SETSCI Conference Indexing System,
Volume 2 (2018), 417-421

ISAS 2018
I[st] International Symposium on Innovative Approaches in Scientific Studies
April 11-13, 2018, Kemer-Antalya, Turkey

Numerical Simulation of A Cavitating Centrifugal Pump

Aydın Hacı Dönmez[1*], Zehra Yumurtacı[1] and Levent Kavurmacıoğlu[2]

[1]*Mechanical Engineering Department, Yildiz Technical University, Istanbul, Turkey*
[2]*Mechanical Engineering Department, Istanbul Technical University, Istanbul, Turkey*
[*]*Corresponding author: adonmez@yildiz.edu.tr*
[+]*Speaker: adonmez@yildiz.edu.tr*
Presentation/Paper Type: Oral / Full Paper

Abstract – Current study is related with numerical simulation of a centrifugal pump impeller with the specific speed of n_q=13. By the help of computational fluid dynamics (CFD) at different operating points, pump performance rsults (Head-Discharge graph) are obtained. Simulations were in good agreement with experimental results. Afterwards, two phase CFD study is carried out in order to examine the cavitation performance of the pump. According to the numerical cavitation tests conducted in selected flow rates by decreasing the inlet pressure gradually; required net positive suction ($NPSH_R$) curve of the pump is obtained. During meshing procedure, single blade passage is considered and tetrahedrons are used. On the other hand, while meshing the vaneless diffuser, hexahedron elements are used. Finer grids are adopted on solid surfaces for a detailed flow observation. Shear Stress Transport (SST) turbulence model is used throughout the study and Rayleigh-Plesset cavitation model is applied in two phase simulations.

Keywords – Centrifugal pump, impeller, cavitation, CFD, turbulence, two phase

I. INTRODUCTION

In any fluid flow process, if the local pressure becomes equal or less than the vapour pressure of the working fluid, vaporization occurs. Afterwards, water vapour bubbles forms and collapse when the local pressure increases again. During this vaporization and condensation cycle pressure fluctuations leads to noise, vibration and wear in the end. This situation is a complex phenomenon with phase changes and defined as cavitation [1-3].

Yokoyama's study is one of the first studies related with cavitation performance of a centrifugal pump. He conducted experiments on an impeller with a sharp, chamfered and blunt vane in order to observe the effect of inlet vane profile on cavitation. As a result, it is seen that as the inlet profile become thinner, better cavitation performance is achieved [4]. Luo et al. improved a centrifugal pump impeller designed by a conventional method. Based on the designed impeller, five new designs are obtained and their cavitation performances are compared with a CFD study. Eventually, the design with both extending leading edge to the suction side and increased inlet blade angle gave the best results for both hydraulic and cavitation performance enhancement [5]. Christopher and Kumaraswamy performed experiments for three different inlet vane profiles: blunt, elliptic and circular in order to compare the cavitation performance of the pumps. Consequently, they showed that as the inlet profile become rounder, cavitation performance of the pump increased [6]. Bonauiti et al. conducted both an experimental and numerical study on a mixed flow pump impeller for a better pump in terms of cavitation and efficiency. Hydraulic variables are used for parameterization of the blade profile in order to achieve both higher suction and efficiency performance increase. Furthermore, the numerical results of the final design is validated by experiments [7]. Liu et al. compared two impellers for a mini turbo-pump with different specifications designed based on different methods, experimentally. As a result, they showed that the increase of outlet blade angle and vane number gave better results in the point of cavitation and hydraulic performance [8]. Caridad et al. conducted two phase CFD studies for a centrifugal pump impeller with varying gas void fractions and compared pump characteristic curves. Also, a sensitivity analysis is carried out according to the gas bubble sizes. The numerical results were in great agreement with the experiments [9]. Zhang et al. performed an experimental and numerical study on the first stage of a centrifugal charging pump which are used in nuclear applications. Cavitation performance tests are actualized at four different flow rates and pump performance and $NPSH_R$ curves are obtained. They also found out that, pressure fluctuations are highly affected from the water vapour formation, growth and collapse [10]. Medvits et al. analysed the performance of a centrifugal pump under cavitating conditions. For this purpose, a homogenous two phase CFD study is conducted and results are compared with the experiments. According to the results, performance curves (head coefficient-flow coefficient and head coefficient-cavitation number) are merged at off-design conditions [11].

This study consists of numerical simulation of a commercial centrifugal pump. Accordingly, CFD simulations are performed for varying flow rates according to the data taken from the company and head discharge pump characteristic curve of the pump obtained numerically. The results were in good agreement with the experimental data. Moreover, a two phase CFD study is conducted in order to obtain the cavitation performance of the pump. Suction performance of the pump is determined for specified flow rates by performing numerical cavitation test. Finally, required net positive suction head ($NPSH_R$) values are calculated according to the results of the numerical cavitation tests. Experimental and numerical results

of the cavitation tests are also compared. All numerical procedure is carried out via Ansys 16.0.

II. Grid Generation and Mesh Independency

A commercial centrifugal pump with total head (H) 90.55 m and 59.9 m3/h discharge is considered rotating at 2975 rpm on its best efficiency point. The impeller consists of 5 vanes and tip diameter is 132 mm. Shaft power is 24.21 kW and required net positive suction head is 5.36 m. Also, the efficiency is 61 % at its best efficiency point. Pump impeller specifications are tabulated in Table 1. All values are taken from the company's test results.

Table 1. Pump specifications

Pump Specifications	Value
Head [m]	90.55
Discharge [m^3/h]	59.9
Rotational speed [rpm]	2975.2
Specific speed	13
Number of vanes	5
Tip diameter [mm]	132
Shaft power [kW]	24.21
Efficiency	0.61
$NPSH_R$ [m]	5.36

Mesh independency process is essential to be carried out in CFD studies. As it is known, the more the computational domain is divided into smaller regions, the more accurate results will be actualized. However, overmuch number of elements will bring about extra time and computational costs. So, grid independent solution with minimum time and computational cost must be achieved together. Six different cases with increasing grid number are demonstrated in Figure 1, which shows the variation of head against total mesh number in the computational domain.

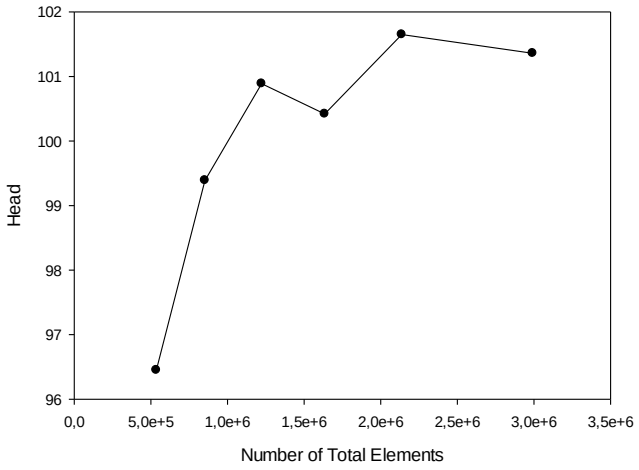


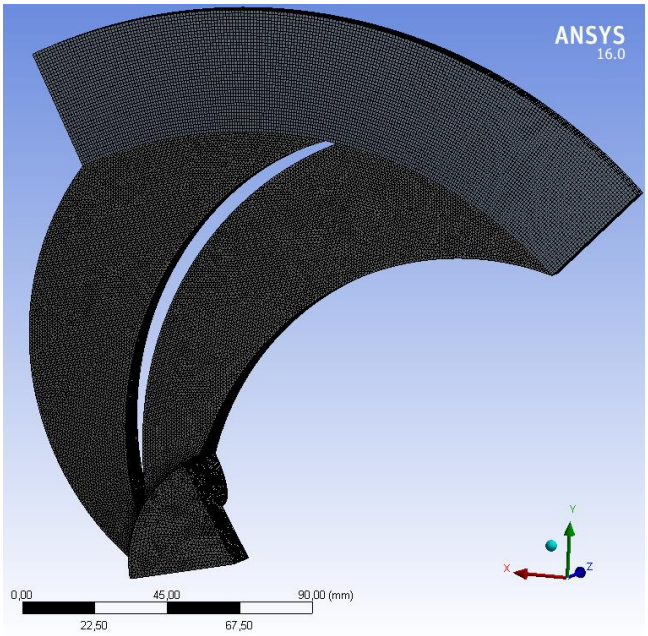
Fig. 1 Variation of head against mesh number

Figure 1 shows that, as the mesh number increases, the head value converges to a value around 101 m. From that point, there is no remarkable change in pump head. Accordingly, scheme 1 and 2 are inappropriate for numerical simulations. On the other hand, case 3 will ensure sufficient results for the computational domain. Detailed values of demonstrated mesh schemes are tabulated in Table 2 including mesh sizing and total number of elements.

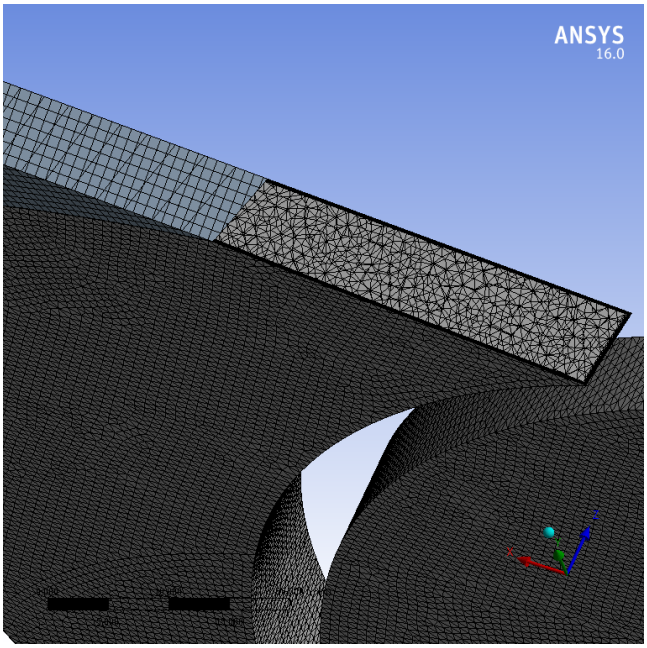
Table 2. Mesh independency study

Case	Sizing [mm]	Num. of El.	Head [m]
1	1.5	535350	96.45
2	1.2	852408	99.39
3	1	1222592	100.89
4	0.8	1634198	100.42
5	0.7	2137597	101.65
6	0.6	2991541	101.36

Accordingly, case 3 with 1 mm face sizing resulted in 1222592 total mesh number is selected in order to carry out the simulations. Mesh details can be seen from Figure 2 below:



(a)



(b)

Fig. 2 Mesh details a) single passage, b) boundary layer over solid surfaces and hexahedral grid on vaneless diffuser

III. Numerical Algorithm

In this section, numerical procedure is explained. In order to conduct a CFD study, a turbulence model and proper boundary conditions must be designated. Moreover, a cavitation model must be introduced for two phase flow analysis.

A. Turbulence Model

All simulations are carried out by Shear Stress Transport (SST) turbulence model. SST turbulence model uses k-ω turbulence model in the boundary layer, which is appropriate for boundary layer simulations. On the other hand, it employs k-ε turbulence model in the free stream. Governing equations of SST turbulence model are [12]:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho U_i k)}{\partial x_i}=\tilde{P}_k-\beta^*\rho k\omega+\frac{\partial}{\partial x_i}\left[(\mu+\sigma_k\mu_t)\frac{\partial k}{\partial x_i}\right] \quad (1)$$

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial(\rho U_i k)}{\partial x_i}=\alpha\rho S^2-\beta\rho\omega^2+\frac{\partial}{\partial x_i}\left[(\mu+\sigma_\omega\mu_t)\frac{\partial\omega}{\partial x_i}\right]+2(1-F_1)\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial\omega}{\partial x_i} \quad (2)$$

where,

$$F_1=\tanh\left\{\left\{\min\left[\max\left(\frac{k^{1/2}}{\beta^*\omega y},\frac{500\nu}{y^2\omega}\right),\frac{4\rho\sigma_{\omega 2}k}{CD_{k\omega}y^2}\right]\right\}^4\right\} \quad (3)$$

$$CD_{k\omega}=\max\left(2\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial\omega}{\partial x_i},10^{-10}\right) \quad (4)$$

$$\nu_t=\frac{a_1 k}{\max(a_1\omega,SF_2)} \quad (5)$$

$$F_2=\tanh\left[\left[\max\left(\frac{2k^{1/2}}{\beta^*\omega y},\frac{500\nu}{y^2\omega}\right)\right]^2\right] \quad (6)$$

$$P_k=\mu_t\frac{\partial U_i}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\rightarrow\tilde{P}_k=\min(P_k,10\cdot\beta^*\rho k\omega) \quad (7)$$

In the equations, F_1 and F_2 are blending functions. S represents the invariant measure of the strain rate, ρ density, $\nu_t=\mu_t/\rho$ turbulent kinematic viscosity, μ molecular dynamic viscosity and y the interval to the closest wall, respectively.

The values of the constants are:

$\alpha_1 = 5/9$, $\alpha_2 = 0.44$, $\beta_1 = 3/40$, $\beta_2 = 0.0828$, $\beta^* = 0.09$, $\sigma_{k1} = 0.85$, $\sigma_{k2} = 1$, $\sigma_{\omega 1} = 0.5$, $\sigma_{\omega 2} = 0.856$

B. Cavitation Model

During two phase simulations, Rayleigh-Plesset cavitation model is used. This model considers bubble dynamics and assumes that cavitation occurs by formation of nuclei, bubble growth and collapse. Governing equation of the cavitation model is presented [13].

$$\frac{p_B(t)-p_\infty(t)}{\rho_L}=R_b\frac{\partial^2 R_b}{\partial t^2}+\frac{3}{2}\left(\frac{\partial R_b}{\partial t}\right)^2+\frac{4\nu_L}{R_b}\frac{dR_b}{dt}+\frac{2\sigma}{\rho_L R_b} \quad (8)$$

In the equation, p_B represents pressure of the bubble surface, p_∞ ambient pressure, ρ_L density of liquid, ν_L kinematic viscosity of liquid, R_b radius of spherical bubble and σ surface tension, respectively.

C. Boundary Conditions

The simulation is conducted by a single blade passage. Wall boundary condition is assigned for solid surfaces like hub, shroud and blade.

On the other hand, atmosphere pressure boundary condition is used at inlet. For outlet, mass flow outlet of a single passage is determined. For two phase flows, it is assumed that 100 % of the fluid entering the pump is liquid. Phase changes are showed up according to the pressure drop at the impeller.

IV. Results and Discussion

In order to compare the experimental and numerical results, pump characteristic curves are considered at first. Afterwards, two phase CFD simulations are carried out to compare the suction performance of the pump. In other words, cavitation test is conducted via finite element approach.

A. Comparison of Pump Characteristic Curves

Pump characteristic curve is determined by conducting experiments on varying discharges. Pressure difference between inlet and outlet is considered as head at a certain flow rate. When these points are combined, pump characteristic curve is formed.

Experimental results are taken from the company and numerical simulations are performed by CFD. Same procedure in the experiment is implemented. Variation of head is calculated with respect to discharge in each simulation. The results are demonstrated in Figure 3.

As it is seen from the figure, the numerical results are in good agreement with the experimental results especially at higher flow rates. As the flow rate increased, the error is increased. This situation can be caused from the reversed flows causing cavitation. It should not be forgotten that, while performing these simulations water is specified as working fluid. Two phase simulations are carried out in numerical cavitation test analysis.

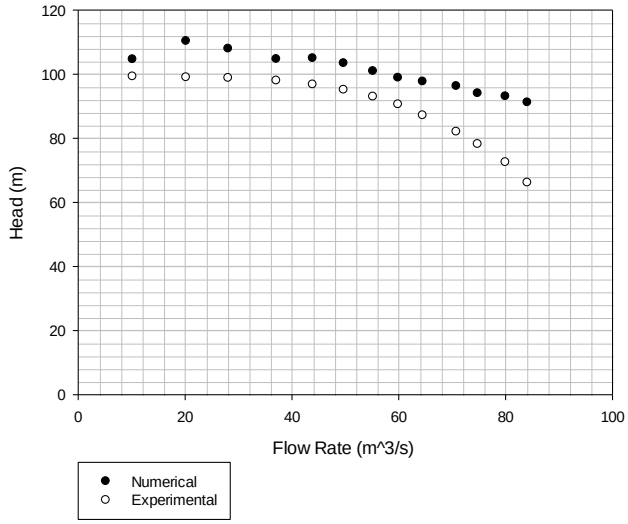


Fig. 3 Comparison of experimental and numerical pump characteristic curves

B. *Two-phase Cavitation Simulations*

In this section, two phase simulations are performed in order to have a knowledge on cavitation performance of the pump. In other words, cavitation test is conducted numerically. For this purpose, inlet pressure is decreased step by step at a certain discharge and the head drop point is determined. This test is conducted at four different flow rates. The results are showed in Figure 4.

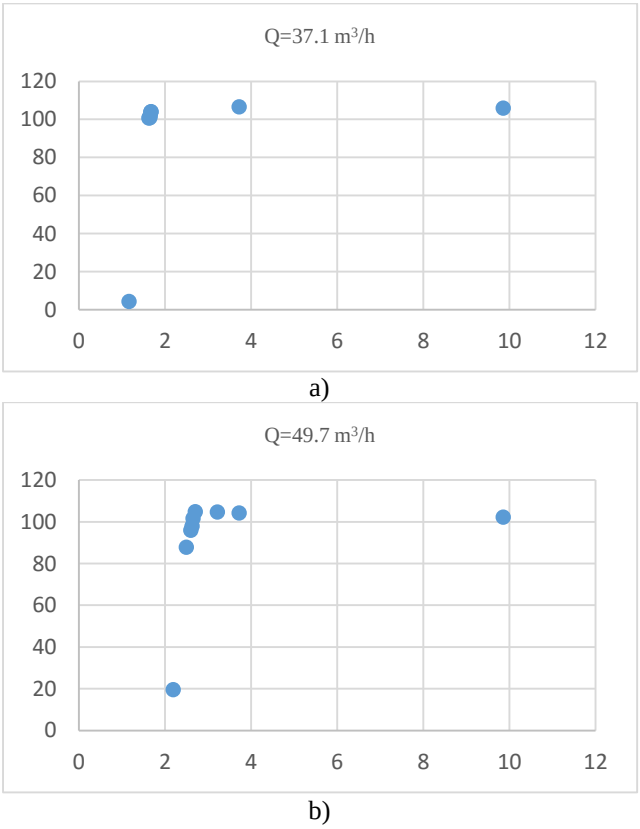


Fig. 4 Cavitation tests conducted numerically for different flow rates a) 37.1 m^3/h, b) 49.7 m^3/h, c) 59.9 m^3/h, d) 74.8 m^3/h

C. *Comparison of Required Net Positive Suction Head Curves*

In order to prevent pump from cavitation, it is essential to obtain the point where head drop is occurred. With this data, required net positive suction head of the pump is calculated. $NPSH_R$ can be defined as total head of the fluid at pump inlet. In practice, it is difficult to determine the point of head drop. So, the pressure where 3 % head drop occurrence is considered as critic pressure.

$$NPSH_R = \frac{P_c - P_v}{\rho g} \tag{5}$$

Here P_c is the pressure where 3 % head drop occurred (critic pressure), P_v is water vapour pressure of the working fluid, ρ is the density of the working fluid and g is the gravitational acceleration. Comparison of experimental and numerical $NPSH_R$ against flow rate is presented in Figure 5 below.

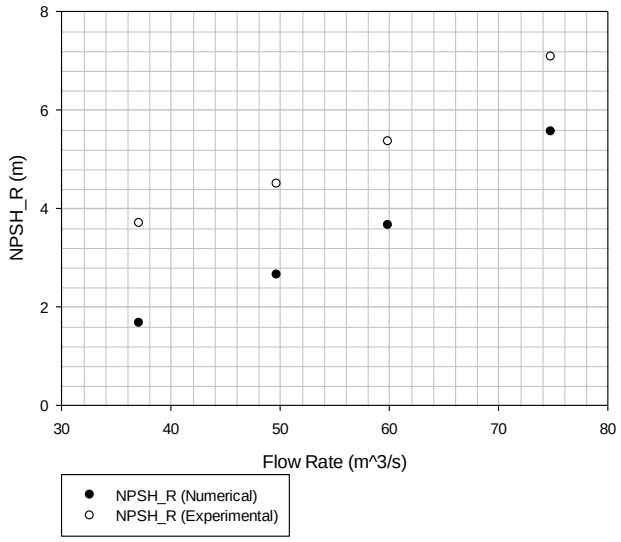


Fig. 5 Comparison of experimental and numerical required net positive suction head curves

V. CONCLUSION

This study is related with numerical modelling of a cavitating pump. In order to figure out this work, CFD calculations are conducted. Accordingly, experimental and numerical pump characteristic and required net positive suction head curves are compared. The main findings and recommendations are:

(1) The numerical results are good agreement with the experiments in pump characteristic curves. At best efficiency point, the error is 9.1 %. Excluding the off-design conditions, the error rate is between 5.3-12 % for varying flow rates.

(2) Two phase cavitation test are conducted at four different flow rates because there were four sets of experimental data. Experimental and numerical $NPSH_R$ curves are somewhat different but the tendencies are similar. Also, the error rate decreases as the flow rate increases.

(3) The study will continue by changing inlet and outlet blade angles, inlet vane profile and wrap angle; which are important parameters for cavitation performance of a centrifugal pump. This will increase the cavitation resistance of the pump.

ACKNOWLEDGMENT

The authors acknowledge the financial support of Yildiz Technical University Scientific Research Projects Coordinatorship (2014-06-01-DOP01) and Teknopark.

REFERENCES

[1] M. Hofmann, B. Stoffel, J. Friedrichs, and G. Kosyna. (2001) "Similarities and Geometrical Effects on Rotating Cavitation in Two Scaled Centrifugal Pumps," [Online]. Available: http://caltechconf.library.caltech.edu/76/1/paper_b8_001.pdf.

[2] Y. Iga, M. Nohml, A. Goto, and T. Ikohaji, "Numerical Analysis of Cavitation Instabilities Arising in the Three-Blade Cascade," *Journal of Fluids Engineering*, vol. 126, pp. 419-429, 2004.

[3] S. D. Kyparissis, and D. P. Margaris, 2012, "Experimental Investigation and Passive Flow Control of a Cavitating Centrifugal Pump," *International Journal of Rotating Machinery*, vol. 2012, Article ID: 248082, 8 pages, 2012.

[4] S. Yokoyama, "Effect of the Tip Shape at Entrance of the Impeller Vane of the Centrifugal Pump on Cavitation," *Bulletin of the Japan Society of Mechanical Engineers*, vol. 3(11), pp. 326-332, 1960.

[5] X. Luo, Y. Zhang, J. Peng, H. Xu, and W. Yu, "Impeller inlet geometry effect on performance improvement for centrifugal pumps,"*Journal of Mechanical Science and Technology*, vol. 22(10), pp. 1971-1976, 2008.

[6] S. Christopher, and S. Kumaraswamy, "Identification of Critical Net Positive Suction Head from Noise and Vibration in a Radial Flow Pump for Different Leading Edge Profiles of the Vane," *Journal of Fluids Engineering*, vol. 135, pp. 121301-1 - 121301-15, 2013.

[7] D. Bonauiti, M. Zangeneh, and R. A. J. Erikson, "Parametric Design of a Waterjet Pump by Means of Inverse Design, CFD Calculations and Experimental Analysis," Journal of Fluids Engineering, *Journal of Fluids Engineering*, vol. 132(3), pp. 031104-1 - 031104-15, 2010.

[8] S. Liu, M. Nishi, and K. Yoshida, "Impeller Geometry Suitable for Mini Turbo-Pump," *Journal of Fluids Engineering*, vol. 123, pp. 500-506, 2001.

[9] J. Caridad, F. Kenyery, A. Tremante, and O. Aguillon, "Characterization of a centrifugal pump impeller under two-phase flow conditions," *Journal of Petroleum Science and Engineering*, vol. 63(1-4), pp. 18-22 ,2008

[10] F. Zhang, S. Yuan, Q. Fu, J. Pei, M. Böhle, and X. Jiang, "Cavitation-Induced Unsteady Flow Characteristics in the First Stage of a Centrifugal Charging Pump," *Journal of Fluids Engineering*, vol. 138, pp. 011303-1 – 011303-13, 2017.

[11] R. B. Medvitz, R. F. Kunz, D. A. Boger, J. W. Lindau, and A. M. Yucom, "Performance Analysis of Cavitating Flow in Centrifugal Pumps Using Multiphase CFD" *Journal of Fluids Engineering*, vol. 124, pp. 377-383, 2002.

[12] F. R. Menter, M. Kuntz, and R. Langtry, "Ten years of industrial experience with the SST turbulence model," *Proc. 4th International Symposium on Turbulence, Heat and Mass Transfer*, 2003.

[13] C. E. Brennen, *Cavitation and Bubble Dynamics*, Oxford University Press, New York, US, 1995.