Asim, Taimoor and Mishra, Rakesh

Numerical Investigations on the Transient Performance of a Centrifugal Pump

Proceedings of the Forty Second National Conference on Fluid Mechanics and Fluid Power

December 14-16, 2015, NITK Surathkal, Karnataka, India

FMFP 2015 – PAPER NO. 116

Numerical Investigations on the Transient Performance of a Centrifugal Pump

Taimoor Asim
School of Computing & Engineering
University of Huddersfield
Huddersfield, HD1 3DH, UK
t.asim@hud.ac.uk

Rakesh Mishra
School of Computing & Engineering
University of Huddersfield
Huddersfield, HD1 3DH, UK
r.mishra@hud.ac.uk

ABSTRACT

Centrifugal pumps are an integral part of plants used in process industries. The flow structure within a centrifugal pump is very complex due to the interaction between the rotating impeller and the geometric features around it. In the present study, numerical investigations on a centrifugal pump have been carried out using a Computational Fluid Dynamics (CFD) based solver. This study employs finite volume technique in order to analyse the influence of variations in the rotational speed of the pump on its head. The instantaneous behaviour of a centrifugal pump with varying rotational speeds is studied using the Sliding Mesh technique. The performance parameters of a centrifugal pump have been predicted using numerical simulations. The results indicate that as the rotational speed of the centrifugal pump decreases, the head developed by the pump also decreases, where the effects of varying rotational speeds are dominant in the volute region. Furthermore, the complex interaction between the impeller blades and the tongue region has been analysed in the present study.

Key words: *Computational Fluid Dynamics, Centrifugal Pump, Sliding Mesh, Head, Rotational Speed*

1. INTRODUCTION

Centrifugal pumps have an important role in many different engineering applications. The design and performance prediction process of centrifugal pumps is quite complex. This is due to the large number of geometric, fluid and flow parameters involved in the design. Furthermore, there is a significant amount of cost and time involved in the routine trial-and-error process of design improvement. This in-turn results in profit margins to decrease, which has a considerable impact on pump manufacturers. For this reason, CFD based analysis is currently being employed for the hydrodynamic design of various pump types [1].

Numerical simulations can offer reasonably accurate information on the fluid behaviour within various systems, assisting the engineers to obtain performance envelope of a particular design [2-8]. To some degree, the pump performance characteristics, i.e. pressure, head, power and efficiency are impacted by the rotational speed of the impeller. Owing to the development of CFD codes, various parameters can now be effectively analysed and the optimum efficiency of the system can be achieved. CFD is a well-established industrial design tool, which assists in decreasing design time scales. Many researchers have conducted various numerical and experimental investigations in the area of centrifugal pumps. These investigations include analysing the effect of parameters on the fluid flow structures. Kim et al. [9] examined the effects of the blade outlet angle on the centrifugal pump's head and efficiency. Two types of volutes were used for the impeller model. The first has a circular-symmetric cross-section, and the second has circular asymmetric cross-section. It was seen that the pump's head increased with the outlet angle. Sidhesware et al. [10] carried out a hydraulic design validation study of a metallic centrifugal pump volute. The authors carried out the flow simulation for a centrifugal pump impeller over a wide range of flow rates. Numerical analysis for this case includes solution of incompressible, three-dimensional, Navier-Stokes equations and k-ε turbulence model with a turbulence intensity of 2%. The authors found that the centrifugal pump's head and efficiency decreased as the flow rates increased. Over the suction and pressure side of the blades, the pressure distribution has been found to be non-uniform. The authors also present pressure contours at mid-span for different flow rates. Rajendran et al. [11] numerically investigated the flow structure in the impeller of a centrifugal pump using ANSYS-CFX. The authors discovered that pressure difference from the pressure side to the suction side of the impeller blade increased from the leading edge to trailing edge. The minimum static pressure inside the impeller is located at the leading edge of the blades on the suction side. The total pressure on the pressure side of the blade is more than that of the suction side. Patel et al. [12] investigated the effect of the impeller blade's exit angle on the performance of a centrifugal pump. The authors analysed the effect of blade exit angle on the pump's efficiency and head. The results indicate that the performance of a centrifugal pump increased with increase in the blade exit angle. Muttalli et al. [13] studied the head and efficiency characteristics to calculate the performance of a centrifugal pump. The authors have found that an increase in mass flow rate causes a decrease in head. Almost the same trends were detected when the operating performance characteristics curves predicted by CFD were compared with the experimental results. The results identified that at a constant operating speed, static head is a strongly dependent on the mass flow rate. Zhang et al. [14] carried out a numerical and experimental study on transient flow in a centrifugal charging pump during a charging process. The internal flow mechanism of a charging centrifugal pump was analysed through the transition process operating between $34m^3/h$ and $110m^3/h$. The authors concluded that the numerical simulations can be employed to clearly capture the transient flow patterns. Cheah et al. [15] used CFD to investigate unsteady fluid flow in a centrifugal pump. Transient flow fields within centrifugal pumps are very complex. Therefore, the authors have extensively investigated the flow phenomena within a centrifugal pump using an impeller having six blades. This particular study was conducted to understand the interactions between the impeller tip and the volute tongue. The numerical setup employed 3D RANS equations with standard k-ε turbulence model. The flow inside the impeller passage was found to follow the blade curvature in a stream-wise direction. The authors concluded that CFD is a useful tool for investigating the complex flow field within the impeller of centrifugal pumps. Furthermore, CFD offers an insight into the fluid dynamics with reasonable accuracy when validated against experimental data.

These studies focus solely on some of the factors that influence the performance of centrifugal pumps. Based on extensive literature review that has been carried out, it can be seen that there is limited amount of work on establishing effects of pump parameters on the transient pressure field within a centrifugal pump. The present study investigates the effect of variations in rotational speed of the impeller blades on the instantaneous head fluctuations in a centrifugal pump. The purpose of this study is to simulate the transient behaviour employing the Sliding Mesh Technique (SMT). The results have been validated against the performance curves provided by the pump manufacturer.

2. NUMERICAL MODELLING

A commercial CFD package has been used in the present study in order to numerically investigate the transient performance of a centrifugal pump. The following sub-sections summarise the various aspects of the numerical modelling employed.

I Geometry

The centrifugal pump that has been numerically modelled is model F32/200AH Perdrollo. This pump has five backward type impeller blades, where the impeller diameter is 215mm. The inlet and outlet of pump have a diameter of 50mm and 32mm respectively. Detailed geometric dimensions of this pump model are available on the web [16]. Figure 1 depicts the numerical model of the pump under consideration. It should be noted that both the inlet and the outlet of the pump have been connected to 0.5m long pipe sections in the numerical modelling in order to mimic real-world scenarios. However, the head calculations presented in this paper are across the pump only, excluding these pipe sections.

Fig. 1 Numerical model of the pump

II Mesh Sensitivity Analysis

Hybrid meshing technique has been employed in the present study, where the inlet and outlet pipes connected to the pump have been meshed using hexahedral elements, whereas the pump itself has been meshed using tetrahedral elements. The mesh element sizing in the inlet/outlet pipes and the pump region are 3mm and 1.2mm respectively. Using these sizing, the flow domain comprises of 2.14million elements. The mesh within the impeller is shown in figure 2.

In order to ensure that the results predicted by CFD solver in the present study are independent of the mesh sizing employed, two further meshes have been generated, having half and double the number of elements than the base mesh of 2.14million elements. The head developed by the pump, using the aforementioned meshes, has been predicted using quasi-steady simulations. Table 1 summarises the predicted results and it can be clearly seen that by increasing the number of mesh elements in the base mesh, negligibly small variations in the pump head have been noticed. Hence, the base mesh, having 2.14million elements, is able to predict the pump head with reasonable accuracy.

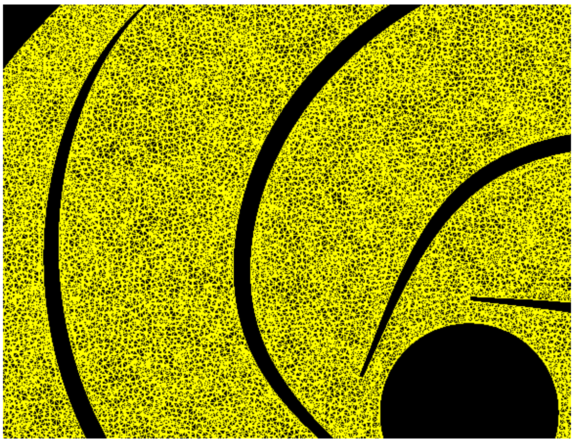


Fig. 2 Mesh in the Impeller region

Table 1 Mesh sensitivity analysis

Number of Mesh Elements	Head	Difference in Head
(million)	(m)	(%)
1.01	52.03	
2.14	55.23	6.15
4.18	55.97	1.34

III Solver Settings and Boundary Conditions

Three dimensional Navier-Stokes equations, along-with the continuity equation, have been numerically solved in an iterative manner for the flow of water within the centrifugal pump in the present study. Both quasi-steady and transient simulations have been carried out for different purposes; quasi-steady for benchmarking and characteristics curves comparison, while transient for time-varying rotational speed of the impeller. Quasi-steady conditions have been employed through the Frozen-Rotor approach [17], while transient conditions have been modelled using Sliding Mesh technique [18]. In both the cases, the rotational speeds of the impeller that have been considered in the present study are:

1. 2900rpm (design rpm)
2. 2700rpm
3. 2900rpm to 2700rpm in one revolution

Frozen-Rotor approach has been used in the present study to obtain CFD predicted characteristic curves of the centrifugal pump. Although same can be obtained through the revolution averaged predictions of sliding mesh technique, frozen-rotor approach is industry-wise preferred turbo-machines modelling approach as it is less time and computational power consuming. Furthermore, it has been shown in the present study that the frozen-rotor approach based CFD predictions are in close agreement with that from sliding mesh technique.
Shear Stress Transport (SST) k-ω turbulence model has been shown to predict flow parameters with reasonable accuracy for similar applications and hence has been employed in the present study [19-20]. Mass flow rate and outflow boundary conditions have been specified at the inlet and outlet boundaries of the flow domain, where the mass flow rates have been computed from the characteristics curves of the pump. Hence, the flow rates specified in the present study corresponds to 6, 9, 12, 15 and 18m^3/hr.

3. Results and Discussion

This study is divided into two segments. The first segment corresponds to the validation of the CFD results against the experimental data (characteristic curve at 2900rpm) available on the web. Furthermore, this segment also includes the formulation of pump's characteristic curve at 2700rpm. This segment uses the quasi-steady approach.

The second segment corresponds to the transient analysis of the pump. For this purpose, the pump has been run at design rpm for one revolution, extracting the pump's head every 3° rotation of the impeller. Then the impeller speed has been decreased to 2700rpm in one revolution. Great care has been taken to ensure that each step corresponds to 3° revolution of the impeller while the speed is reducing. Afterwards, the pump has been run at 2700rpm for one revolution.

I Quasi-Steady Performance of the Pump

The CFD predictions regarding the head developed by the pump have been validated against the experimental findings at the design rpm by the pump's manufacturer [16]. This can be seen in figure 3, where the curves representing Experiment data and CFD predictions both correspond to 2900rpm. It can be seen that CFD predictions very closely matches with the experimental data, providing justification for appropriate numerical settings being used in the study.

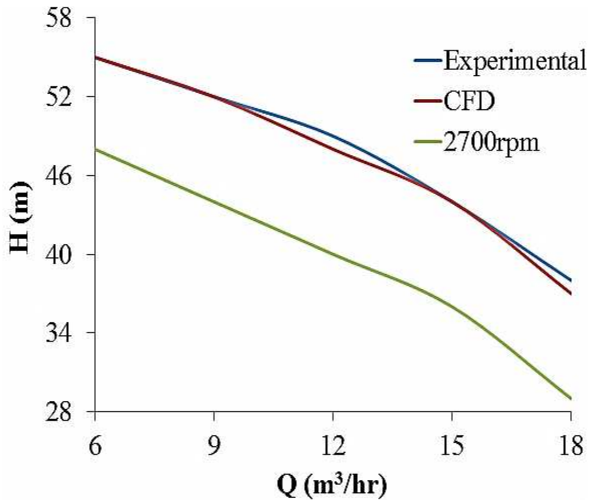


Fig. 3 Characteristic Curves of the Pump

The study is further extended by constructing a characteristic curve, for the same pump, at 2700rpm. It can be seen in figure 3 that for both the speeds, as the flow rate through the pump increases, its head decreases. Furthermore, the pump's head at 2700rpm is lower than at 2900rpm. Both these observations are consistent with the literature. Hence, flow field analysis within the pump at different operating conditions can be carried out. For this purpose, the variations in the pump's head have been shown, in the form of color-filled contours, in figure 4, corresponding to $9m^3/hr$ on both the curves (inlet Reynolds number of 63357).

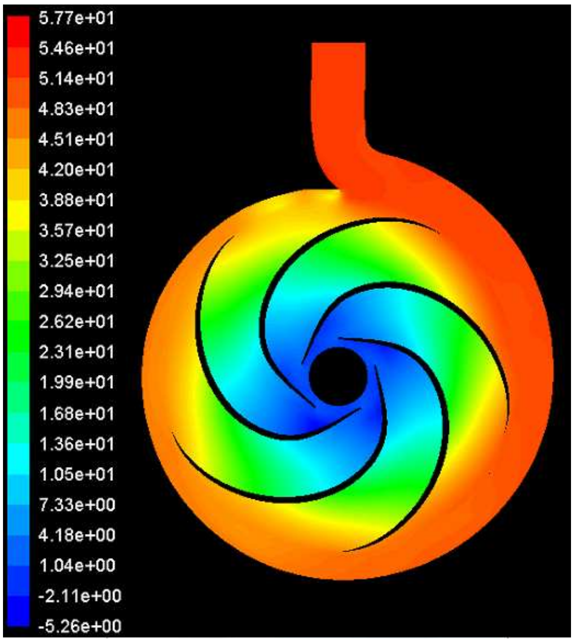
The contours shown in figure 4 have been drawn on the same scale for accurate comparison purposes. It can be clearly observed that the influence of impeller speed is more predominant in the region away from the impeller. In near-impeller zone, this effect is gradually building up. However, the head changes significantly in the volute region of the pump.

In both the contours, it can also been noted that there are some peculiar head variations downstream the tongue region, for both the operating speeds. As this section corresponds to frozen-rotor approach, further analysis on this phenomenon cannot be conducted here, and hence transient analysis, using sliding mesh technique, needs to be employed.

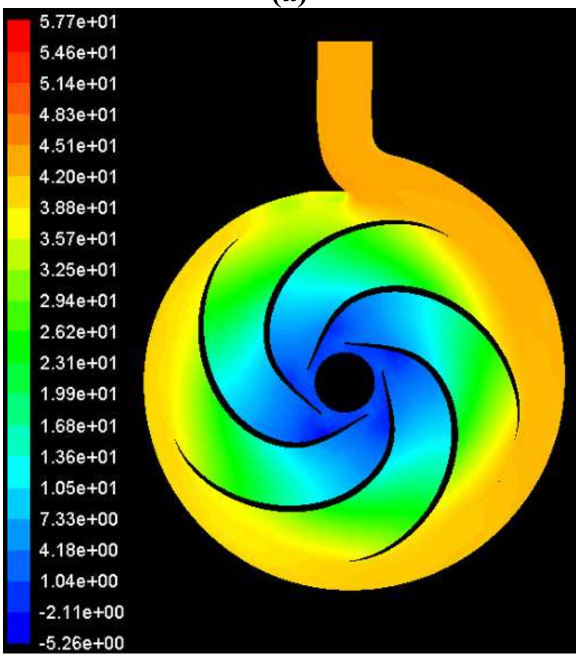
II Transient Performance of the Pump

In the transient analysis, the centrifugal pump has been run at 2900rpm for one revolution. Then its speed has been reduced from 2900rpm to 2700rpm in one revolution's time. Finally, it has been run at 2700rpm for one revolution. Sliding mesh techniques has been employed to carry out this analysis. The instantaneous pump head for the three revolutions is plotted in figure 5. It can be seen that the pump's head is higher for 2900rpm, and decreases as the impeller speed decreases. The instantaneous head's waveform is almost uniform at a particular rpm i.e. the peak-to-peak head remains almost constant at a particular pump speed.

As the pump speed starts to decrease, although the waveform remains the same, but the head gradually decreases until a constant pump speed is attained. It is noteworthy that the head data has been extracted at every 3° revolution of the impeller. This is straightforward in case of a constant rpm. However, as the pump's speed changes, the time required by the impeller blades to rotate 3° also changes, and hence time step size correction is required in CFD. This has been carried out for one complete revolution i.e. from 2900rpm to 2700rpm in 120 steps.



(a)



(b)

Fig. 4 Variations in Pump's Head at $9m^3/hr$ (a) 2900rpm (b) 2700rpm using quasi-steady analysis

Further to the discussions on the effects of speed on the pump's instantaneous head, it can also be observed in figure 5 that the waveform corresponding to a pump's head is very peculiar in nature; comprising of a number of peaks on either side of the average head value.

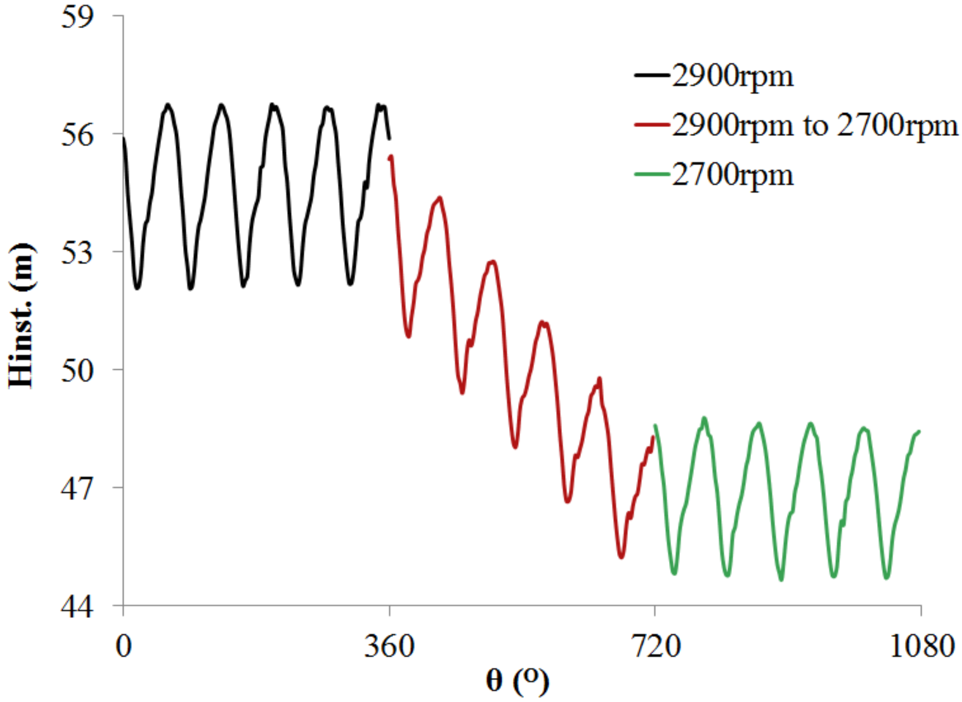


Fig. 5 Instantaneous Head variations in the three revolutions of the pump

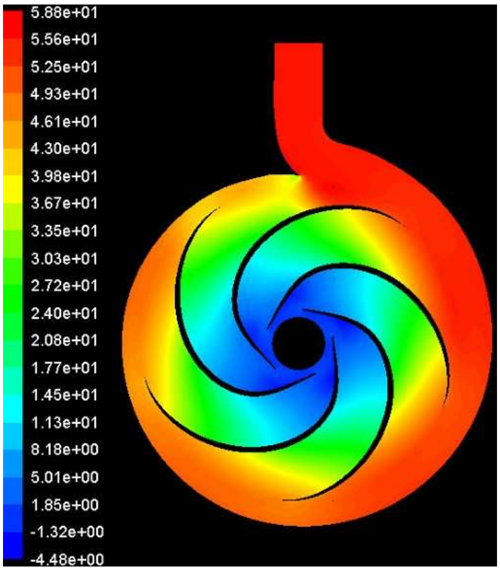
In one revolution, the number of peaks are equal to 10, which is half the number of impeller blades in the pump. Hence, if this waveform is generated by the transient motion of the impeller blades, there occur two peaks (one on each side of the average) when a blade passes through a reference point in the pump. In order to analyse this phenomenon, the highest positive and negative peak points for both 2900rpm and 2700rpm have been chosen. Head variations within the pump are shown on these operating points in figure 6.

The highest positive peak at 2900rpm occurred at 201° angular position (with reference to position shown in figure 4 taken as reference point) of the impeller, while the highest negative peak occurred at 90° angular orientation of the impeller. Similarly, in case of 2700rpm, the highest positive and negative peaks occurred at 66° and 171° angular positions.

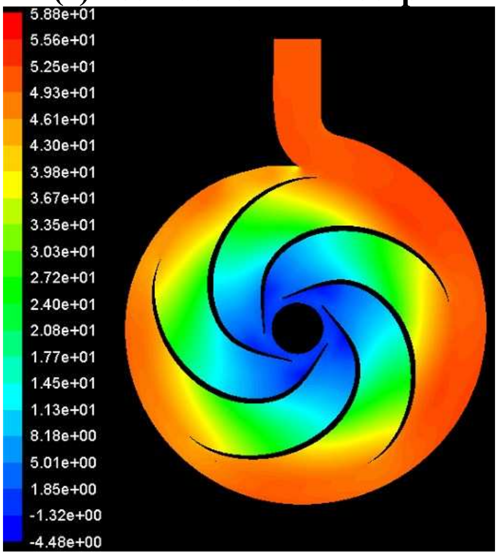
It can be clearly seen in figure 6 that the highest positive peaks in the pump's instantaneous head output occur when the tongue region is roughly in-between two impeller blades (or when one impeller blade has crossed the tongue and the other blade is approaching it). Similarly, the highest negative peaks, for both the rpms, occur when an impeller blade reaches the tongue region. The interaction between the tongue region and the impeller blades is quite complex, which is evident from the fact that all the contours presented here exhibit secondary flow features downstream the tongue region.

It can be further observed in figure 6 that there is no significant change in the pump head in the impeller region, as its speed changes. Hence, the transient effects are more pronounced in the volute of the pump, and to some extent, in the outer regions of the impeller, where blade-tips are present.

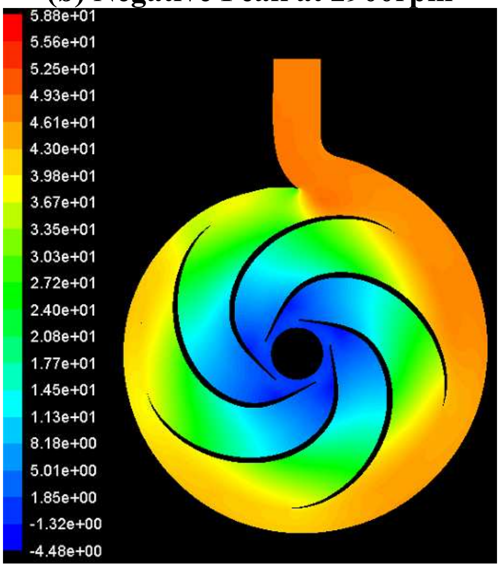
It has been stated earlier that frozen-rotor approach based pump's head predictions are in close agreement with that from sliding mesh technique. In order to analyse this, transient simulations have been run for all the operating points shown in 3, for both the rpms. Table 2 summarises the results obtained. It can be seen that the revolution averaged head is in close agreement with quasi-steady head; hence quasi-steady approach can be used to predict pump's head with reasonable accuracy, to obtain characteristic curves. However, if instantaneous information is required regarding the performance of a centrifugal pump, transient approach is normally used.



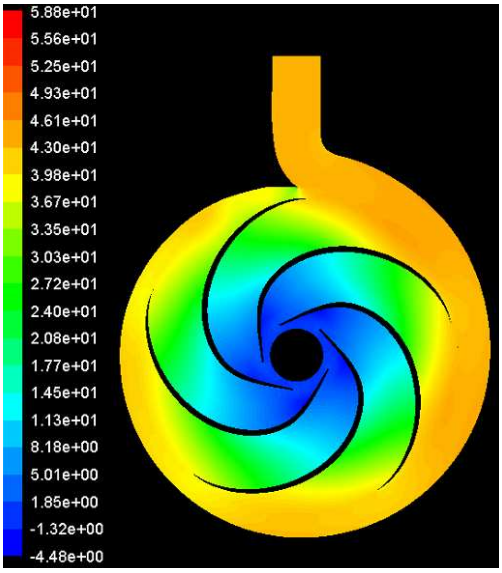
(a) Positive Peak at 2900rpm



(b) Negative Peak at 2900rpm



(c) Positive Peak at 2700rpm



(d) Negative Peak at 2700rpm

Fig. 6 Highest peak's analysis at both operating speeds of the pump

5. CONCLUSIONS

Numerical investigations on the effects of transient rotational speed of a centrifugal pump have been presented in this study. Frozen-Rotor approach has been employed to validate the CFD results, and to formulate the pump's curve at 2700rpm. Flow field analysis has been presented. Furthermore, instantaneous head waveform has been extracted when the pump's rotational speed varies from 2900rpm to 2700rpm in one revolution's time. The waveform comprises of a number of positive and negative peaks, across the average head. Furthermore, it has been observed that speed change of a pump does not show appreciable effect on the flow field in the core impeller region of the pump, and its effects can be observed more clearly in the volute region.

Table 2 Comparison between quasi-steady and transient pump's head

Impeller Speed	Q	Quasi-Steady Head	Revolution Averaged Head	Diff. between the two heads
(rpm)	(m^3/hr)	(m)	(m)	(%)
2900	6	55	57	3.6
	9	52	53	1.9
	12	48	49	2.1
	15	44	44	0.0
	18	37	37	0.0
2700	6	48	49	2.1
	9	44	45	2.3
	12	40	41	2.5
	15	36	36	0.0
	18	29	29	0.0

REFERENCES

1. B. Jafarzadeh, A. Hajari, M. M. Alishahi, and M. H. Akbari, The flow simulation of a low-specific-speed high-speed centrifugal pump, *Applied Mathematical Modelling 35* (2011) 242-249.
2. E. Palmer, R. Mishra, and J. Fieldhouse, An optimization study of a multiple-row pin-vented brake disc to promote brake cooling using computational fluid dynamics, *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering 223* (2009) 865-875.
3. V. Malviya, R. Mishra, and J. Fieldhouse, CFD investigation of a novel fuel-saving device for articulated tractor-trailer combinations, *Engineering Applications of Computational Fluid Mechanics 3* (2009) 587-607.
4. K. S. Park, T. Asim, and R. Mishra, Computational Fluid Dynamics based Fault Simulations of a Vertical Axis Wind Turbines, *Journal of Physics: Conference Series 364* (2012) 012138.
5. G. Colley, and R. Mishra, Computational flow field analysis of a Vertical Axis Wind Turbine, *International Conference on Renewable Energies and Power Quality* (2011) Las Palmas de Gran Canaria, Spain.
6. V. C. Agarwal, R. Mishra, Optimal design of a multi-stage capsule handling multi-phase pipeline, *International Journal of Pressure Vessels and Piping 75* (1998) 27-35.
7. R. Mishra, S. N. Singh, and V. Seshadri, Velocity measurement in solid–liquid flows using an impact probe, *Flow Measurement and Instrumentation 8* (1998) 157-165.
8. R. Mishra, S. N. Singh, and V. Seshadri, Study of wear characteristics and solid distribution in constant area and erosion-resistant long-radius pipe bends for the flow of multisized particulate slurries, *Wear 217* (1998) 297-306.
9. J. H. Kim, K. T. Oh, K. B. Pyun, C. K. Kim, Y. S. Choi, J. Y. Yoon, Design optimization of a centrifugal pump impeller and volute using computational fluid dynamics, *IOP Conference Series: Earth and Environmental Science 15* (2012) 032025.
10. R. Sidhesware, O. D. Hebbal, Validation Of Hydraulic Design Of A Metallic Volute Centrifugal Pump, *International Journal of Engineering Research & Technology 2* (2013).
11. S. Rajendran, and D. K. Purushothaman, Analysis of a centrifugal pump impeller using

ANSYS-CFX, *International Journal of Engineering Research & Technology 1* (2012).
12. M. G. Patel, A. V. Doshi, Effect of Impeller Blade Exit Angle on the Performance of Centrifugal Pump, *International Journal of Emerging Technology and Advanced Engineering 3* (2013) 702-706.
13. R. Muttalli, S. Agrawal, H. Warudkar, Simulation of Centrifugal Pump Impeller Using ANSYS-CFX, *International Journal of Innovative Research in Science, Engineering and Technology* (2014).
14. F. Zhang, S. Yuan, Q. Fu, F. Hong, and J. Yuan, Investigation of Transient Flow in a Centrifugal Charging Pump during Charging Operating Process, *Advances in Mechanical Engineering 6* (2014) 860257.
15. K. W. Cheah, T. S. Lee, and S. H. Winoto, Unsteady fluid flow study in a centrifugal pump by CFD method, *7th ASEAN ANSYS Conference* (2008) Singapore.
16. Pedrollo Pump F32/200AH details accessible at http://www.pedrollopumps.com/f32200hpage.html
17. Multiple Reference Frame Model details accessible at https://www.sharcnet.ca/Software/Fluent6/html/ug/node419.htm
18. Sliding Mesh Model details accessible at https://www.sharcnet.ca/Software/Fluent6/html/ug/node441.htm
17. T. Hua, L. Yi, and Z. Yu-Liang, Numerical Analysis of a Prototype Centrifugal Pump Delivering Solid-liquid Two-phase Flow, *Journal of Applied Sciences 13* (2013).
18. Y. Zhang, Y. Li, Z. Zhu, B. Cui, Computational analysis of centrifugal pump delivering solid-liquid two-phase flow during startup period, *Chinese Journal of Mechanical Engineering 27* (2014) 178-185.