Numerical Investigation of Cavitation Problem on A Specified Geometry

Kutay Yilmaz[1], Cuneyt Yavuz[2]

[1,2] Ph.D. Candidate, Water Resources Lab., Dept. of Civil Engineering, Middle East Technical University, 06800, Çankaya/Ankara, Turkey.

Abstract: *Cavitation can be a significant thread for hydraulic structures. This phenomenon occurs when vapor pressure drops below static pressure. Occurrence of the cavitation can cause cavitation erosion that can damage the parts of the pipes, turbines or pumps. In this study, pipeline system which will be constructed in a part of hydraulic works is investigated numerically using commercially available ANSYS Fluent software in terms of cavitation problem. By varying inlet boundary condition, the maximum velocity that cause cavitation not to occur and the velocity that cause cavitation just to occur is determined. Results of the numerical simulations are visualized. In addition, minimum static pressure and maximum velocity of the pipeline system are determined and compared with the vapor pressure. It is shown that for the given boundary conditions, cavitation does not take place. On the other hand, if the inlet velocity is increased to 5.7 m/s or further, cavitation can be observed depending on flow and structural conditions.*
Keywords: *Cavitation, Static pressure, Vapor pressure, Numerical Model, ANYSIS-FLUENT, GAMBIT,*

I.   Introduction

Cavitation, which occurs when the static pressure of liquid drops below the vapor pressure, is defined as the formation of gas bubbles in the fluid. Cavitation generally occurs in regions near the fast-moving turbine blades and in the outlet zones of the turbines.The liquid enters the turbines under high pressure which is the summation of static and dynamic pressures. [1]

Dynamic pressure is caused by the speed of the fluid where static pressure is the pressure that the liquid actually applies. Static pressure controls the process of formation of air bubbles. As a result, cavitation occurs near the turbine blades such as fast moving objects because the static pressure decreases and the dynamic pressure increases in the regions close to the turbine blade and accordingly the total pressure does not change.

When the bubbles that is formed as a result of cavitation break out, they form high frequency pressure waves, which damages the mechanical parts. For example, in the regions close to the turbine blades, bursting of these bubbles cause damage to the turbine blades. This condition is expressed as cavitation erosion. In order to prevent cavitation, the static pressure should not be allowed to drop below the vapor pressure. The parameters that control the situation are: pressure head, flow rate and outlet pressure of the liquid. [2]

In order to analyze the situation inside the pipeline system that supply water to the turbines, ANSYS FLUENT, a computational fluid dynamics program, is used to model cavitation problem. To obtain a solution of the problem, geometry should be created initially and proper mesh cells should be constructed along the surface and volume of the specified geometry. Despite existing geometry and cell creation programs in ANSYS, the Gambit software is preferred. Because it provides flexibility for users to control in creating cells.

II.   Model Setup

Geometry of the system is generated by using GAMBIT software (Fig.1).Pipeline system consist of two identical vertical pipeswith 6.7 m in length and 1 m in diameter, horizontal extension of vertical pipes with a length of 2.9 m and horizontal pipe which is 5.1 m in length and 1.4 m in diameter. After bending, extension of the vertical pipes lies down on a horizontal plane. Horizontal extensions of the vertical pipes are 2.9 m in length and 1 m in diameter. The horizontal extensions are connected at the junction. Angle between the two-horizontal extension is 120°.Water enter from the two identical vertical pipes with a velocity of 1.91 m/s. Detailed sketch of the geometry can be seen in Figure 1.
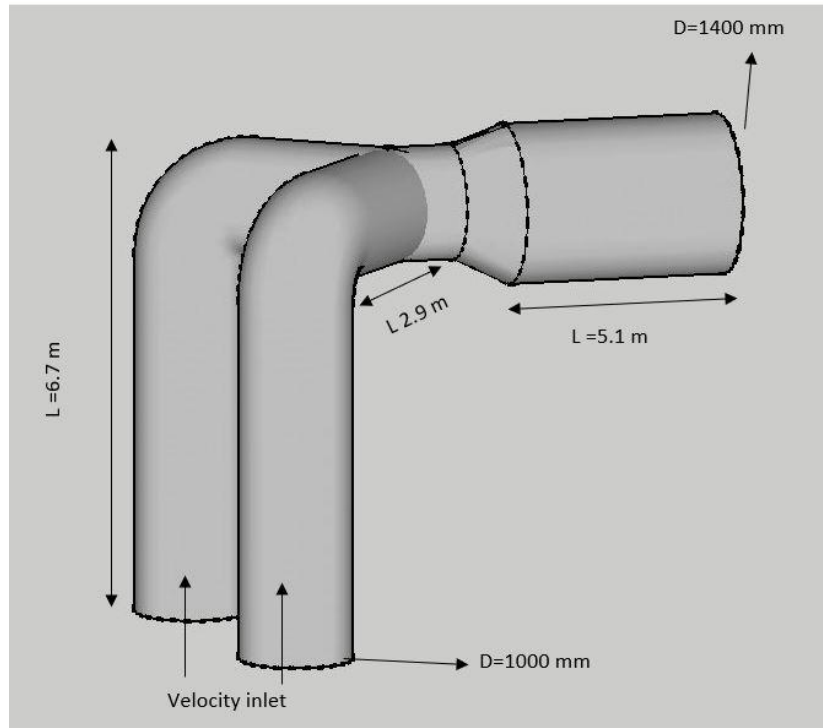
Fig. 1 Detailed sketch of the pipeline system

Cavitation problem can be solved conveniently by creating meshes in an appropriate size. In the mentioned pipeline system, computational domain is covered with unstructured mesh as shown in Figure 2. Cells with skewness greater than 1 make it difficult to get the right solution.Therefore, skewness of the cells is checked via Gambit. After the geometry and cell creation process is completed, the meshed geometry in Figure 2 is embedded into the ANSYS FLUENT software.
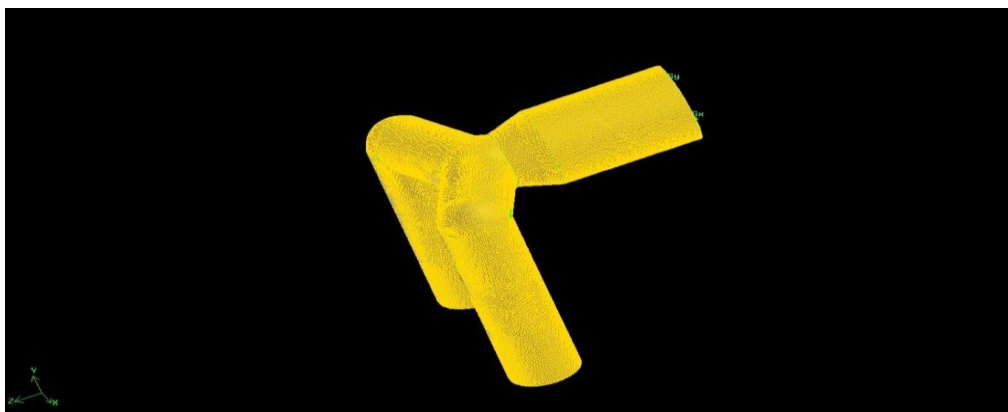

Fig. 2 Mesh generation on GAMBIT

III. Numerical Model

After having imported the geometry into the ANSYS Fluent, the appropriate turbulence model, fluid, boundary conditions are selected on the software and simulation has been initialized. At the end of 300 iterations, the values are found to converge and the program is stopped. The obtained results are visualized with TecPlot 360, one of the most common programs for visualizing computational fluid dynamics results.

After having completed the geometry and meshing process Reynolds number is calculated and found as approxiamely two million by considering the inlet velocity in the pipelines. According to this situation appropriate turbulence model should be chosen. In the simulations k-w turbulence model , solves the turbulence kinetic energy "k" and heat that occurs due to the conversion of turbulence quantities to heat "w", is chosen to simulate the problem. Because k-w model is sensitive to initial conditions, k-e turbulence model is used to obtaion initial conditions. k-w model is proposed for the problems that consist of bending pipes.

IV.   Simulation Results

Because cavitation problem depends on static pressure and velocity, static pressure and velocity in the whole computational domain is checked. Depending on the static pressure and vapor pressure of the water, occurrence of cavitation is examined. Therefore, Figure 3 and 4 show the velocity contours on the different planes for the domain while Figure 5-6-7 show the static pressure of the fluid throughout the domain and planes.

Figure 3 shows distribution of the velocity contours on a cross section at x-z plane. There is a rapid increase in velocity at the junction and as the fluid moves along the pipe, the rate of change of velocity throughout the cross section decreases.
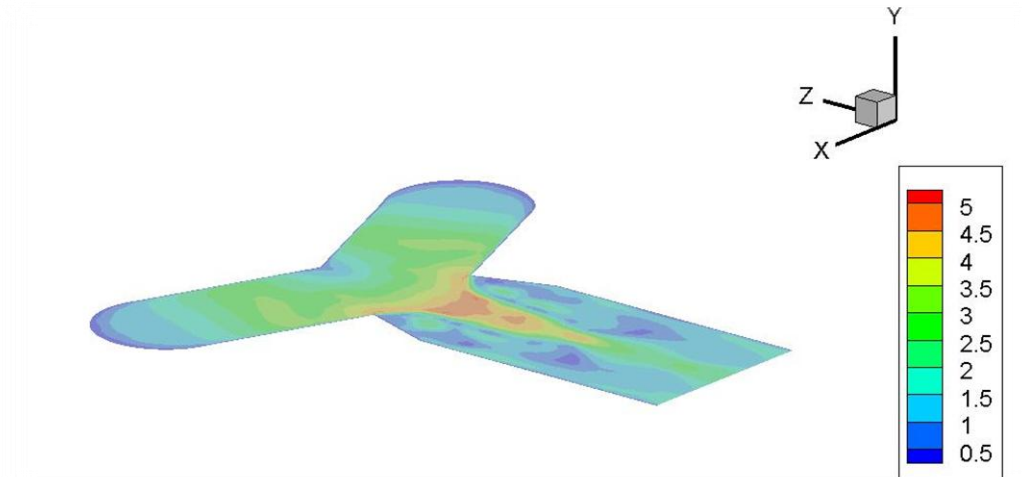

Fig. 3 Distribution of velocity contours on a cross-section at X-Z plane

Figure 4 shows distribution of the velocity contours on a cross-section at y-z plane. As it can be seen from the figure, velocity increases at the bends and velocity decreases as the fluid moves along the pipe through the junction.
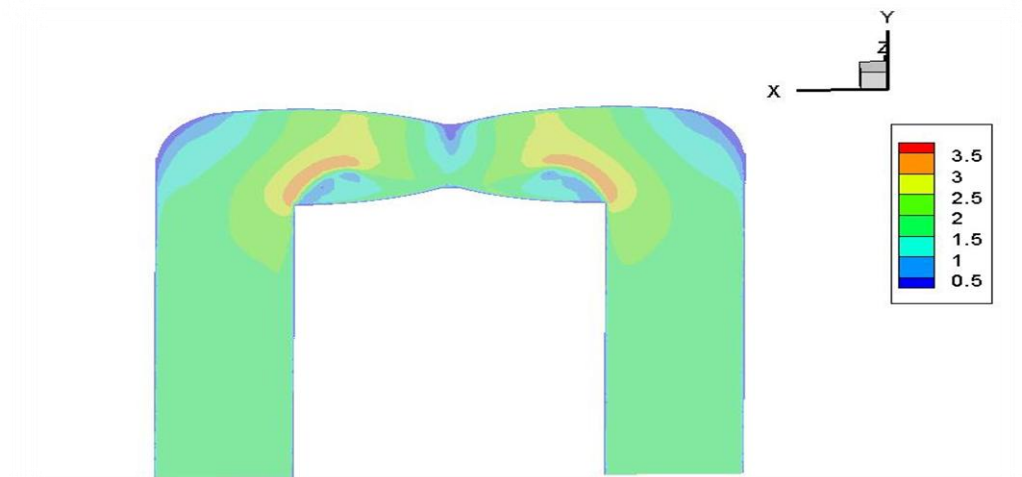

Fig. 4 Distribution of velocity contours on a cross-section at X-Y plane

Figure 5 shows the distribution of the static pressure on a cross section at the x-z plane. From the figure, the static pressure does not fall below 190000 Pascal at the specified cross section.
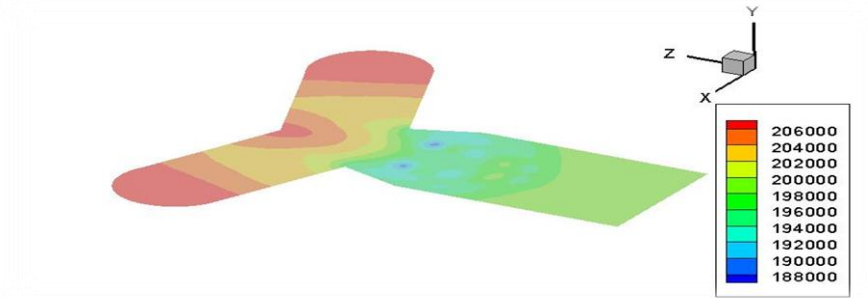
Fig. 5 Distribution of static pressure on a cross-section at X-Z plane

Figure 6 shows the distribution of the static pressure on a cross-section at the x-y plane. As it can be seen from the figure, the static pressure does not fall below 198,000 Pascal.

Fig. 6 Distribution of static pressure on a cross-section at X-Y plane

In Figure 7, distribution of the static pressure over the whole computational domain is shown from the different angles. By considering the total volume, it appears that the static pressure does not fall below 182,000 Pascal.
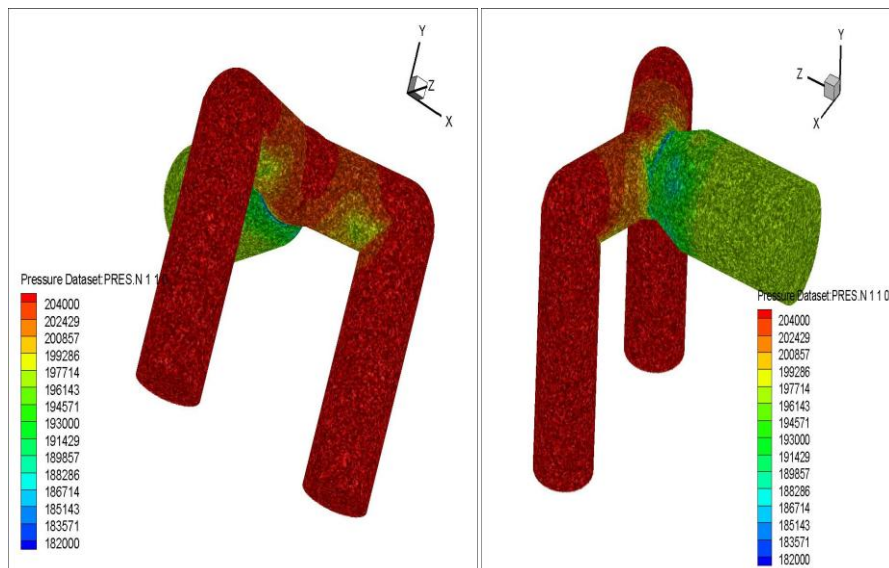

Fig. 7 Distribution of static pressure on the computational domain

Table 1 Water vapor pressure and corresponding temperature[3]

Temperature (°C)	Vapor Pressure (kN/m^2)
0	0.61
5	0.87
10	1.22
15	1.70
20	2.33

48

25	3.16
30	4.24
35	5.62
40	7.38
45	9.59
50	12.35
55	15.76
60	19,94
65	25,03
70	31,19
75	38.58
80	47.39
85	57.83
90	70,14
95	84.55
100	101.33

When the static pressure values of the whole computational domain considered and by examining the cross-sections, it was determined that the static pressure did drops below 182000 Pascal. Vapor pressure of water at different temperatures are presented in the Table 1 and it is determined that 1700 Pa. of water vapor pressure can take place when measured temperature is 15°C. As a result of calculations made with ANSYS FLUENT, the static pressure values remain higher than the vapor pressure even if the water vapor pressure at 100° considered. Under these conditions, the formation of cavitation is not possible for the case where inlet velocities at the vertical pipes are 1.91 m/s. By varying the inlet velocities of the entering water at the vertical pipes, simulations are repeated and static pressure values along the computational domain are obtained. Inlet velocity values and corresponding static pressure values are presented in Table 2.

Table 2 Inlet velocity and corresponding computed static pressure within the computational domain

Velocity (m/s)	Static Pressure (Pascal)
3.5	132332
4.2	99495.6
4.7	78367.9
5.2	46172.4
5.7	22141.9
5.8	-20194.7

V.   Conclusions

In the present study, occurrence of cavitation in a specific geometry is investigated numerically. For a given geometry and inlet velocity at the vertical pipes cavitation is not take place even if the water vapor pressure at 100° considered. In order determine the velocity that cause cavitation in the computational domain simulations are repeated for a range of inlet velocity values. For a velocity values smaller then 5.8 m/s cavitation does not take place if the water temperature is lower then 65°. If the inlet velocity increased to 5.8 m/s or mora, negative pressure can be observed in the computational domain and static pressure drops below the water vapor pressure. Therefore, there is a strong possibility of cavitation occurrence. In addition, results of this study can be compared with the experimental results and numerical model can be validated. Therefore, importance of numerical modelling in the hydraulic computations can be shown.

References
[1].   ''Fundamentals of Hydraulic Dredging'', Thomas M. Turner,Cornell Maritime Press, 1984, p. 95.
[2].   "Fluid Mechanics, Fundamentals and Applications," Y. A. Cengel, J. M. Cimbala,2nd Ed., McGraw-Hill, 2009, p. 42.
[3].   "Fluid Mechanics, Fundamentals and Applications," Y. A. Cengel, J. M. Cimbala,2nd Ed., McGraw-Hill, 2009, p. 924.