

PAPER • OPEN ACCESS

Numerical investigation of the effect of leakage flow on cavitation in centrifugal pump

To cite this article: Yashwant Moganaradjou *et al* 2021 *J. Phys.: Conf. Ser.* **1909** 012016

View the [article online](#) for updates and enhancements.



The banner features a colorful striped border at the top. On the left, the ECS logo is displayed in a green circle. To its right, the text reads: "240th ECS Meeting", "Oct 10-14, 2021, Orlando, Florida", "Register early and save up to 20% on registration costs", "Early registration deadline Sep 13", and "REGISTER NOW" in orange. On the right side of the banner is a photograph of a diverse group of people in professional attire, smiling and clapping.

ECS **240th ECS Meeting**
Oct 10-14, 2021, Orlando, Florida
**Register early and save
up to 20% on registration costs**
Early registration deadline Sep 13
REGISTER NOW

Numerical investigation of the effect of leakage flow on cavitation in centrifugal pump

Yashwant Moganaradjou¹, Anindita Apurbaa Phukan², S Vengadesan^{2,3} and Dhiman Chatterjee^{1,4}

¹ Department of Mechanical Engineering, IIT Madras, Chennai 600036, India.

² Department of Applied Mechanics, IIT Madras, Chennai 600036, India.

³ Adjunct Faculty, Department of Mechanical Engineering, Virginia Tech., Blacksburg, VA 24061, USA.

⁴ dhiman@iitm.ac.in

Abstract. Numerical studies on pumps emphasize mainly on modelling the interactions between the impeller and the volute to obtain an accurate understanding of the physics involved. However, the importance of modelling leakage paths, which is known to have a significant influence on the flow structure in the pump, necessitates an in-depth analysis. This activity is undertaken in this paper by investigating a specific case of a centrifugal pump. Numerical studies have been conducted on the pump modelled with and without leakages for the design condition. The sliding mesh method is used to obtain single phase pressure pulsations data at some important locations in the volute and the leakage path, and transient Multiple Reference Frame (MRF) modelling is utilized to conduct the cavitation analysis. It is observed that for the case under study, the pressure pulsations pattern and the cavitation behaviour varies significantly due to the inclusion of leakage paths in the analysis.

1. Introduction

Numerical studies of pumps are of great help in understanding the physics involved which in-turn helps in designing highly efficient pumps. The closer the model developed for analysis is to real life, the more computationally intensive the numerical studies become. In order to obtain a trade-off, wherein the accuracy of the study is not compromised significantly along with a reasonable computation cost, several details that are deemed unnecessary for the application or purpose of the study are omitted. An aspect that has not been given due importance, while considering the analysis of pumps, is the need for modeling of the leakage circuit (sidewall clearance gaps). The absence of a physical barrier between the suction inlet and the pressure outlet of a centrifugal pump makes leakage flow unavoidable from the high pressure side to the low pressure side. Traditionally a lot of importance was given to the impeller and volute interactions, as this was observed to be the most significant phenomenon in the pump, without the leakage paths being modeled [1]. It is widely acknowledged in literature [2, 3, 4, 5], that modeling leakage paths provides better flow structure prediction as it is closer to the actual pump. The leakage in the sidewall gaps are responsible for a significant portion of the radial and hydraulic forces induced in the pump [6, 7, 8]. Modeling this

⁴ Author to whom any correspondence should be addressed



comes with the trade-off that the simulation becomes computationally intensive with an increase in the number of mesh elements and an added complexity in generating the mesh for the leakage structure. The motivation behind the present study is to address the extent of modeling of leakages required to perform a satisfactory numerical study. This is done by considering a centrifugal pump with an elaborately designed leakage circuit. Two models are created for the pump, one with the leakages modeled and the other modeled without the leakages. This corresponds to a scenario where the differences between the two geometries are quite significant and the differences in the flow physics predictions would be noticeable. This need not be the case with all pump designs. An investigation with other designs of pumps needs to be performed to ascertain the findings of the current study. The impact of leakages is expected to vary depending on the application considered; the main focus here being the flow structure, pressure pulsations and cavitation prediction. We attempt to understand the loss of information associated with neglecting the leakage paths in the analysis of the pump assembly and its impact on the design considerations.

2. Governing equations and computational methodology

For the current numerical study, the commercial code Ansys Fluent v18.1 is used. Water is considered as the working fluid. Two geometries are considered for the same configuration of the pump. The first one is modeled with the leakage paths and the second one is modeled without the leakage paths. The computational domain common to both the cases consists of an inlet pipe, an inducer, an impeller, a volute and an outlet pipe. The geometry modeled with leakages has the leakage circuit (sidewall clearance gaps from the front shroud and back shroud of the impeller) modeled in addition to the above components. The impeller has six vanes and six splitter blades to make the flow structure uniform as it exits the impeller. A block diagram depicting the components is shown in figure 1. The meshing has been done using Ansys ICFM CFD. The geometry without leakages was meshed and the same meshing scheme was utilized to mesh the geometry with leakages. This is done to retain the mesh as similar as possible between the two cases so that the results can be compared with confidence. The mesh generated is unstructured owing to the complex nature of the geometry. The geometry with leakages has about twenty million mesh elements (not shown) and the geometry without leakages has about seventeen million mesh elements and is shown in figure 2. Modeling leakages causes the number of mesh elements to increase by 14%, which directly translates to an increase in the computational time and an increase in the complexity in generating the grid.

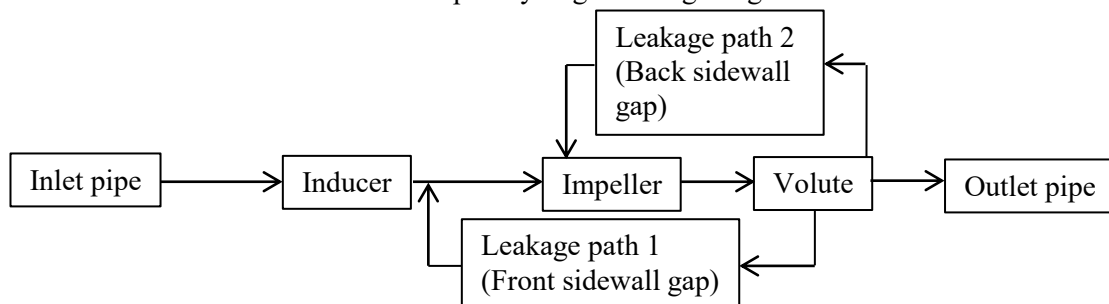


Figure 1. Block diagram showing the components of the centrifugal pump assembly

The single phase continuity and momentum equations are solved for the flow structure prediction and pressure pulsations analysis. $k-\omega$ SST turbulence model, which is known to give accurate predictions for flows involving separation and recirculation [9], is used. For the simulation of cavitation, mixture multiphase modeling is used along with the Schnerr-Sauer cavitation model [10]. The cases as mentioned in table 1 are conducted and the results are presented in the subsequent sections. Cases 1A and 1B refer to steady simulations modeled using the Multiple reference frame (MRF) approach available in Fluent. This is known to give good results for the flow structure predictions and head developed [11]. This is a steady state simulation and the mesh is fixed during this analysis. The equations are solved in a rotating reference frame by including Coriolis and centrifugal

forces in the Navier-Stokes equations for the rotating components such as the inducer and the impeller. Cases 2A and 2B refer to transient simulations modeled using the sliding mesh method available in Fluent. In this approach, the mesh rotates and hence is more accurate than the other available methods. This approach gives satisfactory predictions of unsteady pressure pulsations and is the most accurate method for unsteady analysis of turbomachines [2]. The final set of cases 3A and 3B are modeled using the transient MRF approach for cavitation prediction, which is known to produce satisfactory results [12]. For the transient simulations in the current study, the time step used is such that for a single time step, the rotating components complete a single degree of rotation. The same time step has been used for all the 4 transient cases. The boundary conditions used for the simulations are that of total pressure at inlet and mass flow rate at outlet. The details of the design are not given as this study requires analysis from other centrifugal pump configurations and designs in order to rule out the possibility that the results and the associated trends are dependent on the specific design parameters. It is well understood that the pressure pulsations, cavitation and flow structure are dependent on a lot of design parameters but we intend to conduct further analysis to isolate the general impact on the loss of information or accuracy associated with not modeling leakages. Hence, the current study is only to be considered as a preliminary analysis upon which further research would be done.

Table 1. List of numerical studies conducted

S. No	A. Geometry with leakages	B. Geometry without leakages
1	Single phase – Steady MRF	Single phase – Steady MRF
2	Single phase – unsteady (Sliding mesh)	Single phase – unsteady (Sliding mesh)
3	Multiphase – unsteady (Transient MRF)	Multiphase – unsteady (Transient MRF)



Figure 2. Computational domain of centrifugal pump assembly modelled without leakages

3. Results and Discussion

3.1. Cases 1A and 1B

The head developed in case 1B is greater than that developed in case 1A by about 5% approximately. The head loss in case 1A is expected to occur due to the presence of volumetric leakages. The

difference of 5% is not that significant and the head developed can be predicted even without the modeling of leakages [13]. The difference between the two cases might widen for off-design conditions and it needs to be studied further. The velocity streamline contours of the impeller and inducer to understand the flow structure are shown in figures 3 and 4 respectively.

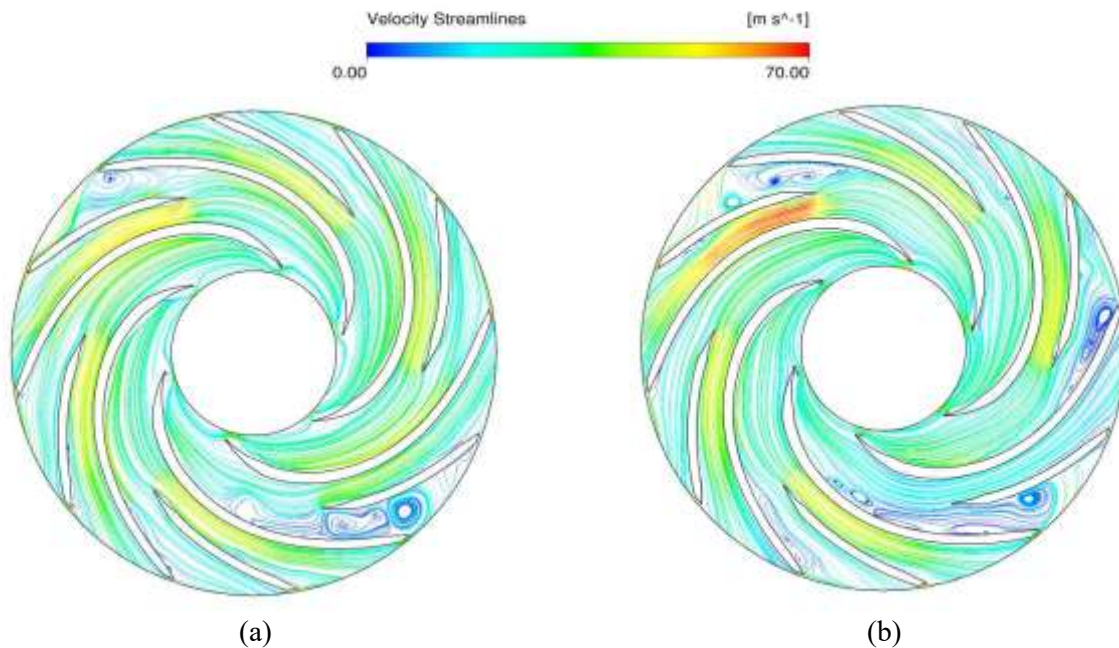


Figure 3. Velocity streamlines contour in relative reference frame for the impeller at the mid-plane (a) with leakages case 1A and (b) without leakages case 1B

The contours in figure 3 are obtained by passing a plane through the mid of the impeller. There are a few impeller passages where flow recirculation is observed in both cases. The phenomenon is more pronounced in case 1B than in case 1A. This could very well be attributed due to the presence of the leakage circuit in case 1A. The fluid from the impeller enters the volute and a fraction of the fluid enters the leakage path 1. The leakage path could be considered as a secondary circuit. The fluid from the leakage path re-enters the primary circuit at the inducer outlet. This causes an increase in the amount of fluid handled by the impeller. In this particular scenario, the case 1A handles about $1.06Q$ quantity of fluid whereas the case 1B handles Q quantity of fluid which corresponds to the design criteria. This 6% increase in mass flow rate makes the velocity streamlines more uniform as compared to that of case 1B, as it is well known that for an impeller the flow structure becomes uniform with an increase in the flow rate [14]. The flow recirculation observed in case 1A could be attributed to the presence of the volute tongue and the splitter, which are designed to be at an angle of roughly 180 degrees apart, in the vicinity of the impeller passages.

From figure 4, it is observed that the flow recirculation is stronger in case 1B. This might also be the reason why, as seen in section 3.3, vapor generation due to cavitation is more pronounced in the geometry without leakages. On analyzing the flow structure through the volute, it is seen that the flow pattern is almost the same for the two geometries. It was expected that the presence of the leakage circuit which removes a fraction of the fluid from the volute would cause strong vortices to be developed for case 1A as compared to case 1B due to the viscosity of the fluid. But it was seen that the vortices developed in case 1B was relatively stronger as compared to that of case 1A. A general trend that is observed is that the flow recirculation is stronger in case 1B for all the components.

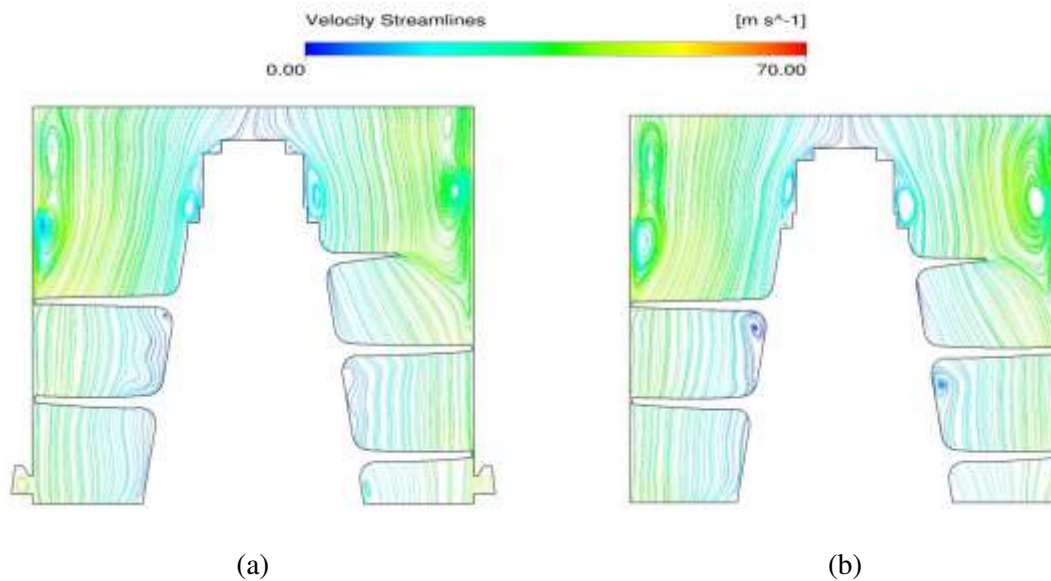


Figure 4. Velocity streamlines contour in relative reference frame for the inducer at the mid-plane (a) with leakages case 1A and (b) without leakages case 1B

3.2. Cases 2A and 2B

It is common to non-dimensionalize the pressure pulsations data [15] so as to be able to make a comparison between data of different size and speed.

$$\Delta p^* = \frac{p - p_{mean}}{0.5 * \rho * U_2^2} \quad (1)$$

In equation (1), p represents the instantaneous value of pressure, p_{mean} denotes the mean pressure over the cycle, ρ denotes the fluid density and U_2 denotes the circumferential velocity at the impeller outlet. The pressure pulsations data presented from figures 5 to 7 follow the notation that the impeller splitter blade aligning with the cutwater corresponds to zero degrees and the data are presented for one cycle of operation or one rotation of the impeller. Points were created at several locations in the geometry to monitor the time-dependent variation of pressure and the results at three locations namely at the volute discharge, near the volute tongue and at the leakage path are given here. It is observed that case 2A has six sharp peaks and other six peaks of lower magnitude. In case 2B, the pressure pulsations do not have such sharp peaks and the pressure peaks appear corresponding to the six vanes and the six splitter blades of the impeller with a much lower magnitude in comparison with the case 2A. The plots for case 2A and case 2B do not have identical co-ordinates for the Non-dimensionalized Pressure pulsations due to the drastic differences in the magnitude of the Pulsations. Case 2A experiences pressure pulsations about 17 times the magnitude for the peaks and about 2.5 times the magnitude for the remaining data as compared to case 2B. In either case, modeling the leakages predict an increased level of pressure pulsations. The peaks for the case 2B are of comparable magnitude with the mean unlike case 2A. It could be speculated that the presence of the leakage structure could have acted as a secondary circuit that interacted with the primary circuit, causing an increase in the magnitude of the pressure pulsations [15]. We expected to see only a marginal increase or decrease in the magnitude of the pressure pulsations with leakage modeling. The modeling of the leakages appears to have caused the pressure pulsations to rise sharply which could be detrimental to the integrity of the mechanical structure itself. Further analysis of the unsteady flow from the impeller outlet is necessary to exactly understand the significance and the difference in the pressure pulsation patterns of the two cases. It is interesting to note that the steep pressure rises are found in the leakage

paths also as can be seen from figure 7. The analysis of pumps of different design is necessary to rule out if such trends are case specific only. There is reason to believe that the present results are case specific as the results of Spence et al. [2] did not observe such steep peaks in pressure pulsations near the volute tongue, where the pressure pulsations are expected to be the maximum, despite modeling the leakage paths. The absence of experimental data for the pressure pulsations is also a shortcoming in the current scenario. It can be recommended at this stage to conduct pressure pulsations analysis while studying the effect of the variation of clearances of the pump geometry in addition to the parameters studied in literature [3, 5].

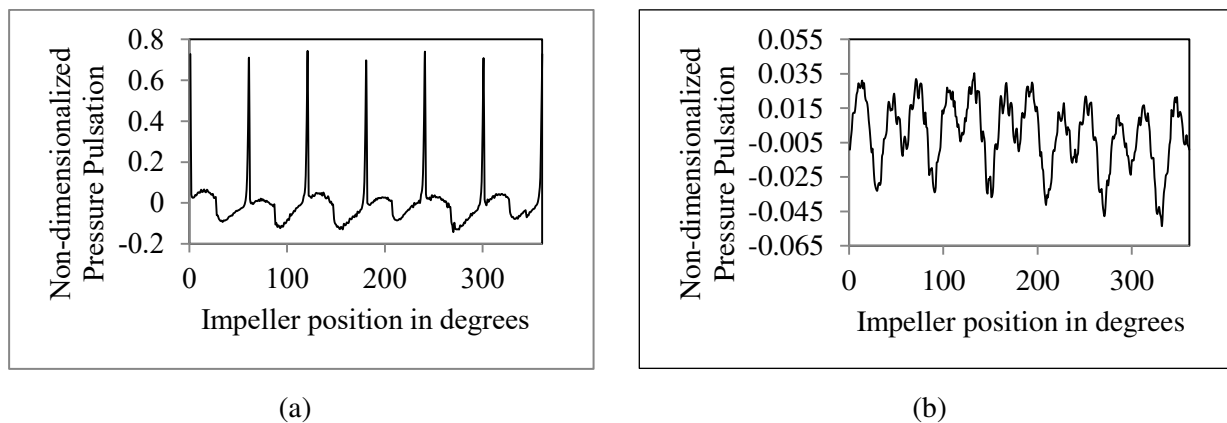


Figure 5. Pressure pulsations plot at a monitor point in the discharge of the volute for (a) case 2A and (b) case 2B

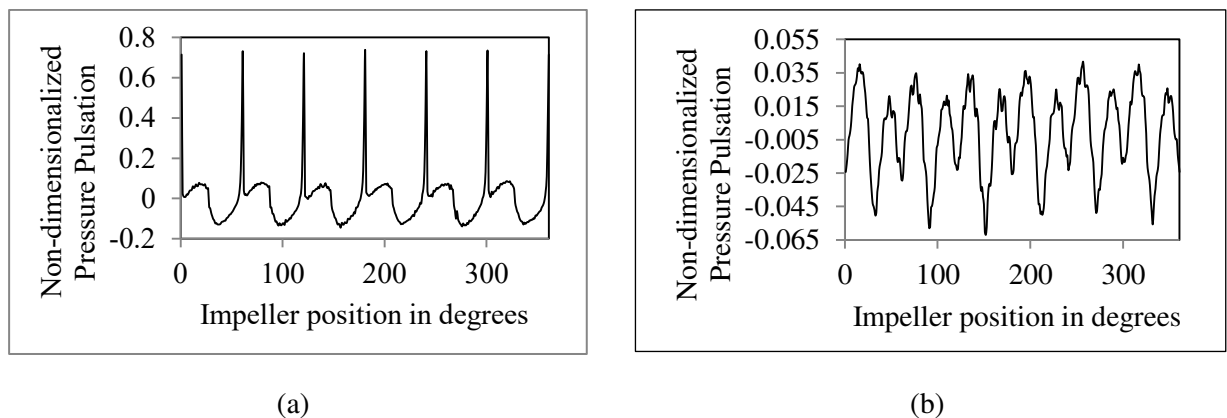


Figure 6. Pressure pulsations plot at a monitor point near the tongue of the volute for (a) case 2A and (b) case 2B

3.3. Case 3A and 3B

It is observed that the extent of cavitation is lesser for case 3A as compared with the case 3B. From figure 8, it is seen that for case 3B, the cavitation developed in the inducer exhibits the following characteristics [16, 17]: Leakage vortex cavitation developing towards the upstream of the inducer and cavitation attached to the suction and pressure side of the blade for the first half of the geometry. For case 3A, the inducer exhibits the same cavitation characteristics as above except cavitation on the pressure side of the blade which is found to be absent. It is to be noted that the tip leakage clearance is the same for both the configurations and hence the type of cavity observed is similar.

The volume of cavity at a particular time step was calculated using the following formulation:

$$V_{cavity} = \sum \alpha_v V_{cell} \quad (2)$$

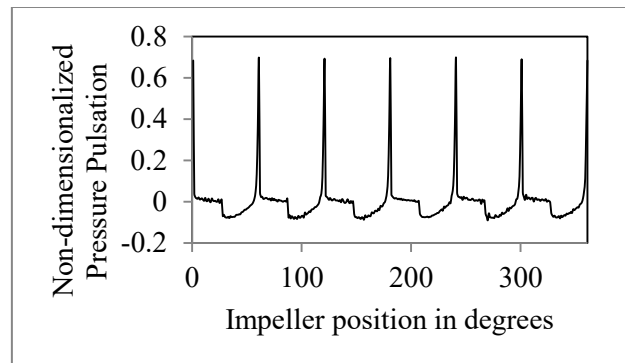


Figure 7. Pressure pulsations plot at a monitor point in the leakage path for case 2A

In equation (2), V_{cavity} is the volume of the cavity, α_v is the vapor fraction in each cell and V_{cell} is the volume of each cell. The summation is over the entire domain to take into account the vapor generated due to cavitation. In order to effectively quantify the difference in the cavity volume predictions, a plot showing the variation of cavity volume with inducer rotation is made. It is observed to exhibit a periodic behavior. From figure 9, it is concluded that the volume of vapor produced in the geometry with leakages is lesser than the volume of vapor produced in the geometry without leakages. This corroborates with the flow structure predictions of the inducer from figure 4. It can be observed that the cavity volume for case 3B takes about 2.3 rotations of the inducer per cycle whereas the cavity volume cycle duration for case 3A is about 2 rotations of the inducer. In this case also, further analysis is required to understand the observed phenomenon.



Figure 8. Cavity volume generated (time-averaged) in the inducer for (a) case 3A and (b) case 3B

Figure 9 has been non-dimensionalized by dividing the instantaneous value of the cavity volume by the maximum value of the cavity volume obtained in case 3B.

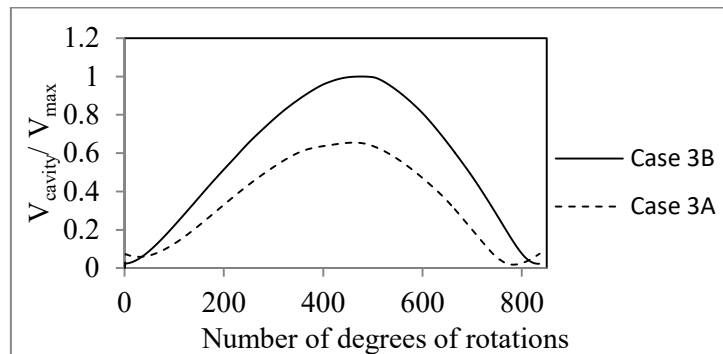


Figure 9. Plot comparing the volume of cavity generated as a function of rotations of the inducer

4. Conclusion

A specific case of a centrifugal pump was chosen and two geometries were created, one with the leakage paths modeled and the other excluding the leakage paths. They were analyzed to obtain the flow structure, pressure pulsations at certain points and cavitation characteristics. The results pointed to significant differences between the two simulations. The pressure pulsations seemed to have increased by several orders of magnitude and the vapor generated was found to be lesser in the case of the geometry with leakages. Since a lot of design parameters influence these predictions, further analysis with several different pump designs is needed to rule out any case-specific trends. We intend to bring out the significance of modeling leakages more quantitatively and the related loss or discrepancy in the predictions obtained with future studies.

References

- [1] González J, Fernandez J, Blanco E and Santolaria C 2002 Numerical simulation of the dynamic effects due to impeller–volute interaction in a centrifugal pump *ASME J Fluids Eng* **124** 348–355
- [2] Spence R and Teixeira J A 2008 Investigation into pressure pulsations in a centrifugal pump by numerical and experimental methods *J. Computer Fluids* **2008** 37 690–704
- [3] Liu H, Ding J, Dai H and Tan M 2014 Investigation into transient flow in a centrifugal pump with wear ring clearance variation *Advances in Mechanical Engineering* **Vol. 2014** 15p
- [4] Chen S X, Pan Z Y, Wu Y L and Zhang D Q 2012 Simulation and experiment of the effect of clearance of impeller wear-rings on the performance of centrifugal pump *Proceedings of the IOP Conference Series: Earth and Environmental Science* **Vol. 15**
- [5] Cao L, Wang Z W, Xiao Y X and Luo Y Y 2016 Numerical investigation of pressure fluctuation characteristics in a centrifugal pump with variable axial clearance *International Journal of Rotating Machinery* **Vol. 2016** 11p
- [6] Brennen C E 2011 *Hydrodynamics of Pumps* Cambridge University Press, Cambridge, UK
- [7] Uy R V and Brennen C E 1999 Experimental measurements of rotordynamic forces caused by front shroud pump leakage *Journal of Fluids Engineering* **Vol. 121** 633–637
- [8] Adkins D R and Brennen C E 1988 Analysis of hydrodynamic radial forces on centrifugal pump impellers *Journal of Fluids Engineering* **Vol. 110** 20–28
- [9] Murugesan C and Rudramoorthy R 2016 Numerical and experimental study of single stage and multistage centrifugal mixed flow submersible borewell pumps *International Journal of Fluid Machinery and Systems* **Vol. 9** 107–118
- [10] Schnerr G H and Sauer J 2001 Physical and numerical modelling of unsteady cavitation dynamics *Fourth International Conference on Multiphase Flow, ICMF New Orleans*
- [11] Dick E, Vierendeels J, Serbrugyns A and Voorde J V 2001 Performance prediction of centrifugal pumps with CFD-tools *Task Q 5 No 4* 579–594

- [12] Pei J, Osman M K , Wang W , Appiah D, Yin T and Deng Q 2019 A Practical Method for Speeding up the Cavitation Prediction in an Industrial Double-Suction Centrifugal Pump *Energies* **12** 2088
- [13] Liu H and Pan H C 2012 The Influence of Turbulence Model Selection and Leakage considerations on CFD Simulation Results for a Centrifugal Pump *Advanced Materials Research* **Vols 594-597** 1940-1944
- [14] Cheah K W, Lee T S, Winoto S H and Zhao Z M 2007 Numerical flow simulation in a centrifugal pump at Design and off-Design conditions *International Journal of Rotating Machinery* **Vol. 2007** 8p (8)Adkins D R and Brennen C E 1988 Analysis of hydrodynamic radial forces on centrifugal pump impellers *Journal of Fluids Engineering* **Vol. 110** 20-28
- [15] Guelich J F and Bolleter U 1992 Pressure pulsations in centrifugal pumps *ASME J. Vibrat. Acoust.* **114** 272–279
- [16] Bakir F, Rey R, Gerber A G, Belamri T and Hutchinson B 2004 Numerical and Experimental Investigations of the Cavitating Behavior of an Inducer *International Journal of Rotating Machinery* **10** 15-25
- [17] Zhang D, Shi L and Shi W 2015 Numerical analysis of unsteady tip leakage vortex cavitation cloud and unstable suction-side-perpendicular cavitating vortices in an axial flow pump *International Journal of Multiphase Flow* **77** 244-259