## 1. Validation

Validation of the FLOW-3D numerical model has been done based on the design of Zarema May Day Dam Spillway in Ethiopia. This verification work was done based on physical model result of Zarema May Day Dam conducted at the PITLAB Laboratory of the University of Pisa, Italy [1], [2]

The comparison was done on measurement of the flow depth along the spillway and with the design data where there is no measured data. The design data is basically computed based on USBR or USACE. Meanwhile, similar design procedure was carried out during research when there is no designed data in the report [1].

Figure 1 shows model output for the validation work of the Zarema May Day Dam Spillway.

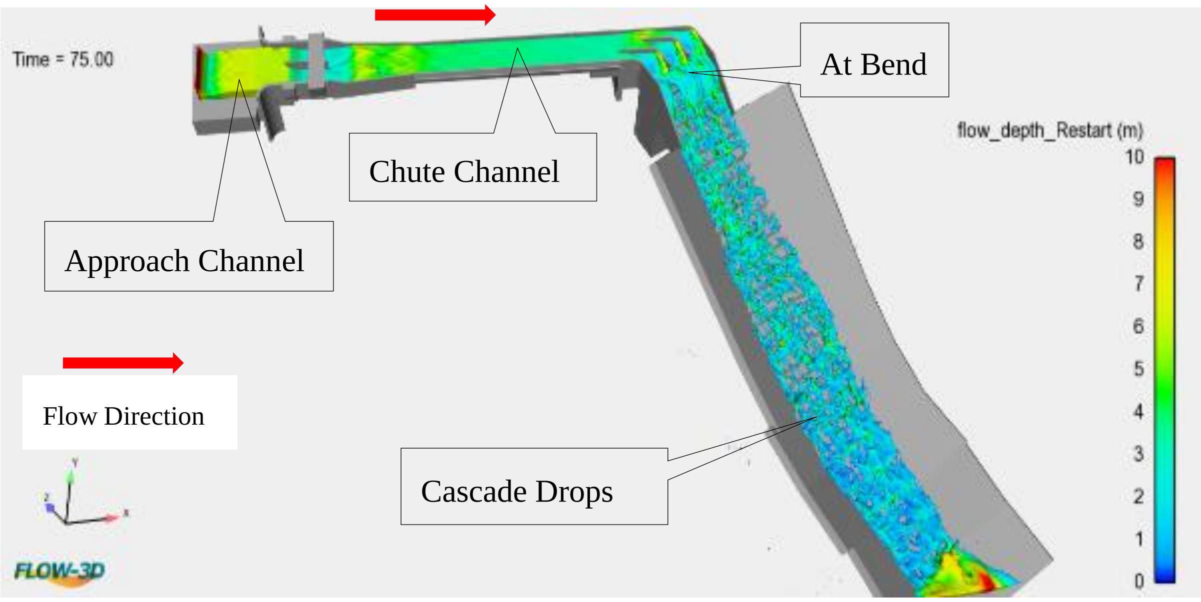


**Figure 1.** 3D model result for the whole part of Spillway in reference to flow depth [1].

### *1.1. Flow Over the Spillway Crest*

There is a physical model result and design result for the rating curve of the spillway. Accordingly, during research, rating curve was verified using USBR and the results are in good agreement. Hence, the three approaches: physical model result, design result and research results are matching with Trend Line $R^2$ 0.9868. Figure 2. Comparison of discharge capacity for Rating Curve of spillway at ogee crest .

shows the show comparison result of the above three approaches.

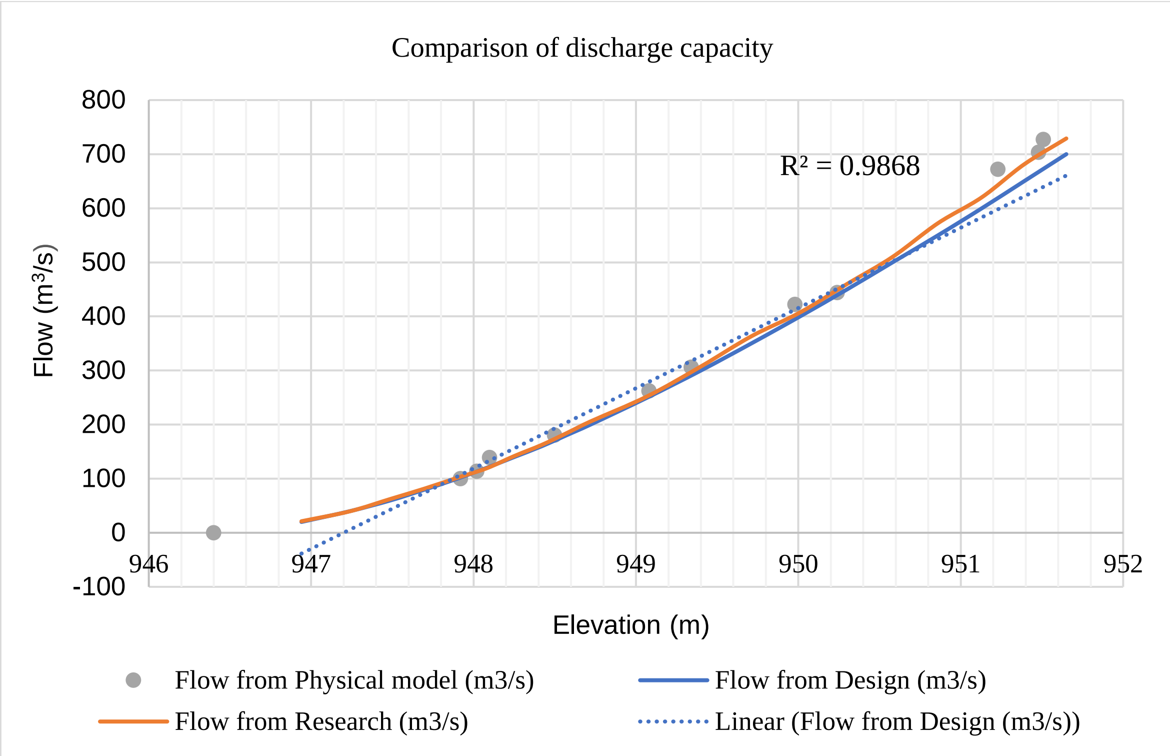


**Figure 2.** Comparison of discharge capacity for Rating Curve of spillway at ogee crest [1].

**Table 1.** Percentage difference of 3D model output with selected model [1]

| Measurement Location | Design | Numerical Method 3D | | Physical Model | Percentage difference between the two selected methods (shaded ones) | Remark |
|---|---|---|---|---|---|---|
| | | Average | Maximum | | | |
| At Approach channel | 5.4 | 5.94 | 6.98 | | 9 | |
| At the Crust of Spillway | 4.99 | 4.46 | 6.31 | 4.88 | -9 | |
| Along the Chute Channel | 3.25 | 3.32 | 4.01 | | 2 | |
| Before Bend | | 6.16 | 6.2 | 6.24 | -1 | |
| At Bend | 6.2 | 6.06 | 6.97 | 6.2 | >11 | As per the design report, the physical model depth result at bend |

| | | | | | | |
|---|---|---|---|---|---|---|
| | | | | | | was less than the depth before bend |
| At Baffle | | 4.35 | 8.42 | | 39 | No design data but depth of channel is 9 m |
| Along Cascade | 8 | 1.9 | 7.55 | | -6 | No design data but depth of channel is 10 m |

NOTE

| | |
|---|---|
| | Recommended model |
| | model for comparison |

*1.2. Summary of the Validation Work*

The concluding remarks for the study showed that FLOW 3D hydrodynamic model can well predicted the flow pattern on the ogee, chute channel and cascade drops. The model shows the overall performance of the spillway system agrees with the measured and designed data. Table 1 shows that FLOW-3D hydrodynamic model has good agreement with the physical model and the designed results [1].

**2. Methods**

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 numerical method of FLOW-3D expressed as follows [3].

The general mass continuity equation is:

$$V_F\frac{\partial p}{\partial t}+\frac{\partial}{\partial x}(\rho u A_x)+\frac{\partial}{\partial y}(\rho v A_y)+\frac{\partial}{\partial z}(\rho w A_z)=R_{DIF}+R_{SOR}---(1)$$

Momentum Equations

$$\frac{\partial u}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial u}{\partial x}+vA_y\frac{\partial u}{\partial y}+wA_z\frac{\partial u}{\partial z}\right\}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+G_x+f_x-b_x-\frac{R_{SOR}}{\rho V_F}(u-u_w-\delta u_s)---(2a)$$

$$\frac{\partial v}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial v}{\partial x}+vA_y\frac{\partial v}{\partial y}+wA_z\frac{\partial v}{\partial z}\right\}=-\frac{1}{\rho}\left(\frac{\partial p}{\partial y}\right)+G_y+f_y-b_y-\frac{R_{SOR}}{\rho V_F}(v-v_w-\delta v_s)--(2b)$$

$$\frac{\partial w}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial w}{\partial x}+vA_y\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial z}\right\}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+G_z+f_z-b_z-\frac{R_{SOR}}{\rho V_F}(w-w_w-\delta w_s)----(2c)$$

In these equations:

($G_x$,$G_y$ ,$G_z$ ) are body accelerations,

($f_x$,$f_y$ ,$f_z$ ) are viscous accelerations,

($b_x$, $b_y$ ,$b_z$ ) are flow losses in porous media or across porous baffle plates, and the final terms account for the injection of mass at a source represented by a geometry component.

The term $U_w$ = ($u_w$, $v_w$, $w_w$) is the velocity of the source component, which will generally be non-zero for a mass source at a General Moving Object (GMO).

The term $U_s$ = ($u_s$, $v_s$, $w_s$) is the velocity of the fluid at the surface of the source relative to the source itself.

$V_F$ = is the fractional volume open to flow
ρ = the fluid density
$R_{DIF}$ = is a turbulent diffusion term
$R_{SOR}$ = is a mass source
$A_x$ = is the fractional area open to flow in the x- direction, $A_y$ and $A_z$ are similar area fractions for flow in the y and z directions, respectively.

The model is considered as incompressible flow and for a free surface, the boundary between water and air, the VOF by function is defined to meet the governing equation. If F(x,y,y,z,t) is equal to 1, the control volume will be full of fluid, and if F is equal to 0, no fluid will exist in a control volume. Furthermore, in the case of a free water surface, F is shown to have the value between 0 and 1.

The Pressure Solution Algorithm in numerical treatment of mass conservation mass equation leads to an algorithm for determining cell pressures and updating velocities. In FLOW-3D this can be done in an automated way [3].

### *2.1. Numerical Simulations Options*

In most cases, the default selections were used; however, in case of momentum advection algorithm, the default momentum advection algorithm is a first-order upwind differencing method. This method is first order accurate in space and time. It is robust and sufficiently accurate in most situations, although, as in any first-order method, it introduces numerical diffusion into the solution. For this it has its own controlling system. When better accuracy is needed for the resolution of flow velocities, e.g., in vortices, then a second-order monotonicity-preserving upwind differencing method can be used by selecting second order monotonicity preserving. This method is second order accurate in space and first order accurate in time. Another second-order method based on central differencing of the advection terms can be used by selecting second order. This method is second order accurate in both space and time. As such, it is the preferred method for minimizing numerical dissipation in swirling flows. This method is the least diffusive of the three, and performs well for circulating flows and free surface waves [3]. In this numerical simulation second order momentum advection was given since there is a sharp bend along the chute channel. Starting from smaller time steps that is from 0.5 seconds upto 100 seconds were given to get the final numerical simulation result. Implicit pressure solver was used.

### *2.2. 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 grid size was given above 0.5 m. Hence, 0.5 m width was given for all hexagonal grid width. Figure 1 shows the plan of the spillway and mesh blocks prepared for the model.

FLOW-3D uses the self-corrective procedure as well as an automatic setting of the convergence criteria that adjusts to whatever is happening during numerical solution process. Generalized Minimum Residual (GMRES) method was preferred due to its rapid convergence and parallel efficiency [3].

### *2.3. Model Geometry*

The numerical analysis was carried out based on original design and modified design. In addition, the designed data obtained from the report which was not done with numerical method, was also compared with the numerical results of design and modified numerical results. For numerical analysis FLOW-3D numerical methods was used. The model for 1D was set up to get overall information at different location along the spillway by giving chainage at 5 m intervals starting from the river (Outlet) and ends at the reservoir (Inlet) with full scale model. The total length of the spillway model is 868 m. Figure 3 and Figure 4 show the model setup.

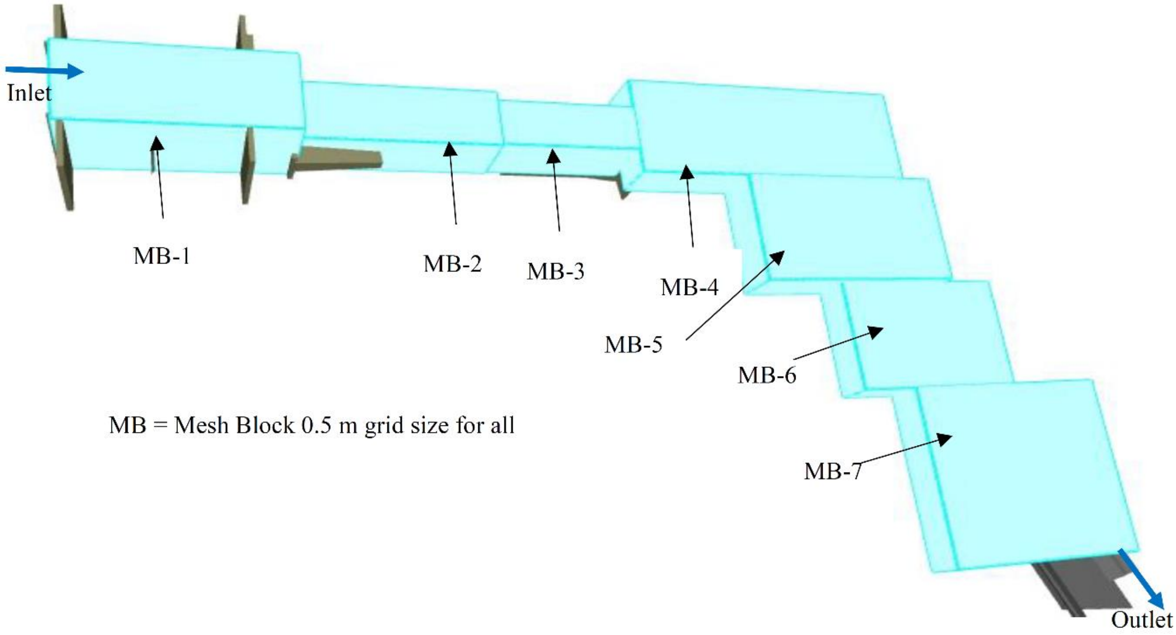


**Figure 3.** Different mesh blocks for the spillway model 0.5 m grid length for all grid.

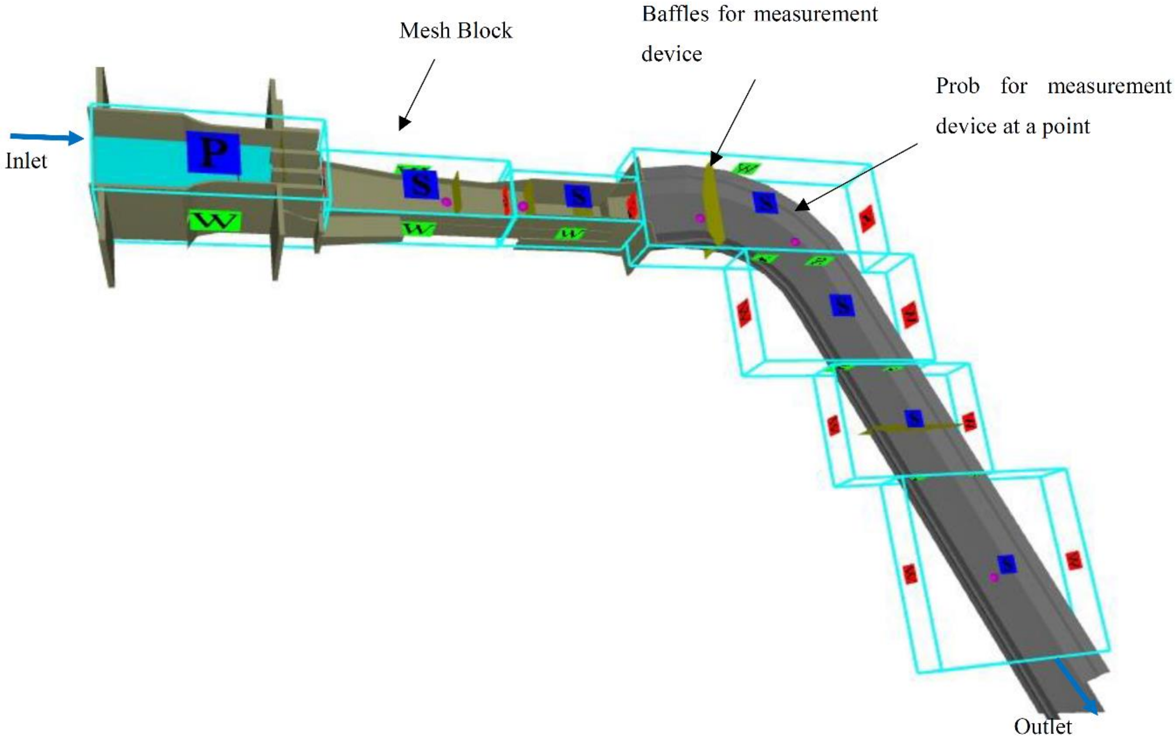


**Figure 4.** Boundary conditions and location of hydraulic measuring devices.

## References

[1] D. Getnet k, H. Dereje and S. Yima, "PERFORMANCE ASSESSMENT OF NUMERICAL 3D HYDRAULIC MODEL USING SPILLWAY PHYSICAL MODEL AND DESIGN RESULTS," 2018.

[2] Studio Galli Ingeneria and Sembenelli Consulting, "Zarema May Day Dam and Auppertenant Structures Detailed Design Spillway," Federal Democratic Republic of Ethiopia Sugar Corporation, Addis Ababa, 2014.

[3] FLOW 3D , "Flow 3D User Manual," 2014.