Numerical Study on the cavitation patterns in a Francis turbine

Mohammad Taghi Shervani-Tabar[1]
Department of Mechanical Engineering, University of Tabriz, Iran

Fatemeh Ghaffari Laleh
Department of Mechanical Engineering, University of Tabriz, Iran

Abstract

Cavitation phenomenon is one of the important issues that must be considered in hydro turbines, including the Francis turbines. Cavitation is the formation of air bubbles in a liquid (such as water) and the creation of small explosions in the water, which can lead to failure and serious damage to turbines. The importance of cavitation in the Francis turbine is due to the fact that maintaining the optimal performance and safety of the turbine is very vital for the production of electrical energy. Also, the failure and damage caused by cavitation can lead to high maintenance and repair costs and even stop the turbine, which can bring serious economic costs for power plants and hydropower production systems. Therefore, cavitation must be given special attention in the design, construction and operation of the Francis turbine in order to guarantee the optimal performance and safety of the turbine. In this paper, cavitation inside the Francis turbine was discussed investigated using OpenFOAM software. Using this open-source software and Q criterion, the location of vortices and bubbles caused by cavitation have been identified and reported. The most powerful cavitation pattern observed belongs to the vortex cavitation at the draft tube inlet, while the weakest cavitation patterns belong to cloud and sheet cavitation and are susceptible to suppression.

Keywords: Cavitation, Francis turbine, numerical study, turbomachine, computational fluid dynamics.

1-Corresponding Author

Introduction

In a Francis turbine, water enters the turbine blades at high speed, and if the pressure and velocity of the water are such that air bubbles form in the water, cavitation occurs. These air bubbles can damage the turbine blades and other internal parts of the turbine due to severe mechanical stresses and small explosions. To prevent cavitation in the Francis turbine, performing accurate calculations to determine the appropriate pressure and speed of the inlet water, optimal design of blades and other parts of the turbine, using methods to reduce mechanical pressure caused by air bubbles, and using systems to prevent cavitation are among the solutions that used in Francis turbine [1]. When fluid passes through an obstacle at high speed, the fluid pressure drops and this can cause cavitation bubbles to form. This type of cavitation may occur around turbine blades, pump impellers, and other places where fluid is passing at high velocity. Also, pressure fluctuations in the fluid can create suitable conditions for the formation of cavitation bubbles. This type of cavitation may occur in systems that have pressure fluctuations [2]. Main types of cavitation in Francis turbines include leading edge cavitation, travelling bubble cavitation, draft tube swirl and inter-blade vortex cavitation. Each of these cavitation patterns can lead to serious consequences for equipment and technical systems, so it is very important to know this phenomenon and take appropriate measures to prevent its occurrence. When cavitation occurs, different flow patterns are created that can help to better understand this phenomenon. The following flow patterns may occur during cavitation: turbulent flow patterns and explosive flow patterns. In some cases, cavitation can cause turbulent flow patterns that may lead to the formation of various bubbles and vorticities in fluid that can increase fluctuations and flow instability. In some other cases, cavitation can cause explosive flow patterns that may lead to the formation of pressure waves and explosions in fluid. These flow patterns may also be created simultaneously in a cavitation process and cause increased risks and unpleasant consequences for systems and equipment. For this reason, it is very important to know and control these flow patterns [3-5].

Theory of the problem

The Q criterion is a parameter used in cavitation analysis to identify and analyze cavitation and vorticities in fluid flow. Cavitation occurs when the local pressure in a fluid drops below the vapor pressure, causing vapor bubbles to form. These bubbles may break violently, resulting in damage to surfaces in contact with the fluid. Vortices, on the other hand, are swirling flow patterns that can have significant effects on the performance and stability of hydraulic machinery. The Q criterion is a dimensionless parameter used to identify and describe cavitation and vortices. This criterion is based on the second inhomogeneity of the velocity gradient tensor and is used to identify regions with high circulation and strain rate in the flow field. These areas are often associated with the formation of cavitation and vortices. In the field of cavitation analysis, the Q criterion can help engineers and researchers visualize and understand the formation and behavior of cavitation bubbles in fluid flow. By identifying areas where the Q criterion is higher than a certain value, it is possible to identify areas that are prone to cavitation, which allows optimization of designs to reduce cavitation-related damage. Likewise, in the analysis of vortices with cavitation, the Q criterion can be used to identify and analyze the formation of vortices in the flow field. By examining regions where the Q criterion indicates high circulation and strain rates, engineers can gain an understanding of the formation and behavior of vortices that is critical to the design and operation of hydraulic machinery. In conclusion, the Q criterion plays an important role in the analysis of cavitation and the analysis of vortices with cavitation. This is a valuable tool for identifying and understanding the formation of cavitation and eddies in fluid flow, which ultimately helps to develop effective and reliable hydraulic systems. As mentioned, the Q criterion is a tool in OpenFOAM that is used to observe bubbles, vortices, and locate cavitation. The value Q comes from the definition of the velocity gradient tensor $\frac{\partial u_i}{\partial x_j}$ as following equations:

$$\frac{\partial u_i}{\partial x_j} = S + \Omega \tag{1}$$

S is known as the strain rate tensor defined by:

$$S = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{2}$$

Ω is known as the rotation rate or vorticity tensor defined by:

$$\Omega = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right) \tag{3}$$

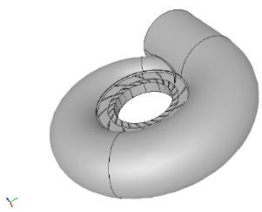
The Q criterion is directly derived based on the second invariant Q of the velocity gradient tensor with the following expression:

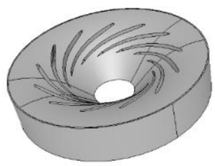
$$Q = \frac{1}{2}\left(\|\Omega\|^2 - \|S\|^2\right) \tag{4}$$

According to the definition, the Q criterion defines vortices as areas where the vorticity magnitude is greater than the magnitude of the rate of strain. It can be clearly seen that Q becomes zero at the wall, which indicates that the strain rate tensor and the vorticity tensor at the stationary wall are equal [6-8].

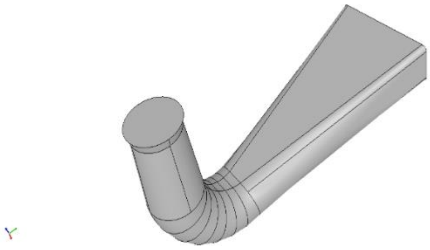
Results and discussions

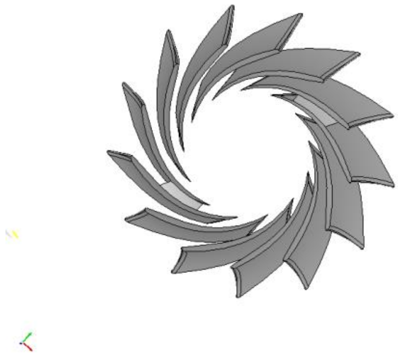
OpenFOAM software is used in this research. OpenFOAM is an open-source computational fluid dynamics (CFD) software used for modeling and simulating fluid flow and heat transfer. This software is widely used in various industries including aerospace, automotive, oil and gas, energy and environment. In the Francis turbine cavitation investigation, OpenFOAM can be used as an effective tool for modeling and simulating the fluid flow in the turbine. This software is able to accurately model the flow of fluids inside the turbine and provide useful information about the performance and optimization of the Francis turbine. Also, OpenFOAM provides the possibility of performing various analyzes to improve the performance of the Francis turbine. Overall, the use of OpenFOAM in Francis turbine cavitation can help improve the performance, efficiency and safety of these turbines, helping to reduce costs and increase productivity in this important industry. For this purpose, the pimpleFoam solver in OpenFOAM was used. This incompressible solver is able to show vortices and bubbles caused by cavitation by Q criterion. The model used in this article, as shown in figure 1, is selected based on the work of Zhang et al [9].



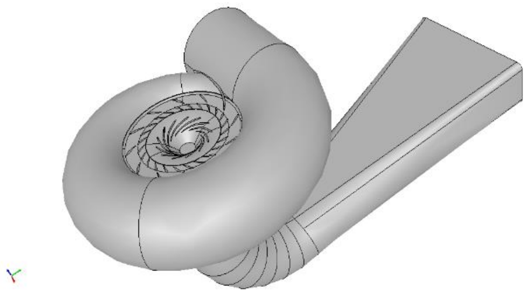


(a) Spiral casing **(b) Runner**





(c) Draft tube **(d) Blades**



(e) Complete assembly

Figure (1) Francis turbine components

In order to reduce computational costs in the current work, simulations have been performed in the runner and draft tube. It should be noted that geometry drawing and meshing were done in Salome software. After entering each of the mentioned geometries in the OpenFAOM software, two meshes are merged with each other using the merge Mesh command. This merging is such that the surface of the adjacent cells in the AMI boundary for the draft tube and the surface of the adjacent cells in the AMI boundary for the runner must be completely coincident. AMI boundaries are the definition of the common boundary or the concept of interface in computational fluid dynamics problems. Next, we will discuss about the cavitation patterns in this study. Cavitation patterns observed in simulation are shown in Figures 2-3. The first pattern belongs to cloud cavitation. Sheet or cloud cavitation refers to the formation of a large, coherent structure of vapor-filled cavities in a fluid flow due to rapid changes in pressure. This phenomenon is common in high-speed flows around underwater vehicles, propellers, or hydrofoils, where the pressure drops are severe enough to induce the formation of these extended vapor structures. Sheet cavitation can affect the performance and stability of marine vehicles and hydraulic systems. Leading edge cavitation occurs when cavitation bubbles form near the leading edge of the turbine blades due to high-speed flow and sharp changes in fluid direction. This phenomenon is particularly common in hydraulic turbines operating under high-pressure differentials. When water flows over the leading edge of the turbine blades, the pressure decreases, reaching levels below the vapor pressure of water, causing cavitation bubbles to form. These bubbles can collapse violently when they move into regions of higher pressure, leading to erosion and damage to the turbine blades. Managing leading edge cavitation is crucial for maintaining the efficiency and reliability of turbine systems. Strategies such as blade design optimization and flow control techniques are employed to mitigate the effects of leading edge cavitation in turbine applications. Vortex cavitation in Francis turbines occurs when vortices form within the flow, particularly in regions of high velocity or adverse pressure gradients. These vortices can lead to the formation of cavitation bubbles, which collapse upon reaching regions of higher pressure, causing erosion and damage to the turbine blades and other components. Factors such as operating conditions, turbine geometry, and flow characteristics influence the occurrence and severity of vortex cavitation in Francis turbines. Effective mitigation strategies include optimizing blade profiles, controlling flow rates, and ensuring smooth flow transitions throughout the turbine system.

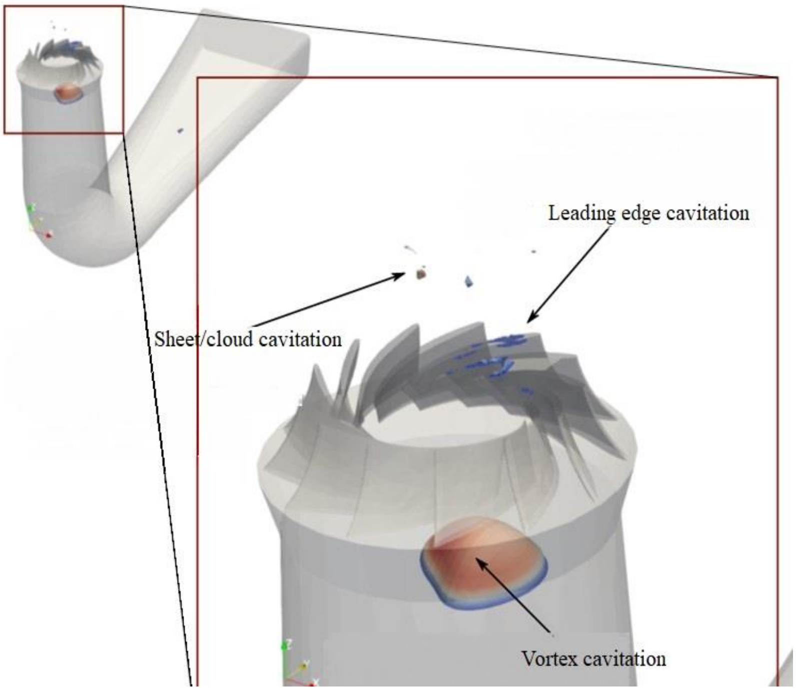


Figure (2) observed cavitation patterns

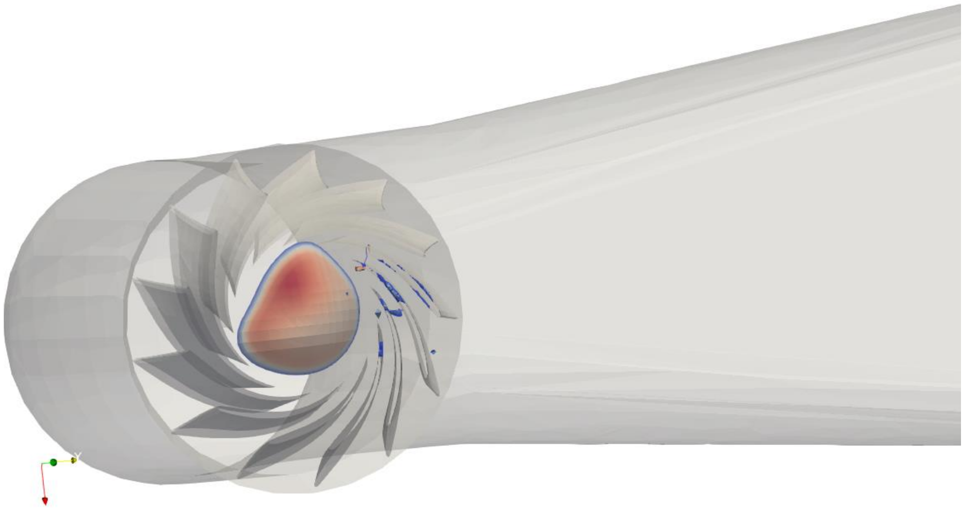


Figure (3) Another view to observe the cavitation patterns

Concluding remarks

Computational fluid dynamics (CFD) simulations are commonly employed to analyze and predict cavitation behavior and inform design modifications aimed at minimizing its impact on turbine performance and longevity. In this study by using OpenFOAM software and Q criterion, the location of vortices and bubbles caused by cavitation have been identified which result in three cavitation patterns. Each of these cavitation patterns can significantly impact turbine performance, efficiency, and durability, necessitating careful design considerations and operational control to mitigate their effects. The most powerful cavitation pattern observed belongs to the vortex cavitation at the draft tube inlet, while the sheet-cloud cavitation generally considered less harmful than others and are susceptible to suppression. By addressing cavitation effectively, turbine designers and operators can enhance performance, extend equipment lifespan, and minimize the environmental impact of turbine operations

References

[1] Kumar, P. and R. P. Saini (2010). Study of cavitation in hydro turbines—A review. Renewable and Sustainable Energy Reviews 14(1): 374-383.

[2] Feng, J., Zhao, N., Zhu, G., Wu, G., Li, Y., & Luo, X. (2024). Cavitation identification in a hydraulic bulb turbine based on vibration and pressure fluctuation measurements. Mechanical Systems and Signal Processing, 208, 111042.

[3] Escaler, X., E. Egusquiza, M. Farhat, F. Avellan and M. Coussirat (2006). Detection of cavitation in hydraulic turbines. Mechanical Systems and Signal Processing 20(4): 983-1007.

[4] Kumar, P., Singal, S. K., & Gohil, P. P. (2024). A technical review on combined effect of cavitation and silt erosion on Francis turbine. Renewable and Sustainable Energy Reviews, 190, 114096.

[5] Arndt, R., G. Balas and M. Wosnik (2005). Control of Cavitating Flows: A Perspective. Jsme International Journal Series B-fluids and Thermal Engineering - JSME INT J SER B 48: 334-341.

[6] Banko, A. J., & Eaton, J. K. (2019). A frame-invariant definition of the Q-criterion. Center for Turbulence Research Annual Research Briefs, 181-194.

[7] Jeong, J., & Hussain, F. (1995). Hussain, F.: On the identification of a vortex. JFM 285, 69-94. Journal of Fluid Mechanics, 285, 69-94.

[8] Liu, C., Gao, Y., Tian, S., & Dong, X. (2018). Rortex—A new vortex vector definition and vorticity tensor and vector decompositions. Physics of Fluids, 30(3).

[9] Zhang, H., & Zhang, L. (2012). Numerical simulation of cavitating turbulent flow in a high head Francis turbine at part load operation with OpenFOAM. Procedia Engineering, 31, 156-165.