M.Sc THESIS

HYDRAULIC PERFORMANCE ASSESSMENT OF ORIFICE SPILLWAYS USING CFD MODELLING: A CASE STUDY OF MANGLA DAM MAIN SPILLWAY

ADVISOR

DR. GHULAM NABI

SUBMITTED BY

ENGR. ZOHAIB NISAR
(2011-MS-WRE-09)

CENTRE OF EXCELLENCE IN WATER RESOURCES ENGINEERING,
UNIVERSITY OF ENGINEERING AND TECHNOLOGY,
LAHORE-PAKISTAN

2015

ABSTRACT

Large capacity outlets placed below the dam crest and controlled by gates are called orifice or submerged spillway. Orifice spillways are designed for dual purpose of flood disposal as well as flushing of sediments. Flow passing through the spillway shows the complex turbulent behaviour. The analytical solutions to even the simplest turbulent flow do not exist. The computation becomes more complex when a turbulence model is introduced in Navier-Stokes equations. To model the effect of turbulence, Reynolds's-averaged Navier-Stokes equations are commonly used which is an expanded form of Navier-Stokes equation. The numerical methods such as computational fluid dynamics (CFD) analysis can solve the Navier-Stokes equations in three dimensions and free surface computation in a significantly improved manner.

Traditionally, reduce scale physical spillways model are used to study hydraulic behaviour. With the use of high performance computer and more efficient CFD codes, hydraulic performance of overflow spillway can be numerically investigated successfully. Flow physics becomes more complex due to short lengths of spillway, large variation in reservoir levels, high flow depths and wide range of Froude numbers varying from 3 to 9 in case of orifice spillways. In view of this background, present study intended to numerically investigate the hydraulic behavior of orifice spillway. For this purpose Mangla Dam's main spillway was selected as a case study.

The objectives of this study include modeling of complex flows over the orifice spillways using CFD model, pseudo validation of numerical model results and to assess the flow parameters at different gate openings and reservoir levels. A three dimensional software package Flow 3D by Flow Science was used for this purpose.

3D Solid geometries were created from available 2D drawings and then imported in Flow 3D after converting them to stereo lithographic (Stl.) file format. Water was selected as fluid and a mesh around spillway geometry was created.

Appropriate boundary conditions were set and RNG method was selected as solver and computed the flow rates, pressures, velocities and water surface profiles at different gate openings and reservoir levels.

Initially partial simulation was performed with 3 m mesh size but results were not accurate so partial simulations were performed with 1 m mesh size to refine the results. However different sets of boundary condition were applied for sensitivity analysis and choose Set No. 3 for further spillways analysis. After sensitivity analysis, model validation was done by passing flow rate through single bay model. Maximum flow passed through single bay of spillway was 1810.91 m^3/sec while calculated value of 2227.48 m^3/sec at full gate opening confirms the design discharge capacity at full reservoir level 384 m. Maximum level of water surface at distance of 51. 72 m from dam axis was 334.24 m which is well below top level of chute wall which indicates no danger of overtopping. Gradual increasing trend in pressure value was observed beyond 175 m distance from dam axis at all operating conditions. Generally, pressure remained positive throughout the length of chute which showed that Mangla dam spillway chute was safe against the cavitation damages. Numerical model also successfully modeled velocity parameters which were in acceptable limit and no danger of downstream erosion.

Results of study depicted that Flow-3D is an appropriate code for simulating the orifice spillway flows. Flow characteristics parameter like flow rates, water surface profiles, pressures and velocities were obtained using this code. Moreover, modelling of aerators are recommended for its performance assessment and simulation of flows using smaller mesh size and nested mesh technique should be applied to improve the results.