OPEN ACCESS Freely available online

Journal of Thermodynamics & Catalysis

Research Article

A Systematic Review on Turbo-Machinery Cavitation Analysis and Thermodynamic Effects

Omer Haitham Kanam[1], Maher Obaid Ahmed[2], Qasim Yahya Rahawi[1]

[1]*Department of Mining Engineering, College of Petroleum and Mining Engineering, University of Mosul 41002, Iraq*

[2]*Department of Petroleum and Refining Engineering, College of Petroleum and Mining Engineering, University of Mosul, 41002, Iraq*

Abstract

Cavitation is a word that widely famous in the field of hydraulic machines design and operation. It is undesirable cause a massive wear effect on the impeller's blades, noise, vibration, and reduction in efficiency. This review article is outlined the development in interpretation of cavitation control and the methods (governing equations, CFD technique, vibration analysis, and obstacle effects) used formerly with its mathematical analysis. Most articles agreed with simulation of basic equations using CFD\CFX software, their results were the rule to optimize the pump's efficiency by reducing cavitation as compared to the studies which were used numerical method only. The pressure and velocity distribution in a volute, height of obstacles and cavitation coefficient parameters are also explained briefly. A thermodynamic effect was found to be interest to cover because it's influencing the cavitation formation directly by controlling the pressure, temperature, and fluid properties. In addition, The NPSHi has thermodynamically calculated.

Keywords: Cavitation, Centrifugal pumps, Obstacles, CFD, NPSHi

INTRODUCTION

Centrifugal pumps are used abundantly in multiple domains and it should works in a better performance when all three parameters are intersect with an operating point on pumps' performance curve. A number of troubleshooting are appear including cavitation. Cavitation occurs when the suction pressure is decreasing to less than vapor pressure, cavities are filled with water vapor. These cavities grow particularly on the impeller blade, where there exists the lowest pressure value, and they tend to increase in size and to be dragged out by the fluid flow. As soon as the vapor bubbles reach regions of higher pressure on their path through the centrifugal pump they collapse by an implosion, producing the whirlpool of the liquid of the neighboring zone [3].

The cavitation is a major issue effected directly to the blades, see Figure No. 1, reduction in pump performance, loss in time and increasing the cost of production; therefore, it's important to do a survey on scientific published papers on cavitation issue deeply.

Since, so many researches had been written on cavitation last decades, It is interesting to look at the dramatically development of cavitation prediction methods through several papers which are listed below.

An investigation of theoretical incipient cavitation model had implemented by A. A. Al-Arabi, andS. M. Selim. The NPSHi was calculated based on thermodynamic considerations (temperature of fluid, flowrate, and etc.) with a standard phenomenon of incipient cavitation. The model was tested at various operation conditions. The predicted results were compared to experimental work and the value of NPSHi calculated is close to the experimental data from previous researches[1].

Correspondence to: Omer Haitham Kanam, Department of Mining Engineering, College of Petroleum and Mining Engineering, University of Mosul 41002, Iraq , Tel: 009647730220669, E-mail: omer.kanam@uomosul.edu.iq

Received: January 07, 2021; **Accepted:** August 28, 2021; **Published:** September 8, 2021

Citation: Kanam O H (2021) A Systematic Review on Turbo-Machinery Cavitation Analysis and Thermodynamic Effects. J Thermodyn Cata. 12:p081

Figure 1: Defected impeller by cavitation

A. R. Al-Obaidi, refers to vibration analysis technique in his Ph.D.'s thesis to investigate the cavitation phenomenon. The mean and the root mean square (RMS) are the attributes of frequency which are domain in vibration analysis to monitor the cavitation. Indeed of fast Fourier transform based on time domain (TD) was also applied in the analysis. The sensors used were recorded the signals accurately at low frequency between 1KHtz to 2KHtz and at different flow rates[2].

T. Capurso et al., have investigated the redesigning in geometry of turbo machinery by introducing of slots nearby it's leading edge of NACA0009 hydrofoil in order to increase the suction pressure to prevent cavitation. They applied several numerical analyses to solve a 2D U-RANS equations. Finally, and after extensive test (both single and multi-phase flow) , they could improve the performance of hydrofoil system by reduce the vapor volume fraction up to -93%[4].

A practical study of hydraulic centrifugal pump to prevent cavitation issue was illustrated by V. Cucit, et al., They were implemented two different control system that acts on a pump rotational speed and fed by inertia sensors. The cavitation was induced by acting on tank's pressure on a special check rig. A set of three vibration sensors had installed on the pump casing to acquit the energy data in terms of binary detector (-1 or +1) and this indicates the incipient cavitation, so, the controlling system set the nominal pump speed to threshold value to avoid cavitation[5].

N. H. Mostafa and M. Adel, suggested a 3D model using Navier-stock equation to study the influence of blade angle on cavitation development inside axial pump. The model was accounted turbulence effect according to K-ε turbulence model and the CFD developed code. The results from CFD software were approximately matching with the results obtained from the experiment carried out by the researchers. The blade angle must be at 60o in position and the tip cavitation refers to the loss in efficiency[6].

H. Kohzadi et al., were proposed a new layout to restrict the cavitation in the centrifugal pump used in the designed model. The model was designed to eliminate cavitation by using heat exchanger before the suction line of the pump to increase temperature and thus increasing the pressure to 3 bars to give a continuous flow before pumping. The results were felicitous by intention sought to eliminate the cavitation and increased the pump performance[7].

A basic study of cavitation was set by Rayleigh in 1917. He put a large block of liquid in a turbo machinery and he held a solution to the issue of the collapse of an empty cavity. He discovered that these tiny bubbles have the ability to focus energy and forces and the capacity to erode any material in any fluid machine[8].

D. Liu, was numerically simulate a new cavitation model and compared to 2D-NACA66MOD hydrofoil system database. The proposed model was based on the equations of mass transport of vapor as well as Rayleigh-Plesset equation. The results were in acceptable limit of cavity on the tip of the blade at different advance ratio and different cavitation number[9].

A new method to rapidly predict incipient cavitation in double suction centrifugal pump was simulated practically by J. Pei et al. Two models were numerically simulated which are sst (k-ω), and ZGB. The results of simulated models were validated with experiment data and predicted a minimal deviation for head (0.0076%) and efficiency (0.0373%) respectively. Indeed, the new method was able to predict the vapour volume fraction efficiently. The time to predicted NPSHr is reduced at different flow rate by the alternative model. This practical research was concerned with optimization of reducing pump cavitation and growing with the pump performance[10].

A. A.B. Al-Arabi, et al., were illustrated a cavitation detecting system on a special rig where designed for this reason. A variety of operating conditions were performed to calculate NPSH level using several correlations. The study show that the cavitation is developed just before dropping the pump head by 3% and beyond this percentage the cavitation becomes highly effected on pump performance. The cavities color is changed with fluid temperature changed, therefore, a porous cavity is developed when flow rate increased and a gaseous type of cavity is developed at low flow rate[11].

One view expressed by Y. G. Shah et al., is a novel approach of numerical simulation of cavitation based on lump size method. The have been modified the Rayleigh–Plesset Equation of bubble dynamics and overcomes several shortcomings present in existing lumped parameter based cavitation modeling approaches. The new developed equation called the Hybrid Rayleigh–Plesset Equation. This equation depends on implicit relationship between bubble radius and the energy consumed by the system. The proposed model gives an optimal design of axial gear machine high performance and reliability in operation[12]

The incipient cavitation phenomenon also covered by D. Somashekar1, and Dr. H. R. Purushothama, on radial flow pump with a specific design parameters (Head=21 m, Capacity=26 lps and Speed=1500 rpm). The study was based on theoretical turbulence k-w model and the model was simulated using ANSYS CFX software. The research ended with a fair results which the head remains constant while NPSHi decreasing until the head becomes 2.5 m at super cavitation point, then it's decreasing speedily[13].

M. Rakibuzzaman, et al., employed a concept of Rayleigh-Plesset and the equations of mass transports to find out the NPSH for inverter centrifugal pump. Ansys-CFX software was used to represent the cavitation under different certain conditions. The NPSH of 3 percentage drop in pump head is gradually increased when the flow rate increased since the NPSH estimated value is fluctuated but it's rising in value above the design pressure in order to increase the NPSH[14].

The Cavitation flow in a centrifugal pump was simulated by developed the renormalized group k-ε turbulence model at low flow rate. The calculated NPSHa show a variation with pump head and it's almost fit with the measurements. This study implemented by TAN Lei and ZHU BaoShan[15].

S. F. Xie, et al., had conducted a study which based on computational fluid dynamic (CFD) technique using Proe software tool to optimize the cavitation performance of centrifugal pump. The model is depends on analysis of the Reynolds Average Navier-Stokes equation and RNG model (k-ω). The vapour volume pressure and the total pressure distribution were represented in the software at different pressure for each type of impeller's geometry. It's noticed that when the width of impeller slot reduced the vapour pressure is rising and the total pressure area decreased that's lead to minimum cavitation formation on the back side of the impeller at low speed[16].

W. Zhao, and G. Zhao, used obstacle as a proposed active method to dominate cavitation in a centrifugal pump. A three dimensional unsteady flow of cavitation was numerically simulated by modifying of SS turbulence transport (k-w) model associate with cavitation model. The study found that a suitable height for obstacles can reduce swirls that effects on pump performance and cavitation formation. The obstacles were minify the cavitation formation up to 1/2 of outer width of pump impeller[17].

THE DESCRIPTION OF INVESTIGATED MODELS

In this review, several different methodologies were constructed based on the literature development survey done in previous section. From the excluding point of view, the most reliable principle can predict the cavitation are as follow:

Governing equations

The continuity equations for two phase mixed flow are given below and it's based on Rayleigh- Plesset equation[14].

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial(\rho_m u_j)}{\partial x_j} = 0 \qquad \ldots (1)$$

$$\rho_m\left(\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j}\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu_m + \mu_i)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_i} - \frac{2}{3}\frac{\partial k_i}{\partial x_k}\delta_{ij}\right)\right] \qquad \ldots (2)$$

where ρm and µm are density and dynamic viscosity, u is the velocity, p is the pressure, and µt is the turbulent viscosity respectively and subscripts i, j, k denotes the axes directions. Mixture of ρm and µm are defined as [14]

$$\rho_m = \rho_l \alpha_l + \rho_v(1-\alpha_l), \mu_t = \frac{\rho_m C_\mu k^2}{\epsilon} \qquad \ldots (3)$$

The volume fractions are concerning with the vapor mass fraction (f). The developing and collapse of a fluid bubble is given by a modified Rayleigh-Plesset equation, which is governed by transport equation prevailing vapor developing and condensation[14],

$$\frac{\partial}{\partial t}(\rho_m f_v) + \nabla.(\rho_v f_m u_i) = \nabla.(\Gamma \nabla f_m) + R_e - R_c \qquad \ldots (4)$$

The source term is the diffusion rate, Re and Rc represents evaporation and condensation respectively. The phase transition are given as [14],

$$R_e = C_e \frac{\sqrt{k}}{\sigma} \rho_l \rho_v \left[\frac{2}{3}\frac{p_v - p}{\rho_l}\right]^{\frac{1}{2}} (1 - f_v - f_g)$$

$$R_c = C_c \frac{\sqrt{k}}{\sigma} \rho_l \rho_l \left[\frac{2}{3}\frac{p - p_v}{\rho_l}\right]^{\frac{1}{2}} (f_v) \qquad \ldots (5)$$

where the rate of the empirical factor constants Ce and Cc for 0.02 and 0.01 the gas mass fraction fg=1.5x10-5, the surface tension coefficient σ=0.0717 N/m, pv is vaporization pressure, and k is the turbulent kinetic energy. if p<pv evaporation exist and if p>pv condensation exist[14].

A computational fluid dynamic technique

It is used to generate the mesh and to simulate the cavitation incipient by changing the flow rate at constant speed with different software like ANSYS; therefore, CFD is in concerned to apply the numerical simulation for cavitation analysis.

Figure No. 2 show a generated mesh and the distribution of pressure and vapor fraction on back of impeller blades.

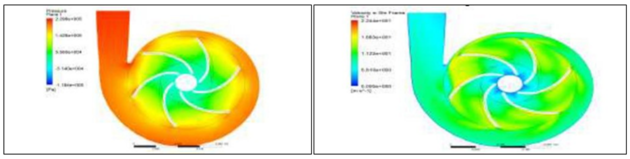

Figure 2: Pressure distribution (left) and velocity distribution at pump (right)[13]

RNG K-ε model in CFD/CfX software has been applied to numerically simulate the NPSHa and compared to experimental work is resulting in Figures No. 3 and 4 respectively. Figure 4 indicates the formation of cavitation when the reduction in NPSHa occur and the vapor bubbles were steeply increased according to reverse relationship of NPSHa and cavitation.

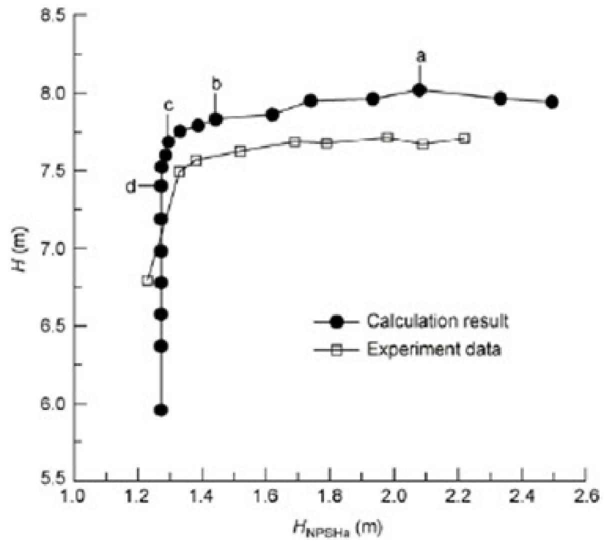

Figure 3: Pressure distribution (left) and velocity distribution at pump (right) [8]

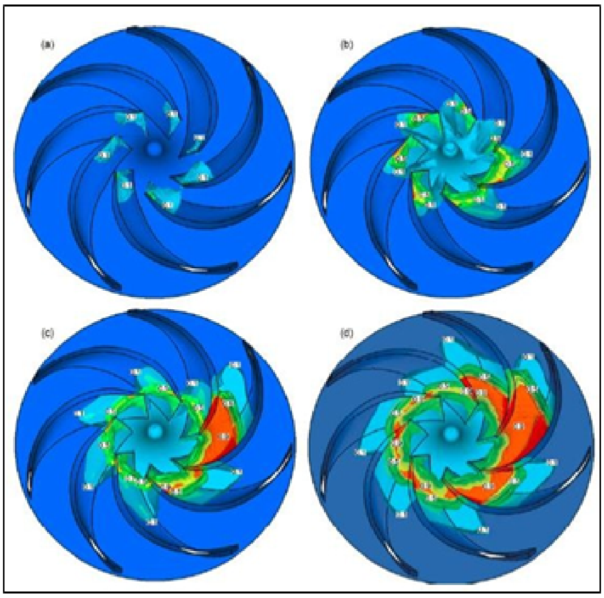

Figure 4: The cavity volume fraction in centrifugal pump impeller. (a) HNPSHa=2.08 m; (b)HNPSHa=1.44 m; (c) HNPSHa=1.30 m; (d) HNPSHa=1.27 m [8]

The Rayleigh-Plesset equation has been also simulated by ANSYS-CFX tool at various flow rates. Similar to the principle calculations of the results showed in figure four, the consequences of changing flow rates have shown in Figure No. 5. The effect of rising bubbles is clearly shown in the red color at low NPSHa but the initial point at 3.7 m predicted no effect on impeller blades.

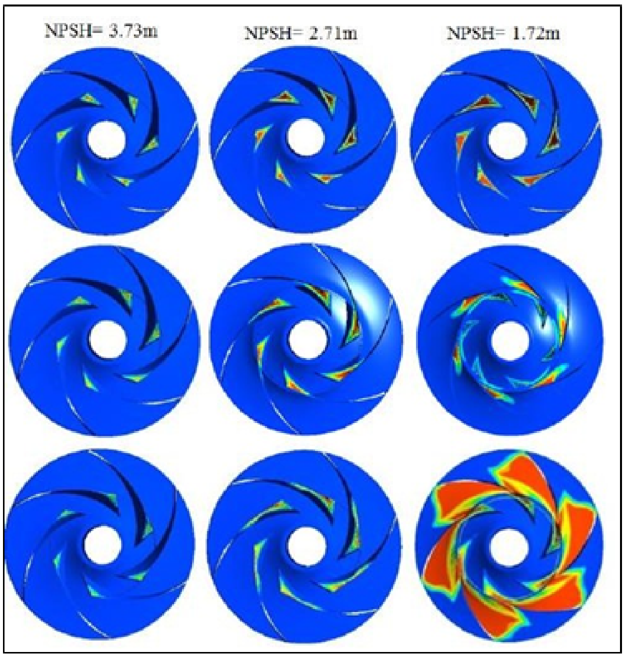

Figure 5: NPSH effects at different flow rates (Q/Qdp=0.8, Q/Qdp=1.0, Q/Qdp=1.13)[14]

Another procedure of simulation using CFD technique is to generated a mesh at the suction of pump case as shown in Figure No. 6. Through mesh test at computational speed and accuracy in concern, it has predicted 4 monitoring points, two are rotating with impeller at the passage (P1 and P2) and the P3 located at the interface while P4 at close to the volute wall, but the last two are at stationary area.

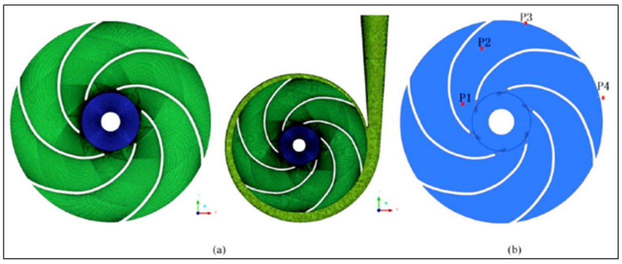

Figure 6: Grids of compute domain and monitoring points[17]

Effect of obstacles

Obstacle is an object which fabricated from material of pump impeller, it blocks the movement of the fluid and help to rise the suction pressure inside pump volute. It effects on the following:

Pressure distribution inside impeller

The mechanism for the cavitation monitoring for the transient flow may be explained as the high pressure induced by obstacles, and Zhang and Cheong thoroughly explained the cavitation pressure waves by a cylindrical object, indeed it's analyzed the transient pressure division around the object, which can also be explained as the monitoring process of cavitation in a centrifugal pump.

Flow structure

When the cavity take place at the leading edge of the blades, the obstacles fixed on the pressure surface raise the function of control the flow field like a reed. As discussed before, the over-high obstacles should block the passages to induce the eddies, which is undesirable to control cavitation in a centrifugal pump. In the developing of cavitation inception, the presence of obstacles may optimize the flow structure and decline the eddies intensity near to the volute. Usually, the cavitation number in hydraulic machinery is defined as[17],

$$\sigma = \frac{p_1 - p_v}{\frac{1}{2}\rho U^2} \qquad \dots (6)$$

where p1 is the static pressure, which is the inlet pressure of the pump; pv is the saturated vapor pressure. The cavitation number is a parameter used to simulate the effect of obstacles on flow structure inside volute using a CFD software

Cavity volume

To better understand the cavitation flow, the overall vapor volume in impeller, Vcav, is defined as[17],

$$V_{cav} = \sum_{i=1}^{N} \alpha_{v.i} \cdot V_i \qquad \dots (7)$$

The vapor volume rise rate , Ve, is defined as[17]

$$V_e = \frac{dV_{cav}}{dt} \qquad \dots (8)$$

where N is the total number of control volumes in the determination domain, is vapor volume fraction in each control volume, and Vi is the volume of each cell.

Transient flow

The fast Fourier transform (FFT) is employed to reveal the frequency characteristics of the internal transient flow in centrifugal pumps. There is an effect of obstacles on amplitude blades passing frequency (ABPF). The obstacles results in a small destabilization, which lead to slightly increase in ABPF. The ABPF of height coefficient (Kh=1/2 ≈ 6 mm) is plainly lower than other obstacles which is tighten to that one without obstacles[17].

Thermodynamic effect

The boiling of liquid in the process of cavitation is a thermal process and is dependent on the liquid properties such as the pressure, temperature, latent heat of vaporization, and specific heat; thus, when hot water and other liquids are pumped, the NPSHrequired will be changed, and cavitation inception will be delayed by the thermodynamic effect. The thermodynamic effect can be understood through the equation of cavitation coefficient (σc), see equation six[3]. The pressure changes due to the effect of thermodynamic can be calculated from the equation below (eq. 9):

$$\Delta P_V = F\,[9.48 \times 10^{-38} T^{15.506}][15.767 \times 10^3 T^{-0.33}][{}^{1}/{}_{C_p}][T^{-n}][{}^{Q}/{}_{Q_{opt}}]^m \qquad \dots (9)$$

The NPSHi has developed based on thermodynamic effect and from the list of equations in[3], the final equation to predict NPSHi is :

$$NPSHi = [1.04 + (0.181(Q_X^{0.67})(N/N_{max})^{-1.547}(T/T_{max})^{-1.11}] + [\left(\frac{[0.69Q_X^{-1.7} \times (3.094Q_X + 0.165)]^2}{2g}\right) + \left[\frac{0.181Q_X^{0.67}(N/N_{max})^{-1.547}(T/T_{max})^{-1.11}U_1^2}{2g}\right] + \left(\frac{[-2.59(\log Q_X) - 0.0427]^2}{2g}\right) - \frac{0.703K[2.936 \times 10^{-35} T^{14.34}][(1.47 \times 10^{16} T^{-6.166}) + 1450.14(Q_X)^{1.015}]}{\frac{1}{2}\rho W_1^2} + \frac{F[9.48 \times 10^{-38} T^{15.506}][15.767 \times 10^3 T^{-0.33}][1/C_P][T^{-n}][Q_X]^m}{\frac{1}{2}\rho W_1^2} \dots (Q_{opt}/Q) > 0.8 \qquad \dots (10)$$

Where, Qx is the ratio Q/Qopt. Equation No. 10 gives the NPSHi as a function of flow condition (flow rate ratio, rotational speed, water temperature), water thermodynamic properties and off-design phenomena (recirculation flow).

The analysis of cavitation had been carried out and developed due time. All researches focused on the basic model equation by Rayleigh-Plesset, and mass transport k-ω SST model, K-ε, but recently, this equation has modified and contribute with the most accurate and efficient tool to hold the complexity of cavitation by using computation fluid dynamic software.

CONCLUSIONS

Through many researches on cavitation analysis, It has including 150 research papers at first look up then it has excluded 120 of them from the abstract scanning and methodologies in their analysis. Thirty papers were included in this review, which have similar analysis procedure and the most important in the investigation of cavitation phenomena. Several points has been concluded from the subject review analysis as following:

- The analysis of cavitation is based on transport equation of mixed flow as listed previously (eq.1 to eq.5)
- Rayleigh-Plesset equation has been modified to be as in form of equation 4.
- K-ε turbulence and K-ω stress turbulence transport models were used in numerical analysis of cavitation.
- A better interpreting of cavitation behavior is to simulate the transport equation and any of turbulence model using computational fluid dynamics (CFD\CFX) software tool to generate a mesh of pump impeller and volute which are related in the analysis.
- NPSHa must be calculate at different flow rates cause it's a driving parameter in the analysis of cavitation.
- Obstacle is an object fitted on the leading edge of the impeller vane and it gives best running conditions for the pump with less cavitation because it works on increasing the suction pressure and this leads to rising the NPSHa.
- The height of obstacles is playing a major rule in distribution of pressure, flow structure, cavity volume, and transient flow.
- A cavitation number is necessary in analysis because it is limiting the flow structure passing on the impeller blades (eq. 9).
- An important methodology used to analyze the cavitation concept in turbo-machinery is to use the vibration analysis through sensors. This method is concerning with transient flow frequency which then analyze with respect to time to detect the cavitation.
- Another view in the analysis is to consider a thermodynamic effect, since there is a temperature and pressure changes inside

the pump volute to find the pressure changes as in equation 9 and NPSHi as in equation 10.

References

1. Al-Arabi, A. A. B. & Selim, S. M. A. A Theoretical Model to Predict Cavitation Inception in Centrifugal Pumps. 5th Int. Conf. Heat Transf. Fluid Mech. Thermodynamic. (2007).
2. Al-Obaidi, A. R. Investigation of effect of pump rotational speed on performance and detection of cavitation within a centrifugal pump using vibration analysis. Heliyon (2019), 5, e01910.
3. Baldassarre, A. et al. Real-Time Detection of Cavitation for Turbomachines, (1998), 416, 403–416
4. Capurso, T. et al. Investigation of a passive control system for limiting cavitation inside turbomachinery under different operating conditions. Energy Procedia, (2018), 148, 416–423.
5. Cucit, V. et al. A control system for preventing cavitation of centrifugal pumps. Energy Procedia, (2018), 148, 242–249
6. H. Mostafa, N. Effect of Blade Angle on Cavitation Phenomenon in Axial Pump. J. Appl. Mech. Eng. (2012), 01, 1–6
7. Kohzadi, H., Shadaram, A. & Hoseinzadeh, S. Improvement of the centrifugal pump performance by restricting the cavitation phenomenon. Chem. Eng. Trans. (2018), 71, 1369–1374
8. Li, S., Brennen, C. E. & Matsumoto, Y. Introduction for amazing (cavitation) bubbles. Interface Focus, (2015), 5, 1–2
9. Liu, D. The Numerical Simulation of Propeller Sheet Cavitation with a New Cavitation Model. Procedia Engineering vol. 126 (Elsevier B.V., 2015).
10. Pei, J. et al. A practical method for speeding up the cavitation prediction in an industrial double- suction centrifugal pump. Energies 12, (2019).
11. Selim, S. M. A., Saidur, R., Kazi, S. N. & Duffy, G. G. Detection of Cavitation in Centrifugal Pumps Department of Mechanical Engineering , Faculty of Engineering , University of Malaya,. (2011), 5, 1260–1267
12. Shah, Y. G., Vacca, A. & Dabiri, S. Air release and cavitation modeling with a lumped parameter approach based on the Rayleigh–Plesset equation: The case of an external gear pump. Energies 11, (2018).
13. Somashekar, D. & Purushothama, R. Numerical Simulation of Cavitation Inception on Radial Flow Pump. IOSR J. Mech. Civ. Eng. (2012), 1, 2278–1684
14. Rakibuzzaman, M., Suh, S.-H., Kim, H.-H. & Jung, Y.-H. Relationship between Cavitation Incipient and NPSH Characteristic for Inverter Drive Centrifugal Pumps. KSFM J. Fluid Mach. (2015), 18, 76–80
15. Tan, L., Zhu, B. S., Cao, S. L. & Wang, Y. M. Cavitation flow simulation for a centrifugal pump at a low flow rate. Chinese Sci. Bull. (2013), 58, 949–952
16. Xie, S. F. et al. Optimization of centrifugal pump cavitation performance based on CFD. IOP Conf. Ser. Mater. Sci. Eng. (2015), 72,
17. Zhao, W. & Zhao, G. An active method to control cavitation in a centrifugal pump by obstacles. Adv. Mech. Eng. (2017), 9, 1–15