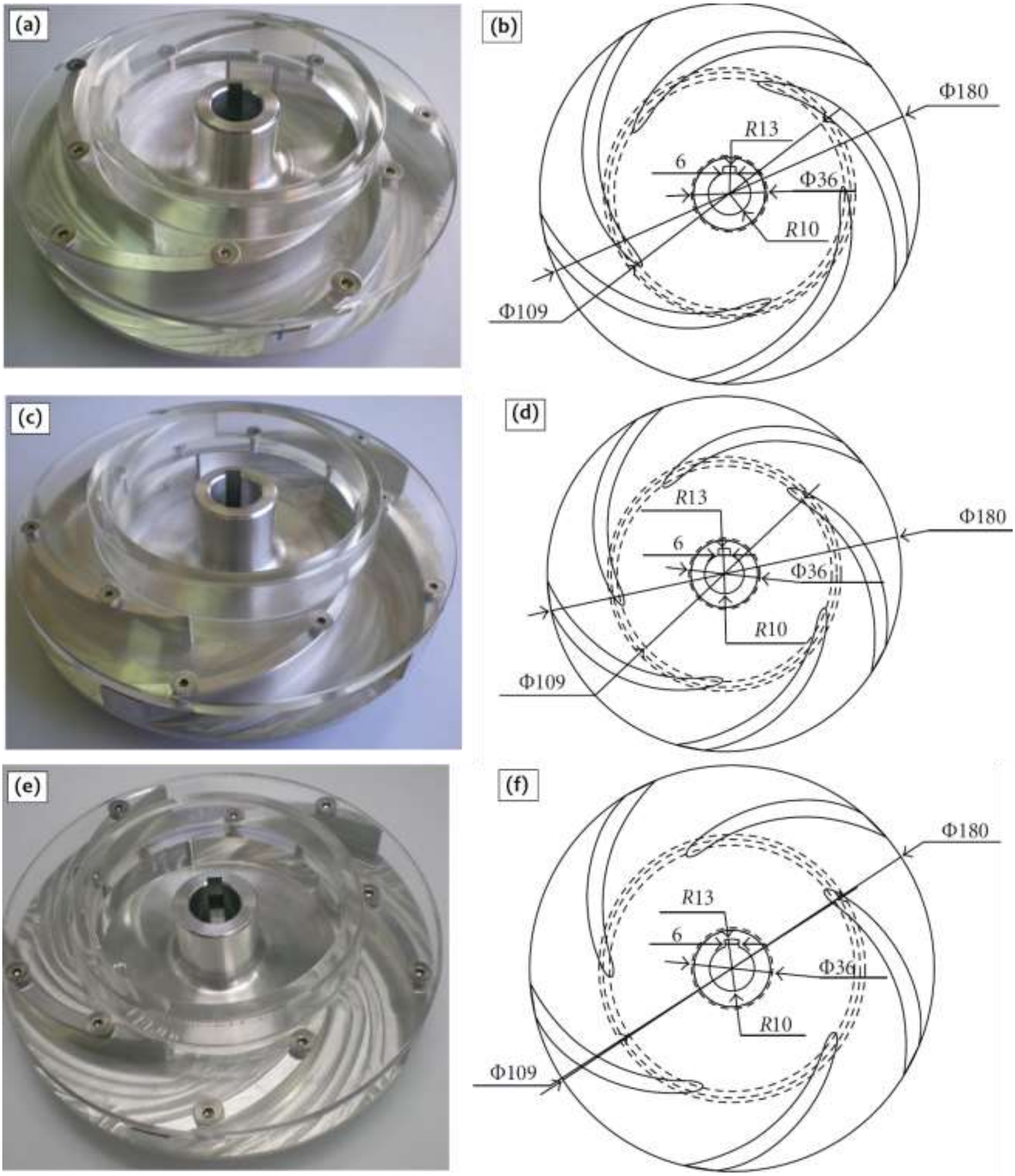

Figure 3: Designed impellers for experimental analysis having blade leading edge angles of (a) 9º, (b) 15º and (c) 21º

[Kyparissis and Margaris, 2012].

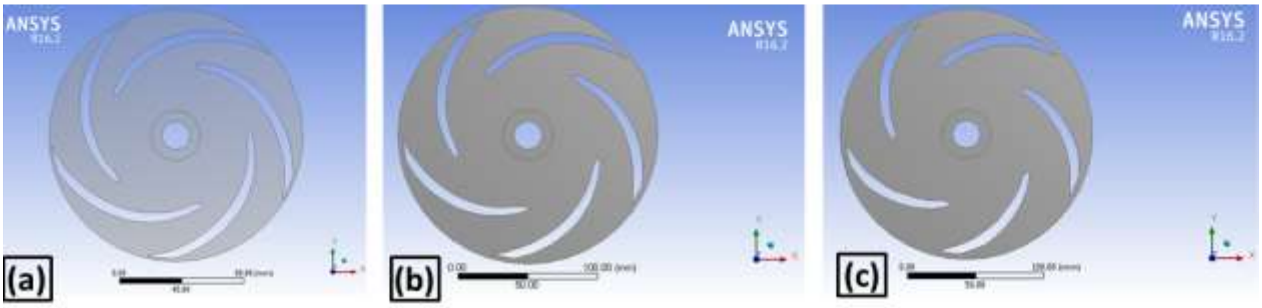

Figure 4: Designed impellers for computational analysis having blade leading edge angles of (a) 9º, (b) 15º and (c) 21º.

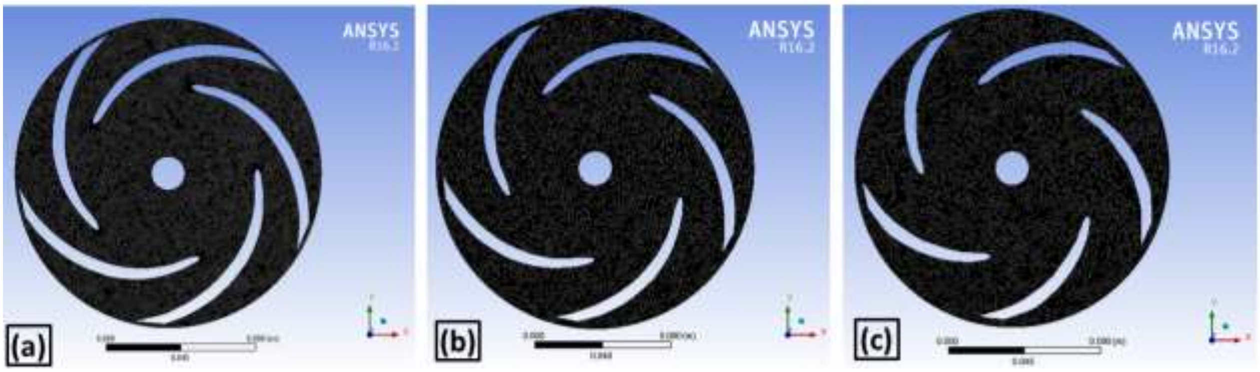

Figure 5: Unstructured mesh of designed impellers having blade leading edge angle of (a) 9º, (b) 15º and (c) 21º.

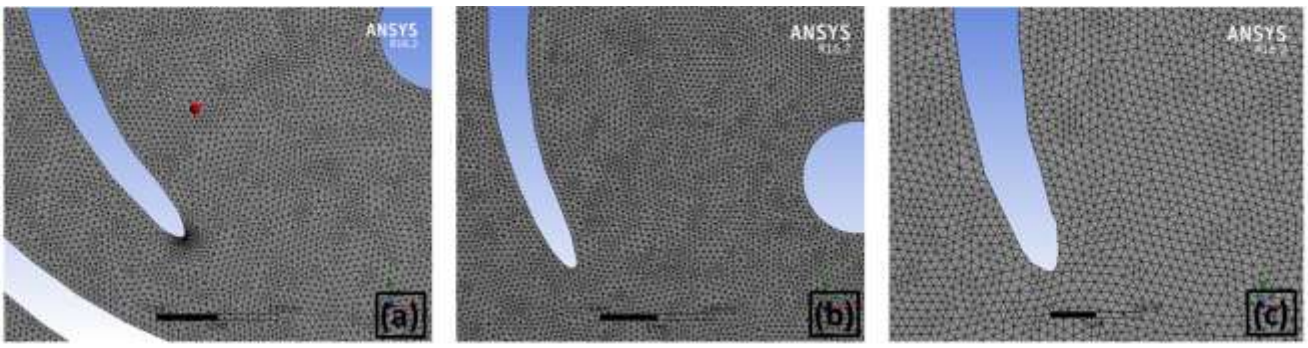

Figure 6: Illustrates the mesh refinement around the blade surface and inflated layers impellers having blade leading edge angles of (a) 9º, (b) 15º and (c) 21º.

Three different impellers, having inlet blade angles of 9, 15 and 21º, are considered with a rotating frame of reference with a rotational speed of 1200 rpm. The working fluid through the pump is water at 25^{o}C. k-ε Turbulence model with turbulence intensity of 2% is considered. Non-slip boundary conditions are applied on impeller blades, hub and shroud sections. Inlet relative pressure and outlet mass flow rate are given as boundary conditions. Three-dimensional incompressible N-S equations are solved by using Ansys-CFX Solver. For each inlet-bladed angle, the pressure contours, velocity vectors, and velocity contours are drawn and analyzed. Cavitation models are imposed by using Ryley Plessey Equations. Here, we have considered seven different mass flow rates. The simulations are carried out both without cavitation and with cavitation conditions.

Table 2: Mesh Statistics for all three impellers

	Impeller 1	Impeller 2	Impeller 3
Mess Sizing			
Smoothing	Fine	Fine	Fine
Min Face Size	3.8e-5 m	3.59e-5 m	3.84e-5 m
Max Face Size	1.e-3 m	1.e-3 m	1.e-3 m
Min Edge Length	4.9e-4 m	3.7e-4 m	6.5e-5 m
Inflation			
Transition Ratio	0.77	0.77	0.77
Maximum Layers	8	8	8
Statistics			
Nodes	591405	515217	586700
Elements	3264708	3028733	3264650

3. Results and Discussion

In this section, we are presenting the CFD results obtained for three impellers having inlet blade leading edge angles of 9, 15 and 21º, operating at 1200 rpm at seven different mass flow rates. For conducting CFD simulations, we are considering the experimental results from Kyparissis and Margaris, 2012. The experimental results of total head and total efficiency as a function of flow rate are shown in Figure 7(a) and (b), for a rotational speed of 1200 rpm. From these results, the best efficiency point is approximately 22 m^3/h for 9º, 34 m^3/h for 15º, and 43 m^3/h for 21º.

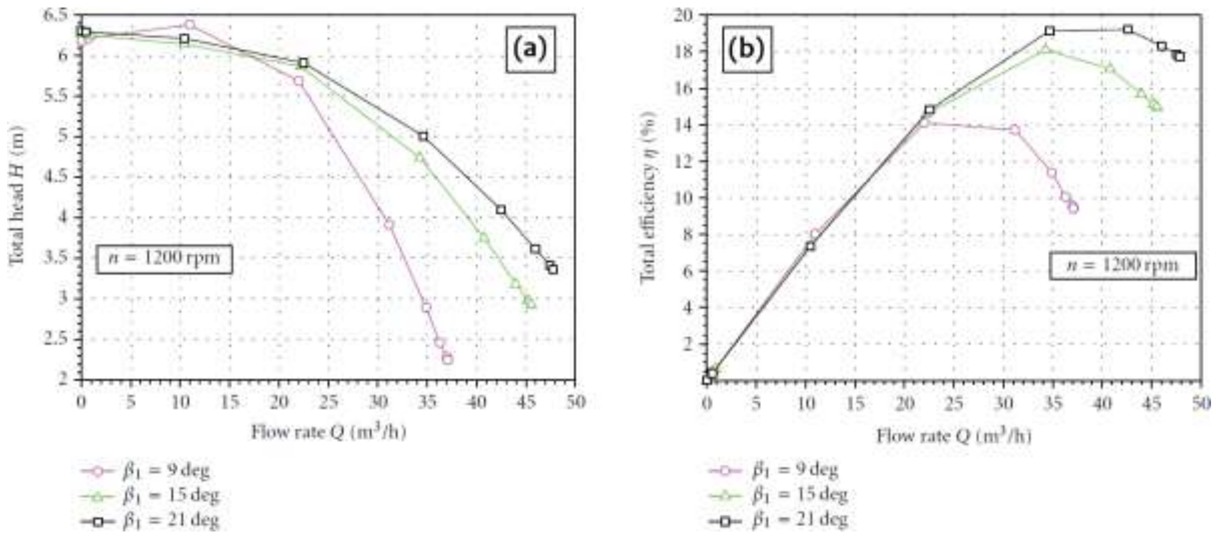

Figure 7: Experimental results showing (a) Total head Vs. Flow rate, (b) Total efficiency Vs. Flow rate for non-cavitating.

For numerical analysis, we have considered seven different mas flow rate values, such as 2.28 kg/s, 6.16 kg/s, 8.42 kg/s, 9.8 kg/s, 10.12 kg/s, 10.36 kg/s and 10.5 kg/s.

ISSN- 2394-5125 VOL 7, ISSUE 19, 2020

Studies on Pressure Contours on Mid-plane- Without Cavitation:

The total pressure contours are taken at the mid-span location. From these contours, Figures 8 to 10, there are very low pressures observed at the hub location and the blade leading edge. The pressures keep on increasing from these locations due to the increase of dynamic head due to impeller rotation. At the tailing edge, total pressure losses are observed, which may be due to wake formation.

Impeller with Leading Edge Angle 9^{O}- Without Cavitation

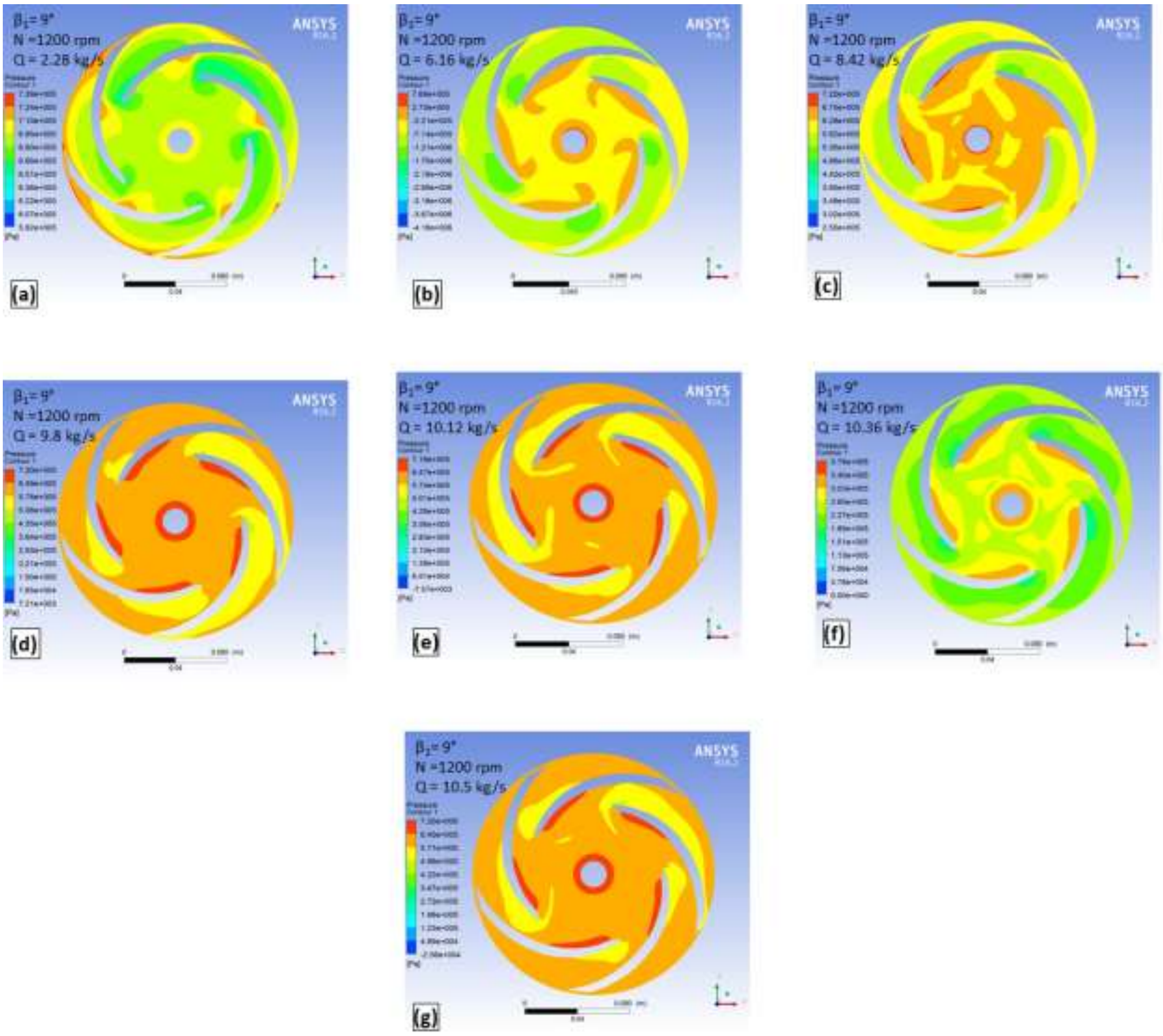

Figure 8: Pressure contours at non-cavitation conditions for the blade leading edge angle of 9^{o} operating at 1200 rpm, having different mass flow rates of (a)2.28 kg/s, (b) 6.16 kg/s, (c) 8.42 kg/s, (d) 9.8 kg/s, (e) 10.12 kg/s, (f) 10.36 kg/s and (g) 10.5 kg/s.

ISSN- 2394-5125 VOL 7, ISSUE 19, 2020

Impeller with Leading Edge Angle 15^{O}- Without Cavitation

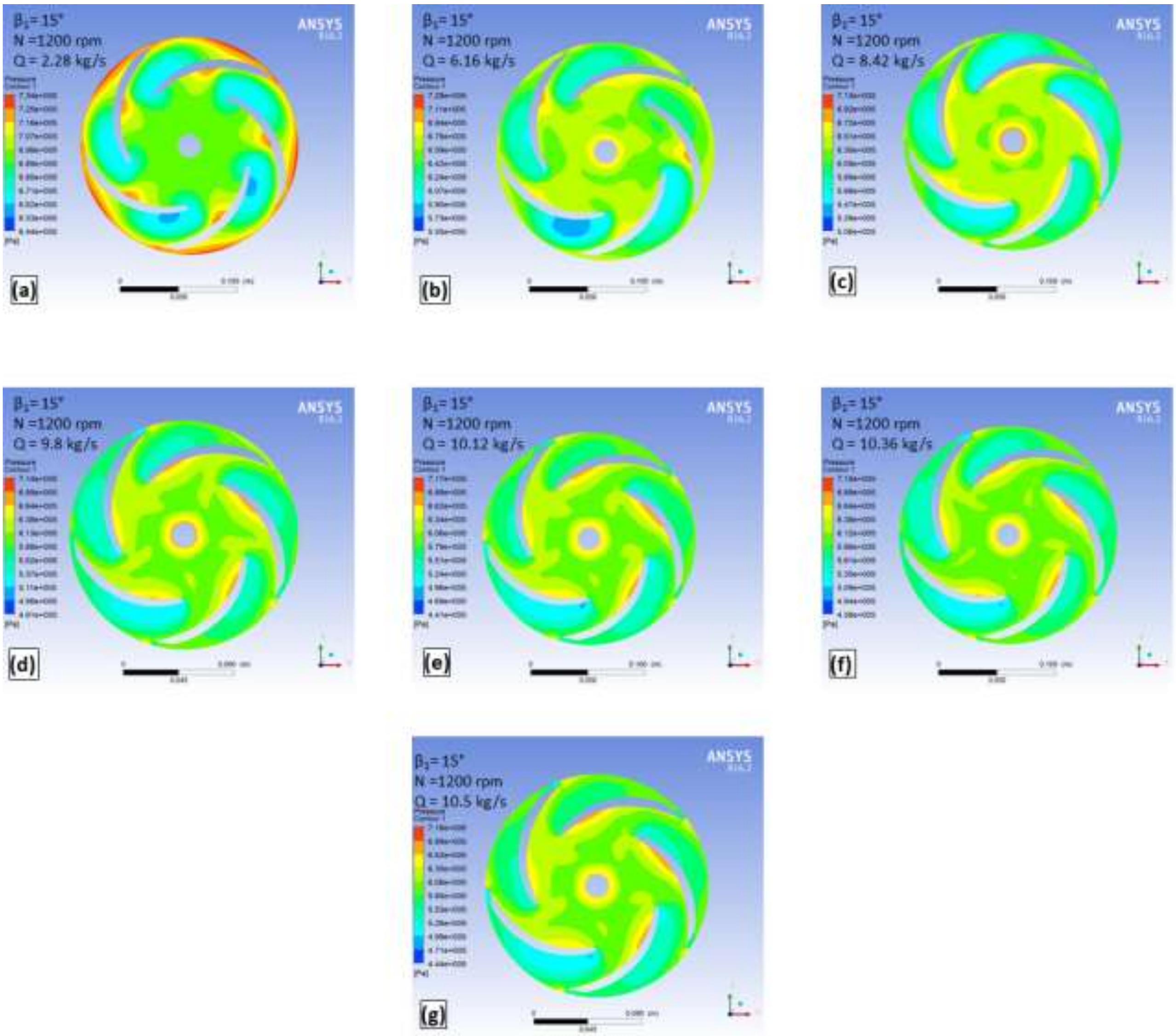

Figure 9: Pressure contours at non-cavitation conditions for the blade leading edge angle of 15º operating at 1200 rpm, having different mass flow rates of (a)2.28 kg/s, (b) 6.16 kg/s, (c) 8.42 kg/s, (d) 9.8 kg/s, (e) 10.12 kg/s, (f) 10.36 kg/s and (g) 10.5 kg/s.

ISSN- 2394-5125 VOL 7, ISSUE 19, 2020

Impeller with Leading Edge Angle 21^{O}- Without Cavitation

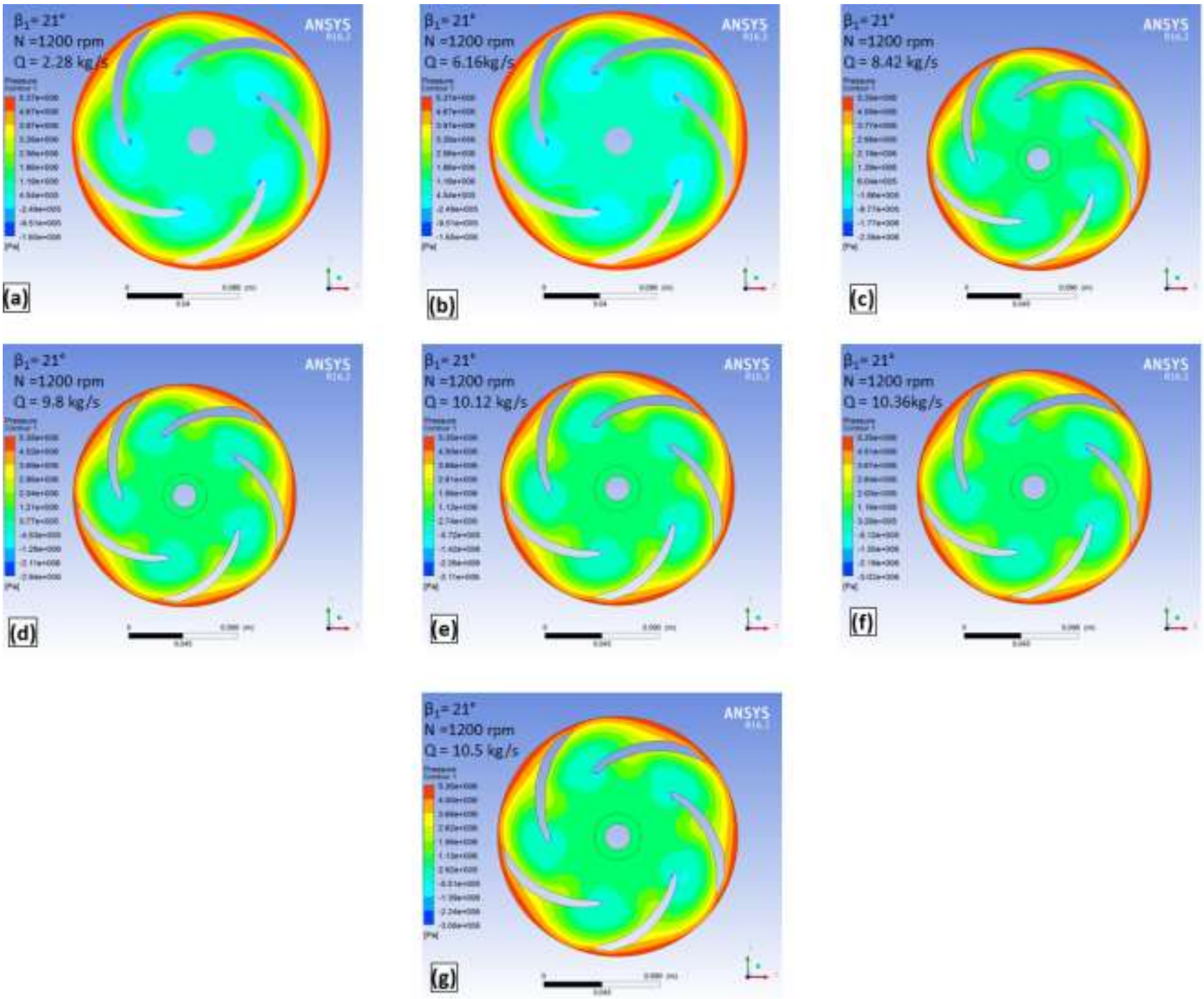

Figure 10: Pressure contours at non-cavitation conditions for the blade leading edge angle of 21^{o} operating at 1200 rpm, having different mass flow rates of (a)2.28 kg/s, (b) 6.16 kg/s, (c) 8.42 kg/s, (d) 9.8 kg/s, (e) 10.12 kg/s, (f) 10.36 kg/s and (g) 10.5 kg/s.

Studies on Pressure Contours on Mid-plane- With Cavitation:

We all know that cavitation is a main problem in the case of pumps and turbines. It results in the modification of hydrodynamic flow due to the presence of cavities formation. In cavitation, the cavitation bubbles will occupy the flow space, resulting in the reduction of the flow rate. The collapse of cavitation bubbles results in erosion and material losses. Due to cavitation, we can also experience the noise and vibrations in most of the cases. The pressure contours on mid-place with cavitation conditions are shown in Figures 11 to 13. All these Figures clearly show that low pressure will occur at leading-edge locations.

ISSN- 2394-5125 VOL 7, ISSUE 19, 2020

Impeller with Leading Edge Angle 9º- With Cavitation

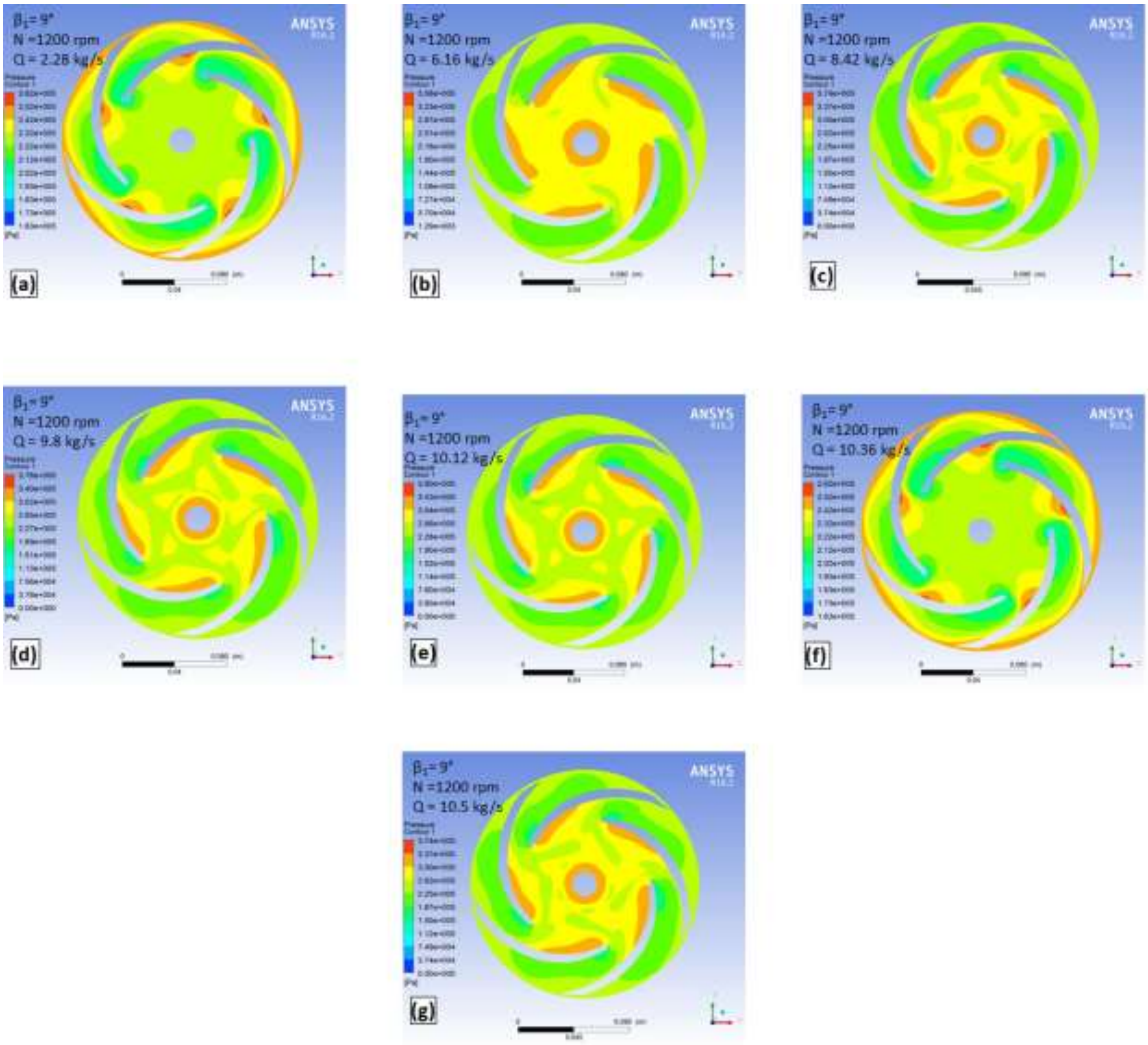

Figure 11: Pressure contours at cavitation conditions for the blade leading edge angle of 9º operating at 1200 rpm, having different mass flow rates of (a)2.28 kg/s, (b) 6.16 kg/s, (c) 8.42 kg/s, (d) 9.8 kg/s, (e) 10.12 kg/s, (f) 10.36 kg/s and (g) 10.5 kg/s.

ISSN- 2394-5125 VOL 7, ISSUE 19, 2020

Impeller with Leading Edge Angle 15º- With Cavitation

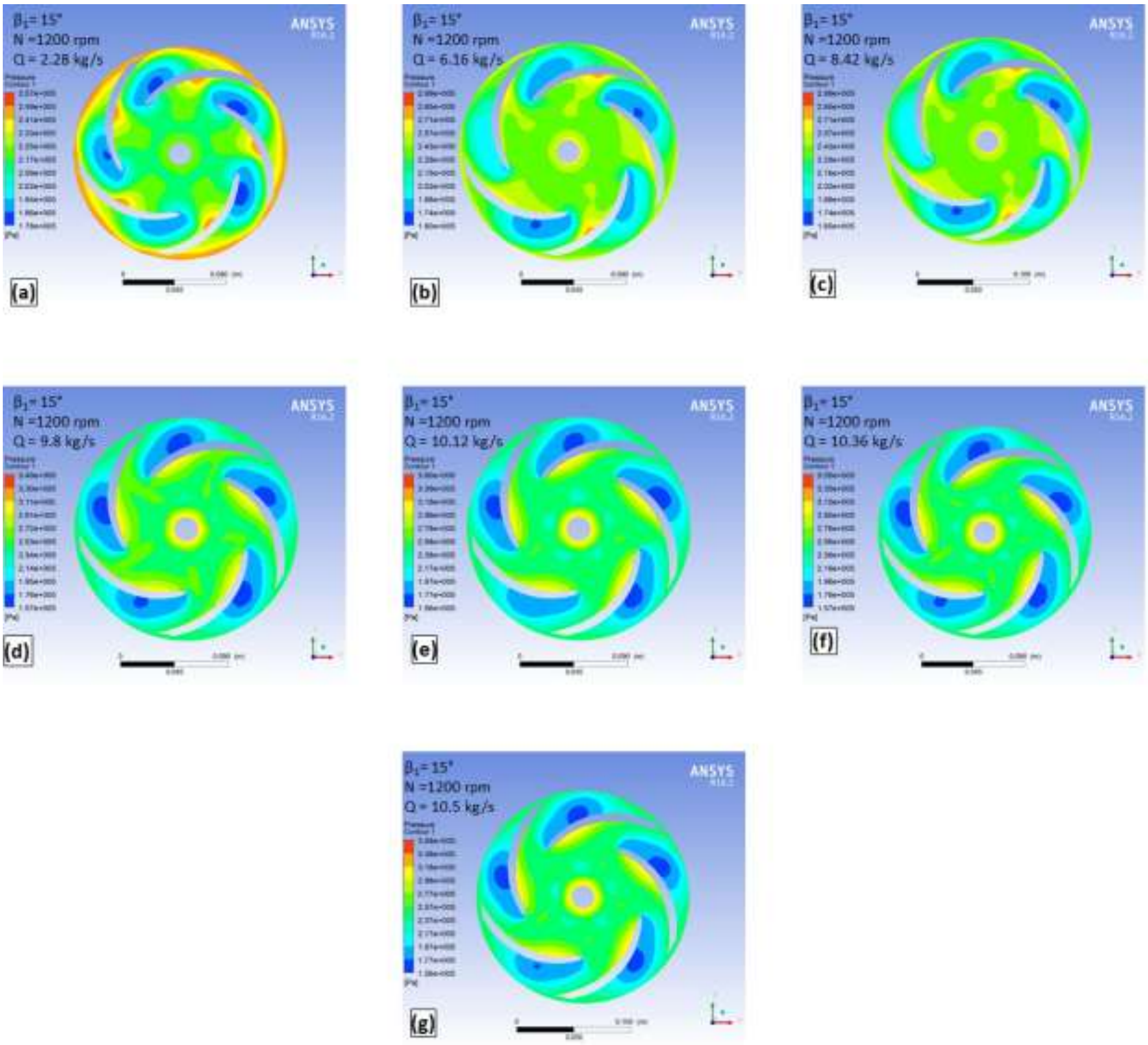

Figure 12: Pressure contours at cavitation conditions for the blade leading edge angle of 15º operating at 1200 rpm, having different mass flow rates of (a)2.28 kg/s, (b) 6.16 kg/s, (c) 8.42 kg/s, (d) 9.8 kg/s, (e) 10.12 kg/s, (f) 10.36 kg/s and (g) 10.5 kg/s.

ISSN- 2394-5125 VOL 7, ISSUE 19, 2020

Impeller with Leading Edge Angle 21^{O}- With Cavitation

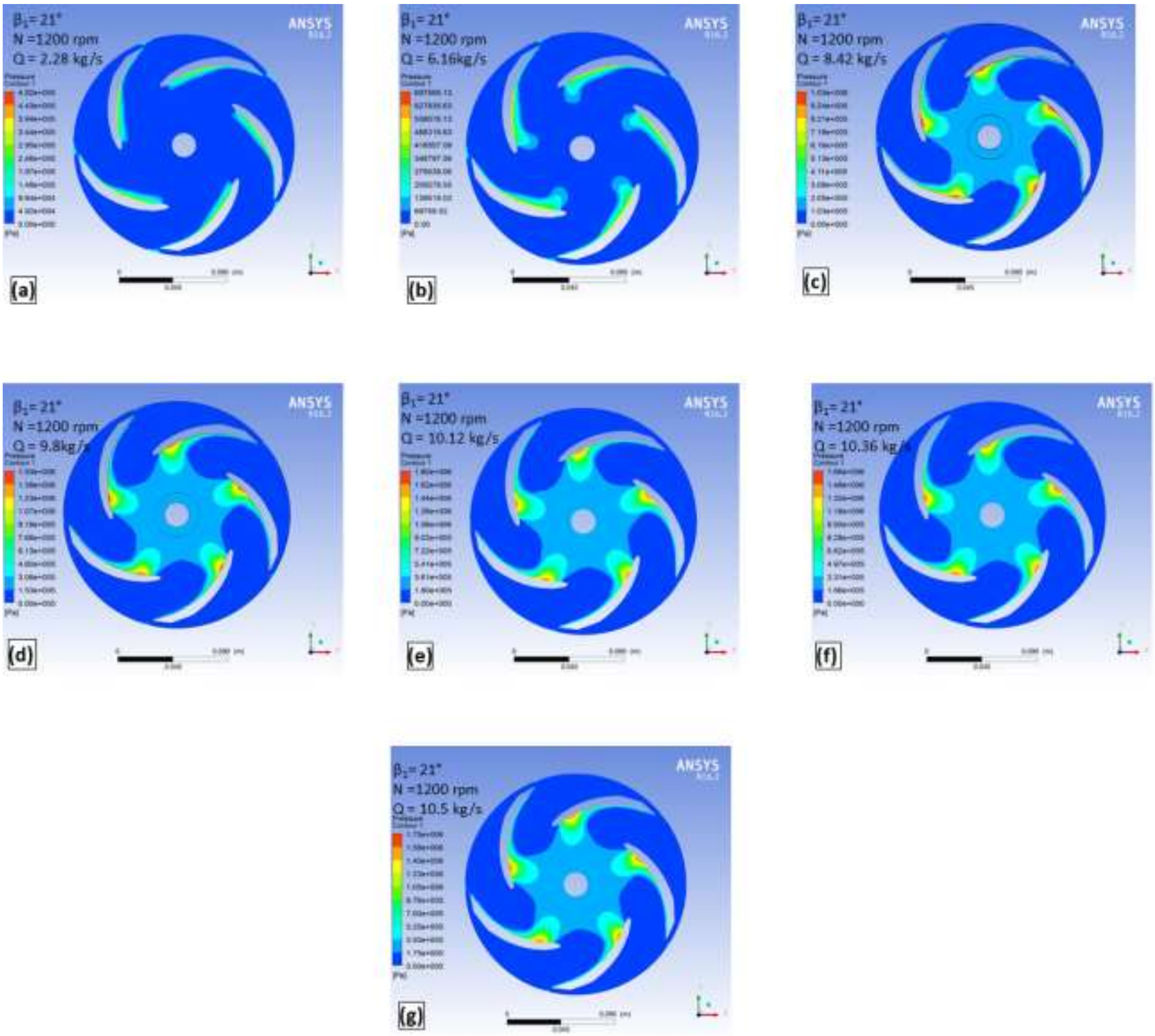

Figure 13: Pressure contours at cavitation conditions for the blade leading edge angle of 21° operating at 1200 rpm, having different mass flow rates of (a)2.28 kg/s, (b) 6.16 kg/s, (c) 8.42 kg/s, (d) 9.8 kg/s, (e) 10.12 kg/s, (f) 10.36 kg/s and (g) 10.5 kg/s.

The main function of any centrifugal pump is to lift the liquids/water from lower levels to higher levels. Due to losses in the suction pipe and suction portion, the pressures at the section portion will still be reduced. When the fluid enters the inlet of the impeller vane, the relative velocity accelerates. Still, the total energy remains constant, resulting in the reduction of static pressure, which further leads to cavitation. It is also found that just the minimum total pressure not only occurs at the blade leading edge but also slightly inside. When this minimum pressure is less than vapor pressure, cavitation will occur.

Thus, due to the height through which the liquid is lifted to the impeller vane leading edge and or due to the losses in the suction pipelines, the static pressure at the leading edge is reduced. It has been found that in a region just inside the vane inlet edge, the static pressure is lower than the static pressure at the inlet edge. At the inlet to the impeller vane, as the flow enters over the leading edge, the relative velocity accelerates. Tip clearance cavitation mostly occurs in pumps. This is mainly due to leakage flow at the blade tips due to high flow velocities and, consequently, low static pressure. It results in cavitation, and blade tips will get eroded. Fig. 11 to 13 clearly shows the location of the cavitation zones for varying flow conditions. At off-design conditions, there are alterations of flow velocities at leading and trailing edges. With increasing mass flow rates, cavitation will begin at the middle portion of the suction surface, and cavitation zones will develop further. It also results in wake formations. With further increasing the mass flow rates, the cavitation will develop at the pressure side also.

4. Conclusions

Three centrifugal pump impellers having leading edge angles of 9º, 15º, and 21º operating at 1200 rpm, having different mass flow rates of 2.28 kg/s, 6.16 kg/s, 8.42 kg/s, 9.8 kg/s, 10.12 kg/s, 10.36 kg/s and 10.5 kg/s are modeled with CFD.

The main conclusions are:

- The result of cavitation is that it increases the blade loading and its performance.
- When the intensity of cavitation increases, it increases vortices. These two reasons result in a head drop. At minimum mass flow rates, cavitation mainly occurs at the suction side of the blades. However, when the flow rate is designed one, there is a minimum cavitation will occur. As the mass flow rates increase, the cavitation at the blade suction side disappears, but it occurs at the blade pressure side.
- The increase in the designed mass flow rate causes cavitation.

5. References

1. Grindoz, Lois de Similitudes dans les Essays de Cavitation des Turbines Francis, Ph.D. Thesis 714, EPFL, Lausanne, 1991.
2. Hammit, Cavitation erosion: the state of the art and predicting capability, Applied Mechanics Reviews 32 (6)(1979) 665–675.
3. Knapp, Daily, F.G. Hammit, Cavitation, McGraw-Hill, New York, 1970.
4. McNulty, Pearsall, Cavitation inception in pumps, Journal of Fluids Engineering 104 (1982) 99–104.
5. Philipp, Lauterborn, Cavitation erosion by single laser-produced bubbles, Journal of Fluid Mechanics 361(1998) 75–116.
6. Li (Ed.), Cavitation of Hydraulic Machinery, vol. 1, Imperial College Press, London, 2000.
7. Xavier Escaler, Eduard Egusquiza, Mohamed Farhat, Franc-oisAvellan, Miguel Coussirat, “Detection of cavitation in hydraulic turbines”, Mechanical Systems and Signal Processing 20 (2006) 983–1007
8. Kapil Pandya, Chetankumar Patel, A Critical review on CFD Analysis of centrifugal pump impeller, International Journal of Advance Engineering and Research Development, Vol.1, Issue 6,June 2014.
9. Rakesh joshi; Computation investigation of flow field in a centrifugal slurry pump; Mechanical engineering department; Thapar University; Patiala
10. Sujoy chakraborty, Kishan choudhary, Pransenjit datta, Bishop Debbarma; Performance prediction of centrifugal pumps with variations of blade number; Journal scientific & industrial research, 2014.
11. Shah, Jain, Patel, Lakhera, CFD for centrifugal pumps: a review of the state-of-the-art, Procedia Engineering 51 (2013) pp.715 – 720

12. Usha, Syamsundar, C.,2010. “Computational analysis on the performance of a centrifugal pump impeller,” Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power. Chennai, India, paper#TM-07.
13. Patel, Satanee, M., 2006. “New development of high head Francis turbine at jyoti ltd. for small hydro power plant,” Proceedings of Himalayan Small Hydropower Summit. Dehradun, India, paper, 13.
14. Croba, Kueny, J., 1996. Numerical Calculation of 2D Unsteady Flow in Centrifugal Pumps: Impeller and Volute Interaction. International Journal for Numerical Methods in Fluids22, pp. 467.