



Design of Venturi for Hydrodynamic Cavitation using CFD

Melwin Mathai Pothan ¹, Thomas Joy Kuttukaran ¹, Vinayak A Ashok ¹, Vishnu K C ¹,
John Paul C D ², James Mathew ²

U.G. Student, Department of Mechanical Engineering, Rajagiri School of Engineering and Technology, Kakkanad,
Kerala, India¹

Assistant Professor, Department of Mechanical Engineering, Rajagiri School of Engineering and Technology,
Kakkanad, Kerala, India²

ABSTRACT: Hydrodynamic cavitation is a process in which formation of bubbles occurs when the fluid pressure falls below vapour pressure of the liquid and these bubbles collapse when the pressure recovers above the vapour pressure. The cavities are formed by passing it through a geometric constriction, such as orifice, venturi, throttling valve. In this paper, CFD analysis of venturi based constriction is done to determine the optimum geometry for cavitation to take place. The optimization of venturi was done by varying the divergence angle and keeping the other parameters constant at a time.

KEYWORDS: Hydrodynamic cavitation, venturi, divergence angle, Computational Fluid Dynamics

I. INTRODUCTION

Cavitation is the phenomenon of sequential formation, growth and rapid collapse of micro-bubbles or cavities in liquid medium. Bubbles generate and grow when the external pressure over the liquid decreases to an inception value, usually around the vapour pressure of the liquid. After reaching a maximum size, and as pressure recovery takes place, the non-equilibrium state gives rise to bubble collapse, which under certain conditions, can be considered implosive. During collapse, extremely high pressures on the order of thousands of atmospheres and extremely high temperatures on the order of thousands of degrees kelvin are generated in the vapor phase inside the bubble.

In hydrodynamic cavitation, the cavities or bubbles are generated due to the sudden pressure drop and increase in a flowing liquid when passed through the geometric constrictions such as venturi, orifice or rotating blades and results in sudden and violent collapse. This violent collapse of bubbles or cavities results in the generation of OH- free radicals due to the generation of shock waves. This hydroxyl free radical causes the development of localized forces which have destructive effects on the solids in dispersions, emulsions and slurries affecting particle size and distribution resulting in the degradation of pollutants.

One of the major advantage of HC is that it can be easily scaled-up compared to other cavitation methods. When hydrodynamic cavitation is used alone for the treatment of wastewater, efficiency obtained will be less compared to HC combined with AOPs. In order to achieve desired level of COD reduction, number of conventional processes is available but because of their low efficiency these processes are nowadays being replaced by advanced oxidation processes (AOPs). Venturi meter and orifice meter are widely used for HC in the industry and also used for generating cavitation with improved design. A venturi, has advantage over the orifice due to its smooth converging and diverging sections and it can also generate a higher velocity at the throat for a some pressure drop to achieve a lower cavitation number. The flow dynamics at the throat depends on the number of cavities that can be generated. In this work, we numerically analyze various designs of the cavitating reactor by varying operating conditions and geometrical parameters. CFD (ANSYS Fluent software) was used to observe the hydrodynamic behaviour inside the geometric constriction i.e. venturi and to optimize the geometry.

In the CFD (Computational Fluid Dynamics) approach, the equations that govern the process of interest are solved at discrete locations in the domain numerically in an iterative manner. CFD predicts the fluid flow, heat and mass transfer, chemical reactions and related phenomenon by solving numerically the set of governing mathematical equations.

Paper is organized as follows, Section II discusses the parameters and governing equations that has to be considered while performing the simulation and also to investigate the effect of geometrical parameters for the design of venturi. After designing of venturi, the parameters of meshing and conditions used in the solver part for simulating cavitation is given in Section III. Section IV presents simulation results showing results of cavitation effect in venturi using CFD. Finally, Section V presents conclusion.

II. RELATED WORK

Y. Tao et. al. [1] the hydrodynamic cavitation has been an effective means to destruct contaminants in wastewater than acoustic cavitation because it is much more energy efficient and also easy to scale up. Also, the theoretical and experimental aspects of hydrodynamic cavitation applied to wastewater treatment are discussed. Kuldeep et. al. [2] reports about the optimization of various geometrical parameters of different hydrodynamic cavitating geometries such as slit, circular and elliptical venturi. Also observed that venturi has advantage over the orifice due to its smooth converging and diverging sections and it can generate a higher velocity at the throat for a given pressure drop to achieve a lower cavitation number. Under ideal condition cavitation number should be less than or equal to 1 for cavity generation. Therefore, the optimization of venturi was carried by taking three important parameters such as divergence angle, diameter to length ratio and inlet pressure. It was observed that circular venturi is an optimum geometry for best cavitation activity. Saharan et. al [3] observed that venturi type geometry was more efficient as higher cavitation effect can be obtained due to the smooth divergence angle. The study of different geometries of venturi type HC reactor (like slit, circular and elliptical) shows that 1:1 of the ratio of throat height/diameter to length and 6.50 of divergence angle is an optimum geometry for best cavitation activity. For venturi type hydrodynamic cavitation the divergence angle affects the intensity of collapse of active cavities by controlling pressure recovery rate at cavitation zone. By increasing the divergence angles, the cavitation zone is decreasing but the intensity of active cavity collapse increases which cause a decrease in cavitation zone. Kuldeep et. al. [2] discusses about the flow characteristics inside the hydrodynamic cavitating device using Computational Fluid Dynamics (CFD) software to simulate the flow phenomena in various dimensions. Cavitation model and Turbulence model was used to study the CFD of cavitating device. In the CFD (Computational Fluid Dynamics) approach, the equations that govern the process of interest are solved at discrete locations in the domain numerically in an iterative manner. According to C.R. Sanghani et. al. [4] for model solving density, momentum and vapour are discretized using first order upwind discretization scheme and turbulent kinetic energy and turbulent dissipation rate are discretized using second order upwind discretization scheme. The governing equations are the mass conservation and the momentum balance equations have solved using the SIMPLEC algorithm and in turbulence model standard k- ϵ is used. H. Shi et. al. [5] after experimental and numerical studies, found that cavitation model used was Schnerr-Sauer. The Schnerr-Sauer cavitation model is used to calculate the phase transition between the water liquid phase and the vapor phase. This model is derived on the assumption of isothermal, incompressible, and zero-slip velocity between the liquid phase and vapor phase. G. G. Dastane et. al. [6] investigates that cavitating flow in a venturi is simulated using Computational Fluid Dynamics (CFD). Optimum mesh size and suitable turbulence model for such a flow were selected using a preliminary study. The simulations are carried out considering a single homogeneous fluid. Initial simulations were carried out assuming liquid water as the single phase, while the subsequent simulations were carried out considering a homogeneous mixture of liquid water and water vapor.

III. METHODOLOGY

The main need of a constriction device (venturi) was to introduce the phenomena of cavitation within it allowing the formation of bubbles and the subsequent collapsing of bubbles resulting in degradation of the waste water. Circular Venturi was chosen as the constriction device which essentially consists of a converging section, a throat and a diverging section; in reference with the research journals. Three different geometries were designed by varying the divergence angle of the venturi. The venturi is designed keeping the throat diameter to inlet diameter ratio as 2:5 and the throat diameter to width ratio as 1:2. Designing of the venturi was done in ANSYS using the Design Modeler workbench. The Fig. 1(b) shows the 2-Dimensional sketch of the venturi and Fig. 1(a) represents the 3-Dimensional geometry of the venturi generated after performing revolve feature on the 2-D sketch in ANSYS version 2020 R2.



The dimensions of the 3 geometries used for simulation are mentioned in the Table1 given below;

DESIGN	V1(mm)	V2(mm)	H3(mm)	H4(mm)	L5(mm)	$\alpha(^{\circ})$	$\beta(^{\circ})$
1	12.7	12.7	20	40	5.08	15	5.5
2	12.7	12.7	20	40	5.08	15	6.5
3	12.7	12.7	20	40	5.08	15	7.5

Table 1. Dimensions of Venturi

where, α – Convergence angle and β – Divergence angle

The 3 geometries of the venturi were studied and analyzed in ANSYS software using the FLUENT tool from the ANSYS Workbench. Computational Fluid Dynamics can be used as an effective tool to analyze the flow behavior and cavity dynamics inside the venturi at different operating and geometrical parameters. In CFD analysis, the governing equations for the process of interest (flow through cavitating venturi) are solved at discrete locations in the domain numerically in an iterative manner. In all the cases 2-Dimensional axis-symmetric analysis was done due to the limitation in creation of number of elements during meshing (up to 5,00,000 elements) in the software. Each geometry was analyzed separately and the optimum geometry providing better cavitation was chosen as the design. After designing the geometry in the Design Modeler tool, meshing of the geometry was performed. Meshing was done such that the number of elements crossed 1,00,000 mark. It was done so as to improve the accuracy of the results and also because it was ideal to maintain the number of elements above that mark as the results won't differ much even when the number of elements varied after crossing the 1,00,000 mark. Face meshing was done and element shape was set to quadrilateral to enhance the quality of the mesh. It was observed that keeping the element size below 1mm was appropriate, so the element size was chosen as 0.1mm for all the geometries as it did not exceed the limit in each case generating between 2,50,000 cells and 3,00,000 cells in all the 3 geometries. Mesh quality factors were also looked into while fixing the element size such as aspect ratio, skewness and element quality. It is understood that for a mesh to be of good quality, skewness should be below 0.90, aspect ratio should not be beyond 5 and maximum value of element quality should not exceed beyond 1 and looking into these conditions element size of 0.1mm proved to be a good quality mesh. The mesh quality characteristics are shown in Table 2.

DESIGN	SKEWNESS		ASPECT RATIO		ELEMENT QUALITY	
	MIN	MAX	MIN	MAX	MIN	MAX
1	1.2017×10^{-5}	0.16206	1	2.4655	0.63152	0.99929
2	3.9038×10^{-5}	0.24073	1	2.663	0.57065	0.99944
3	4.3298×10^{-5}	0.23496	1	2.6562	0.5738	0.99944

Table 2. Mesh quality characteristics

Mesh obtained for Design 1 ($\beta=6.5^\circ$) is shown in Fig. 1(c) and a zoomed view of the mesh obtained is shown in Fig. 1(d) where it is clearly visible that the shape of each element resembles a quadrilateral.

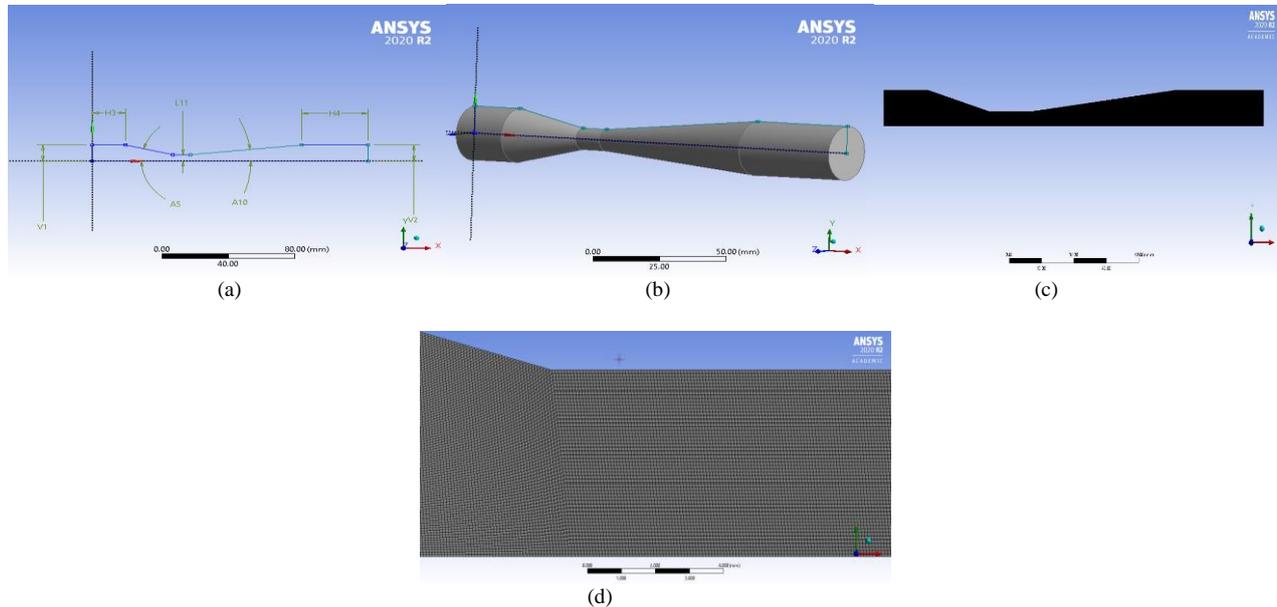


Fig. 1. (a) 2-D sketch of venturi (Design 1) (b) 3-D view of venturi (c) Mesh of Design 1 (d) Zoomed view of the mesh

The next stage after meshing is applying the various conditions pertaining to the requirements for simulation of cavitation phenomena in the flow through the venturi. The simulations are carried out considering a single homogeneous fluid. Initial simulations were carried out assuming liquid water as the single phase, while the subsequent simulations were carried out considering a homogeneous mixture of liquid water and water vapor (as vaporous cavities) with water being the primary phase whereas the water vapor was defined as the secondary phase. Mixture model in FLUENT is based on Euler–Euler approach. In this model, the phases are treated as interpenetrating continuum. The mixture model solves for the mixture momentum equation and prescribes relative velocities to describe the dispersed phases. This model is commonly used in simulations of bubbly flows. The mixture model can also be used without relative velocities for the dispersed phases to model homogeneous multiphase flow, which is the case in cavitating flows. The secondary phase is dispersed in liquid phase and forms a homogeneous multiphase flow. In this study, mixture multiphase model is used without considering any relative velocity (no slip velocity condition) between the two phases, to model the cavitating flow as a homogenous flow. The simulation was performed using ANSYS FLUENT 2020 R2. The 2D axisymmetric computational geometry to be simulated is discretized with quadrilateral type meshes using a face meshing method with 2,49,345 (Design-1), 2,76,783 (Design-2) and 2,62,824 (Design-3) computational cells (elements). To simulate cavitation, mixture model was chosen as the multiphase model as the mixture model is capable to predict cavitation phenomena in water flows by solving a set of transport equations governing the mixture continuity, momentum, energy and the disperse phase for the volume fraction equation. The standard $k-\epsilon$ turbulence model was applied for modelling both liquid and vapor phase with standard wall function as it can provide more accurate solutions in the near wall boundary regions. The Schnerr-Sauer cavitation model is used to calculate the phase transition between the water liquid phase and the vapor phase and the saturation vapor pressure (P_v) was set to 3540 Pa as the primary phase is water.

The solution of governing equations was found after applying the specific boundary conditions for given geometrical configurations. The inlet and outlet boundary conditions type were set to pressure-inlet type and pressure-outlet type. As the inlet boundary condition, total pressure in the range of 4,00,000 Pa is set while the outlet boundary condition was set to 1,01,325 Pa of total static pressure. It was also verified that at both inlet and outlet of the venturi the vapor volume fraction (VVF) was initialized as 0 as vapor is absent at both the inlet and outlet. In model solving density, pressure is discretized using Pressure Staggering Option(PRESTO!) scheme as owing to the geometry design the flow field is expected to have a steep pressure gradient at the cell face, momentum and vapor are discretized using First Order Upwind discretization scheme and turbulent kinetic energy and turbulent dissipation rate are discretized using



Second Order Upwind discretization scheme. The governing equations such as the mass conservation and the momentum balance equation were solved using the Semi-Implicit Method for Pressure Linked Equations-Consistent (SIMPLEC) algorithm.

IV. SIMULATION RESULTS

From this study, simulation of vapor formation within the venturi during cavitation is analysed for the 3 geometries. Cavitation results in intense vapor bubble formation after the change in area that is within the region of the throat of the venturi. From various reference papers it was understood that the divergence angle of the venturi most affected the cavitation phenomenon. Hence we choose to vary the design by varying the divergence angle and keeping the rest of the dimensions constant. After applying the various boundary conditions and modelling methods, before calculation it is mandatory to initialize the values once again in the software computation has to be done from the inlet so choosing the standard initialization technique, compute from inlet condition was selected and initialized. Later before beginning the calculations, number of iterations is set to 2000. Initially single-phase simulations were performed. It was done by disabling the volume fraction and slip-velocity conditions before initialization and allowed to calculate until the solution converged. It was done so as to allow smooth convergence during multiphase simulations and to get a rough idea about when and at which iteration the solution would converge. After the solution converged for single phase simulation, it was verified whether only singlephase flow occurred by observing the vapor volume fraction of vapor phase which would be entirely blue indicating the absence of vapor in the flow as shown in Fig.2(a).

The multiphase simulation was carried out after this by enabling the volume fraction and slip velocity condition and was allowed to calculate the problem. The solving began where it had stopped during single phase simulation. After the solutions converged, vapor volume fraction contour of vapor phase was observed for each of the 3 geometries as seen in Fig. 2(b) representing design 1, Fig.2(c) representing design 2 and Fig.2(d) representing design 3. It was observed that maximum vapor volume fraction was for design 2 (VVF=0.997) indicating that vapor formation was highest for venturi with divergence angle of 6.5° (Design 2).As a result, Design 2 was opted among the 3 designs as it produced enhanced cavitation bubble formation during the simulation.

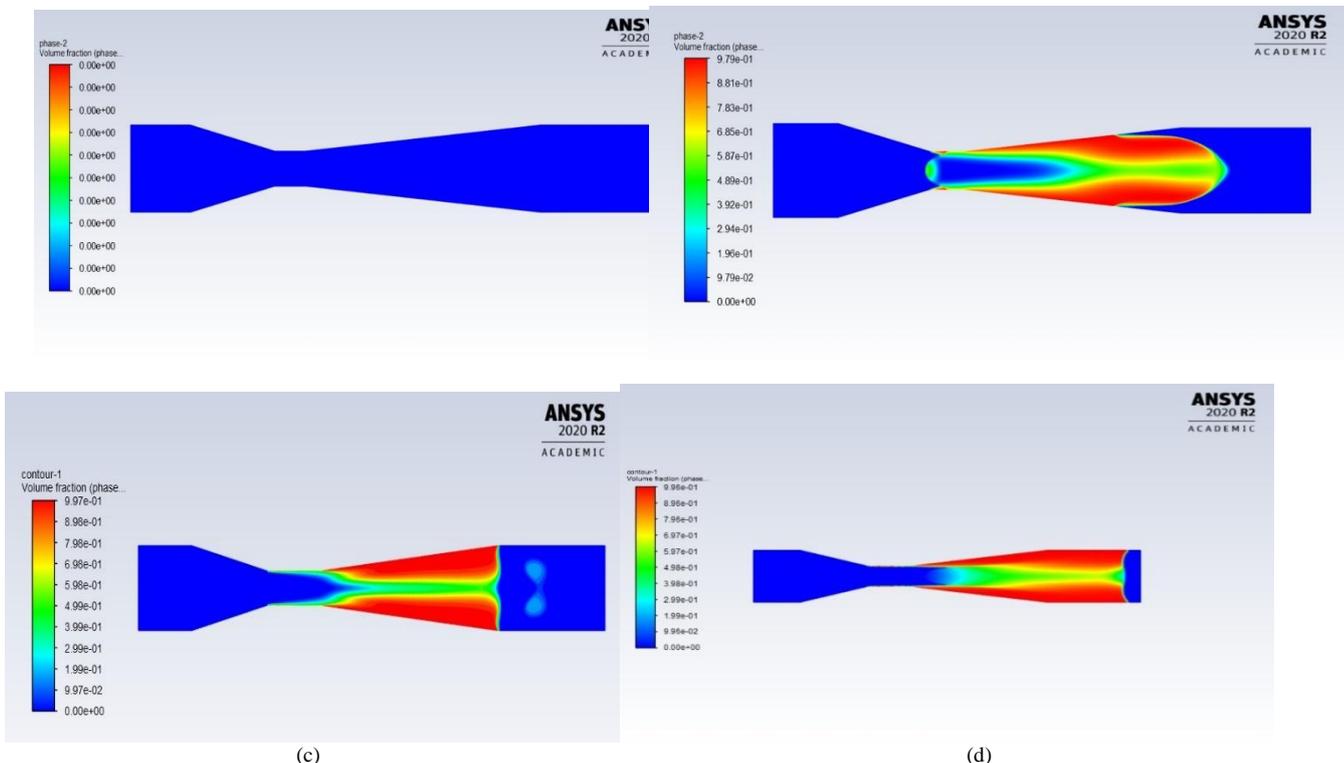


Fig. 2.(a)Vapour volume fraction contour of Design 1 after single phase simulation(b) Vapour volume fraction contour of Design 1 (c) Vapour volume fraction contour of Design 2 (d) Vapour volume fraction contour of Design 3



V. CONCLUSION

We have simulated cavitation using ANSYS Fluent and it was concluded that optimum cavitation was obtained for venturi with divergence angle of 6.5° . It can be concluded that with the increase in divergence angle, cavitation effect increases upto a certain extent as further increase of divergence angle leads to decrease in cavitation effect.

REFERENCES

- [1] Yuequn Tao, Jun Cai, Xiulan Huai, Bin Liu, Zhixiong Guo, Application of hydrodynamic cavitation to wastewater treatment, *Chemical Engineering & Technology*, Vol. 39, Issue 8, 1363-137, 2016.
- [2] Kuldeep, Jitendra Carpenter, Virendra Kumar Saharan, Study of Cavity dynamics in a Hydrodynamic Cavitation Reactor, *Energy Technology & Ecological Concerns: A Contemporary Approach*, Vol. 3, 37-43, 2018.
- [3] Kuldeep, Virendra Kumar Saharan, Computational study of different venturi and orifice type hydrodynamic cavitating devices, *Journal of Hydrodynamics*, Vol. 28, 293-305, 2015.
- [4] C. R. Sanghani, D. C. Jayani, N. R. Jadvani, H. N. Dobariya, K. R. Jasoliya, Effect of Geometrical Parameters of Venturimeter on Pressure Drop, *Engineering and Technology*, Vol.2, Issue 2, 865-868, 2016.
- [5] Hongbo Shi, Mingda Li, Petr Nikrityuk, Qingxia Liu, Experimental and numerical study of cavitation flows in venturi tubes: From CFD to an empirical model, *Chemical Engineering Science*, Issue 207, 672-687, 2019.
- [6] Gaurav G. Dastane, Harsh Thakkar, Rushabh Shah, Sivaramakrishna Perala, Janhavi Raut, A.B. Pandit, Single and multiphase CFD simulations for designing cavitating venturi, *Chemical Engineering Research and Design*, Issue 147, 1-12, 2019.