water

MDPI

Article

3D Hydrodynamic Modelling Enhances the Design of Tendaho Dam Spillway, Ethiopia

Getnet Kebede Demeke *, Dereje Hailu Asfaw and Yilma Seleshi Shiferaw

School of Civil and Environmental Engineering, Addis Ababa University, Addis Ababa P.O. Box 385, Ethiopia; dereje_hasfaw@yahoo.com (D.H.A.); yilma.seleshi@gmail.com (Y.S.S.)
* Correspondence: getnetas@yahoo.com

Received: 13 November 2018; Accepted: 27 December 2018; Published: 4 January 2019

check for updates

Abstract: Hydraulic structures are often complex and in many cases their designs require attention so that the flow behavior around hydraulic structures and their influence on the environment can be predicted accurately. Currently, more efficient computational fluid dynamics (CFD) codes can solve the Navier–Stokes equations in three-dimensions and free surface computation in a significantly improved manner. CFD has evolved into a powerful tool in simulating fluid flows. In addition, CFD with its advantages of lower cost and greater flexibility can reasonably predict the mean characteristics of flows such as velocity distributions, pressure distributions, and water surface profiles of complex problems in hydraulic engineering. In Ethiopia, Tendaho Dam Spillway was constructed recently, and one flood passed over the spillway. Although the flood was below the designed capacity, there was an overflow due to superelevation at the bend. Therefore, design of complex hydraulic structures using the state-of- art of 3D hydrodynamic modelling enhances the safety of the structures. 3D hydrodynamic modelling was used to verify the safety of the spillway using designed data and the result showed that the constructed hydraulic section is not safe unless it is modified.

Keywords: velocity distributions; water surface profiles; computational fluid dynamics (CFD); 3D numerical models; spillway and superelevation

1. Introduction

Water is the world's most important natural resource; however, it is estimated that by 2030, the world is projected to face a 40% global water deficit. Research development and modern technology enhances and provide an improved and optimized hydraulic structures for water resource projects worldwide [1].

In Ethiopia, large-scale water resources development is at its beginning stage. Currently large-scale projects such as Tendaho Dam, Kesem Dam, Gidabo Dam, Ribb Dam, Megech Dam, Welkait (Zarima May Day) Dam, Kuraz Barrage, Jema Dam, Gigel Abbay Dam, Gibe 1 Dam, Gibe 3 Dam, Renaissance Dam, Upper Guder Dam, etc are under design, under construction or recently constructed. Huge water resource development projects such as the aforementioned have a significant impact on the country's economy; hence, the successful implementation of water resource projects require special attention from envision to completion. Therefore, proper design and analysis of the hydraulics structures is essential [2]. Tendaho dam and appurtenant structures were designed and constructed in 2008 in Ethiopia. The spillway was designed based on Indian Standards [3–5] which is similar to U.S. Bureau of Reclamation (USBR). However, the design safety for its operation has not been confirmed either with physical model or 3D numerical model. The estimated design inflow discharge into reservoir is 3042 m^3/s [6] and in 2011, there was an inflow discharge of 2100 m^3/s and the water spilled accordingly. The designed flow depth over the spillway crest is 9.2 m and during this flood time, 5.5 m depth of water passed over the spillway crest. At the time of flooding, the gates were not installed. The flow of water leaving the stilling basin and entering the downstream channel was turbulent, high velocity and wavy since the water in the stilling basin was not properly dissipated.

The downstream channel was constructed with cemented stone pitching and concrete blocks for 110 m downstream of the stilling basin. The downstream spillway channel has a bend and after the bend it crosses the main canal where the main canal crosses under the spillway channel.

During the flood time mentioned above, the concrete blocks and stone pitching were taken away by the flood. In addition, the flow had superelvation at the bend and due this superelevation, the water overtopped the spillway channel and entered the main canal at a location where the spillway channel crosses over the Main Canal. Because of this problem there was significant losses occurred in the main canal. The researcher had an opportunity to see the problem while it was occurring, and this problem inspired him to conduct research to investigate the hydraulic problems and propose solutions. In addition, the maintenance work after the problem was carried out based on the original design which shows that still the spillway is in danger of being unsafe.

The main objective of this research is to design a hydraulically safe structure for the complex section using the state- of- art of 3D hydrodynamic modelling in a case study of Tendaho Dam Spillway, Ethiopia.

Verifying spillway safety using state-of art 3D numerical model by FLOW-3D has been done. The comparison of the results showed that the numerical solution can predict well the existing parameters in the hydraulic jump, such as the velocity and water surface profile across and along the spillway channel. In this numerical solution, the RNG turbulence model presented better results. FLOW-3D results showed that the RNG turbulence closure, when combined with the volume of fluid (VOF) surface tracking method, can accurately predict separation [7] zones as well as 3-D patterns of the fluid motion. The agreement is remarkable for a field situation. Results are satisfactorily accurate, as they confirm the experimental findings from the physical models [8].

FLOW-3D is commercially available CFD package developed by Flow Science Inc. (Santa Fe, NM, USA) which has been broadly applicable for the past decades [9–11], capable of solving a wide range of fluid flow problems. FLOW-3D uses the finite volume method to solve the Navier–Stokes system of equations in three dimensions to simulate the flow of fluid. Two-fluid and one-fluid solvers are available, both of which use a proprietary volume-of-fluid (VOF) method to track the free surface. The VOF technique was developed mathematically by the founder of Flow Science and was described by Nichols and Hirt [12] and more thoroughly detailed by Hirt and Nichols [13]. FLOW-3D also has the ability to calculate solutions using various implicit and explicit solver options. The ability of using multiple and nested meshes as well as the re-run capability available in FLOW-3D are other options that make the numerical model suitable for spillway modelling [14].

The model includes many optional models that add to or modify the basic Navier–Stokes equations. Additional models that are used frequently in hydraulics include options for describing the effects of turbulence, surface tension, heat transfer, fluid solidification, sediment scour, Lagrangian particles, granular flows, moving solids, solid deformation, air entrainment, cavitation, and porous media [15].

A unique feature of FLOW-3D is the FAVOR (Fractional Area/Volume Representation) method, which permits true representation of complex geometry in a simple Cartesian mesh. As a result, FLOW-3D can be used to simulate flow in complex hydraulics structures accurately and efficiently [16].

This study uses FLOW 3D verification work which has been conducted for Zarema May Day Dam Spillway in Ethiopia. The verification work has been undertaken based on physical model result [17]. Accordingly, numerical modelling of the spillway was carried out for a Tendaho Dam, Ethiopia, which is constructed with a routed half PMF (probable maximum flood) design flow rate of 1700 m^3/s. The routed hydrograph is characterized by the attenuation of flood peak of the inflow hydrograph. The design results were compared with FLOW-3D model results. As per the result, redesign of the hydraulic section at stilling basin was proposed and verified with the 3D numerical model.

2. Background

Tendaho dam is constructed across the river Awash, nearly 7 km upstream of its confluence with Logia tributary. The location of the dam is Latitude 11.3° (North), Longitude 41.0° (East). Figure 1 shows the Location of Tendaho Dam. The dam site lies across a gap in rock ridge adjacent to and immediately east of the present Addis Ababa–Assab highway. The dam is as an earth dam with impervious clay core [18].

Tendaho Dam spillway structure comprises an approach channel leading the water to the control structure, a control structure, and an inclined chute terminating into a stilling basin. Energy dissipation is by means of the hydraulic jump forming in the stilling basin [19]. A dentated sill is provided at the end of stilling basin to facilitate formation of the hydraulic jump for a large range of discharge passing over the control structure. After the dissipation of energy in the stilling basin, the flow will be conveyed back to the river through an outlet channel (downstream discharge channel). After the stilling basin, the downstream discharge channel has a bend with deflection of 68°. Just downstream of the bend, the downstream discharge channel crosses over the main canal at chainage (ch) 0 + 390 m.

The bend along the downstream discharge channel represents the key element of the spillway design. Difficulty in the design arises because of the complexity of the flow around a curved path, which is not readily subject to any analytical solutions [20]. Refer Figure 2.

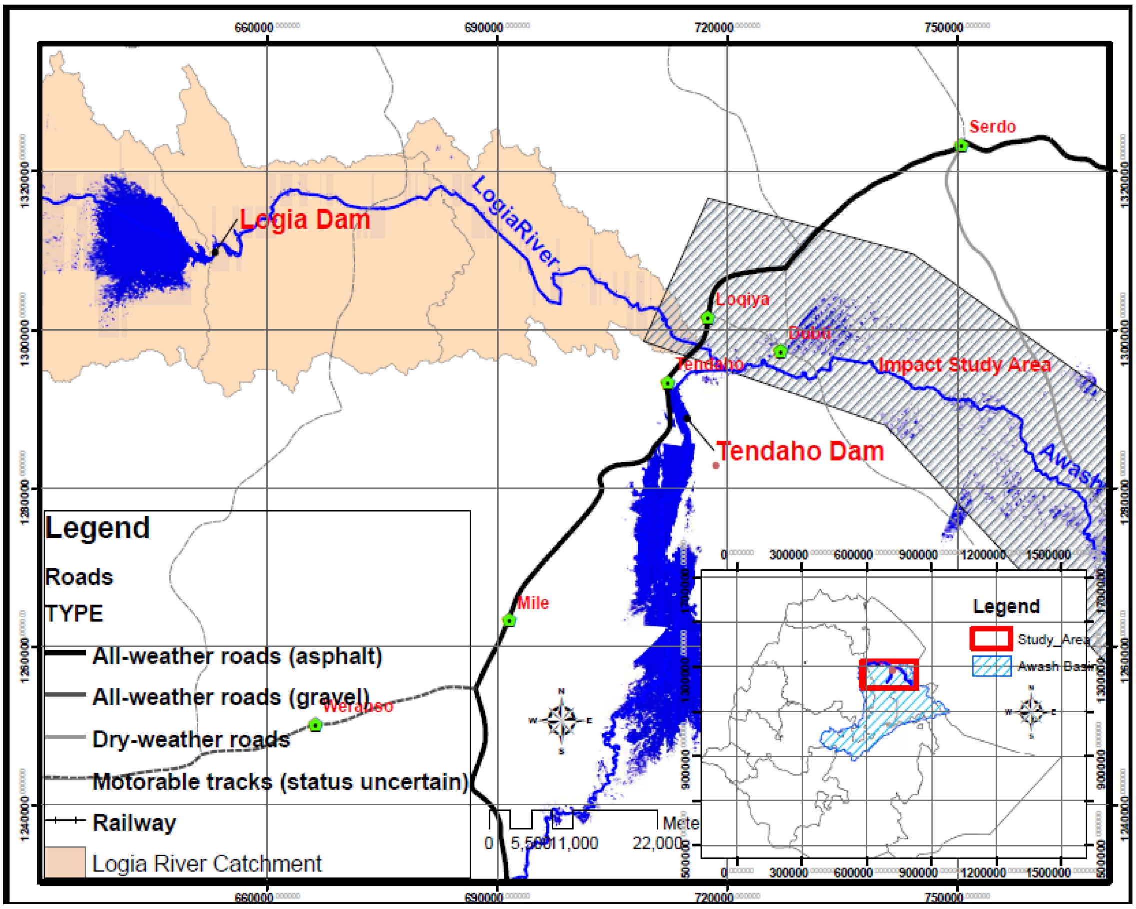


Figure 1. Location of Tendaho Dam [21].

Hence, the purpose of the model is to assess the global behavior of the spillway such as:

- The discharge capacity of the ogee crest.
- To study the general appearance and performance of structures and determine hydraulic flow conditions throughout the spillway section.
- To find a solution and improve the performance of hydraulic structures for the hydraulic problems using 3D numerical models.

The model was set up by dividing the length of spillway at 5 m intervals starting from the river (outlet) up to the reservoir (inlet) with full-scale model. The total length of the spillway model is 868 m. Figures 2 and 3 show the overall information of the plan. The spillway has 30.0 m clear crest length with (3 bays of 10.5 each), the depth of water above the spillway crest of 400 m will be 9.2 m which corresponds to maximum water level of 409.2 m and the routed or attenuation of flood peak discharge is 1694 m^3/s which is 1700 m^3/s [18].

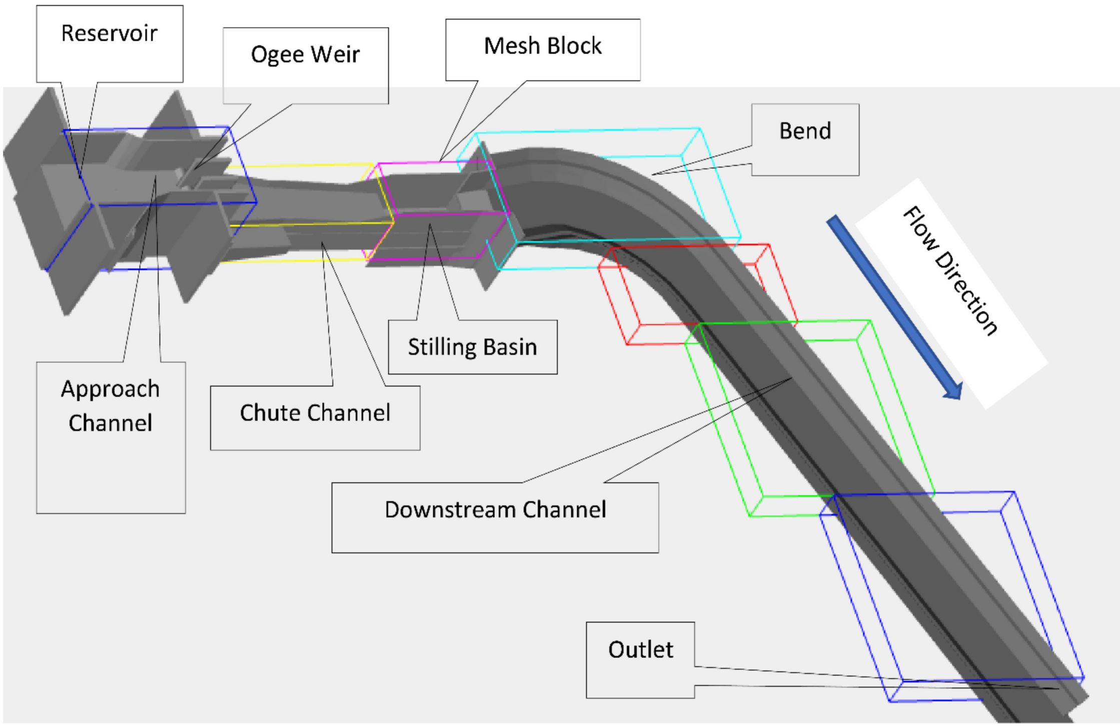


Figure 2. Plan of the spillway and mesh blocks prepared for the model.

In this paper, FLOW-3D numerical model is used with full-scale of spillway for simulating the hydraulics characteristics of the flow over the spillway, along the chute channel, at the stilling basin, and at the bend and downstream channel. The spillway model exported from Auto CAD to FLOW-3D and seven mesh blocks are developed for simulation.

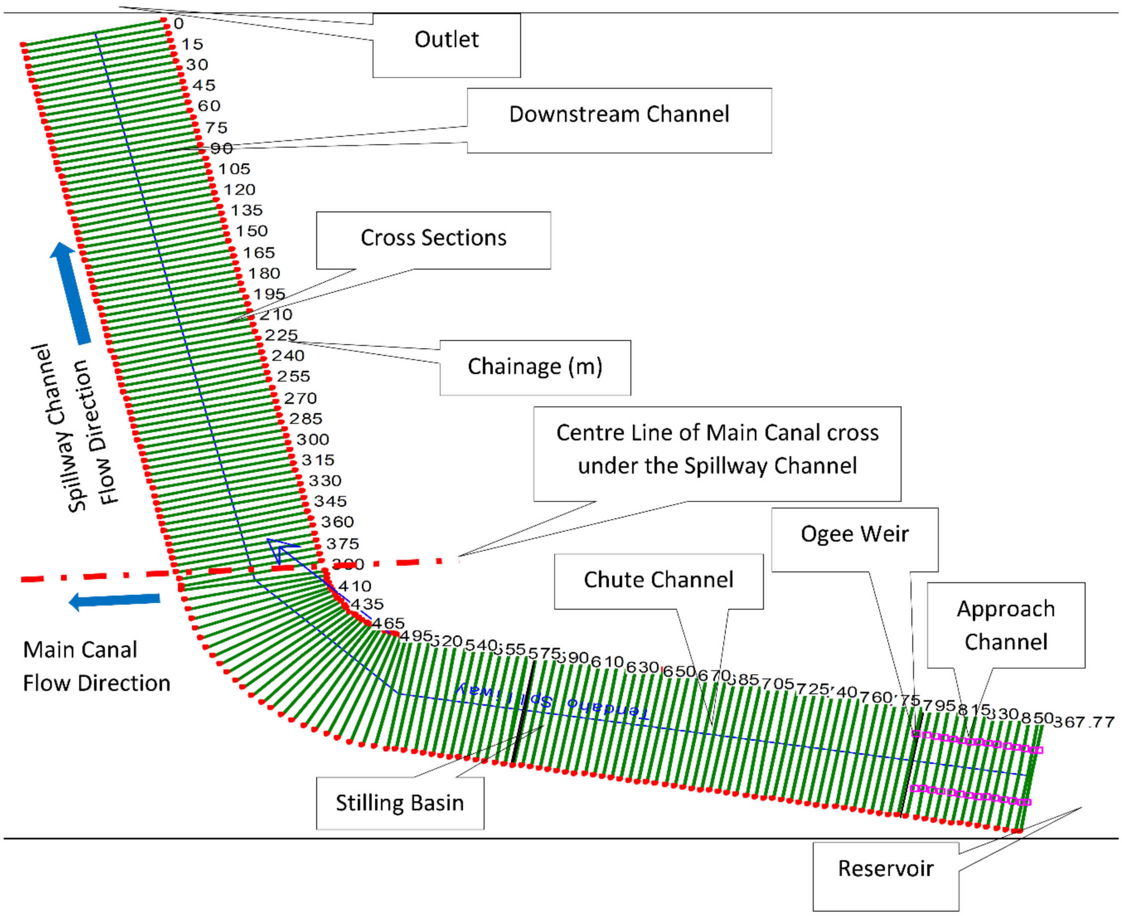


Figure 3. Spillway chainage distance from downstream to upstream in meter.

3. Methods

Tedaho Dam spillway was designed and constructed without the physical model or 3D hydrodynamic model being verified due to its complexity of flow conditions. As a result, significant loss occurred during the flood season. Therefore, the objective of this study is to verify the design and propose a solution which will enhance the design of the spillway using 3D hydrodynamic model. Accordingly, FLOW-3D hydrodynamic model has been used for verification and modification of the Tendaho spillway design. For easy checking, different trials were done to obtain approximated section and level with the 1D model. This trial is based on experience and checking the depth of flow in sequential depth. As per the design section, the depth of water was raised up to 19.85 m when it was verified by 3D model. The downstream flow depth was 3 m. Therefore, more than 15 m depth of water needs to be dissipated before it joins to the downstream channel. Accordingly, the stilling basin level was proposed to be lowered by 15 m and maintaining the same design width. After deciding the proposed section and dimensions, using the AUTO CAD 3D model, the section was prepared and exported to the FLOW-3D model in stl format.

The FLOW-3D program subdivides the Cartesian computational domain into a grid of hexagonal cells. For each cell, the program calculates average and maximum values for the flow parameters (depth, velocity, pressure) at discrete times.

The equation of motion for the fluid velocity components (u, v, w) in the three x, y and z coordinate directions are the Navier–Stokes equations with some additional terms in a numerical method [15].

This study uses FLOW 3D verification work which has been conducted for the Zarema May Day Dam spillway in Ethiopia [17]. This verification work was done based on physical model result of Zarema May Day Dam conducted at the Laboratory of Hydraulic Protection of the Territory (PITLAB)

Laboratory of the University of Pisa, Italy [22]. The result showed that FLOW-3D is capable of simulating the hydraulics characteristics of the flow over the spillway, along the chute channel, bend, baffle structures and cascade drops for verification of the model obtained from the physical model. FLOW-3D model has good agreement with the physical model especially at the crest, after the guide wall, along the chute channel, before the bend, and at the baffles and cascade drops [17]. Refer to Supplementary File S1.

3.1. Boundary and Initial Conditions

Since the flow domain is defined as a hexagonal in the Cartesian coordinates, there are six different boundaries on each mesh block. The boundaries on the mesh and their coordinate directions were set as follows.

For optimal simulation, the spillway was modeled in seven mesh blocks. The boundary condition for flow data in Block 1, the upstream boundary condition (Xmin) was defined as volume flow rate 1700 m^3/s at the inlet of the spillway, the downstream boundary condition (Xmax) was defined as symmetry. The bottom boundary (Zmin) was defined as the wall (no-slip condition) where the approach channel invert elevation is 396 m and the spillway crest elevation is 400 m. The top boundary (Zmax) was set as specified pressure (with fluid fraction = 0) that was specified with water elevation 410 m which is flow depth over the crest is 10 m. In Block 7 at the outlet, Ymin was given as an outflow. All the other blocks between Block 1 and 7 were either symmetry or wall condition. Symmetry means a zero-gradient condition which is continuative at the boundary. Wall means a no-slip condition at the boundary as well as a zero-velocity condition normal to the boundary [15].

The initial conditions that represent on the upstream sides of the spillways at the same level as the specified fluid heights were proposed fluid elevation. The free surface water elevation option was given as an initial condition at the Block 1. The Zmax-coordinate of the free surface was fixed for the estimated initial condition which is 410 m. Refer to Supplementary File S1.

3.2. Discretization Approach

FLOW-3D uses the finite volume method to solve the Navier–Stokes system of equations in three dimensions to simulate the flow of fluid.

In this numerical model seven numerical hexagonal meshes blocks were built to properly capture the geometry of the structure and correctly simulate the water flow from the reservoir to the outlet. Various computational trials are conducted with different grid cells size in x, y and z directions and it was observed that the cells at the ogee, near the baffles and along the curve were not resolved well when the grid size was given above 0.5 m. Hence, 0.5 m width was given for all hexagonal grid widths. Figure 2 shows the plan of the spillway and mesh blocks prepared for the model.

A larger time step will lead to numerical instabilities and non-physical results. The first durations which is less than 1 s out of 100 s, showed a slight instability condition; however, other simulations were stable.

3.3. Model Geometry

The numerical analysis was carried out based on the original design and modified design. In addition, the designed data obtained from the report which was not done with the numerical method, was also compared with the numerical results of design and modified numerical results. For numerical analysis, FLOW-3D numerical methods were used. The model for 1D was set up to get overall information at different location along the spillway by dividing the length of spillway at 5 m intervals starting from the river (outlet) up to reservoir (inlet) with a full-scale model. The total length of the spillway model is 868 m.

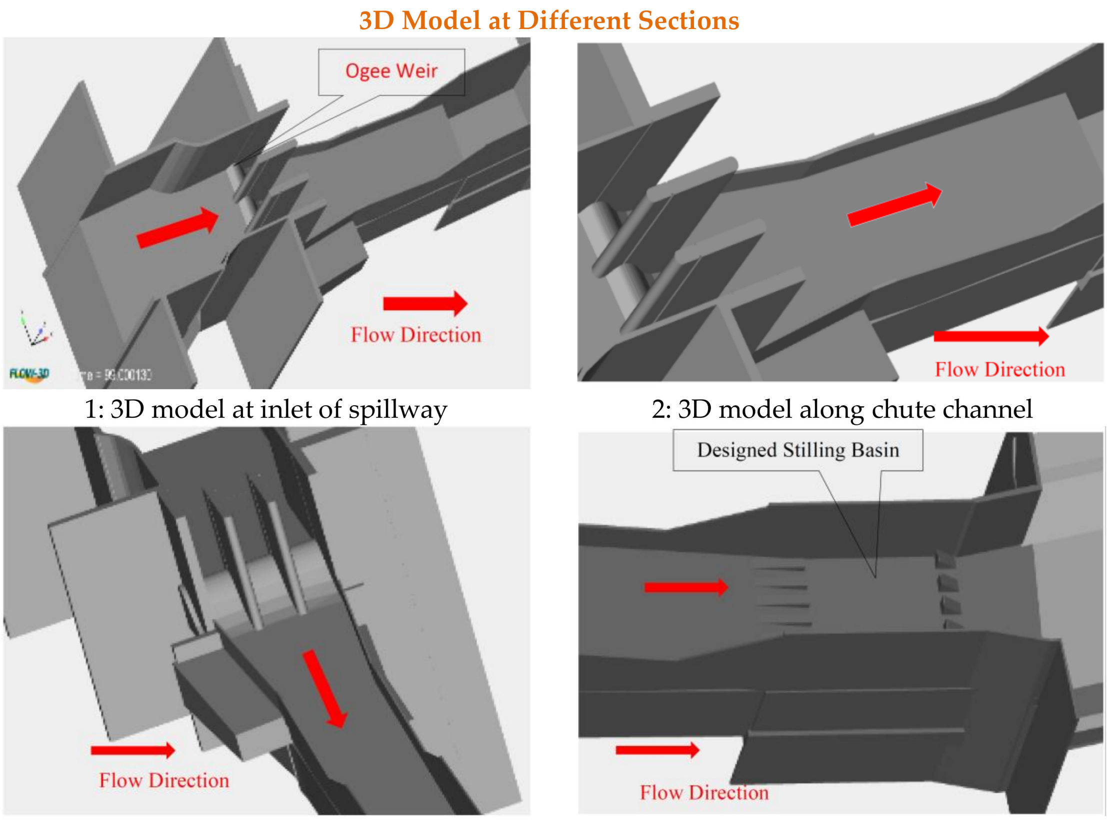
Another geometry option that remained constant for all spillway modeling completed as part of this study, was the inclusion of surface roughness value applied on the surface of all spillway geometry. The roughness friction factor = 0.016 for a concrete-lined channel and 0.025 for the downstream channel were utilized in the model. This was done in FLOW-3D by specifying a surface roughness value, equal to the average height of surface imperfections to the desired components in the meshing and geometry tab since Manning roughness was converted to FLOW-3D's surface roughness. Also included in the geometry tab for all simulations was a baffle, which is a plane that was defined as a flux surface and specified to be 100% porous so that it would have no effect on the flow. This baffle was normally located in a plane at different sections along the spillway which was responsible for providing the discharge measurement. The numerical model comprises the approach channel, ogee crest, crest pier, chute channel, stilling basin, channel bend, downstream channel [18]. Refer Figure 4.

The 3D numerical model has different types of options for the output result that includes hydraulic data of flow depth, maximum flow depth, velocity, specific hydraulic head and total head, pressure, etc. Refer Figure 5.

The runs were developed in a personal computer with a Pentium 4 Ghz processor and 16 GB of RAM; the evolution in time was used as a relaxation to the final steady state. Simulation time was given as 100 s and accordingly 8 days were taken to finalize simulation. The required output memory size was 138 GB carried out with eight multi processors parallel.

3.4. Model Output

FLOW 3D is a preprocessor of the major model setup and run performance. For post-processer, the FlowSight application is the main element to analyze the required output. FLOW-3D in Collaboration with FlowSight provides a powerful and simple method for understanding and sharing simulation results. Results can be compared by viewing both numerical and visual formats, analyzing isosurfaces from all six degrees simultaneously, and linking and viewing separate cases together in the same viewport, plots, and output [15].



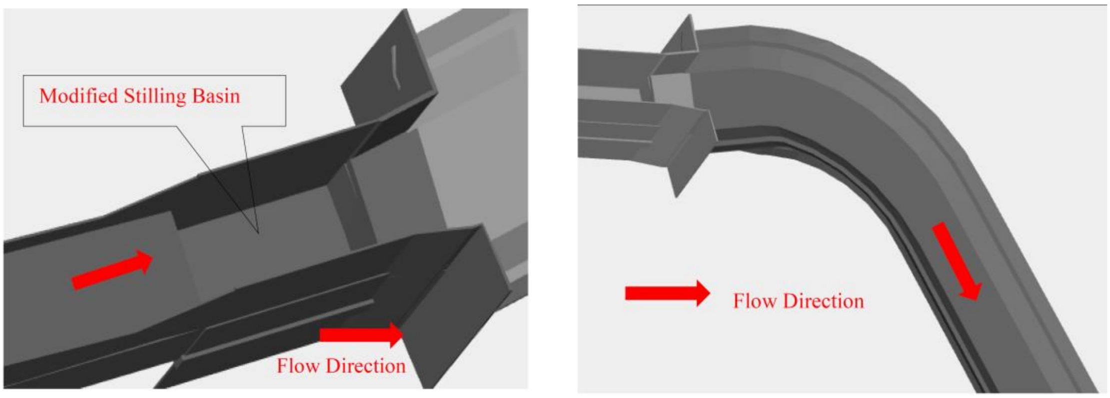
1: 3D model at inlet of spillway

2: 3D model along chute channel

3: 3D model at converging section

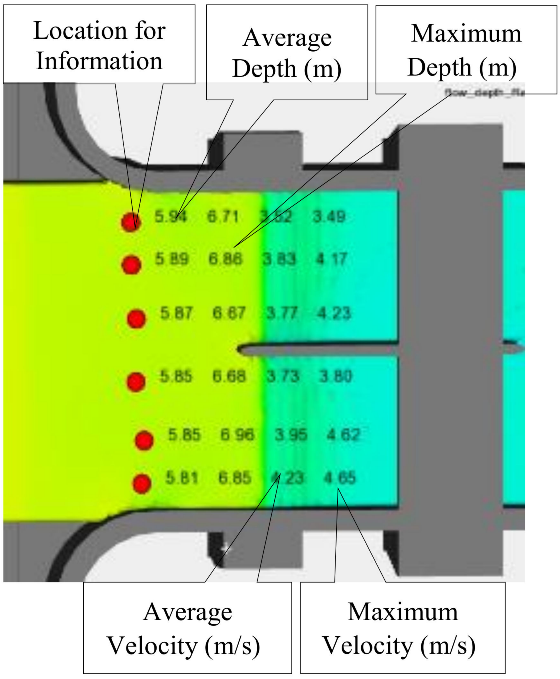
4: 3D model at stilling basin as per design

Figure 4. *Cont.*



5: 3D model at stilling basin as per modification 6: 3D model at bend and downstream channel

Figure 4. Shows 3D model at different sections of the spillway.



At seven locations, hydraulics data were taken across the flow direction and the results are:

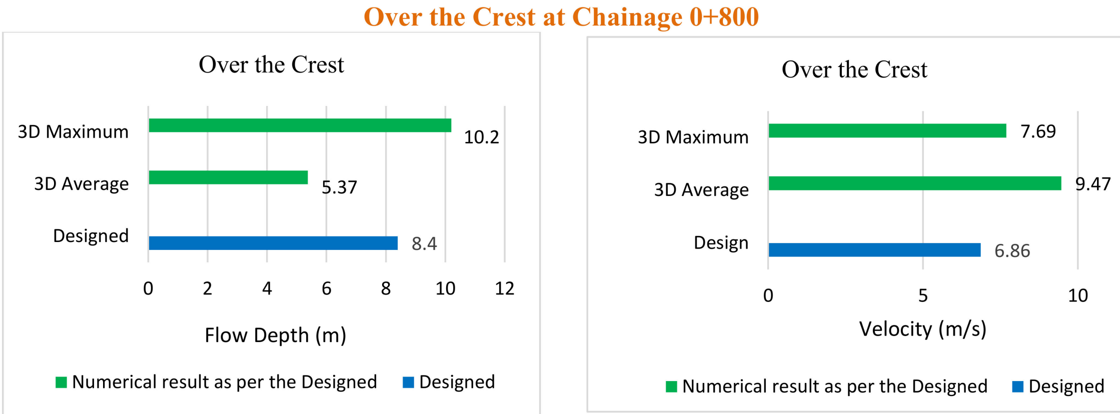
average depth, maximum depth, average velocity and maximum velocity.

Figure 5. 3D model result information.

4. Results

4.1. Flow over the Spillway Crest

At chainage 0 + 800 m, the routed 1/2 PMF flow which is 1700 m^3/s was considered as flow over the spillway crest. In addition, the initial boundary condition for the FLOW-3D model was given as the top boundary (Zmax) and was set as specified pressure that was specified with water elevation of 410 m at which the flow depth over the crest was 10 m. As per the design, flow depth was 8.4 m with a velocity of 6.86 m/s. The 3D average flow depth was 5.37 m with average velocity of 9.47 m/s and the maximum flow depth was 10.2 m with velocity 7.69 m/s. The designed result has less than 10% difference with 3D maximum depth. The maximum depth over the crest was seen due to the initial condition; however, the average flow depth over the crest was 5.37 m. Refer Figure 6.



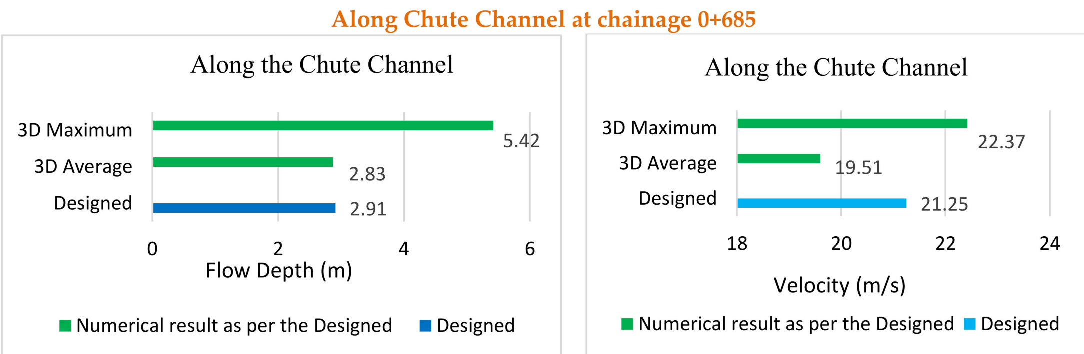
1: Depth comparison 2: Velocity comparison

Figure 6. Model result over spillway crest.

4.2. Flow Along Chute Channel

The results of flow Along Chute Channel at ch 0 + 685 are shown in Figure 7. All the flow depth in this section were similar with the 3D model. The design depth and 3D average depth are almost the same. The designed result had less than 5% difference with 3D average depth. Similarly, the velocity for the design was less than 10% from the 3D model. Generally, there was good agreement between the design and 3D model.

The design depth of the chute channel is 4.7 m. The 3D maximum depth along the cross section has more than 4.7 m. The average maximum flow depth at this section was 5.42 m which is higher than the channel depth. The channel depth along the chute channel from the toe of the spillway to the stilling basin is 4.7 m. The average maximum depth of flow due to turbulence or cross wave exceeds the wall height. The designed velocity and the 3D model are almost similar. Refer Figure 7.



1: Depth comparison 2: Velocity comparison

Figure 7. Models result along chute channel.

4.3. At Stilling Basin

The flow results at the stilling basin ch 0 + 600 are as follow. As per the design, the depth of flow was 16.1 m. The numerical results as per the designed and modified 3D average 2.81 m and 19.85 m. For 3D maximum flow depth as per the designed and modified were 4.03 m and 26.88 m. Meanwhile, there was no designed velocity. The numerical result of velocity as per the design and modified 3D average were 19.7 m/s and 5.81 m/s. For 3D, maximum flow velocities as per the design and modified were 22.8 m/s and 19.27 m/s. The energy in terms of hydraulic head has decreased from 23 m to 20 m as per design and as per the modification decreased from 23 m to 13 m which is nearly by half.

Asper the design report, the depth of flow in the stilling basin stated 16 m, however in the 3D numerical model the result showed that no such depth of flow at stilling basin. As per the design, the depth of flow was 4.3 m for the 3D average depth. In addition, there is no data for the velocity; however, the velocity as per the designed and constructed section using 3D numerical model showed that, it is more than 19 m/s.

Therefore, to dissipate the energy and reduce the flow, the researcher suggests the stilling basin type needs to be modified and verified with the 3D numerical model. Accordingly, the stilling basin was modified by lowering the stilling basin floor to a depth of 15 m. After modification, it was verified with the 3D numerical model. Accordingly, the velocity was reduced from 20.15 m/s to less than 5.81 m/s. The designed results are discussed above, and the results after modification are shown in Figure **??**. In addition, the overall comparisons before modification and after modification are shown in Figure **??**.

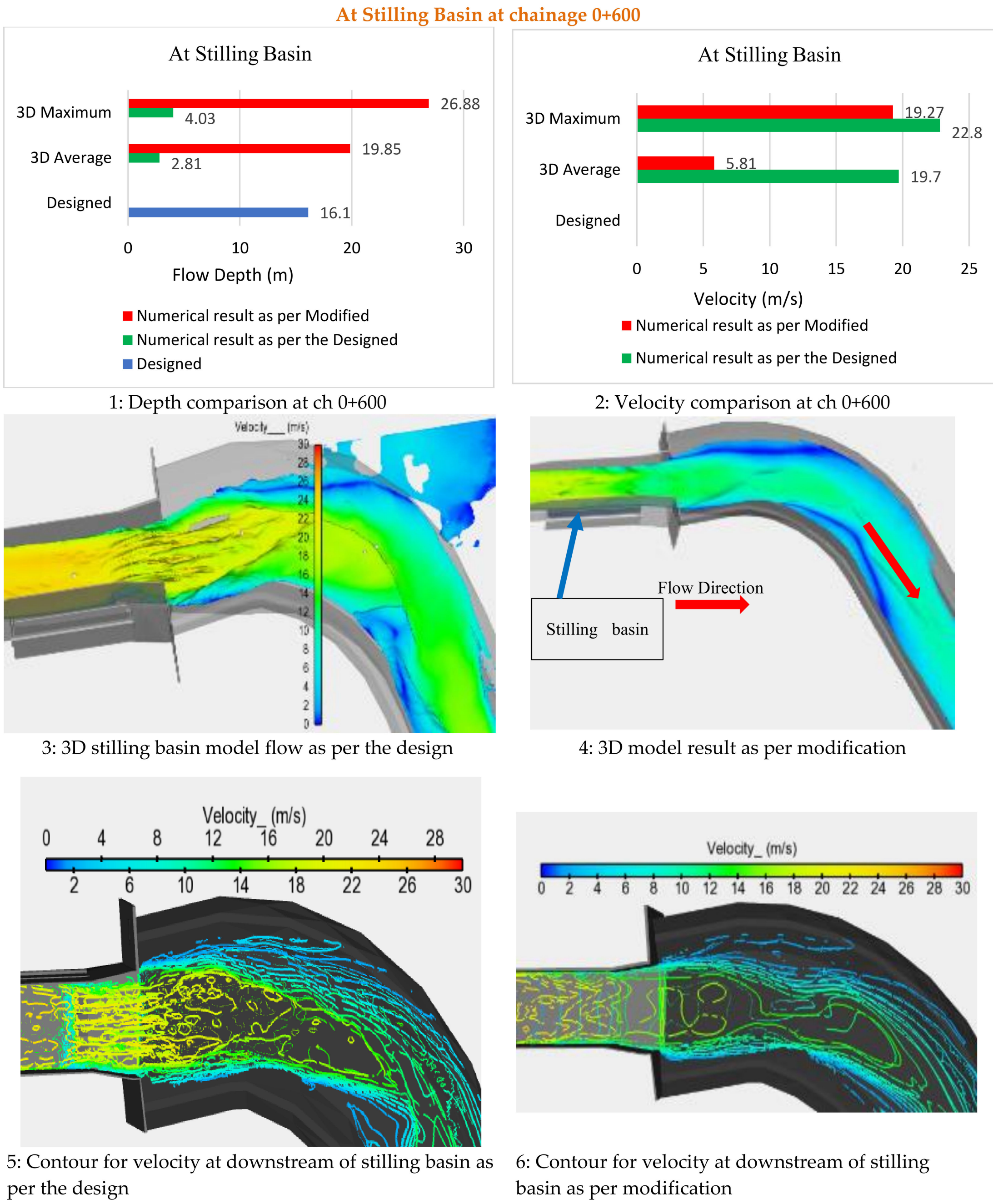


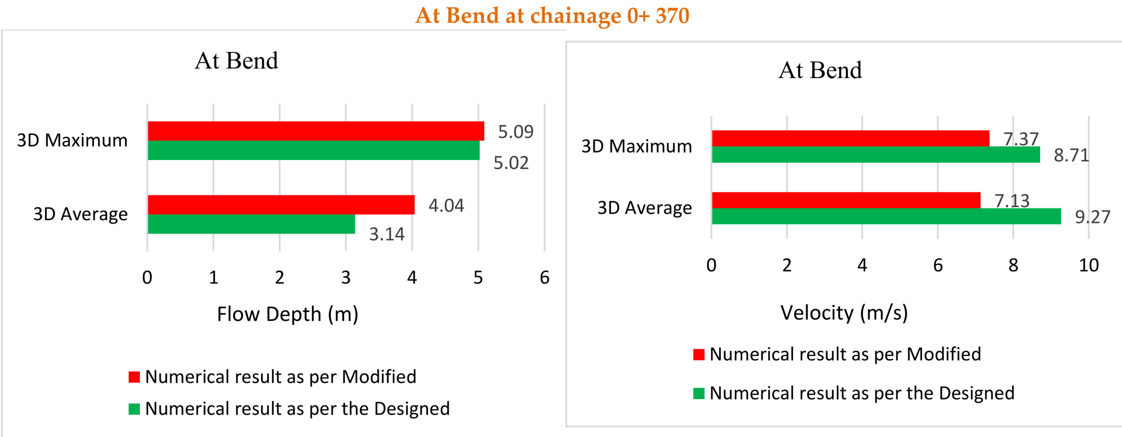
Figure 8. 3D model results at stilling basin.

4.4. Flow along Channel Bend

Flow along the bend at ch 0 + 370 has different features than expected [23,24]. There is no design data for the depth and velocity. The numerical result as per the designed and modified 3D average were 3.14 m and 4.04 m. 3D maximum flow depths were 5.02 m and 5.09 m for the design and modified conditions. Meanwhile, there were no designed velocity data.

The incoming velocity towards the bend was more than 20 m/s before modification. Therefore, the water overtopped the channel due to superelevation. Thus, modification at the stilling basin was undertaken and after modification, verification with 3D hydrodynamics was done. Accordingly, the numerical result of velocity at bend as per the design was 9.27 m/s and as per modified section 7.13 m/s. For 3D maximum flow velocity as per the design and modified were 8.71 m/s and 7.37 m/s, respectively. Refer Figure 9.

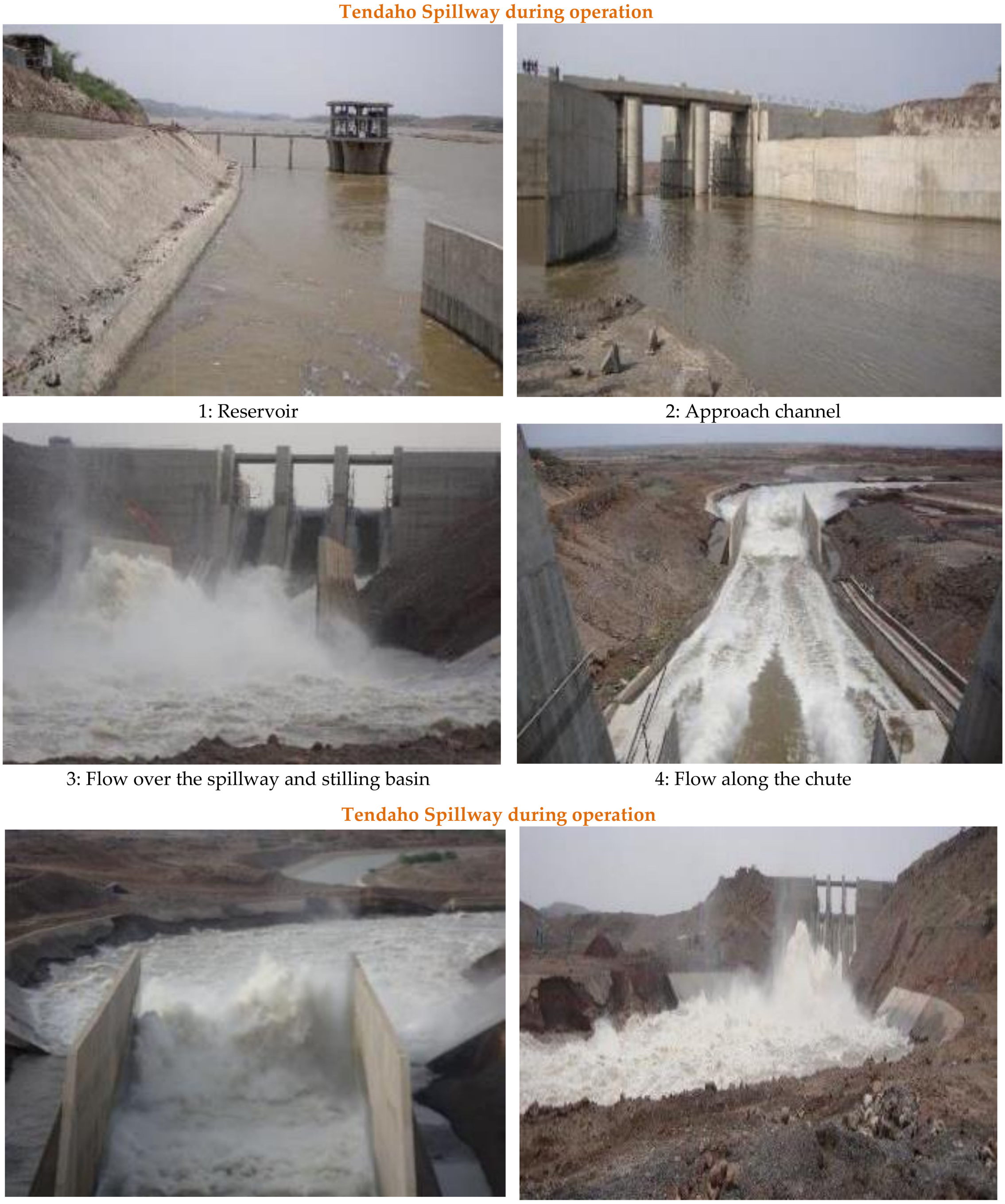
As per the design, shown in Figure **??** at number 3, due to superelevation the water overtopped the channel. This is one of the advantages of using a 3D hydrodynamic model to identify whether a section is safe for its operation or not. Such result can only be identified by the 3D numerical model. Meanwhile, similar phenomena occurred during the flood time as shown in Figure 10. Depth across the cross section varies significantly with turbulence and wavy flow. On the other hand, after modification, there was less turbulence, wavy flow and relatively uniform depth. Figure **??** at number 4 shows detailed information of the results.



1: Depth comparison 2: Velocity comparison

Figure 9. 3D model result along bend.

Tendaho Spillway during operation



1: Reservoir

2: Approach channel

3: Flow over the spillway and stilling basin

4: Flow along the chute

Tendaho Spillway during operation

5: Flow after stilling basin

6: Flow after stilling basin

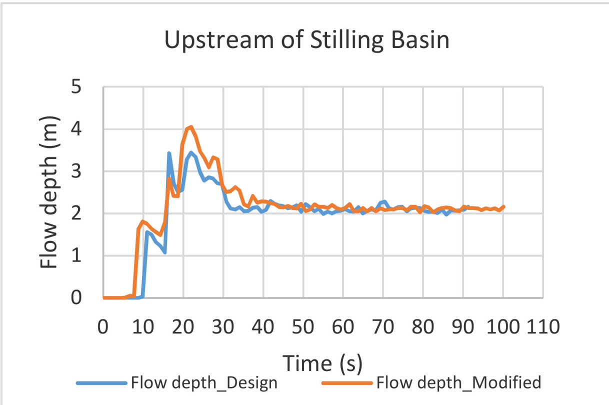
Figure 10. Tendaho spillway during operation.

5. Discussions

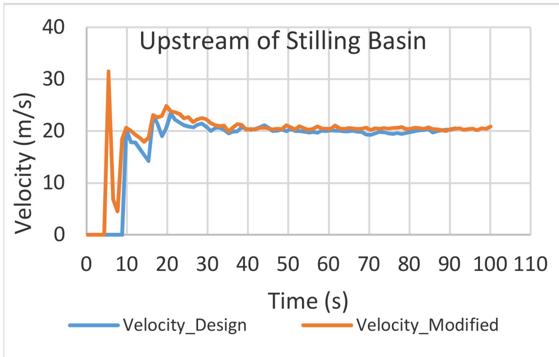
A complex hydraulic structure such as a spillway of dam design must be verified by 3D hydrodynamic numerical model rather than finalizing its design with only 1D model. The study undertaken at the University of Queensland shows the numerical limitations of 1D hydraulic models [25]. Typical limitations of 1D hydraulic models has been practically seen in Ethiopia Tendaho Dam spillway where, the design was finalized with only 1D model and was not verified with a physical model or 3D numerical model. As a result, significant losses occurred during its operation. Therefore, verification of the original design for Tendaho Dam spillway was done using FLOW-3D hydrodynamic numerical model. According to the original design, the spillway sections including the approach channel, ogee crest and chute channel are safe for the spillway operation. However, stilling basin, downstream channel, along bend and after bend, the spillway is not safe while in operation. Therefore, modification of the spillway section at the stilling basin was undertaken and after modification the design was verified using FLOW-3D hydrodynamic model and the result showed that the spillway is safe for operation.

The designed and constructed stilling basin cannot dissipate the energy of the flow which was also seen in numerical model. Accordingly, modification was done at stilling basin by lowering the bed level to a depth of 15 m. For checking, different trials were undertaken to obtain the approximated section and level with a 1D model. This trial is basically based on experience and checking the depth of flow in sequent depth. After dissipation in stilling basin, the depth of water was raised up to 19.85 m as per 3D model. The downstream flow depth was 3 m. Therefore, more than 15 m depth of water needs to be dissipated before it joins the downstream channel. Accordingly, the stilling basin was lowered by 15 m in order to reduce the hydraulic head in the stilling basin. After deciding the proposed section, using the AUTO CAD 3D model the section was prepared and exported to the FLOW-3D model. The selected level with section was analyzed by the 3D model and the results showed that after modification, the velocity was considerably reduced from 19 m/s to 6 m/s. Furthermore, the energy dissipated before modification was decreased from 23 m to 20 m. On the other hand, the energy dissipated after modification decreased from 23 m to 13 m. This shows that modification will dissipate the energy significantly. Figures 11 and 12 show more information.

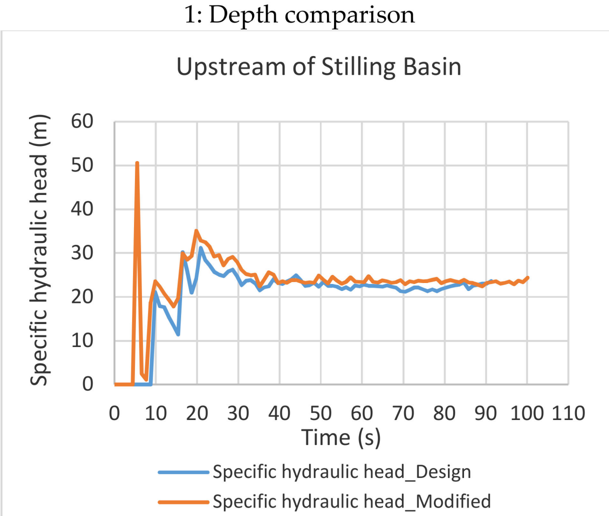
Upstream of Stilling Basin at chainage 0+620



1: Depth comparison



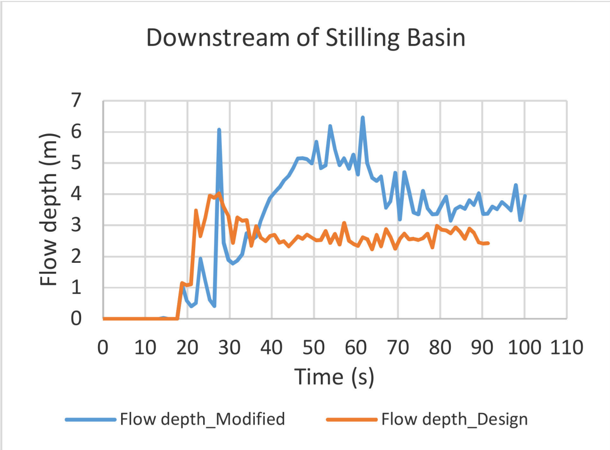
2: Velocity comparison



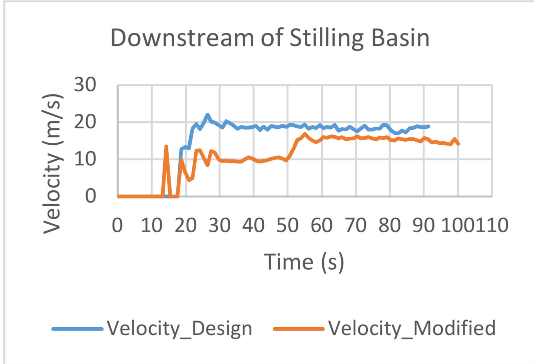
3: Specific hydraulic head comparison

Figure 11. 3D model comparison upstream of stilling basin.

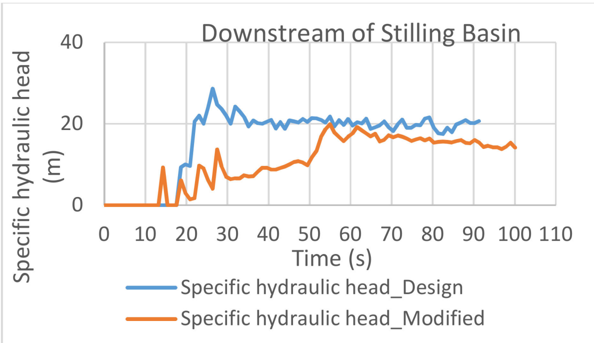
Downstream of Stilling Basin at chainage 0+520.



1: Depth comparison



2: Velocity comparison



3: Specific hydraulic head comparison

Figure 12. 3D model comparison downstream of stilling basin.

Without modification, after the stilling basin which is at the beginning of downstream channel, the flow had wavy and high turbulence with a velocity of 20 m/s. Such a result was also seen during the problem and became the cause of destruction along the bed and sides of the channel where the concrete blocks and structures were taken away by the flood. On the other hand, after modification, the numerical model showed that, the turbulence was significantly decreased, and the velocity was reduced from 19 m/s to 6 m/s.

During the flooding time, there was an overtopping channel at the bend and this condition could not be identified during design. On the other hand, the FLOW-3D numerical model showed that what happened in the time of flood where there was an overtopping due to superelevation. Meanwhile, no overtopping was seen after modification of the stilling basin. This is one of the advantages of using a 3D hydrodynamic model in order to identify whether a section is safe for its operation or not. Whenever, it is not safe, another hydraulic section should be proposed by the designer and verified by the 3D hydrodynamic model.

The 3D model has the ability to show the turbulence condition [26] in the model and gives the outputs in both average depth, maximum depth, etc. at different points across and along the channel.

6. Recommendations and Conclusions

The Tendaho Dam spillway was constructed over 10 years ago. However, the design safety for its operation has not been confirmed either with a physical model or 3D numerical model. In these durations, one flood season has occurred, and it caused significant loss. On the other hand, after damages, maintenance on the spillway has been undertaken based on the original design. Therefore, the following recommendations are put forward to minimize further loss:

- The stilling basin must be modified by at least lowering the floor level by 15 m, otherwise the spillway is not safe. As per the design, the flow will not dissipate its energy and will have high velocity that can create erosion at the downstream channel. In addition, due to superelevation at the bend, the water will overtop the channel and enter into the main canal and will create a significant loss.
- From the stilling basin up to the bend, the channel bed and side slopes need to be constructed with concrete blocks that can resist a high velocity and increase the roughness of the channel.
- Due to significant superelevation, the water overtopped the channel. This result can only be identified by a 3D numerical model since flow at such curve has a three-dimensional feature. Therefore, a 3D numerical model must be used to check if there is superelevation so as to control water overtopping the channel.
- It is a significant technological development to use a 3D hydrodynamic numerical model for the design of complex hydraulic structures. Flow condition at the curve or complex section cannot be seen by a 1D numerical model. Therefore, any complex hydraulic structure design such as spillway of the dam must be verified with a 3D hydrodynamic model.
- 3D hydrodynamic modeling must be used because it helps the designers to improve the hydraulic performance of existing or new hydraulic structures.

Due to the development of science and technology, the emergence of the 3D numerical model is one of the base lines for the design of the complex hydraulic structures that can operate safely. Modeling flow in a complex geometry using the state-of-art 3D numerical model will not only help to enhance and recognize the complex hydrodynamic flow conditions but it also helps to ensure uniform velocity distribution, depth variation, energy dissipation, etc. This will enable the engineers to provide the required sections with greater accuracy, appropriate energy dissipation methods and safe hydraulic structure. Therefore, for a complex flow structure, the design or verification must be done with a 3D numerical model.

Supplementary Materials: The following are available online at http://www.mdpi.com/2073-4441/11/1/82/s1, File S1: Validation and Methods.

Author Contributions: G.K.D. was responsible for this current research article in the framework of his PhD program and initially wrote the manuscript. D.H.A. and Y.S.S., directed the study by helping to interpret the results, editing and organizing of this research article.

Funding: This research received no external funding.

Acknowledgments: The first author would like to give his thanks to Addis Ababa University School of Civil and Environmental Engineering for giving him the opportunity of a PhD program. We acknowledge FLOW-3D created by Flow Science Inc. (Santa Fe, NM, USA) and their support. In addition, we acknowledge the data support from WWDSE for the Tendaho dam.

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

References

1. Tran, M.; Koncagul, E.; Connor, R. *The United Nations World Water Development Report Water and Jobs: Facts and Figures*; United Nations World Water Assessment Programme: Perugia, Italy, 2016.
2. Water Resources and Irrigation Development in Ethiopia. 2016. Available online: https://www.researchgate.net/publication/42765483_Water_Resources_and_Irrigation_Development_in_Ethiopia (accessed on 15 November 2018).
3. Bureau of Indian Standard. *Hydraulic Design of High Ogee Overflow Spillways Recommendations*; IS 6934-1998; Bureau of Indian Standard: New Delhi, India, 1998.
4. Bureau of Indian Standard. *Criteria for Design of Hydraulic Jump Type Stilling Basin with Horizontal and Sloping Apron*; IS 4997-1995; Bureau of Indian Standard: New Delhi, India, 1995.
5. US. Bureau of Reclamation (USBR). *Design of Small Dams*; U.S. Bureau of Reclamation: Washington, DC, USA, 1987.

6. Water Works Design and Supervision Enterprise (WWDSE/WAPCOS). *Tendaho Dam & Irrigation Project Main Report Final*; Water Works Design and Supervision Enterprise: Addis Ababa, Ethiopia, 2005.
7. Gacek, J.D. *Numerical Simulation of Flow through a Spillway and Diversion Structure*; McGill University: Montréal, QC, Canada, 2007.
8. Babaali, H.; Shamsai, A.; Vosoughifar, H. Copmutational Modeling of Hydraulic Jump in Stilling Basin with Convergence Wall Using CFD Cods. *Arab. J. Sci. Eng.* **2015**, *40*, 381–395. [CrossRef]
9. Kim, D.G.; Park, J.H. Analysis of Flow Structure over Ogee-Spillway in Consideration of Scale and Roughness Effects by Using CFD Model. *KSCE J. Civ. Eng.* **2005**, *9*, 161–169. [CrossRef]
10. Kim, S.-D.; Lee, H.-J.; An, S.-D. Improvement of hydraulic stability for spillway using CFD model. *Int. J. Phys. Sci.* **2010**, *5*, 774–780. Available online: http://www.academicjournals.org/IJPS (accessed on 4 October 2018).
11. Date, V.; Dey, T.; Joshi, S. Numerical Modeling of Flow over an Ogee Crested Spillway under Radial Gate: VOF and MMF Model. *J. Appl. Mech. Eng.* **2017**, *6*, 287.
12. Hirt, C.W.; Nichols, B.D. Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. *J. Comput. Phys.* **1981**, *39*, 201–225. [CrossRef]
13. Burnham, J.P.E. Modeling Dams with Computational Fluid Dynamics: Past Success and New Directions. Available online: https://www.flow3d.com/wp-content/uploads/2014/08/Modeling-Dams-with-Computational-Fluid-Dynamics-Past-Success-and-New-Directions (accessed on 1 October 2018).
14. Chanel, P.G.; Doerin, J.C. Assesment of Spillway Modeling Using Computational Fluid Dynamics. *Can. J. Civ. Eng.* **2008**, *35*, 1481–1485. [CrossRef]
15. FLOW-3D. *FLOW-3D V11.0.3 User Manual*; Flow Science, Inc.: Santa Fe, NM, USA, 2014.
16. Erduran, K.S.; Seckin, G.; Kocaman, S.; Atabay, S. Erduran. 3D Numerical Modelling of Flow around Skewed Bridge Crossing. *Eng. Appl. Comput. Fluid Mech.* **2012**, *6*, 475–489.
17. Getnet, D.; Dereje, H.; Yilma, S. Performance Assessment of Numerical 3D Hydraulic Model Using Spillway Physical Model and Design Results. Ph.D. Thesis, Addis Ababa University, Addis Ababa, Ethiopia, 2019.
18. Water Works Design and Supervision Enterprise (WWDSE/WAPCOS). *Design of Dam and Appurtenant Works Final Report*; Water Works Design and Supervision Enterprise: Addis Ababa, Ethiopia, 2005.
19. Chow, V.T. *Open Channel Hydraulics*; McGraw-Hill: New York, NY, USA, 1976.
20. Ghahfarokhi, G.S.; van Gelder, P.H.A.J.M.; Vrijling, J.K. Evaluation of superelevation in open channel bends with probabilistic analysis methods. In Proceedings of the World Environmental and Water Resources Congress, Honolulu, HI, USA, 12–16 May 2008.
21. Water Works Design and Supervision Enterprise (WWDSE). *Emergency Action Plan for Tendaho and Logia Dam*; WWDSE: Addis Ababa, Ethiopia, 2017.
22. Studio Galli Ingeneria and Sembenelli Consulting. *Zarema May Day Dam and Appurtenant Structures Detailed Design Spillway*; Federal Democratic Republic of Ethiopia Sugar Corporation: Addis Ababa, Ethiopia, 2014.
23. Davidsen, T.S. *Numerical Studies of Flow in Curved Channels*; Department of Mathematics University of Bergen: Bergen, Norway, 2007.
24. Constantinescu, G.; Koken, M.; Zeng, J. Simulation. In *River Flow 2010*; Dittrich, A., Koll, K.A., Aberle, J., Geisenhainer, P., Eds.; Bundesanstalt für Wasserbau: Karlsruhe, Germany, 2010; ISBN 978-3-939230-00-7.
25. Toombes, L.; Chanson, H. Numerical Limitations of Hydraulic Models. In Proceedings of the 34th IHAR World Congress—Balance and Uncertainty, Brisbane, Australia, 26 June–1 July 2011.
26. Kamel, B.; Ilhem, K.; Ali, F.; Abdelbaki, D. 3D simulation of velocity profile of turbulent flow in open channel with complex geometry. *Phys. Procedia* **2014**, *55*, 119–128. [CrossRef]