*Proceedings of the Pakistan Academy of Sciences:*
*A. Physical and Computational Sciences* 53 (2): 177–185 (2016)

ISSN: 0377 - 2969 (print), 2306 - 1448 (online)

Pakistan Academy of Sciences

Research Article

# Application of Computational Flow Dynamics Analysis for Surge Inception and Propagation for Low Head Hydropower Projects

**M. Mohsin Munir[1 *], Taimoor Ahmed[2], Javed Munir[3], and Usman Rasheed[4]**

[1,2,3]Water Resources Division, National Engineering Services, Pakistan, Pvt, Ltd, Lahore, Pakistan
[4]Civil Engineering Department, University of Management and Technology, Lahore, Pakistan

**Abstract:** Determination of maximum elevation of a flowing fluid due to sudden rejection of load in a hydropower facility is of great interest to hydraulic engineers to ensure safety of the hydraulic structures. Several mathematical models exist that employ one-dimensional modeling for the determination of surge but none of these perfectly simulate real-time circumstances. The paper envisages investigation, inception and propagation of surge for a Low-Head Hydropower project using Computational Fluid Dynamics (CFD) analysis in FLOW-3D software. The fluid dynamic model utilizes Reynolds' Averaged Navier-Stokes Equations (RANSE) for surge analysis. The CFD model is designed for a case study at Taunsa Hydropower Project in Pakistan which has been run for various scenarios keeping in view the upstream boundary conditions. The prototype results were compared with the results of physical model and proved quite accurate and coherent. It is concluded that CFD Model gives an insight of the phenomenon which are not apparent in physical model and shall be adopted in future for the similar low head projects. Its application will be helpful in limiting delays and cost incurred in the physical model testing.

**Keywords**: Surge, FLOW-3D, numerical model, Taunsa, RANSE

## 1. INTRODUCTION

Maximum elevation that the flow will achieve due to a sudden rejection of load in a hydropower facility is an important parameter to hydraulic engineers. The information is required in setting the maximum height of side walls to prevent overflow in the headrace channel as well as to understand the surge propagation upstream in headrace channel to schedule the opening of gates of the main barrage and balance the discharge through the barrage and power channel.

In Pakistan, physical model studies are the only practical medium available to understand and analyze the three dimensional and time-dependent complexities of the fluid flow phenomenon. Physical models can only be setup at the final design stage and their execution is expensive. However, the avant-garde computational flow dynamics has emerged not only as an alternative analysis and design tool but also an approach to analyze phenomenon that are not possible to evaluate with physical testing.

## 2. LITERATURE REVIEW

In a hydropower project, a monoclinal wave exhibiting a rapidly varying flow may generate due to sudden closure or opening of the powerhouse control structure such as the sluice gates or wicket gates. This hydraulic transient analysis is of immense importance in hydropower projects.

Audrius et al. [2] concluded in his study that recent advances in Computational Fluid Dynamics (CFD) has made it possible to simulate highly turbulent multiphase flows in reasonable time with good accuracy. The study explored recent development in hydraulic design of Pelton and Turgo Impulse turbines and highlights the opportunities for future expansion. Shojaeefard

Received, January 2016; Accepted, May 2016
*Corresponding Author: M. Mohsin Munir:Email: mohsin_uett@yahoo.com

et al. [3] investigated 3D behavior of axial-flow type microhydro turbines by utilizing an open source computational fluid dynamic (CFD) code, to investigate the rotor-stator interaction and losses occurring in the turbine.

Several mathematical methods exist for one-dimensional analysis, e.g., Saint Venant Equations and the Johnson method. The first method is simplification of the original problem and is based on several assumptions. These methods do not truly represent the original problem and these conditions are rarely met in practice. Similarly, the Johnson method can be arduous as the computation proceeds because it involves the production and propagation of numerous surges. Hence it becomes inaccurate and difficult to assess the surge using these methods.

Other time-dependent analysis methods increase the complexities of the problem by introducing additional independent variable of time since the resulting equations become partial differential equations instead of ordinary differential equations. Method of Characteristics and Implicit and Explicit Finite-Difference Methods have better accuracy and rigor but are time-consuming and cannot be applied appropriately to original conditions. Implicit Methods are preferred in special cases, such as analysis of hydropower projects comprising of open headrace channel with tailrace tunnel (Closed conduit). Such complexities can only be modeled using a contemporary software like FLOW-3D, based on numerical solution schemes that can accurately predict fluid flow using the concept of fluid volume tracking.

FLOW-3D was developed at the Los Alamos National Laboratory in the 1960s and 1970s as a general purpose computational fluid dynamics simulation package. FLOW-3D uses an Eulerian framework in which volume tracking technique models the free surface. It is based on volume of fluid method in which fraction analysis of fluid in each cell is carried out. FLOW-3D is a robust software that can handle the breakup and coalescence of fluid masses.

FLOW-3D uses several models to numerically simulate turbulent flows. For this case study, the Renormalization group (RNG) k-epsilon turbulence model was used with a no slip or partial slip wall shear boundary condition. The RNG turbulence model uses statistical models to solve the turbulent kinetic energy (k) and the turbulent kinetic energy dissipation rate (Є), renormalizing the Navier Stokes Equations to cater for the effects caused by smaller scale motion.

## 3. CASE STUDY OF 135MW TAUNSA HYDROPOWER PROJECT

The Taunsa Hydropower Project has been sited along right bank of Taunsa Barrage across Indus River in Pakistan. It is a run of the river, low head hydropower scheme envisaged to provide power to the national grid. Table 1 summarizes the salient

**Table 1.** Salient features of 135MW Taunsa HPP.

| | |
|---|---|
| Type of Turbine | Horizontal Bulb |
| Mode of Operation | Run of the River |
| Installed Capacity | 135 MW |
| Gross Head | 6.0 m |
| Rated Head | 5.8 m |
| Design Discharge | 3,155.5 $m^3$/sec |
| Turbine Units | 9 Units |
| Headrace Width | 203 m |
| Headrace Length | 1100 m |
| Vertical Gate Height | 16 m |

features of the project.

FLOW-3D analysis is often limited by computation power available. For numerical simplicity in this case, the three-dimensional model was setup based on only one hydropower unit and not the complete power-station. FLOW-3D can effectively use the symmetrical approach conditions of the project layout without affecting the accuracy of the surge analysis. A 3D model was used since flow has three-dimensional characteristics.

### 3.1 Model Preparation

The model for Taunsa hydropower project was prepared in AutoCAD and imported in stereo-lithographic format which is used for rapid prototyping, 3D printing and computer-aided manufacturing. Three dimensional models were prepared for the headrace, transitions, powerhouse and tailrace. Fig. 1 illustrates the 9-unit powerhouse with gates open. Fig. 2 shows the detail of transitions of a single unit with bulb turbine housed inside.

### 3.2 Meshing

Meshing is a consequential part of the analysis process which not only determines the numerical accuracy of the model but also the memory and time required for the simulation. Meshing was done and refined in FLOW-3D model setup options. FLOW-3D has an advantage of FAVOR; the Fractional Area/Volume Obstacle Representation Method, which allows modeling of complex geometries based on equations formulated as functions of the area and volume porosity.

As specified before, for simplicity as well as increased accuracy, meshing was done for one unit. This is a common practice since for large models no of total active cells are limited by computer memory. For Taunsa Hydropower Project, the

**Table 2.** Summary of meshing in FLOW-3D.

| **Total active cells** | 2320000 | | |
|---|---|---|---|
| **Dimension** | **Extent (m)** | **No of cells** | **Cell size (m)** |
| X | 400 | 400 | 1.00 |
| Y | 18.5 | 50 | 0.37 |
| Z | 29 | 116 | 0.25 |

**Table 3.** Flow conditions for Taunsa HPP.

| **Flow Condition** | **U/S water level (m)** | **D/S water level (m)** |
|---|---|---|
| Normal Flow | 135.94 | 130.25 |
| Flood Flow | 136.25 | 133.26 |

**Table 4.** Summary for normal conditions.

| **Location** | **Max Surge** | **Surge Elevation** | **Time (sec)** |
|---|---|---|---|
| **Unit Entrance** | 2.25 m | 138.19 m | 7 |
| **85m U/S** | 2.21 m | 138.15 m | 14 |
| **Headrace Entrance** | 0.53 m | 136.47 m | 260 |

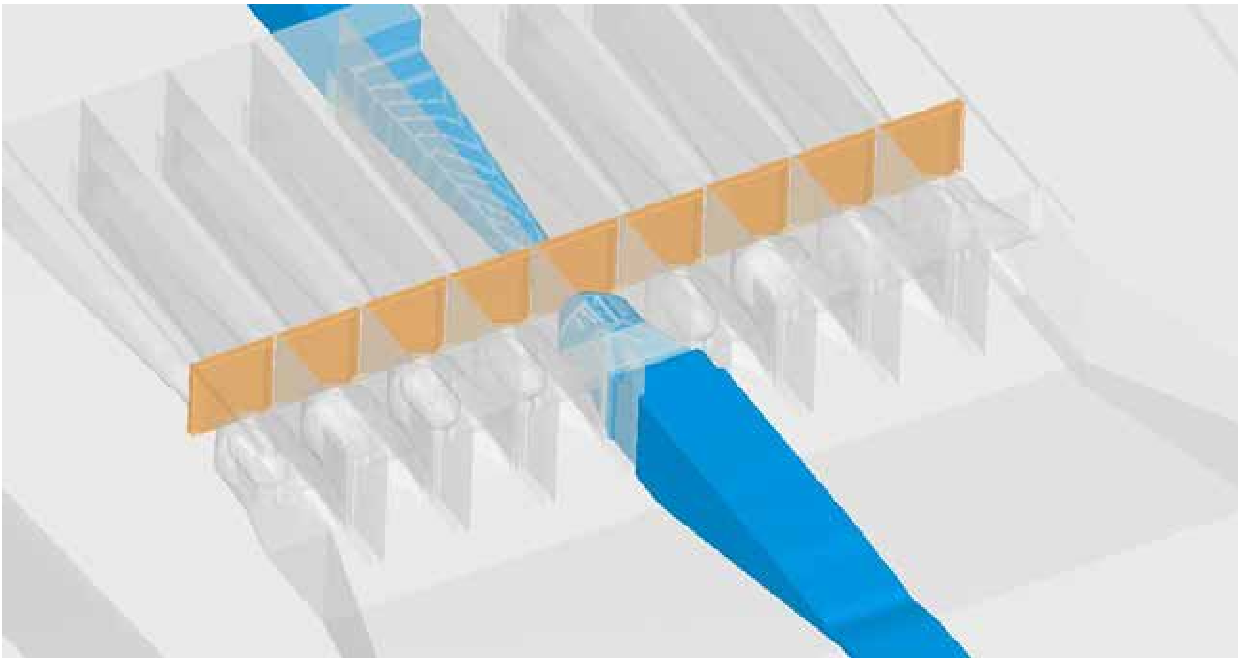

**Fig.1.** Isometric view of Taunsa HPP model.

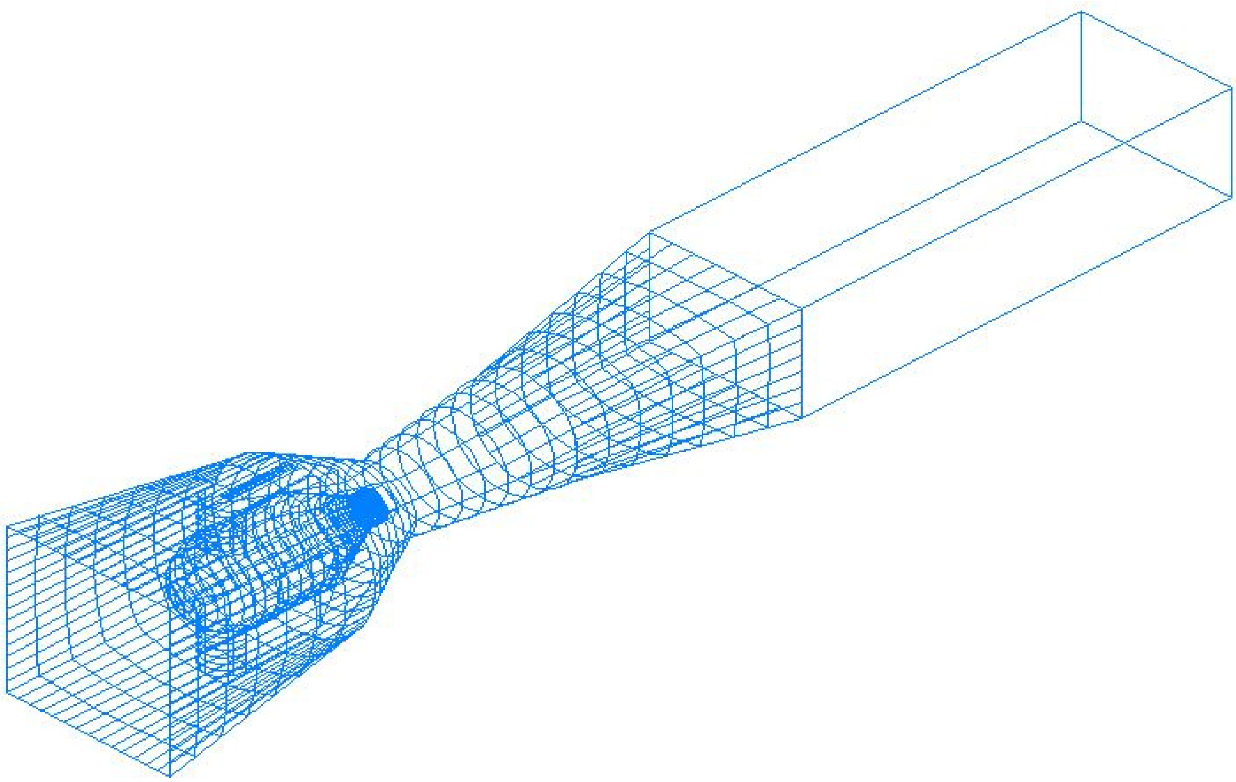

**Fig.2.** Wireframe of a single in-take and draft tube with bulb turbine.

extents, no of cells and cell sizes are tabulated in Table 2.

Cell sizes for the mesh were selected based on two factors. First consideration in this regard is the model accuracy. In case of surge analysis, height of surge (z-direction) is the most important. Hence, smallest cell size has been selected for z axis. Second consideration is CPU memory and computation time. Increasing the number of cells requires greater computation power and longer simulation time. Average simulation time for Taunsa HPP was 15 hours. Keeping in view the aforementioned two factors, minimum cell size was selected so that full computation power can be employed.

### 3.3 Boundary Conditions

Boundary conditions have a huge impact on the final results of the simulation. It is necessary to assess the boundary conditions that best replicate the real-time conditions and actual simulation. Boundary conditions applied for the problem are specified below:

| | |
|---|---|
| X minimum | Specified Pressure Boundary |
| X maximum | Outflow Boundary |
| Y minimum | Symmetry Boundary |
| Y maximum | Symmetry Boundary |
| Z minimum | Wall Boundary |
| Z maximum | Symmetry Boundary |

The upstream boundary condition consisted of a specified pressure to maintain a prescribed reservoir elevation. The downstream boundary utilized the FLOW-3D outflow boundary condition. A symmetry boundary condition was applied along right and left side of the mesh section to take advantage of the inherent symmetry in the problem and thereby decrease computational time while maximizing spatial resolution. Boundary conditions used have been explained below:

#### *3.3.1. Symmetry Boundary*

There is no mass flux (flow) through a symmetry boundary. There is also no shear stress or heat transfer applied at this boundary type. It is useful for reducing the size of a simulation when symmetry exists by cutting the simulation at the symmetry plane.

#### *3.3.2. Wall Boundary*

The wall boundary is similar to the symmetry boundary in which mass flux across the boundary is not allowed. However, with a wall boundary, heating and viscous stresses can be applied.

#### *3.3.3. Specified Pressure Boundary*

This boundary type sets a pressure condition. The pressure can be constant by setting a value in the dialog box, or time dependent by selecting the pressure button.

#### *3.3.4. Outflow Boundary*

This boundary type is useful for surface waves because they are able to leave the flow region without reflecting back into the domain. This boundary type looks at flow conditions just inside the mesh and matches them to allow fluid to freely dissipate through the mesh extent.

### 3.4 Case Study Scenarios

Two flow scenarios have been analyzed for the case study as tabulated in Table 3. During normal flow operation while all 9units are in operation, initial water level elevation of 135.94 m will be maintained in the headrace channel and 130.25m in the tailrace channel. The model is allowed to operate freely for 30 seconds after which the gates are shut at 1.4 m/s to close the orifice in 5 seconds in order to replicate sudden closure conditions. Type B surge or the rejection surge occurs as a result of sudden decrease in power output. Similarly during flood flow operation, initial water level elevation of 136.25 m will be maintained in the headrace channel and 133.26 m in the tailrace channel. Other conditions are kept same as in normal flow conditions.

## 4. RESULTS

The rejection surge occurs as a result of sudden decrease in power output. Fig. 3, 4 and 5 illustrate the surge inception and propagation at different times after load rejection in the extent just upstream of the gates. The points of flow direction reversal

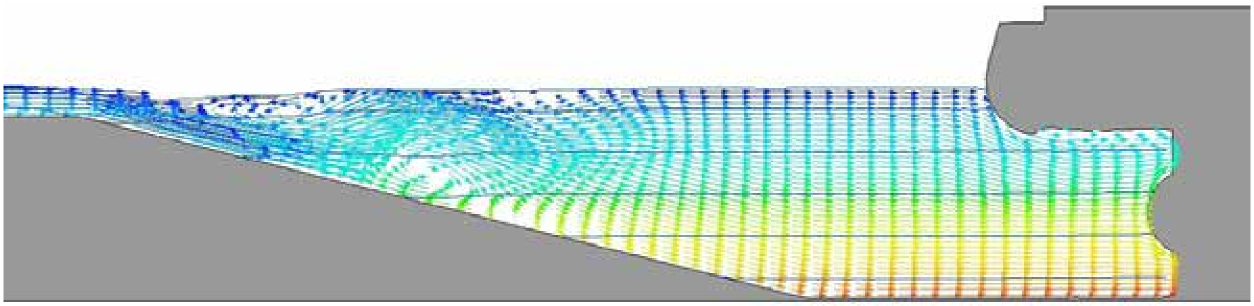

**Fig. 3.** Flow through the power unit at full operation.

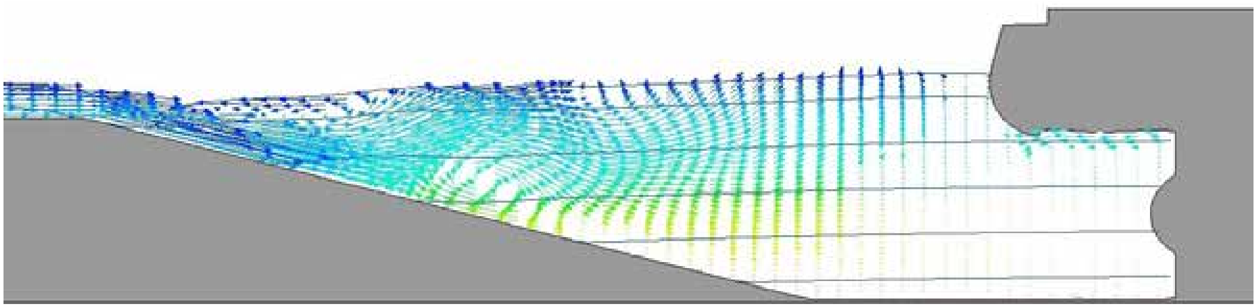

**Fig. 4.**Surge inception at time of load rejection.

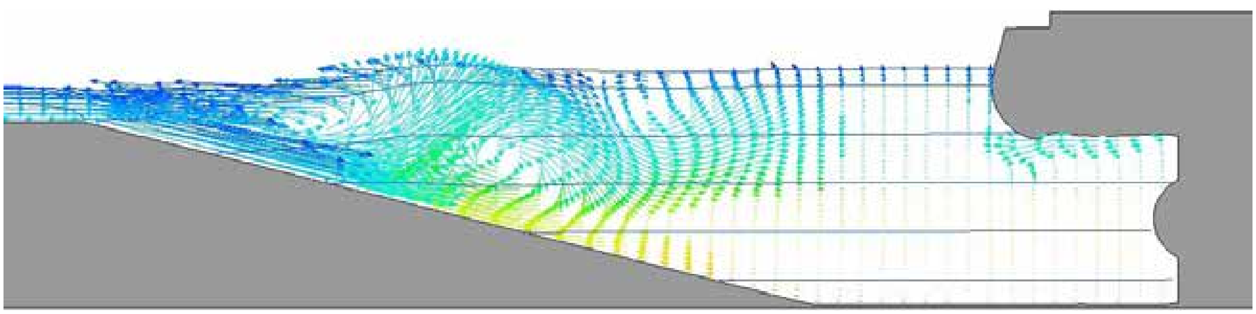

**Fig. 5**. Flow reversal after 10 seconds of load rejection.

can easily be tracked for the upstream advancing surge. This is a feature that is only possible with computational flow dynamics analysis since it is not possible to observe these phenomena in physical model testing.

In Fig. 3, flow vectors at full flow conditions are illustrated by arrows. This shows an uninterrupted movement of fluid. After load rejection, a *region of fluid immobility* starts to develop as observed from Fig. 4. This region acts a cushion against incoming discharge and reverses its direction which results in surge inception. Fig. 5 shows the region of fluid immobility develops upstream with time as surge wave begins to achieve greater elevation. The phenomenon holds true for both normal flow conditions and flood flow conditions.

### 4.1 Normal Flow Conditions

As the surge moves upstream, change in flow depth will ensue. FLOW-3D can track the change in elevation of free surface efficiently for any spatial location of the model. Changes in surface elevation with time were observed for three locations, i.e., at turbine unit entrance, 85 m upstream of the entrance and at headrace entrance.

The temporal and spatial changes in free surface elevation at 3 locations have been shown in Fig. 6. Table 4 shows the summary of results which depict a maximum surge of 2.25 m at unit entrance. Time of arrival of first wave has also been tabulated. The surge is expected to reach the headrace entrance in 260 seconds after gate closure with an average velocity of 4.84 m/s. As the surge wave moves upstream, the height of wave dampens.

### 4.2 Flood Flow Conditions

In case of flood flows in Taunsa hydropower, an upstream water level of 136.25 m is maintained in the headrace with a corresponding tail-water level of 133.26 m in tail-race. Other conditions are kept same as in normal flow conditions. The surge is expected to reach the headrace entrance in 206 seconds with an average surge velocity of 6.7 m/s. Maximum surge elevation of 138.4 m (2.21 m) is observed at the power unit entrance. The results have been tabulated in Table 5. Fig. 7 shows the plot between the change in free surface elevation at the three selected locations and time after load

**Table 5.** Summary for flood conditions.

| Location | Max Surge | Surge Elevation | Time (sec) |
|---|---|---|---|
| **Unit Entrance** | 2.21 m | 138.39 m | 6 |
| **85 m U/S** | 2.00 m | 138.20 m | 12 |
| **Headrace Entrance** | 0.32 m | 136.52 m | 206 |

**Table 6.** Salient features of physical model.

| Features | Symbol | Conversion Factors | Scale Ratio |
|---|---|---|---|
| **Length** | $L_r$ | $L_r$ | 1/36 |
| **Depth** | $Y_r$ | $Y_r$ | 1/36 |
| **Area** | $A_r$ | $L_r Y_r$ | 1/1296 |
| **Time** | $T_r$ | $L_r / Y_r^{1/2}$ | 1/6 |
| **Velocity** | $V_r$ | $Y_r^{½}$ | 1/6 |
| **Discharge** | $Q_r$ | $L_r Y_r^{3/2}$ | 1/7776 |
| **Roughness Co-efficient** | $n_r$ | $Y_r^{2/3} / L_r^{1/2}$ | 1/1.8193 |

**Table 7.** Comparison of parameters between numerical and physical model.

| Parameters | Max Surge | Surge Elevation | Time (sec) |
|---|---|---|---|
| **Numerical Model** | 2.25 m | 138.19 m | 260 |
| **Physical Model** | 2.05 m | 138.00 m | 207 |

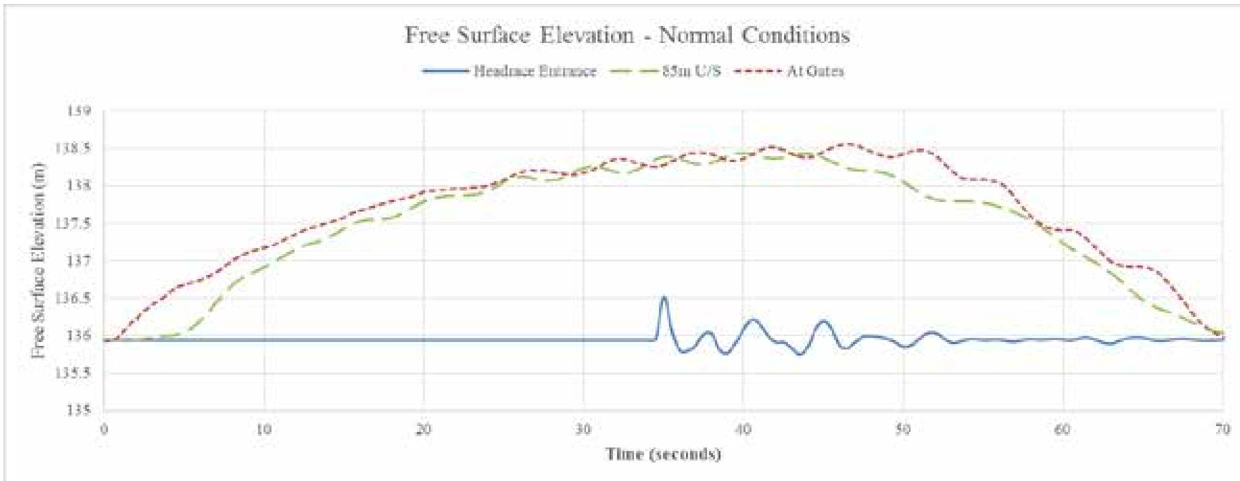


**Fig. 6.** Change in water elevation vs time after load rejection for normal flow conditions Taunsa headrace.

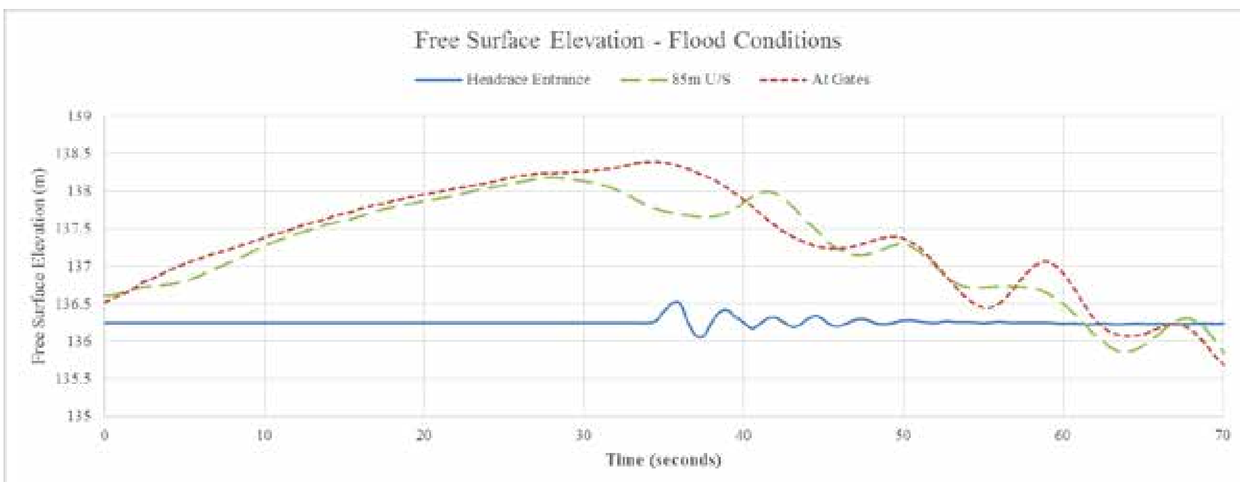


**Fig. 7.** Change inwater elevation vs time after load rejection for flood flow conditions Taunsa headrace.

rejection from the power unit.

It is observed that the surge travels faster and achieves a lower maximum free surface elevation for the flood flow conditions compared to the Normal Flow Conditions.

### 4.3 Physical Model Testing

Geometrical model of the proposed power channel along with appurtenant structures were constructed at a scale of 1:36 to compute maximum surge height and travel time to the start of headrace channel. The salient features of the model tray are presented in Table 6. The simulation results were compared with the mathematical formulae mentioned above as well as the physical testing model carried out at Irrigation Research Institute, Nandipur, Punjab. The results of the physical model testing were within ±10% of the results achieved with simulations in FLOW-3D and presented in Table 7.

## 5. CONCLUSIONS

Observations of the surge inception and propagation are in line with the theory and assumptions specified by VenTe Chow and other authors. For upstream advancing surge, when the surge wave reaches any point in the headrace, the water elevation behind the wave approximately equals the maximum elevation of the surge wave.

The simulation results were compared with the mathematical formulae mentioned above as well as the physical testing model carried out at Irrigation Research Institute, Nandipur, Punjab. The surge height computed at physical model is 2.05 m with maximum surge elevation of 138.00m. The time at which surge wave enters the pond is computed as 207 sec. The results obtained from physical model are found within ±10% of the results achieved with simulations in FLOW-3D. This has built greater confidence in modeling using computational flow dynamics especially for modeling in field of power generation.

Applications of computational flow dynamics have been increasing in engineering applications over recent times. The computer numerical models are a cost-effective alternative to physical modeling techniques offering more flexibility during design and analysis. However, CFD analyses limited by computational power. The present 3D detailed analysis of a single unit required more than 24 hours of computation per simulation to examine a 70-second time-history. A longer time-history is often more desirable.

The analysis of Taunsa hydropower headrace channel using computational flow dynamics is a step-forward in supplementing the results from mathematical modeling and conventional physical model testing in Pakistan. It is not only a robust analysis software solution but can be employed as efficacious tool in the design process of hydraulic structures where it can offer extensive flexibility to assess and compare different proposed designs and their efficiencies. This design support can save a lot of time and money by the optimization process before final designs are physically tested.

## 6. REFERENCES


1. Chow, VenTe. *Open Channel Hydraulics*. McGraw-Hill Book Company, USA, p. 561-565 (1959).
2. Zidonis, A., D.S. Benzon, & G.A. Aggidis. Development of hydro impulse turbines and new opportunities. *Renewable and Sustainable Energy Reviews* 51: 1624-1635 (2015).
3. Shojaeefard, M.H., A. Mirzaei, M.S. Abedinejad, & Y. Yassi. Fluid flow modeling through an axial-flow microhydro turbine, Tehran, Iran. *Technical Gazette 22*(6): 1517-1526 (2015).
4. Rich, G.R. *Hydraulic Transients*, 2nd ed. Dover Publications, New York, NY, USA (1951).
5. Chaudhry, M.H. *Applied Hydraulic Transients*, 3rd ed. Springer, London: p. 450-474 (2014).
6. Yakhot, V., S,A. Orszag, S. Thangam, T.B. Gatski, & C.G. Speziale. Development of turbulence models for shear flows by a double expansion technique. *Physics of Fluids A*. 4 (7): 1510-1520 (1992).
7. *FLOW-3D®SoftwareUser's Manual Version 9.3.* Flow Science Inc. Los Alamos, New Mexico, USA (1997).
8. Hirt, C.W. & B.D. Nichols. Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics* 39(1): 201-225. (1981).
9. Teklemariam, E., B.W. Korbaylo, J.L. Groeneveld, & D,M. Fuchs. Computational fluid dynamics: Diverse applications in hydropower project's design and analysis. In: *Proceedings of the CWRA 55th Annual Conference, Winnipeg, Man,* p. 11-14 (2002).