

Article

Hydraulic Performance of Howell–Bunger and Butterfly Valves Used for Bottom Outlet in Large Dams under Flood Hazards

Grigorios L. Kyriakopoulos ¹ , Younes Aminpour ², Omid Aminoroayaie Yamini ^{3,*} , Azin Movahedi ³, S. Hooman Mousavi ³  and Mohammad Reza Kavianpour ³ 

¹ School of Electrical and Computer Engineering, National Technical University of Athens, 15780 Athens, Greece

² Department of Hydraulic, Hydro-Environmental Engineering, Water Research Institute, Ministry of Energy, Tehran 16589-54381, Iran

³ Laboratory of Hydraulic Models, LHM-Research Center, Department of Civil Engineering, K. N. Toosi University of Technology, Tehran 19967-15433, Iran

* Correspondence: o.aminoroaya@mail.kntu.ac.ir

Abstract: Floods control equipment in large dams is one of the most important requirements in hydraulic structures. Howell–Bunger valves and butterfly valves are two of these types of flow controls that are commonly used in bottom outlet dams. The optimal longitudinal distance (L) between the two Howell–Bunger and butterfly valves is such that the turbulence of the outlet flow from the butterfly valve should be dissipated before entering the outlet valve. Subsequently, the flow passing through the butterfly valves must have a fully developed flow state before reaching the Howell–Bunger valve. Therefore, the purpose of this study was to evaluate the optimal longitudinal distance between the Howell–Bunger and butterfly valves. For this purpose, different longitudinal distances were investigated using the Flow-3D numerical model. The ideal longitudinal distance obtained from the numerical model in the physical model was considered and tested. Based on the numerical study, the parameters of flow patterns, velocity profiles and vectors, turbulence kinetic energy, and formation of flow vorticity were investigated as criteria to determine the appropriate longitudinal distance. In addition, the most appropriate distance between the butterfly valve and the Howell–Bunger valve was determined, and the physical model was evaluated based on the optimal distance extracted from the numerical simulation. A comparison of the results from the numerical and the laboratory models showed that the minimum distance required in Howell–Bunger valves and butterfly valves should be equal to four times the diameter of the pipe ($L = 4D$) so as not to adversely affect the performance of the bottom outlet system.

Keywords: turbulence kinetic energy; dam; bottom outlet; flood hazards; flow patterns



Citation: Kyriakopoulos, G.L.; Aminpour, Y.; Yamini, O.A.; Movahedi, A.; Mousavi, S.H.; Kavianpour, M.R. Hydraulic Performance of Howell–Bunger and Butterfly Valves Used for Bottom Outlet in Large Dams under Flood Hazards. *Appl. Sci.* **2022**, *12*, 10971. <https://doi.org/10.3390/app122110971>

Academic Editor: Francesca Scargiali

Received: 9 October 2022

Accepted: 25 October 2022

Published: 29 October 2022

Publisher's Note: MDPI stays neutral with regard to jurisdictional claims in published maps and institutional affiliations.



Copyright: © 2022 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<https://creativecommons.org/licenses/by/4.0/>).

1. Introduction

Large dams, considered as “structures of public utility”, play an essential role in controlling and harvesting benefits from floods, and always pose potential risks to human life and property on their downstream side in the event of a flood. Flow control structures in large dams are one of the most vulnerable pieces of equipment during operation. The main tasks of these structures are to drain the excess flood of the dam, drain the incoming and settled sediments in the reservoir of the dam, control the volume of the reservoir, supply energy, and also supply downstream water for agricultural, drinking, and industrial uses. Howell–Bunger and butterfly valves are examples of the structures commonly used in bottom outlet dams and outlet intakes. This type of valve is usually installed at the end of the large pipe diameter or tunnels. Howell–Bunger valves show advantageous operation while radially discharging the flow into a conical expanding spray, since such a design does not require the valve to overcome excessive hydrostatic forces to open or close, enabling superb flow control via a moveable sleeve or gate that sits against the cone and is sealed against the valve body [1–3]. Howell–Bunger valves are widely used in flow regulation and

control in dam construction. The outlet flow from these valves collides with a fixed cone at a 90-degree angle mounted at the end of the valve they result in a 45-degree deflection. In the absence of a terminal hood, the outlet flow can be distributed as a fountain, divergent and hollow over a large area. In cases of limited outside space of the valve, the use of a fixed outer cylinder or hood is common, and the conical flow becomes axial due to the impact on the external hood. The Howell–Bunger valve is simple, inexpensive, and relatively safe from cavitation; therefore, it is used in dams and structures with many heads up to about 200 m. In Howell–Bunger valves water spreads like a fountain, so the energy of water particles is reduced due to high dispersion in space, resulting in significant energy loss [4,5]. Figure 1 shows the components of a Howell–Bunger valve along with the outlet flow conditions of this type of valve in the presence or absence of a hood. Under the conditions of using the hood in the flow path, the amount of spraying is reduced and, as a result, the flow rate coefficient is reduced. These structures are designed at a discharge coefficient of about 0.85 (about 0.78 with a hood) in 200–2500 mm sizes. Large valves operate for heights equivalent to heads of about 140 m while small valves operate for heights up to 300 m [6].

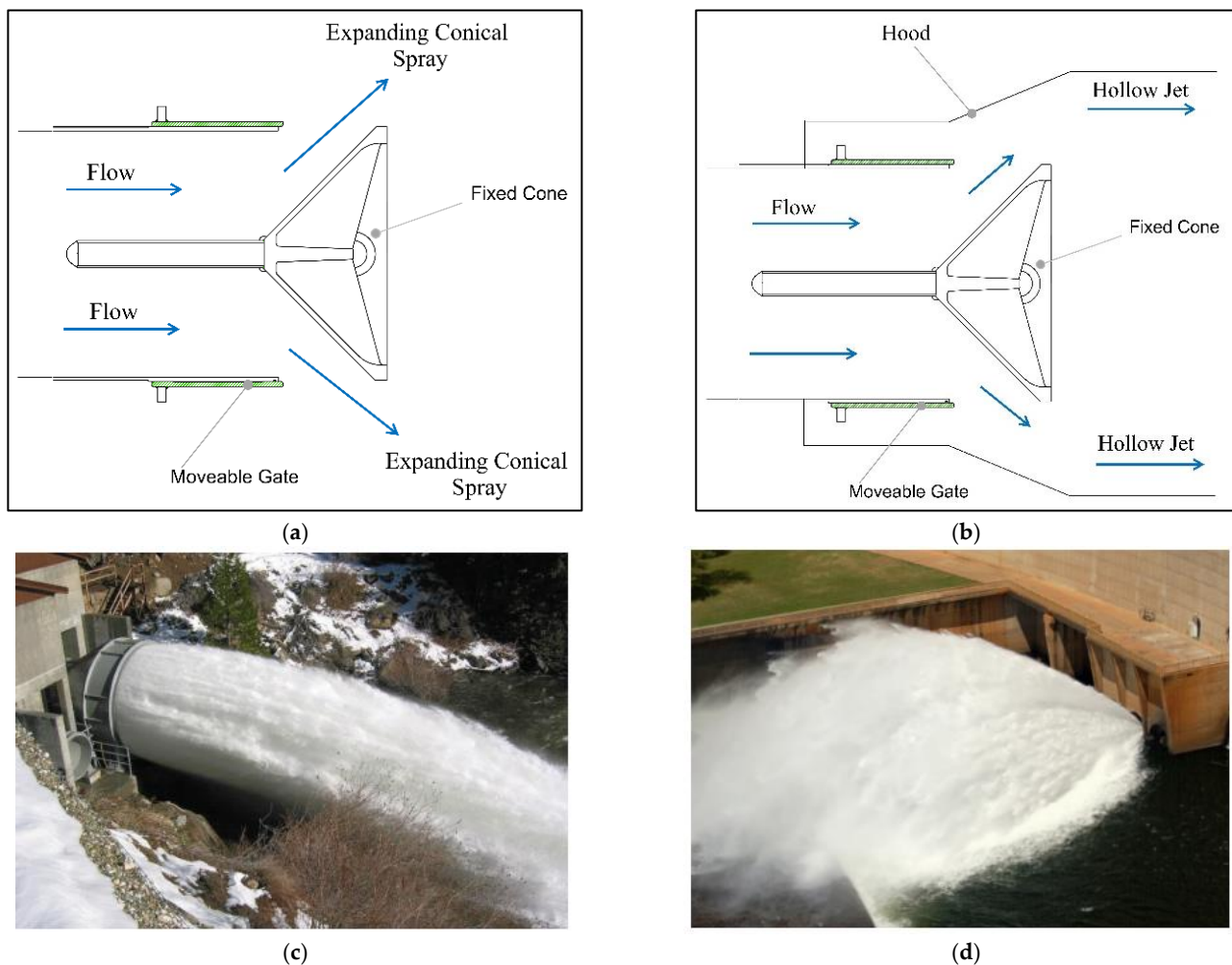


Figure 1. The Sectional view of typical (a) Howell–Bunger with hood; (b) Howell–Bunger without hood; (c) Howell–Bunger with hood of Salt River and Lower Bear Reservoirs, CA, USA; and (d) Howell–Bunger without hood of Jozini Dam, South Africa.

Butterfly valves are another type of valve widely utilized for flow regulation and control. Butterfly valves consist of three main parts: body, disc, and shaft. The advantageous characteristics of butterfly valves include a simple installation, low costs of installment, operation, and maintenance, as well as high flow capacity compared to other types of

valves [7,8]. Usually, when using Howell–Bunger valves, in the upstream area a butterfly valve is also considered for repair, inspection, operation and maintenance conditions in case of emergency. The flow through the butterfly valves can cause unfavorable turbulence conditions, which adversely affect the operation of the Howell–Bunger valves. This study reviewed previous studies in the field of Howell–Bunger valves and butterfly valves, while the innovation features and the research necessity of this study were also discussed. Unfavorable hydraulic/turbulence conditions in a valve cause a variety of negative effects such as cavitation, K factor head loss, noise, vibration, imprecise control, and excessive wear. Cavitation is particularly troublesome in some valves such as Howell–Bunger. In time, enough metal erodes from the valve surfaces that the device begins to leak, and control properties (ability to precisely regulate flow and/or pressure) are diminished. Eventually, the valves must be reconditioned or replaced. In high-performance applications, this may require the expensive interruption of important processes (e.g., hydraulic applications in large dams). Unfavorable hydraulic/turbulence conditions also cause noise and vibration that increase over time as valve integrity deteriorates. Consequently, these inefficiencies drive valve designs to be larger and less effective than they should be if turbulence did not enter the equation.

Computational fluid dynamics (CFD) techniques are now routinely employed to determine the cavitation behavior and the flow patterns in valves. The effects of cavitation on the performance of a proportional directional valve were investigated by Amirante et al. (2014) [9]. Nzombo (2017) [10] investigated the feasibility of obtaining optimal ball and butterfly valve projects with a reduced cavitation potential. Liu et al. (2020) [11] investigated cavitation in diaphragm valves with various apertures. Tabrizi et al. (2014) [12] further used pressure drop and vortex generation to study cavitation in ball valves at various apertures. Lee et al. (2016) [13] determined the optimization of a section of a globe valve geometry to reduce cavitation effects. Wu et al. (2003) [14] focused on cavitation in the Hollow-Jet valve utilized in Wang's experiments by using three-dimensional computational fluid dynamics (1999) [15]. These authors deployed pressured-based algorithms to simulate turbulent cavitation flows with turbulence closure accomplished by the turbulence equations and linked with a cavitation model. Subsequently, cavitation occurrences in the geometry's valve tip were determined. The applicability of turbulence and cavitation models to solve the set of fundamental equations is a common practice in running simulations in similarly reported studies. The axial fluid flow in a fixed-cone valve was simulated using the vortex method by Tsukiji et al. [16]. Cao [17,18] developed a boundary element method by integrating equations and the finite element method. This method enabled the performance of a numerical analysis of flow fields in a fixed-cone valve while conducting a special experimental equipment to investigate the pressure distribution on the conical surface of the fixed-cone valve and internal fluid power in the fixed-cone valve. Gao [19] also used the Galerkin finite-element approach to calculate the hydraulic pressure on the core surface of a fixed-cone valve and the internal flow fields in a fixed-cone valve under varied apertures. Jalilet al. [20] applied the Re-Normalization Group (RNG) turbulence model to simulate a cavity flow of a fixed-cone valve.

Cavitation and vibration inside a relief valve were experimentally explored and their interactions were discussed by Yi et al. [21]. Moreover, Li et al. [22] investigated the cavitation behavior of an electrohydraulic servo-valve by using both experiment and CFD and discovered cavitation at the nozzle tip and the flapper leading edge. Adamkowski and Lewandowski [23] introduced a numerical solution to solve the inline valve's cavitation problem. Considering the experimental and numerical approaches published, Zeng et al. [24] investigated the vibration and the noise in control valves by adjusting the pressure ratio. The dynamic feature of a shut-off valve was studied by Saha et al. [25]. Towards selecting cost-effective check valves, Tran [26] investigated pressure transients while the check valve was closed. Beune et al. [27] determined the opening statistical characteristics of high-pressure safety valves. Høset al. [28,29] investigated the dynamic behavior of pressure relief valves in the presence of gas and provided a mathematical model as well as a novel reduced-order mathematical model in order to capture the start

sites of instability and chatter. Similarly, Posa et al. [30] deployed the Direct Numerical Simulation (DNS) approach to investigate a directional hydraulic valve.

As discussed in the aforesaid technical literature overview, there is no comprehensive numerical and affirmative laboratory study to jointly influence the performance of butterfly valves and Howell–Bunger valves. The turbulence of the outflow flow from the butterfly valve, before the Howell–Bunger valve, disappears, and the flow reaches the Howell–Bunger valve with full development. Therefore, this study evaluated the optimal distance between the Howell–Bunger and the butterfly valves. The research objective was numerically simulated in various numerical scenarios and the research outcomes were confirmed using a physical model.

Previous works on bottom outlets CFD analysis primarily focused on valve design optimization, investigation of outlet flow patterns, streamline flow, and what measures should be taken to reduce vortex flow, but only a few were related to the prediction of flow patterns around valves and the calculation of turbulence kinetic energy for Howell–Bunger valves and butterfly valves. There are many numerical models on bottom outlet structures, but it was uncommon to look at the efficiency of the bottom outlet system that often uses Howell–Bunger and butterfly valves at the same time. This study was conducted to ensure that the flow in the pipe is suitable and creates good operating conditions for valves and bottom outlets systems. According to the hydrodynamic limits and other restrictions design, one of the most important goals of this study was to determine the optimum dimensions for the distance between Howell–Bunger and butterfly valves in bottom outlets systems.

2. Materials and Methods

The deployed hydraulic model of this study is the physical model of the downstream water supply system of Khodaafarin Dam on the Aras River in East Azarbaijan Province (Iran). The Aras River is among the most significant rivers of Iran in the Caspian Sea. The principles of the operational systems for the Howell–Bunger valves used in the experimental study and model set up are shown in Figure 2. In order to meet the water needs downstream of Khodaafarin Dam, four branches of the bottom outlet tunnel with Howell–Bunger valves (each with a maximum flow of $56 \text{ m}^3/\text{s}$) they were considered in parallel (with a central axis distance of 6 m from each other) and each with a diameter of 2 m. Each branch pipe has a butterfly valve with a diameter of 2.5 m in front of the Howell–Bunger valve. It is denoted that the stream's outflow is transmitted downstream through the stilling basin. A physical model with a scale of 1:15 of the mentioned system it was created and studied by the Water Research Institute of the Ministry of Energy (Tehran, Iran). The reason for constructing a physical model was to determine the optimal longitudinal distance between the butterfly valves and the Howell–Bunger valve so that the flow conditions passing through the butterfly valve do not affect the inlet flow to the Howell–Bunger valve, while the flow enters the Howell–Bunger valve in a fully developed manner. For this purpose, first, different longitudinal distances were deployed in the numerical model Flow 3D, Ver. 12.2 [31], and finally, the suitable distance obtained from the numerical model was considered and tested in the physical model. The pointed distances were considered in the numerical model as ratios of pipe diameters (2D, 3D, 4D, and 5D), and the numerical model calibration was performed using flow rate. Figure 2 shows images of the hydraulic model developed in laboratory conditions.



Figure 2. Preparation, construction, and installation of Howell-Bunger and other details of downstream system in the physical model: (a) the geometric form of the plexiglass hood; (b) the geometric form of the plexiglass Howell-Bunger; (c) installation of the bottom outlets in the stilling basin; and (d) views from the physical model of the stilling basin.

Flow-3D is a CFD software that handles the most difficult free-surface flow problems with accuracy, speed, and reliability. The three-dimensional geometry (*.stl file format) may then be imported into the computational fluid dynamics algorithm to analyze the hydrodynamic interactions. Previous research indicates that computations using Flow-3D were successful [32,33]. Flow-3D, like other commercial software, uses numerical methods to trace the location of fluid and solid surfaces and apply the proper dynamic boundary conditions; however, it has some unique features, such as the FAVOR (fractional area volume obstacle representation) method for defining complex geometric regions within rectangular cells and multiblock meshing. In the fractional area volume obstacle representation approach, a surface is permitted to cut through an element, and its location is recorded in terms of the fractional face areas and fractional volume of the element that is not

covered by the solid, rather than repositioning the edges of the element. The mathematical approximation consequences of this fractional area for establishing solid boundaries are the same as those of a deformed (i.e., body-fitted coordinates) mesh technique. The FAVOR method's most essential feature is that approximations of fluid-dynamic variables are limited to the open areas of components [34,35].

Numerical modeling of the hydraulic conditions of the flow in the area between the butterfly valve and the Howell–Bunger valve was performed using Flow-3D software. The total solid body of the bottom outlets' system, including the four sub-branches pipeline, Howell–Bunger valves, butterfly valves, aeration system, and stilling basin with all its details, was made in 3D by a geometric simulation software such as AutoCAD, CATIA, SolidWorks [36]. This simulation was based on the SolidWorks 2012 software, and an STL file was created. As seen in Figure 3, the model's geometry was designed and constructed with the highest accuracy, while supporting complete and informative details on the flow field.

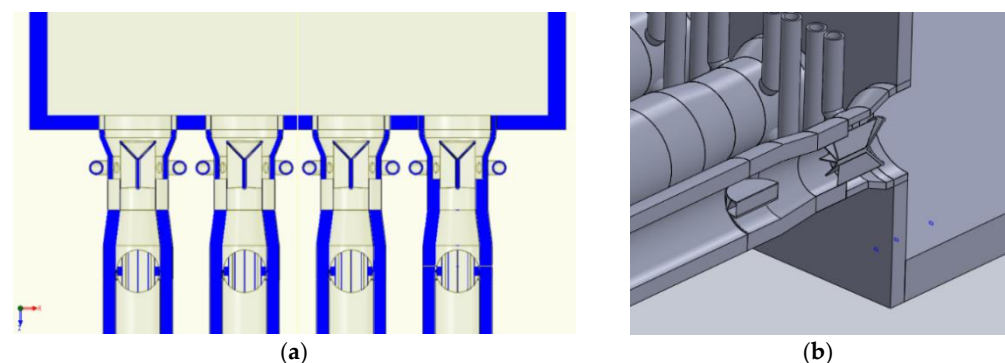


Figure 3. The SolidWorks model of stilling basin and Howell–Bunger valves: (a) top view with a schematic illustration of the section areas; and (b) details of the Howell–Bunger valve and butterfly valve in 3D Model.

The whole space was created by the mesh block and divided into cells of a certain size during the meshing stage under fixed points in specific headings of the solid body. Then, the desired physically shaped structure was achieved by utilizing obstacles called baffles. Adding a baffle with specific coordinates inside the block was recognized by the model as a barrier against fluid flow. It should be noted that for the baffle to work properly, the solid option must be selected at the coordinate definition. In order to cover the whole solid body, a single branch pipe with a stilling basin was selected in two mesh blocks. The selection of the range and size of blocks was materialized on the basis that there was as little space as possible in the branch area as empty space. In Figure 4, the solid geometry was added to the Flow3D numerical model, and was created in the Flow3D numerical model using two mesh blocks and a computing domain for the downstream water supply system of Khodaafarin Dam. It is noteworthy that the cells of the Flow3D numerical model are of the rectangular cube element type.

The FLOW-3D mesh generation technique employs a structured, rectangular, and Cartesian mesh that is independent of the geometry being used, providing the user with convenience and flexibility. The size of the cell used is 27 mm, and the simulation is run with a mesh block (each cell was 27 mm in the x -, y -, and z -axes). The block mesh contained 801,783 cells (Figure 4a). Boundary conditions should be appropriately defined. Based on the flow characteristics, the specific pressure at the top of the basin was set to atmospheric pressure. The entrance area of the inlet pipe cross section was set at a flow rate (Q). The far end of the basin outlet was defined as an outflow, the boundary between the adjacent two-blocks is symmetry (S), and the remaining boundaries and the flow domain envelope were defined as a wall (W) (Figure 4b).

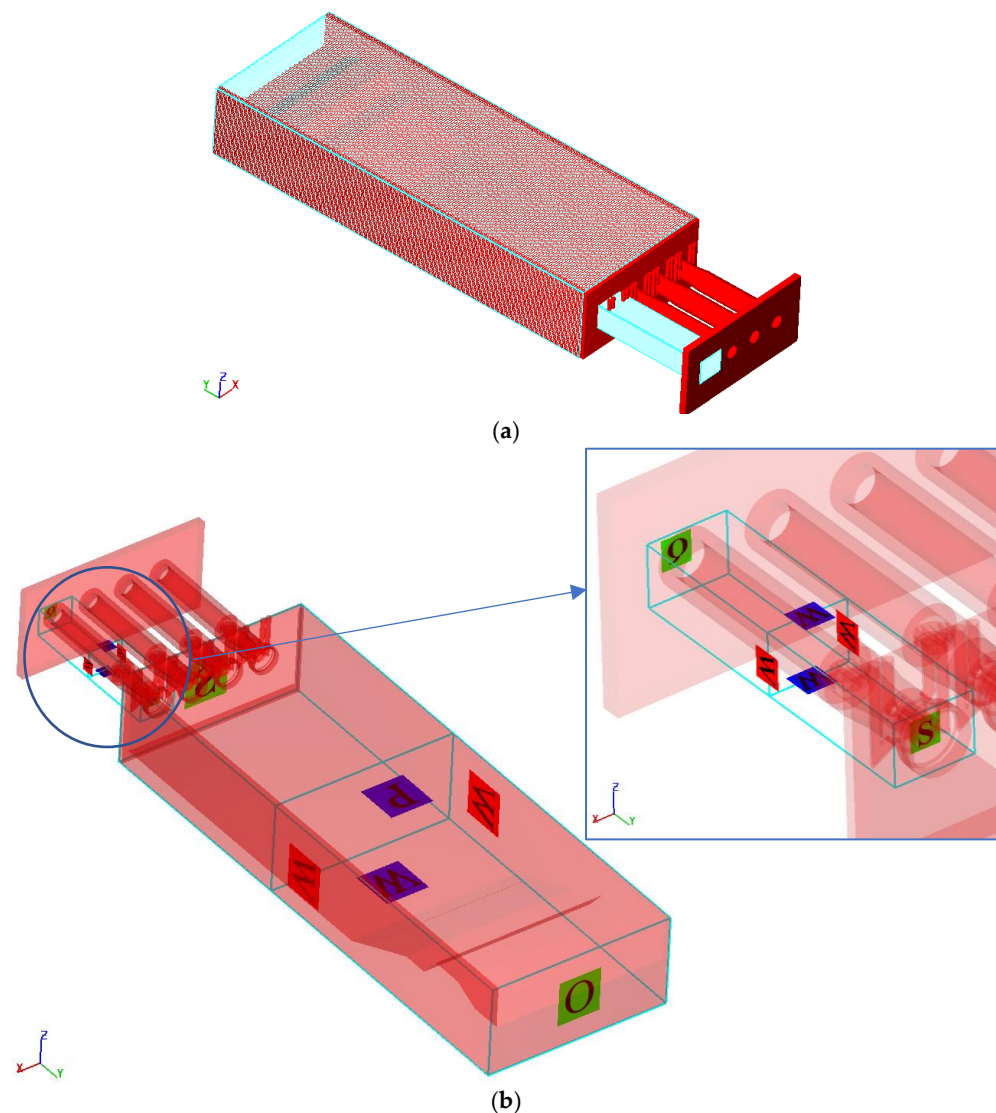


Figure 4. (a) Flow3D-generated orthogonal mesh associated with STL; and (b) domain and boundary conditions in the Flow3D numerical model.

The steady-state simulation used an iterative scheme to progress to convergence. Persistent oscillations in the residuals plot (solver diagnostics) and/or oscillations in a key monitor, such as volume flow rate, with increasing iterations, is a good indicator that the flow may be unsteady (transient) and the simulation needs to be run as an unsteady simulation. When there is uncertainty as to whether our simulation is unsteady or steady-state, it is always worth running a steady-state simulation first because it typically takes an order of magnitude less CPU (central processing unit) time to complete. If the steady-state simulation is sufficient, time will be saved over running an unsteady simulation. The steps in the simulation are to calibrate, to correctly extract the results and accurately validate these model results derived from the numerical model. This means that the effects of external factors should be minimized, and the model conditions can closely represent the prototypical conditions. However, it is necessary to achieve stable conditions of calibrating a numerical or laboratory model. In this study, the derived results of the numerical model were verified in the full opening of the valves for maximum flow, where the outlet flow of the numerical model can be equal to the flow applied in the boundary conditions. In Figure 5, the values of the outlet flow in the steady state conditions in the numerical model were compared with the applied boundary conditions. It was calculated that the flow rate for numerical modeling was equal to $56.25 \text{ m}^3/\text{s}$ and the opening of the valves was 100%.

In this research, after running numerous different models with the existing numerical model, the acceptable time to extract the results was determined to be 4 s (Figure 5).

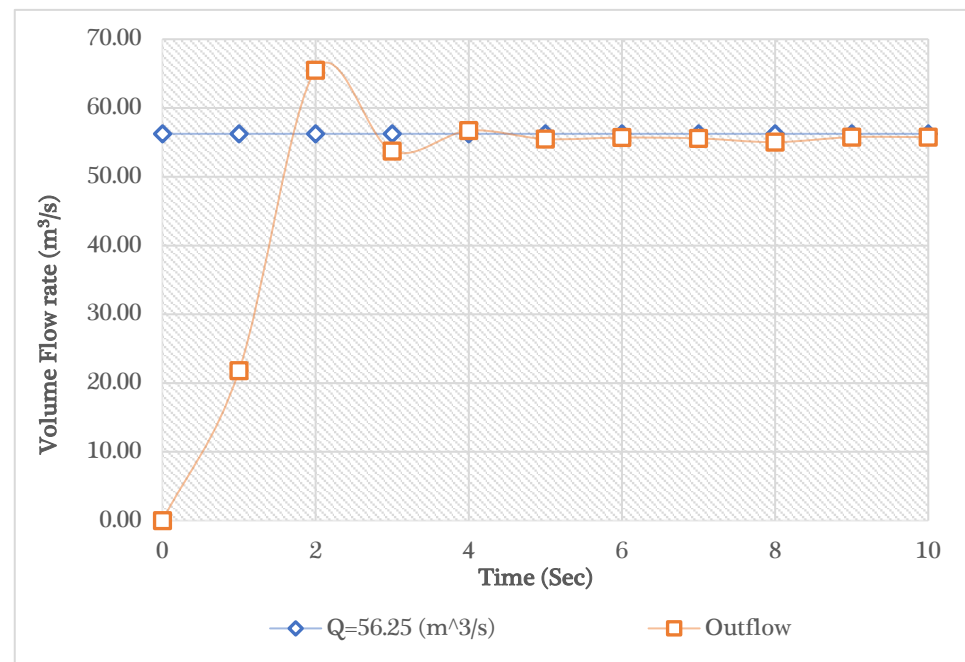


Figure 5. Comparison of the flow rates between inflow and outflow from Flow3D numerical model.

In the Table 1, the values of the simulation of the hydrodynamic parameters including average velocities (V), pressure (h), and flow rate (Q) in the Flow3D numerical model have been calibrated and validated in relation to the experimental model. The relative error values of the following equation are used:

$$\text{Error \%} = 100 \times \frac{X_{Exp} - X_{Num}}{X_{Exp}} \quad (1)$$

where, X_{Exp} is the actual value of the parameter (laboratory values) and X_{Num} is the simulated value of the parameter. Based on the findings of numerical modeling, the maximum relative error of the numerical model in the parameters of average velocities (V) is equal to 5.95%. Additionally, the maximum error of simulating the pressure (P) compared to the experimental modeling state is calculated to be 5.15%.

Table 1. Comparison of validation and accuracy results and numerical model calibration for hydrodynamic parameters.

Hydrodynamic Parameters	V (m/s)	Pressure (Pa)	Q (m³/s)
Experimental Modeling	17.8	24,571	56.25
Numerical Modeling	16.74	25,837	54.83
Error %	5.95	5.15	2.52

3. Results and Discussion

3.1. Scenario I: Longitudinal Distance between Two Valves $L = 2D$ (4 m)

Based on the numerical model results it was reported that the flow conditions in the area between the butterfly valve and the Howell–Bunger valve (contractions with a length of 4 m from a diameter of 2.5 to 2 m) were investigated. For this purpose, the numerical model outputs -being based on the distribution of velocity values in the cross-section- they must be firstly considered. Besides, a velocity distribution was extracted for the mentioned region, as it is shown in Figure 6a. According to Figure 6, the presence of a

butterfly valve in the pipeline as well as the presence of a contraction downstream they caused sudden changes in velocity, and velocity profiles showed that these immediately observed changes were intensified at the Howell–Bunger valve and do not create suitable conditions at this point for fluid flow. The short distance between the butterfly valve and the Howell–Bunger valve prevented the flow from developing upstream of the Howell–Bunger, which was inappropriate for the bottom outlet system. Figure 6b shows the stream-wise velocity component in the section before the flow entered the valves and downstream of the butterfly valve, the intensive turbulence of the flow, and the formation of cross flows in this area.

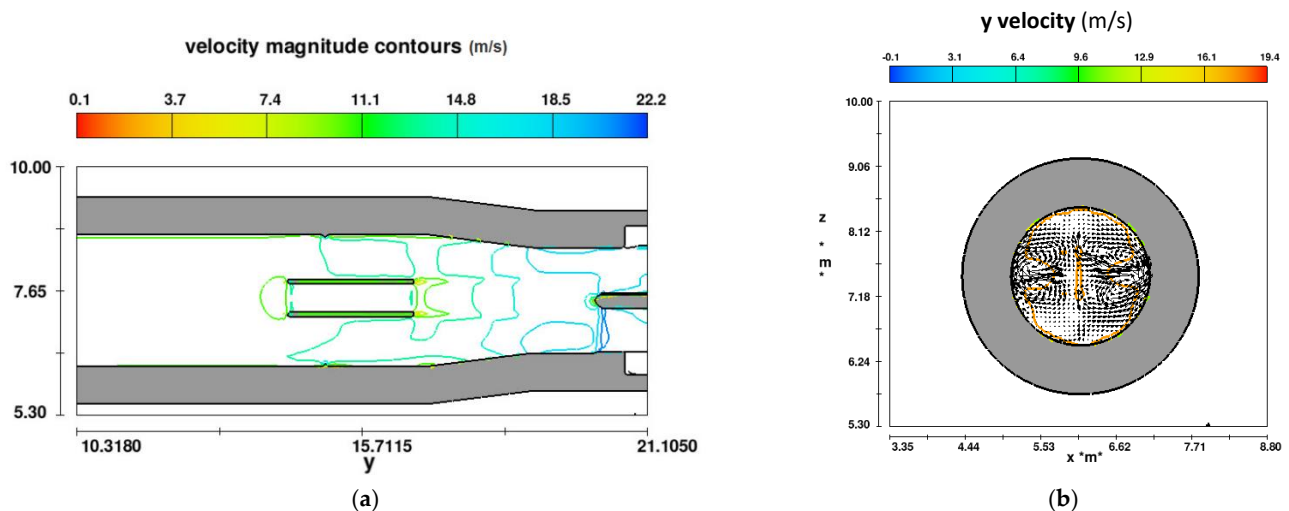


Figure 6. (a) sectional view of velocity magnitude contours for the longitudinal distance between two valves = $2D$; and (b) cross-flow vector and contours of the stream-wise velocity component.

Another significant parameter considers the effect of the presence of a butterfly valve at close distances to the Howell–Bunger valve in the studied model, as well as the changing ways of the turbulence kinetic energy. Sudden and cross-sectional changes in this parameter can lead to unfavorable hydraulic conditions. Based on the numerical model results, the changes in the turbulence kinetic energy parameter in the area between the butterfly valve and the Howell–Bunger valve were investigated and presented in Figure 7a. As shown in Figure 7, the presence of a butterfly valve and a contraction in the upstream area created the conditions for increased turbulence kinetic energy until the inlet flow to the Howell–Bunger valve. Due to the short distance between these two valves, the turbulence kinetic energy created in this path was not dissipated and penetrated the Howell–Bunger valve. Another examining parameter of the numerical model is related to the determination of the hydraulic conditions of the flow between the butterfly valve area and the Howell banger is how the vorticity changes after the flow passes through the butterfly valve (Figure 7b). According to Figure 7b, the vorticity flow due to velocity changes can be amplified after the current passes through the butterfly valve and then it can be transferred to the Howell–Bunger valve. The vorticity flow formed in the cross-section before the current can enter and then it flowed in the contraction region, at the direction of the current into the Howell–Bunger valve, which can cause adverse effects on the system. Considering the evaluation of the numerical model results, and taking into consideration some of the most important hydraulic parameters, including velocity profiles, turbulence kinetic energy, and vortex or vorticity formation, it can be concluded that the proposed distance of 4 m ($2D$) with one contraction was not desirable; therefore it should be supplemented with numerical studies. Based on these results, flow patterns can be examined based on the longitudinal distance of three to five times the pipe's diameter.

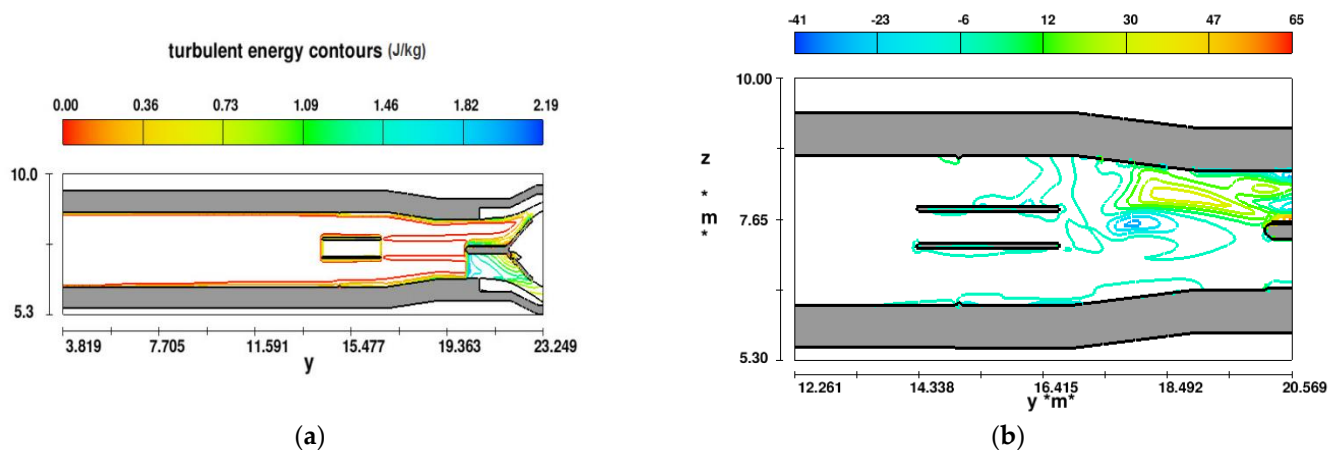


Figure 7. (a) sectional view of turbulent energy contours for the longitudinal distance between two valves = 2D; and (b) vorticity contours.

3.2. Scenario II: Longitudinal Distance between Two Valves $L = 3D$ (6 m)

Using the Solid Works 2020 design tool, the pipeline was re-prepared and executed for each of the geometric conditions of the location of the butterfly valve from the Howell–Bunger numerical model. In addition, the hydraulic conditions of the flow in the pipeline for a distance of 6 m (3D) were investigated. It is noteworthy that the increase in the distance between the butterfly valve and the Howell–Bunger valve was created in a pipe with a diameter of 2 m. Actually, the contraction did not change from 2.5 m to 2 m in diameter, and this increase was applied between the contraction and the Howell–Bunger valve. Velocity profiles in the area between the butterfly valve and the Howell–Bunger valve were extracted, as shown in Figure 8. Figure 8 velocity profiles revealed that by increasing the distance between the butterfly valve and the Howell–Bunger valve from 4 m to 6 m more conditions were offered for the development of flow in the pipe after passing the butterfly valve. A comparison of the velocity distribution at this point in 2D and 3D showed that by increasing the distance between the butterfly valve and the Howell–Bunger valve, it can result in the better formation of velocity profiles before the butterfly valve. However, it seems that the flow conditions for the complete stability of the velocity distribution cannot be formed at the inlet section of the Howell–Bunger valve, thus, it could be more feasible researchers to consider the numerical model in terms of 4D and 5D conditions.

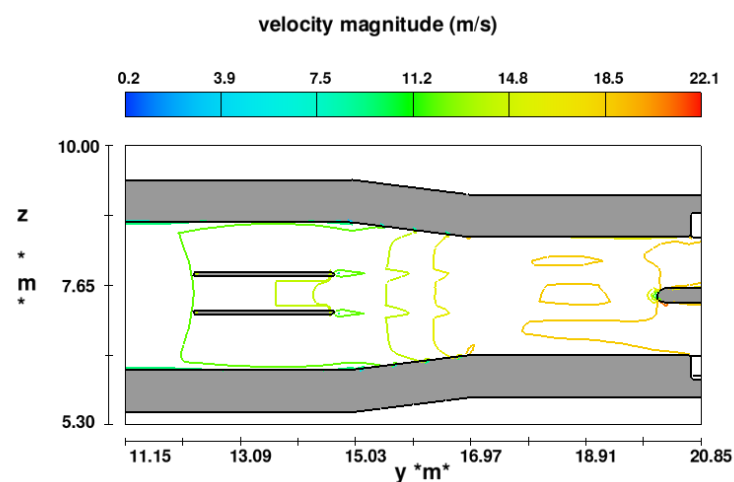


Figure 8. The Sectional view of velocity magnitude contours for the longitudinal distance between two valves = 3D.

Another parameter of research interest was to determine the ways under which the turbulence kinetic energy changes between the two valves. As previously noted, sudden and cross-sectional changes in this parameter indicate that flow expansion cannot occur in the enclosed and pressurized pipes and can induce unfavorable hydraulic conditions. Moreover, utilizing the numerical model results it can be a precise and comprehensive understanding of how the turbulence kinetic energy parameter changes in the areas between the butterfly valve and the Howell–Bunger valve for 3D conditions (Figure 9a). Figure 9 disclosed that the perturbation energy continued to the areas close to the Howell–Bunger valve due to the presence of a butterfly valve in the pipe downstream of the valve. Moreover, this energy was dissipated during contractions. According to the numerical model results, distance increased between the butterfly valve and the Howell–Bunger valve from 2D to 3D, and was essential in order to completely dissipate the turbulence energy. Figure 9b further showed that the vorticity profile changes at a distance of 6 m, which can be interpreted since, as the flow ran through the cross-section of the butterfly valve, then the vorticity created by the butterfly valve it continued in the 6 m long pipe to the location of the valve. However, the formation of vorticity in 3D conditions was reduced compared to 2D conditions, thus, supporting a more regular distribution pattern. However, depending on the other conditions, this distance can also be checked to ensure proper duct function for 4D and 5D conditions.

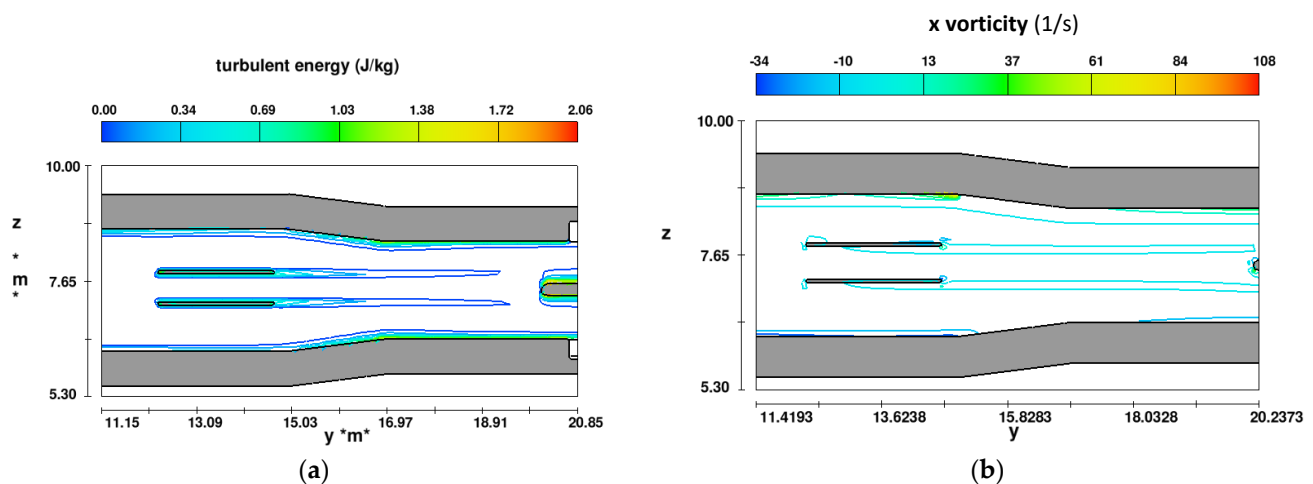


Figure 9. (a) sectional view of turbulent energy contours for the longitudinal distance between two valves = 3D; and (b) vorticity contours.

3.3. Scenario III: Longitudinal Distance between Two Valves $L = 4D$ (8 m)

The comparison of flow field conditions in the areas between the butterfly valve and the Howell–Bunger valve for 4 m (2D) and 6 m (3D) showed that the hydraulic conditions, especially in terms of velocity profiles, turbulence kinetic energy, and vorticity, with increasing longitudinal distance, they were all improved. However, the unstable hydraulic conditions in the mentioned parameters in the areas close to the Howell–Bunger valve, they proved the necessity of modeling while increasing the distance between the butterfly valve and the Howell–Bunger valve. Subsequently, the numerical modeling of the flow field in the butterfly and Howell–Bunger valve areas was investigated based on 4D (8 m) distance conditions. The numerical model results regarding the velocity distribution in the areas of the butterfly valve and Howell–Bunger valve for conditions of 8 m at an opening of 100 valves, they were presented in Figure 10. As shown in Figure 10, the velocity profiles downstream of the butterfly valve they were fully developed, and stable hydraulic conditions were observed in the numerical model. Comparing the results between 3D and 4D modes, it was proven that the pipe length increase in this area can significantly improve the velocity distribution conditions. Subsequently, the positioning of a butterfly

valve on the velocity profiles in the sections before the flow inlet, it can only slightly affect the Howell–Bunger valve.

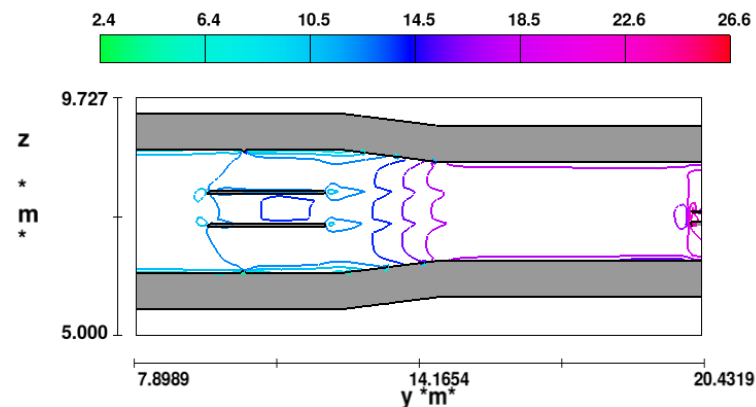


Figure 10. The Sectional view of velocity magnitude contours for the longitudinal distance between two valves = 4D.

At this point it is that critical researchers to ensure the hydraulic performance of the areas between the butterfly valve and the Howell–Bunger valve, while considering the results of the turbulence kinetic energy for the distance of 8 m (4D), as depicted in Figure 11. The numerical model results stressed that the amount of turbulence kinetic energy in the lower part of the butterfly valve decreases sharply with increasing the distance between the butterfly valve and the intake valve, because the fully developed flow conditions can be well provided by increasing this distance. The distribution of perturbation energy confirmed the correct hydraulic operation of the flow in the pipe due to the proper longitudinal distance of the butterfly valve from the Howell–Bunger valve. For ease of comparability, the formation conditions of vorticity in the area between the butterfly valve and the Howell–Bunger valve were extracted from the numerical model, as shown in Figure 11. It was argued that the vorticity caused by the butterfly valve decreased in the section before the flow entering the Howell–Bunger valve.

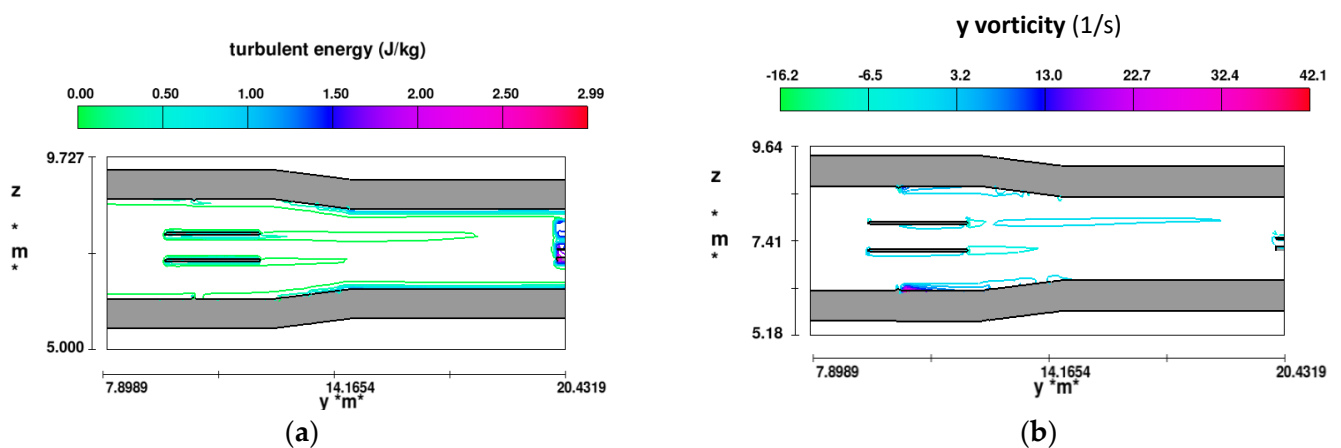


Figure 11. (a) sectional view of turbulent energy contours for the longitudinal distance between two valves = 4D; and (b) Vorticity contours.

3.4. Scenario IV: Longitudinal Distance between Two Valves $L = 5D$ (10 m)

Based on the results presented so far it can be stated that the hydraulic performance of the pipe in the case of using a distance of 8 m ($L = 4D$) compared to 6 m ($L = 3D$) can be significantly improved, resulting in a fully developed flow. In the scenario IV the difference in increasing this path it was considered by modeling similar conditions for a distance of 10 m ($L = 5D$) between the butterfly valve and the Howell–Bunger valve. Moreover, the optimal distance was selected. For this, the results of the numerical model simulation regarding the velocity distribution parameter in the area between the butterfly valve and the Howell–Bunger valve for a distance of 10 m ($L = 5D$), as depicted in Figure 12. Indeed, Figure 12 revealed that the distance of 10 m ($L = 5D$) between the butterfly valve and the Howell–Bunger valve can provide the fully developed flow conditions and the trend of changes in the velocity profile distribution at this point with the distance conditions of 8 m did not change noticeably. Therefore, flow velocity changes in this section can be considered the same at intervals of 8 and 10 m. In order to compare the turbulence energy conditions of the flow in the pipe, the results of the numerical model for this parameter have also been considered (Figure 12). Figure 12 showed that the amount of turbulent energy was completely dissipated after the flow passes through the butterfly valve in the pipe by creating a suitable distance from the Howell–Bunger valve, and the conditions were the same as that before the butterfly valve section in the pipe. The Such distance conditions prevalence between the two valves, they confirmed the correct hydraulic operation of the flow for a distance of 10 m between the valves. However, there was no significant difference between the results for two distances of 8 m and 10 m between the two valves. Thus, at a distance of 10 m, only the distance of the formation of turbulent energy from the location of the Howell–Bunger valve can be safely considered. In order to investigate the formation of vorticity due to the presence of a butterfly valve in the pipe, the numerical model results at the last two cases they were also investigated. The simulation results were extracted from the model, analyzed, and represented at Figure 12, revealing that the vortex due to (a) the presence of a butterfly valve in the pipe, (b) the distance of 10 m from the Howell–Bunger valve, it was at a completely stable condition and confirmed the development and non-formation of vorticity flow.

3.5. Evaluate the Results of Numerical

In this co-evaluation section, after checking (a) the optimal distance between the butterfly valve and the Howell–Bunger valve, as well as (b) the difference between the output results of the two distances of 8 m and 10 m, in order to save operating costs, then, the distance between the two valves was set at 8 m (equivalent to four times the diameter of the pipe). Besides, the desired distance with the butterfly valve was set and installed in one of the branches of the physical model. For the quantitative evaluation of the flow conditions in the distance between the butterfly valves and the Howell–Bunger valve, a dye injection flow visualization was added at a distance of 20 m from the howler valve (133.3 cm in the model). Figure 13 showed images of the flow situation in the area. In these images of Figure 13 it was concluded that the dye injection flow visualization at the inlet section of the intake valve can cover the entire section of the pipe and enter the Howell–Bunger valve uniformly and in a steady state. Laboratory results also confirm that, at low flow velocities, the fluid motion within a pipe is smooth and laminar, while at high velocity, the motion quickly becomes complex and turbulent. In some intermediate range of flow velocities, the flow is neither fully laminar nor fully turbulent, but rather a complicated combination of these two that varies over both space and time in a highly intermittent and unpredictable fashion.

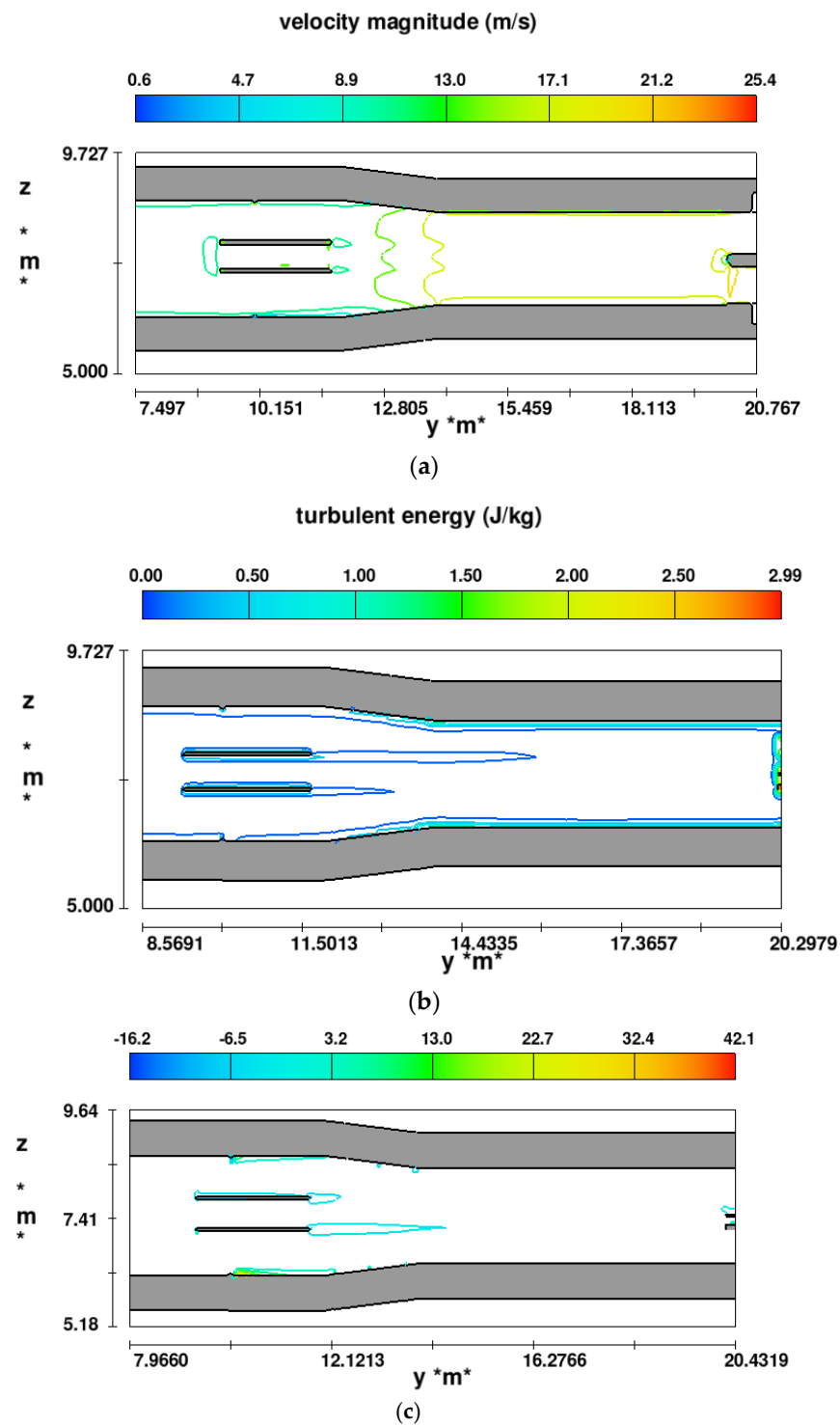


Figure 12. (a) The Sectional view of velocity magnitude contours for the longitudinal distance between two valves = 5D; (b) The Sectional view of turbulent energy contours for the longitudinal distance between two valves = 5D; and (c) Vorticity contours.

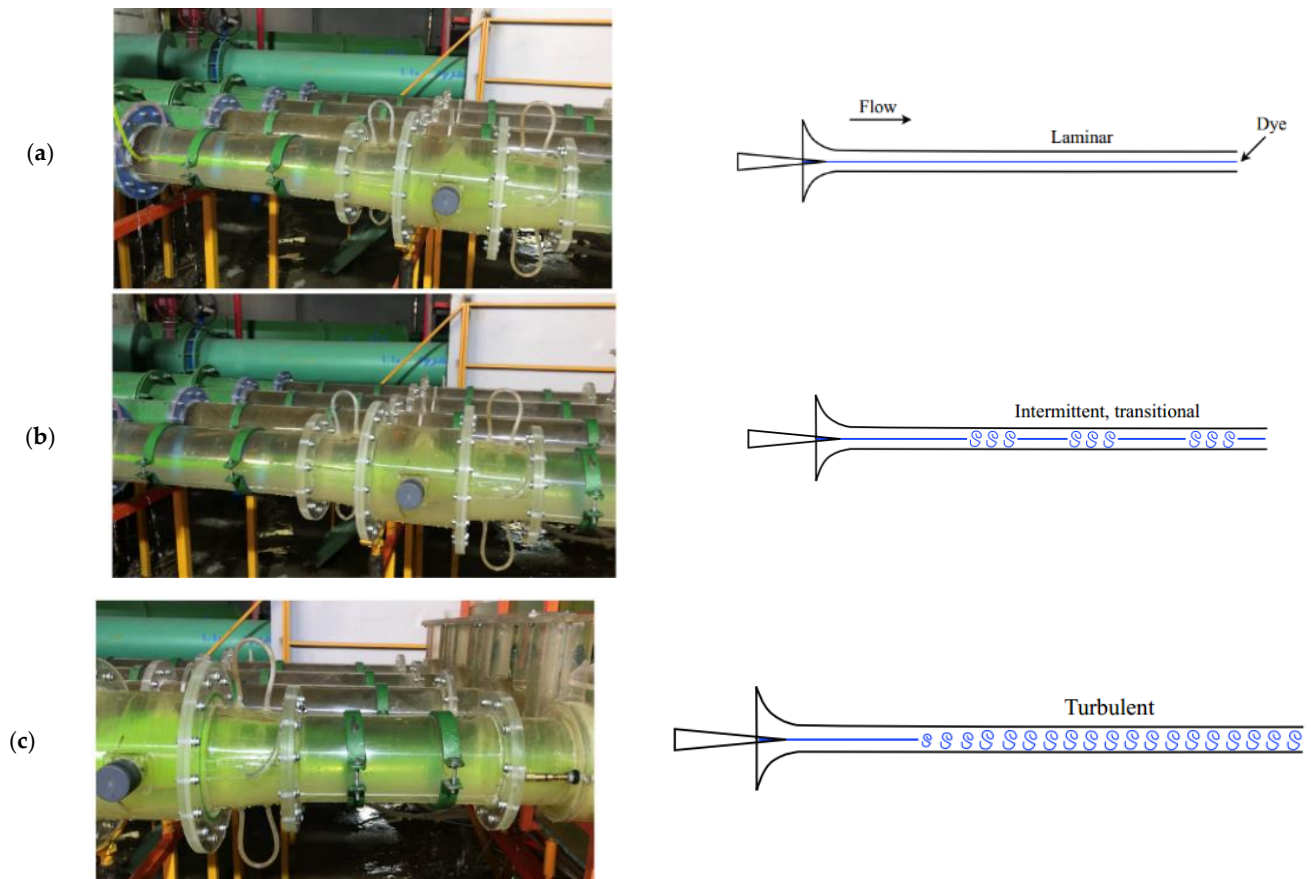


Figure 13. The regimes of pipe flow are illustrated by experimental modelling and sketches modeled after those in Reynolds (1883): (a) at low flow velocities, the fluid motion is laminar and dye injected at the pipe inlet produces a straight streamline; (b) the transitional regime occurs at intermediate flow velocities where the flow exhibits irregular, intermittent turbulence; and (c) at high flow velocities, swirling motions of injected dye reveal that the fluid becomes turbulent within a short distance of the pipe inlet.

In Del Toro's (2012) [37] study based on numerical simulation of butterfly valves, the required values for the velocity inlet boundary when using the k- ϵ turbulence model, such as velocity vectors, turbulent kinetic energy, Reynolds Number, and turbulent dissipation rate, were extracted from the periodic simulations and considered for the inlet for the 2.5D-long upstream pipe. In the Del Toro (2012) study, a distance of 2.5D times the diameter of the pipe has been confirmed to create conditions for the fully developed flow. Considering the existence of the Howell–Bunger valve and also the local expansion after the butterfly valve in the present study, it seems necessary to consider a distance of at least 5D times the diameter of the pipe.

Tao et al. (2022) [38] researched the DN50 butterfly valve and studied the flow situation of butterfly valves with different shafts. The resistance characteristics, flow state characteristics, Reynolds Number, and flow instability of butterfly valves with different structures were studied by experiments and numerical simulations. The results of the study showed that after 4D, the velocity in the valve is basically equal to the outlet velocity. The absence of velocity fluctuations shows that the developing flow becomes a fully developed flow, where flow characteristics no longer change with increased distance along the pipe. According to the location of the Howell–Bunger valve near the butterfly valve in the present study and the evaluation of the results presented in Tao et al. (2022) studies, the minimum distance between the butterfly valve and the butterfly valve was confirmed to be 5D times the diameter of the pipe.

4. Conclusions

Flood hazard control is linked to the bottom outlets as a secondary function of the dam's operation since, apart from its main operation, its discharge contributes to the attenuation of flood peaks. The three-dimensional modeling of the flow field in different conditions of distances between the butterfly valve and the Howell–Bunger valve, it revealed that the proposed distance conditions were not suitable in the case of sources positioning at distances equal to twice the cross-section diameter ($L = 2D = 4\text{ m}$). In such a case there were not fully shaped conditions. Under these conditions the presence of a butterfly valve affects the flow conditions of the Howell–Bunger valve and provides the conditions of forming undesirable and destructive hydraulic phenomena such as vibration, cavitation, and flow rate reduction. This study further investigated the most important hydraulic parameters of the flow, including velocity profiles and vectors, turbulence kinetic energy, and vortex and vorticity formation as the criteria that determine the appropriate longitudinal distance. Respectfully, the most appropriate distance between the butterfly valve and the Howell–Bunger valve in the bottom outlet dam was proposed to be at least four times the pipe diameter ($L = 4D$). However, it is necessary to signify that in the absence of special conditions and limitations of the project and in order to ensure maximum hydraulic operation of the flow, it is plausible and realistic to consider a distance equal to five times the diameter of the pipe ($L = 5D$) between the two valves.

Author Contributions: Conceptualization, O.A.Y., Y.A., A.M. and M.R.K.; methodology, O.A.Y., Y.A., A.M., S.H.M. and G.L.K.; software, O.A.Y. and A.M.; validation, O.A.Y., Y.A., A.M. and S.H.M.; data curation, O.A.Y. and Y.A.; writing—original draft preparation, O.A.Y., Y.A., A.M., S.H.M. and G.L.K.; writing—review and editing, O.A.Y., Y.A., G.L.K. and S.H.M.; supervision, M.R.K. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: All data used to support the findings of the study are included within the article.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Johnson, M.C.; Dham, R. Innovative Energy-Dissipating Hood. *J. Hydraul. Eng.* **2006**, *132*, 759–764. [\[CrossRef\]](#)
2. Sazonov, Y.A.; Mokhov, M.A.; Gryaznova, I.V.; Voronova, V.V.; Tumanyan, K.A.; Frankov, M.A.; Balaka, N.N. Development and Prototyping of Jet Systems for Advanced Turbomachinery with Mesh Rotor. *Emerg. Sci. J.* **2021**, *5*, 775–801. [\[CrossRef\]](#)
3. Liu, Y.L.; Lv, B.; Wei, W.L. Large Eddy Simulation of Gas-Liquid Two-Phase Flow for a Nested Type Fixed-Cone Valve. In *Applied Mechanics and Materials*; Trans Tech Publications Ltd: Stafa-Zurich, Switzerland, 2012; Volume 170, pp. 2458–2463.
4. Lewin, J. *Hydraulic Gates and Valves: In Free Surface Flow and Submerged Outlets*; Thomas Telford: London, UK, 2001.
5. Yamini, O.A.; Movahedi, A.; Mousavi, S.H.; Kavianpour, M.R.; Kyriakopoulos, G.L. Hydraulic Performance of Seawater Intake System Using CFD Modeling. *J. Mar. Sci. Eng.* **2022**, *10*, 988. [\[CrossRef\]](#)
6. Ortiz, J.P.; Xavier, T.C.L. Three-Dimensional Simulations and Economical Solutions for Cavitation in Hollow-Jet Dispersive Valves. *J. Appl. Fluid Mech.* **2021**, *14*, 1399–1410.
7. Alrwashdeh, S.S.; Ammari, H.; Madanat, M.A.; Al-Falahat, A.M. The Effect of Heat Exchanger Design on Heat transfer Rate and Temperature Distribution. *Emerg. Sci. J.* **2022**, *6*, 128–137. [\[CrossRef\]](#)
8. Aminoroayaie Yamini, O.; Mousavi, S.H.; Kavianpour, M.R.; Safari Ghaleh, R. Hydrodynamic performance and cavitation analysis in bottom outlets of dam using CFD modelling. *Adv. Civ. Eng.* **2021**, *2021*, 5529792. [\[CrossRef\]](#)
9. Amirante, R.; Distaso, E.; Tamburrano, P. Experimental and numerical analysis of cavitation in hydraulic proportional directional valves. *Energy Convers. Manag.* **2014**, *87*, 208–219. [\[CrossRef\]](#)
10. Nzombo, D. A numerical Investigation of Cavitation in Valves and Techno-Economic Analysis of Pinewood Solvent Liquefaction. Ph.D. Thesis, Iowa State University, Ames, IA, USA, 2017.
11. Liu, X.; Wu, Z.; Li, B.; Zhao, J.; He, J.; Li, W.; Zhang, C.; Xie, F. Influence of inlet pressure on cavitation characteristics in regulating valve. *Eng. Appl. Comput. Fluid Mech.* **2020**, *14*, 299–310. [\[CrossRef\]](#)

12. Tabrizi, A.S.; Asadi, M.; Xie, G.; Lorenzini, G.; Biserni, C. Computational fluid-dynamics-based analysis of a ball valve performance in the presence of cavitation. *J. Eng. Thermophys.* **2014**, *23*, 27–38. [\[CrossRef\]](#)
13. Li, J.-Y.; Gao, Z.-X.; Wu, H.; Jin, Z.-J. Numerical Investigation of Methodologies for Cavitation Suppression Inside Globe Valves. *Appl. Sci.* **2020**, *10*, 5541. [\[CrossRef\]](#)
14. Wu, J.; Utturkar, Y.; Senocak, I.; Shyy, W.; Arakere, N. Impact of turbulence and compressibility modeling on three-dimensional cavitating flow computations. In Proceedings of the 33rd AIAA Fluid Dynamics Conference and Exhibit, Orlando, FL, USA, 23–26 June 2003; p. 4264.
15. Wang, G.; Liu, S.; Shintani, M.; Ikohagi, T. Study on cavitation damage characteristics around a hollow-jet valve. *JSME Int. J. Ser. B Fluids Therm. Eng.* **1999**, *42*, 649–657. [\[CrossRef\]](#)
16. Tsukiji, T.; Noguchi, E.; Yoshida, F. Development of Oil Hydraulic Components Using a Flow Visualization Technique. *Int. J. Autom. Technol.* **2012**, *6*, 410–417. [\[CrossRef\]](#)
17. Cao, B.; Guo, M. Analysis of boundary element of flow field in poppet valve. *Mach. Tool Hydraul.* **1991**, *2*, 2–10.
18. Cao, B.; Shi, W. Experimental study on flow force and pressure distribution on the poppet of a converged flow poppet valve. *J. Xi'an Jiaotong Univ.* **1995**, *7*, 7–13.
19. Gao, D. Investigation of Flow Structure Inside Spool Valve with Fem and Piv Methods. *Int. J. Fluid Power* **2004**, *5*, 51–66. [\[CrossRef\]](#)
20. Jalil, J.M.; Ahmed, S.T.; Xue, Y.; Ghadhbani, S.A. Experimental and numerical investigation of fluid flow of truncated conical poppet valve. *Int. J. Fluid Power* **2015**, *16*, 25–34. [\[CrossRef\]](#)
21. Yi, D.; Lu, L.; Zou, J.; Fu, X. Interactions between poppet vibration and cavitation in relief valve. *Proc. Inst. Mech. Eng. Part C J. Mech. Eng. Sci.* **2015**, *229*, 1447–1461. [\[CrossRef\]](#)
22. Li, S.; Aung, N.Z.; Zhang, S.; Cao, J.; Xue, X. Experimental and numerical investigation of cavitation phenomenon in flapper-nozzle pilot stage of an electrohydraulic servo-valve. *Comput. Fluids* **2013**, *88*, 590–598. [\[CrossRef\]](#)
23. Adamkowski, A.; Lewandowski, M. Cavitation Characteristics of Shutoff Valves in Numerical Modeling of Transients in Pipelines with Column Separation. *J. Hydraul. Eng.* **2015**, *141*, 04014077. [\[CrossRef\]](#)
24. Zeng, L.F.; Liu, G.W.; Mao, J.R.; Wang, S.S.; Yuan, Q.; Yuan, H.; Wang, K.G.; Zhang, J.J.; Xu, Y.T. Flow-induced vibration and noise in control valve. *Proc. Inst. Mech. Eng. Part C J. Mech. Eng. Sci.* **2015**, *229*, 3368–3377. [\[CrossRef\]](#)
25. Saha, B.K.; Chattopadhyay, H.; Mandal, P.B.; Gangopadhyay, T. Dynamic simulation of a pressure regulating and shut-off valve. *Comput. Fluids* **2014**, *101*, 233–240. [\[CrossRef\]](#)
26. Tran, P.D. Pressure Transients Caused by Tilting-Disk Check-Valve Closure. *J. Hydraul. Eng.* **2015**, *141*, 04014081. [\[CrossRef\]](#)
27. Beune, A.; Kuerten, J.; van Heumen, M. CFD analysis with fluid–structure interaction of opening high-pressure safety valves. *Comput. Fluids* **2012**, *64*, 108–116. [\[CrossRef\]](#)
28. Hős, C.J.; Champneys, A.; Paul, K.; McNeely, M. Dynamic behavior of direct spring loaded pressure relief valves in gas service: Model development, measurements and instability mechanisms. *J. Loss Prev. Process Ind.* **2014**, *31*, 70–81. [\[CrossRef\]](#)
29. Hős, C.; Champneys, A.; Paul, K.; McNeely, M. Dynamic behaviour of direct spring loaded pressure relief valves in gas service: II reduced order modelling. *J. Loss Prev. Process Ind.* **2015**, *36*, 1–12. [\[CrossRef\]](#)
30. Posa, A.; Oresta, P.; Lippolis, A. Analysis of a directional hydraulic valve by a Direct Numerical Simulation using an immersed-boundary method. *Energy Convers. Manag.* **2013**, *65*, 497–506. [\[CrossRef\]](#)
31. Flow Science, Inc. *Flow-3D User Manual v11. 2*; Flow Science, Inc.: Santa Fe, NM, USA, 2018.
32. Imanian, H.; Mohammadian, A. Numerical simulation of flow over ogee crested spillways under high hydraulic head ratio. *Eng. Appl. Comput. Fluid Mech.* **2019**, *13*, 983–1000. [\[CrossRef\]](#)
33. Yang, J.; Teng, P.; Zhang, H. Experiments and CFD modeling of high-velocity two-phase flows in a large chute aerator facility. *Eng. Appl. Comput. Fluid Mech.* **2018**, *13*, 48–66. [\[CrossRef\]](#)
34. Khalifehei, K.; Azizyan, G.; Shafai-Bajestan, M.; Chau, K.W. Experimental Modeling and Evaluation Sediment Scouring in Riverbeds around Downstream in Flip Buckets. *Int. J. Eng.* **2020**, *33*, 1904–1916.
35. Ghaleh, R.S.; Yamini, O.A.; Mousavi, S.H.; Kavianpour, M.R. Numerical Modeling of Failure Mechanisms in Articulated Concrete Block Mattress as a Sustainable Coastal Protection Structure. *Sustainability* **2021**, *13*, 12794. [\[CrossRef\]](#)
36. Del Toro, A. Computational Fluid Dynamics Analysis of Butterfly Valve Performance Factors. Master's Thesis, School of Mechanical Engineering, Utah State University, Logan, UT, USA, 2012.
37. Tao, J.; Lin, Z.; Zhang, G.; Su, J.; Zhu, Z. A Numerical and Experimental Study of the Time Averaged and Transient Flow Downstream of a Butterfly Valve. *J. Fluids Eng.* **2022**, *144*, 051202. [\[CrossRef\]](#)
38. Reynolds, O. An experimental investigation of the circumstances which determine whether the motion of water shall be direct or sinuous, and of the law of resistance in parallel channels. *Phil. Trans. R. Soc. Lond. A* **1883**, *174*, 935–982.