Yüzüncü Yıl Üniversitesi Fen Bilimleri Enstitüsü Dergisi/ Journal of The Institute of Natural & Applied Sciences 20 (1-2) 22-33, 2015

Araştırma Makalesi/Research Article

Numerical Investigation of Cavitation on Different Venturi Models

Sedat YAYLA[1], Shakhwan YASEEN[1], Ali Bahadır OLCAY[2]

ABSTRACT: When the pressure in the fluid falls below the liquid's saturation pressure, Cavitation phenomena can severely damage devices or machine parts for example pumps, propellers and impellers. In the current study two-dimensional computational fluid dynamics (CFD) venturi models with variety of inlet pressure values, the throat lengths and vapor fluid contents were employed to examine how cavitation was influenced. In this study two different venturi models were used at various inlet pressures of 2, 4, 6, 8 and 10 atm, throat lengths of 5, 10, 15 and 20 mm and three different vapor contents of 0%, 5% and 10% to unveiled the influence of every parameter on the cavitation number. The investigation unveiled that pressure inlet and vapor fluid content and cavitation number have a positive relationship. It was also discovered that at the inlet pressures of 6, 8,10atm, velocity remains almost constant, however increasing length of throat led to the sharp increase in the velocity of throat at inlet pressures of 2 and 4 atm. Additionally, there was negative correlation between velocity and cavitation number. The outcomes of the cavitation number were different from 0.092 to 0.495 relying on the velocity values of the throat.

Keywords: Cavitation number, CFD, Mixture of fluid, Two-phase flow, Velocity of throat,

Farkli Venturi Modelleri Üzerinde Oluşan Kavitasyonun Numerik Olarak İncelenmesi

ÖZET: Akışkan basıncı doyma basıncının altına düştüğünde oluşan kavitasyon, pompalar ya da pervaneler gibi makine parçalarına zarar vermektedir. Bu çalışmada, farklı giriş basıncı ve boğaz uzunluğu değerlerine sahip sıvı - buhar içeriği değişken olan venturi modellerinin kavitasyonu nasıl etkilediği iki boyutlu hesaplamalı akışkanlar dinamiği (CFD) metodu kullanılarak araştırılmıştır. Giriş basınçları 2, 4, 6, 8 ve 10 atm, boğaz uzunlukları ise 5, 10, 15 ve 20 mm ve % 0, % 5 ve % 10 oranları arasında değişen üç farklı buhar içeriği ile kavitasyonu etkileyen her parametreyi iki farklı venturi modeli üzerinde denenmiştir. Araştırma neticesinde, giriş basıncı, buhar içeriği ve kavitasyon sayısının birbiriyle doğru orantılı olduğu belirlenmiştir. Giriş basıncı 6, 8, 10 atm için akış hızı sabit kalmakta fakat boğaz uzunluğunun artırılması ile 2 ve 4 atm giriş basıncında boğazdaki hız değerinde ciddi bir düşüş gözlemlenmiştir. Hız ve kavitasyon sayısı arasında ise ters orantı bulunmaktadır. Boğazdaki hız değeri üzerinde kavitsasyon sayısı 0.092 ile 0.495 arasında değişmektedir.

Anahtar Kelimeler: CFD, İki fazlı akış, Boğazdaki hız, Karışık akış, Kavitasyon sayısı

Introduction

Cavitation has a widespread utilization in a variety of field like chemical, mechanical engineering, medical and nuclear utilization and cleaning purposes. Cavitation is a dynamic phenomena since it is concerned with the growth and collapse of cavities in fluid flow . There are two stages of the cavitation process as described in (Knapp et al., 1970). The first stage is called "incipient stage", where cavitation is just barely detectable. In this stage the discernible bubbles of incipient cavitation are small and occupy a limited zone of cavitation. The second stage is named as "develop Stage", where evaporation rates increased and cavitation grows due to the changes in conditions such as pressure, velocity and temperature. The occurrence of the inception and development of cavitation depend on the condition of the liquid, including the presence of contaminations, either solid or gaseous, and on the pressure field in the cavitation zone. (Randy, 2001)

Rayleigh Plesset equation provide an important aspect of any cavitating flow since the phase change from liquid to gas turns the flow from being single phase the multiphase. Cavitation is usually considered to consist of microscopic nuclei carried by the flow.

These nuclei are known to be the point of weakness for the liquid from which macroscopic cavities are generated and grow in the low pressure regions of the flow. These small voids in the liquid are induced when the local tension exceeds the tensile strength of the liquid, or as impurities of the liquid such as air bubbles. There are several ways to assess the bubble dynamics in a cavitating flow and the most prominent is the Rayleigh-Plesset equation, derived by Lord Rayleigh (1842-1919) and further developed by Milton Plesset (1908-1991). The Rayleigh-Plesset equation assumes that the nuclei`s start out as spherical micro-bubbles of typically a few microns in diameter that contains a gaseous mixture of vapor for the liquid and possibly some non-condensable gas (e.g., air). Some air is usually present in most liquids, especially when the liquid has been subjected to degassing. Pressure is main driving parameter for bubble dynamics namely bubble growth; besides , bubble collapse is controlled by the pressure divergence between the pressure inside the bubble, usually set equal to the vapor pressure and the ambient pressure. The bubble nuclei is transported with the moving fluid with the same velocity ;therefore, the pressure divergence between the bubble and the local pressure is time dependent for "cavitation" Implying the partial evaporation of liquid in a flow system. A cavity filled with vapor is created when the static pressure in a flow locally drops below the vapor pressure of the liquid as shown in figure 1. When this happens some fluid evaporates and a two-phase flow occures. The vapor condenses suddenly ("implodes") as soon as it reaches an area where the static pressure again exceeds the vapor pressure where the size of the cavitating zone gets large enough (Herland, 2011).

Materials Methods

Cavitation phenomena in the two different venturi models were investigated by changing various geometrical parameters and working conditions in the present study. Mainly, influence of throat length, vapor fluid content and inlet pressure on cavitation was examined in this study. The present work utilized a computational fluid dynamics (CFD) model employed cavitation and mixture models to examine the cavitation.

Specifically, the local pressure below the vapor pressure of the fluid indicating bubble formation. Then, formed bubbles travel with the fluid flow until the throat of the as the bubbles move through the diverging section, they start to collapse and bubble disintegration can be clearly observed.

This study used ANSYS Workbench to produce the venturi geometries as shown in Figure 2 and 3. Design Modular module of ANSYS was utilized for divergent and convergent angles. There were briefly two different models investigated in this work and both models had half divergent and convergent angles of 6º and 10.5º, respectively. This study investigated the effect of venturi shapes on cavitation. Model 1 was exhibited in Figure 2 and this model was obtained from straight lines to define diverging and converging sections. However, Model 2 was produced from curved lines for the converging and diverging sections as shown in figure 3. The throat of 5 mm, 10 mm, 15 mm, and 20 mm were used for both models. In addition, the lengths of upstream and downstream sections were maintained at 55 mm and 27 mm, respectively.

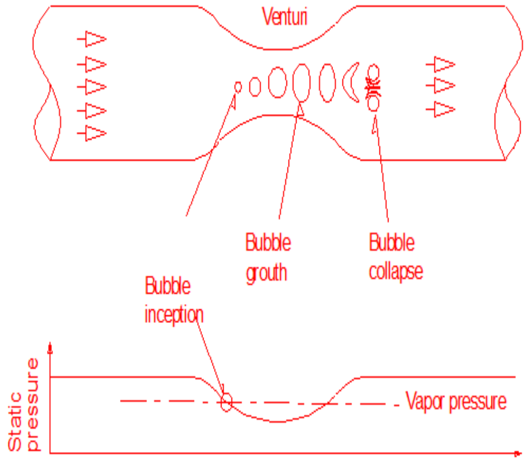


Figure1: Sketch of cavitation formation in a venture venture.

The optimal mesh size was determined based on mesh independence test. The minimum size of mesh was set to be fewer than the first layer thickness near the wall and the minimum sizes of mesh were defined to be 0.000013, 0.0000138, 0.0000145 and0.0000152 m to avoid incorrect automatic mesh production in the solver. The full size mesh elements were described for both models in figure 4 and 5. Tiny bubbles were formed when the pressure of liquid fell below the saturation pressure. In this research, the commercial software ANSYS FLUENT was applied to solve hydrodynamic cavitation computationally with the help of full cavitation model. Generally, two-phase cavitation model consists of a standard turbulence model, namely k-ϵ model, and a conventional viscous stream equation undergo the mixture transfer for example mixture model in the modeling of multiphase cavitation. (Ansys, 2009)

When the computational domain was formed, left and right sides of the domain were defined as pressure inlet and outlet, respectively. Top and bottom sides were identified to be walls as shown in Figure 2 and 3. The diesel fuel was employed to the computational domain of the models and properties of both phases were denoted in Table 1.

A non-dimensional number used to characterize the flow regime is Reynolds number and it is defined based on the characteristic length of geometry, average velocity and fluid properties. In this study, Reynolds number (Re) was expressed as

$$\mathrm{Re} = \frac{\mathrm{VD}}{\mathrm{v}} = \rho \mathrm{vD}/\mu$$

Here, μ was viscosity of fluid, ρ was density of fluid, D was diameter and V was average velocity of fluid.

Cavitation number (σ) is a dimensionless quantity which plays an important role in characterization of cavitation in the fluid flow. It accounts for the difference between the energy head of liquid at outlet and inlet. Mathematically, it was defined as

$\sigma = \frac{2(P_2 - P_v)}{\rho V^2}$ in this study.

Here, P_2 was discharge pressure of bulk fluid, P_v was vapor pressure of fluid, ρ was density of fluid and V was velocity of fluid at the throat.

The governing equations for the two phase flow can be given as

$$\partial(\alpha\rho_v) / \partial t + \nabla \cdot (\alpha\rho_v \overline{V}_v) = R_e - R_c \quad (1)$$

Here, α was vapor volume fraction, ρ_υ was vapor density, $\overline{V}_v$ was vapor phase velocity, R_e and R_c are mass transfer source terms connected to the growth and collapse to the vapor bubbles, respectively [9, 10]. Similarly, the transport equation which governs the vapor mass fraction is given in equation (2).

$$\partial(f_v\rho)/\partial t + \nabla \cdot (f_v\rho\overline{V}v) = \nabla \cdot (\Gamma\nabla f_v) + R_e - R_c \quad (2)$$

In here, f_v represents vapor mass fraction, ρ was mixture density, $\overline{V}_v$ was vapor phase velocity and $\Gamma$was diffusion coefficient for density of fluid smaller than the fluid's saturation density, equation (3) was used when fluid density was larger than the saturation density of the fluid.

$$R_e = C_e \frac{\sqrt{K}}{\sigma} \rho_v \rho_l \left[\frac{2}{3} \frac{\rho_v - \rho}{\rho_t}\right]^{\frac{1}{2}} (1 - f) \quad (3) \qquad R_c = C_c \frac{\sqrt{k}}{\sigma} \rho_v \rho_l \left[\frac{2}{3} \frac{\rho_v - \rho}{\rho_t}\right]^{\frac{1}{2}} f \quad (4)$$

In here, K was turbulence kinetic energy, f is vapor mass fraction, σ was cavitation number and ρ_t was total pressure. The value of constants, namely C_e and C_c, are determined to be 0.02 and 0.01, respectively.

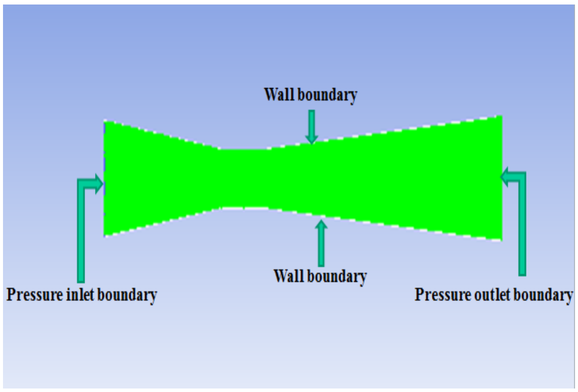


Figure 2. Boundary conditions of model 1

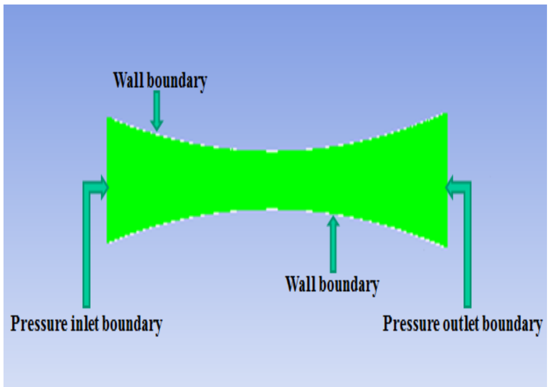


Figure 3. Boundary conditions of model 2

Table.1. The fluid properties of each phase.

Property	Diesel fuel-liquid	Diesel fuel-vapor
Density (kg/m^3)	832	0.1361
Viscosity kg/ms	0.0065	$5.953.10^{-6}$
Vapor pressure pa		5400

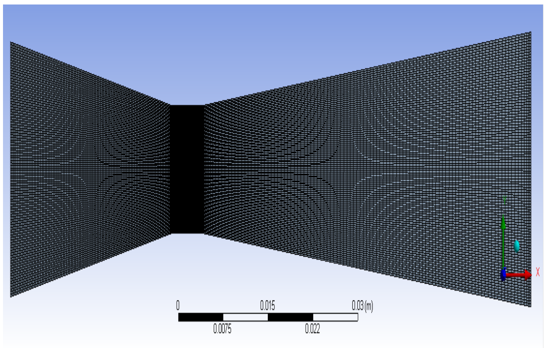


Figure 4. Computational mesh for Model 1

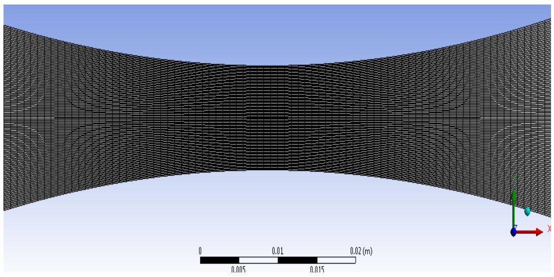


Figure 5. Computational mesh for Model2

Results and Discussion

Effect of the Inlet Pressure

In any flow system, a single flow or a two-phase flow, pressure is a very crucial factor that influences hydraulic design. In this study, pressure gauge was a significant element in determining the inlet and outlet boundary pressure especially in the venturi. It was discovered that the inlet pressure and velocity of the throat increased when the cavitation number decreased. It was also found that (Yan and Thorpe, 1990) the increase in the discharge pressure at the inlet side lead to the increase in the cavitation formation. (Senthilkumar et al., 2000) explained that the cavitational zone augments with the pressure gauge at inlet side except for the ideal condition.

The influence of venturi shape on cavitation was also examined in this work. Moreover, the effect of flow conditions on cavitation was also analyzed utilizing CFD simulation outcomes. When inlet pressure was 4 atm, 10 mm throat length and zero vapor fluid content at the inlet, velocity of throat and cavitation number were 31.281 m/s and 0.235, respectively. Figure 6 exhibits that the pressure decreases progressively from inlet to the throat position. However, after the flow passed the throat the pressure starts to raise gradually to the downstream as monitored by (Jain et al., 2014).

The influence of venturi shape on cavitation was examined in this study. When inlet pressure was 6 atm, 15 mm throat length and zero vapor fluid content at the inlet, velocity of throat and cavitation number were 38.391 m/s and 0.154, respectively. Figure 7 indicates that the pressure similarly shows a decline progressively from inlet to the throat position. However, after the flow passed the throat the pressure again exhibits an increase to the downstream as observed by (Jain et al., 2014).

Effect of The Stream Line

The most important parameter that influences the design in any flow system is Reynolds number. It is one of the factors that defines the kind of flow (laminar, transition and turbulent) and it also influence the transition flow pattern. Turbulent flow stream lines and laminar flow stream lines are the kinds of liquid flow or fluid gas that are noticed. The term streamlines can be defined as continuous phase particles of the fluid have smooth motion along continuous pathways. Relying on the flow of the fluid, streamlines might be either straight or curved. Furthermore, this particular movement can be also termed as a turbulent flow. Because of the diameter of this section is smaller than converge and diverge section, stream line velocity extended to it is maximum value at throat section. However, stream line velocity descended to it is minimum value at the end of the diverge section near the walls. In addition, vortices occurred in the same location.

The streamline velocity of throat was 31.200 m/s, denoted in figure 8 when inlet pressure, length of throat and vapor fluid content were 4atm, 10 mm and 0.05 respectively. It is explicitly shown that Streamline velocity of throat increased when the inlet pressure increased.

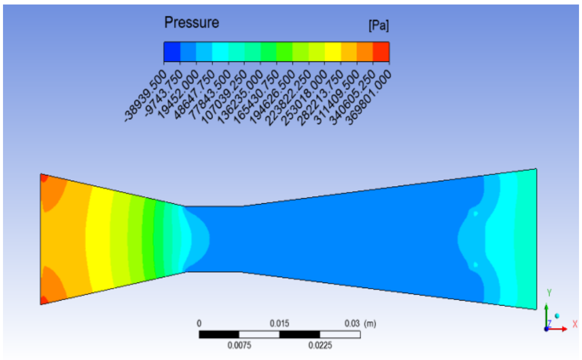


Figure 6. Variation of pressure when vapor fluid content was 0 and the length of throat was 10 mm.

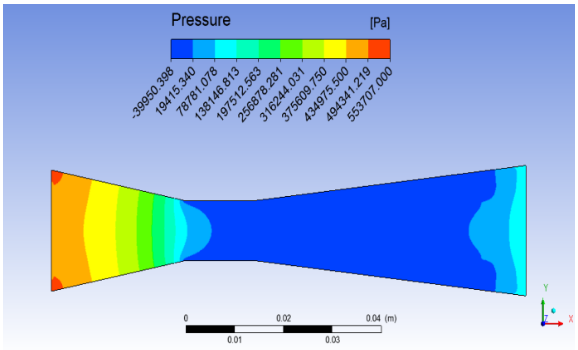


Figure 7. Variation of pressure when vapor fluid content = 0.1 and the length of throat=15 mm.

Figure 9 denotes streamline velocity when it was 21.682 m/s and inlet pressure was 2atm. It is explicitly shown that streamline velocity of throat increased when the inlet pressure increased.

Effect of Vapor Volume Fraction

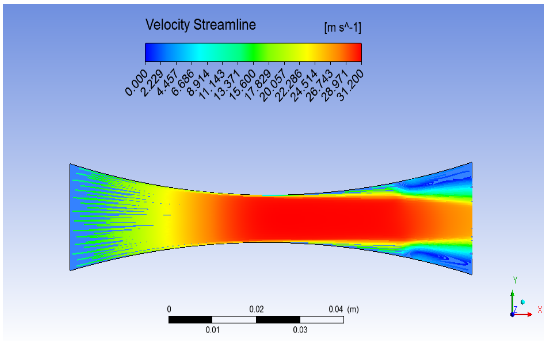
Vapor volume fraction contour displays the gas phase diffusion and formation in the throat section of the venturi. The gas phase thickness area in the throat exhibits the coefficient discharge of particular venturi geometry. The gas phase formation at the walls during the throat section led to reduce the influence of boundary layer on the core of liquid. As a result in the velocity profile to be more uniform during the throat section. This gives the opportunity to utilize simple equation of Bernoulli on such complicated stream fields. As it is shown in vapor volume fraction figure the cavitation length of the venture at the throat and divergence section is increased with the increase of inlet pressure.

Figure 10 explains the vapor volume fraction of 0.991 while the other variables such as length of throat, inlet pressure and vapor fluid content were 10 mm, 4 atm and 0.05, respectively. It was observed that the cavitational zone was prolonged from the throat section to the diverging section.

Figure 11 indicates the vapor volume fraction of 0.99 when the other parameters like length of throat, inlet pressure and vapor fluid content were 20 mm, 6 atm and 0, respectively. It was observed that the cavitational zone was prolonged from the throat section to the diverging section.

The effects of the inlet pressure on the cavitation number in various vapor fluid content such as 0, 0.05 and 0.1 was examined. Shortly, five inlet pressure values (2, 4, 6, 8 and 10 atm) were utilized as boundary condition to unveil its impact on the cavitation number. Figure 12 illustrates that the increase in inlet pressure directly reduced the cavitation number and the vapor content insignificantly effect on cavitation number. It was also observed that the reduce in cavitation number was very significant when the inlet pressure was altered from 2 atm to 4 atm. When the inlet pressure values increased from 6, 8 and 10 atm the rate of decrease in the Cavitation number diminished. The impact of the increase in the pressure inlet is more severe when the inlet pressure increased from 2 to 4 and 6 atm for the vapor fluid content of 0, 0.05 and 0.1. In this situation the cavitation number decreased from 0.485 to around 0.235 causing more than 50% reduction. However, this impact was eased when the pressure inlet raised from 6 to 8, 10 and 12 atm and the vapor fluid content was again 0, 0.05 and 0.1, the cavitation number reduced from 0.235 to less than 0.093 producing almost 50% decrease. Though, it can be concluded that increasing the inlet pressure has the similar impact on the Cavitation number when the vapor fluid contents are 0, 0.05 and 0.1.

In table 1 it can be observed that the alteration in the inlet pressure value can affect both velocity of the throat and cavitation number. Increasing the inlet pressure results in an increase of the velocity of the throat and a decrease in the cavitation number. Therefore, it can be stated that inlet pressure has positive correlation with the velocity of the throat however it has negative correlation with cavitation number using the same length of the throat and vapor fluid content which are equal 15 mm and 0 respectively.



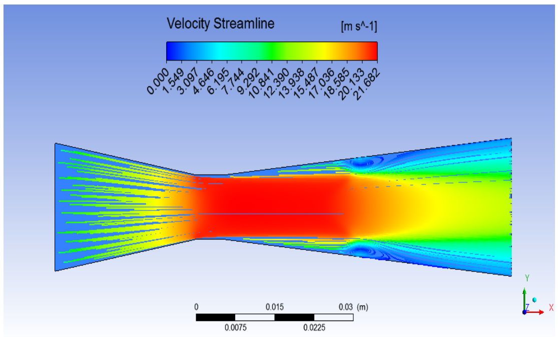
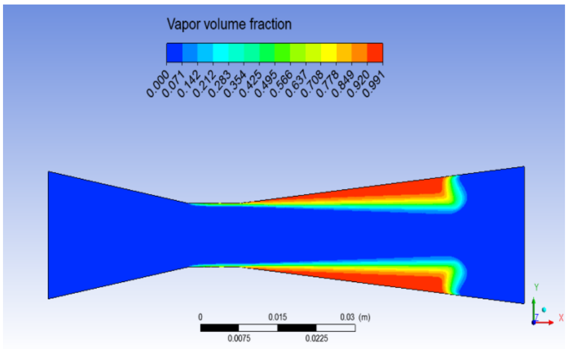


Figure 8. Variation of velocity streamline when the length of throat =10 mm and vapor fluid content 0.05.

Figure 9. Variation of velocity streamline when the length of throat was 5 mm and vapor fluid content was 0.



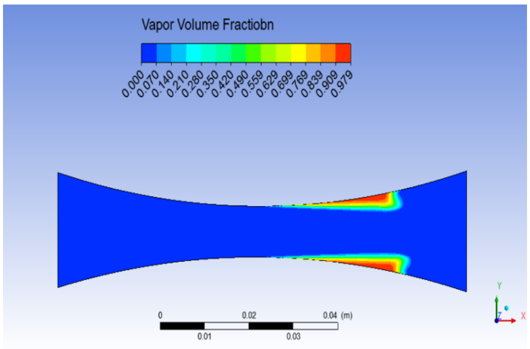


Figure 10. Vapor volume fraction when vapor fluid content= 0.05 and the length of throat=10 mm.

Figure 11. Vapor volume fraction when vapor fluid content was 0.05 and the length of throat was 10mm.

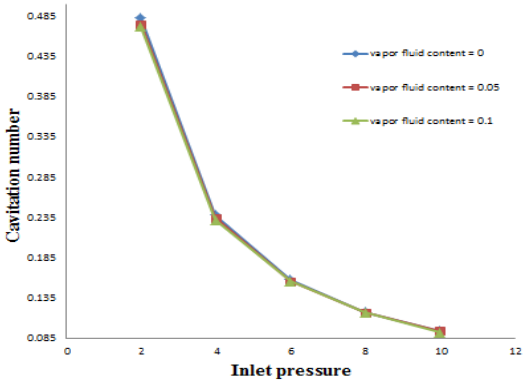


Figure12. Effect of inlet pressure on Cavitation number when the length of throat was 5 mm of model 1.

In table 2 it can be observed that the alteration in the inlet pressure value can affect both velocity of the throat and cavitation number. Increasing the inlet pressure results in an increase of the velocity of the throat and a decrease in the cavitation number. Therefore, it can be stated that inlet pressure has positive correlation with the velocity of the throat however it has negative correlation with cavitation number using the same length of the throat and vapor fluid content which are equal 15 mm and 0.05 respectively.

In Table 3 it can be observed that the alteration in the inlet pressure value can affect both velocity of the throat and cavitation number. Increasing the inlet pressure results in an increase of the velocity of the throat and a decrease in the cavitation number. Therefore, it can be stated that inlet pressure has positive correlation with the velocity of the throat however it has negative correlation with cavitation number using the same length of the throat and vapor fluid content which are equal 15 mm and 0.1 respectively.

Conclusion

This study examined the cavitation event happening in the venturi tube. The present study aimed to discover a novel shape of the venturi with lowest cavitation activity because cavitation has a widespread utilization in a variety of field. ANSYS FLUENT was applied in fluid flow simulations and both cavitation and mixture CFD models were utilized for the two different models of the venturi. The results of stream line velocity, cavitation number, pressure contour, vapor volume of fraction and velocity vector were gained from CFD simulations models. The simulation of cavitation in venturi was performed for inlet pressure of 2, 4, 6, 8 and 10 atm for both models. Additionally, four different lengths of throat 5, 10, 15 and 20 mm, were utilized for both models. Moreover, this research utilized three different vapor fluids content 0, 0.05, and 0.1 with liquid stream at the inlet boundary. It was discovered that when the inlet pressure was raised velocity of throat increased; however, cavitation number decreased. Increasing the pressure gauge at inlet for all the studied cases resulted in the increase in the cavitational zone. It was also discovered that the increase in the discharge pressure at the inlet side leads to the increase in the formation of cavitation. The geometrical parameters played an important role reaching the maximum cavitation production. While velocity was almost unchanged at the inlet pressures of 6, 8 and 10 atm., the velocity was increased progressively by increasing length of throat at inlet pressure from 2, to 4 atm. It was discovered that the velocity of the throat and the vapor fluid content rise led to a decrease in the cavitation number. In all situations, increasing the vapor fluid content led to an increase in the cavitational zone. Overall, the study's outcomes found that model 2 was more useful than model 1. As discussed in the vapor volume fraction figures, cavitation length in model 2 was shorter than in model 1. Moreover, vorticity zone in the divergent section near the walls of the model 2 was smaller than in model 1 as exhibits in velocity vector and velocity stream line figures for all cases.

Table.2. Result when length of throat=15 mm and vapor fluid content 0 of the Model 1.

Inlet pressure (atm.)	Length of throat (mm)	Velocity throat (m/sec)	Cavitation number
2	15	21.831	0.483
4	15	31.092	0.238
6	15	38.127	0.158
8	15	44.067	0.118
10	15	49.294	0.094

Table.3. The results when length of throat=15 mm and vapor fluid content 0.05 of the Model1.

Inlet pressure (atm.)	Length of throat (mm)	Velocity throat (m/sec)	Cavitation number
2	15	21.003	0.476
4	15	31.293	0.235
6	15	38.394	0.156
8	15	44.371	0.117
10	15	49.337	0.094

Table.4. Results when length of throat=15 mm and vapor fluid content 0.1 of the Model 1.

Inlet pressure (atm.)	Length of throat (mm)	Velocity throat (m/sec)	Cavitation number
2	15	22.100	0.472
4	15	31.513	0.232
6	15	38.391	0.156
8	15	44.668	0.117
10	15	49.998	0.092

References

Ansys fluent guide (ANSYS FLUIENT), theory and guide in (2009)

Bashir, T., Soni, Mahulkar, A., and Pandit, 2011. The CFD driven optimization of a modified venturi for cavitation activity. Can. J. Chem. Eng. 1366–1375.

Herland, F., 2011. Numerical Simulation of the Flow in Fuel Nozzles for Two-Stroke diesel Engines MEK-FM-EP-2011-05.

Jain, T., Carpenter, C., Saharan, K.,2014. CFD Analysis and Optimization of Circular and Slit Venturi for Cavitational Activity Journal of Material Science and Mechanical Engineering Volume 1, ISSN: 2393-9095.

Knapp, R., 1970. Cavitation. J.W. Daily, and F.G Hammitt. McGraw-Hill Bo Company.

Moholkar, V., Senthilkumar, P., and Pandit, A., 1999. Hydrodynamic cavitation for sonochemical effects, Ultrason. Sonochem. pp. 6, 53-65.

Randy, S., 2001. Modeling and simulation of Multiphase / Multi component. M.S., The University of New South Wales, Australia.

Senthilkumar, P., Sivakumar, M., and Pandit, 2000. Experimental quantification of chemical effects of hydrodynamic cavitation. Chem. Eng. Sci. 1633–1639.

Saharan, k., Mandar , P., Badve, B., and Pandit, 2011. Degradation of Reactive Red 120 dye using hydrodynamic cavitation.

Yan, Thorpe. 1990. Flow Regime Transitions due to Cavitation in the Flow through an Orifice. Int. J. Multiphase Flow 16 1023–1045.