Numerical Investigations of Downpull Forces for leaf gate in high dams

Bashir Thabit Mohammed*
*Professor in Department of Dam and Water Resources Engineering, College of Engineering, Salahaddin University-Erbil

Binahi Mohammad Amin Said Ali**
**Lecturer in Department of Dam and Water Resources Engineering, College of Engineering, Salahaddin University-Erbil

INTRODUCTION:

Downpull forces are defined as the unbalanced vertical hydrodynamic forces resulting from the difference between the downward force on the gate top due to the flow passing over the top gate surface through the upstream and downstream clearance and the upward force results from pressure exerted on the bottom gate surface by the flow issuing beneath the gate. The net force is termed downpull when it is in the downward direction and uplift when it is in the upward direction. Many studies are conducted to determine the magnitude of downpull and investigated some relevant parameters. The turbulent condition near separation points on the gate bottom by using a laser Doppler-Anemometry was specified. The study concluded that the variation of downpull and discharge coefficient was influenced by the sensitivity of the separated flow pattern near bluff bodies to free stream turbulence and to changes in the mean flow incidence,(Thang et.al.,1983).A one-dimensional of the discharge passing under a tunnel gate and of the hydraulic downpull acting on it was presented and shown that the downpull force was significantly affected not only by the geometry of the gate bottom but also by the rate of flow passing over the top of
the gate through the gate well, (Naudascher et. al., 1986). One-dimensional analyses was presented for estimating the discharge passing over the gate top, the total discharge, top downpull coefficient, the effective piezometric head on the gate top, and predicting the flow condition downstream the gate shaft. Downpull forces acting on the bottom were estimated by predicting the mean pressure and the velocity distribution using two finite element models, one with constant eddy viscosity, and the other of variable eddy viscosity, (Alkadi, 1997). A two-dimensional CFD model was applied to predict a downpull coefficient. The results illustrate that the inclined gate lip shape with an angle of (θ = 35°) gave minimum positive values of downpull force. Moreover, the downpull coefficient depends mainly on the magnitudes and the distribution of (kb) for a given value of gap width ratio. A general statistical model was built to predict the bottom downpull coefficient for any gate lip geometry as well, (Almaini, Al-Kifae and Alhashimi, 2010). Finite volume multiphase flow model using standard (k-ε) turbulence model for the case of high Reynolds number was used. The model was verified by backward-facing step flow and results had been compared with experiments founded by (Durst and Schmitt, 1985), on the other hand, air demand ratio had been determined as a function of Froude number at contracted section, (Shmsai and Soleymanzadeh, 2006). The pressure distribution around the outlet channels was evaluated and calculated the hydrodynamic forces. The physical model results were compared with numerical model results by “Fluent” software based on finite volume method, (Naderi and Hadipour, 2013). Two and three dimensional “FLUENT” software were applied. The simulation revealed the effects of many geometric and hydraulic parameters on downpull coefficient and Experimental investigations at the hydraulic laboratory were also conducted so as to calibrate and verify the numerical results. The evaluated model was used to predict the distributions of piezometric head on the inclined gate lip surface, top and bottom tunnel walls, and predict the distributions of bottom downpull coefficient in the gate lip, (Alkadi and Ali, 2015). In the current study, the water flow controlled by leaf gate in dams under high pressure had been modeled using “FLUENT” software to estimate the effects of the gate well on downpull coefficients.

2. THEORETICAL CONSIDERATIONS

2.1. Hydraulic downpull Force

The downpull force is influenced by various parameters, which may be classified into three groups, (Sagar, 1978). The first group: The flow characteristics which include the operating head on the gate, flow conditions which contain whether free or submerged flow exists, and aeration downstream the gate. The second group: Includes the flow properties such as the specific weight of water, dynamic viscosity, and vapor pressure. The third group: Is the geometry of gate installation, including conduit height upstream the gate shaft, gate opening, gate thickness, gate shaft dimensions, angle inclinations of gate bottom with horizontal, geometry of other lip shapes, and location and thickness of the skin plate. See Fig. (1).

![Fig.(1).General Sketch of the Welled gate.](image)

2.2. Bottom Downpull and Discharge Coefficient

The bottom downpull coefficient along the gate width (k_b) that can be computed by following equation:
The Navier Stokes equations can be closed using
3. GOVERNING EQUATIONS
In the present study, the numerical model is based on the time-averaged conservation equations (mass and momentum) for two-dimensional (2D), turbulent, steady and incompressible flow. The governing conservation equations of continuity and momentum for an incompressible flow with constant viscosity can be expressed as (in tensor form), Piradeepan, (2002):
1. Continuity equation
\[
\frac{\partial}{\partial x_i} (\rho \bar{u}_i) = 0
\]
2. Momentum (Navier-stokes) equation
\[
\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{u}_i \bar{u}_j - \tau_{ij}) + \frac{\partial p}{\partial x_i} - S_{ui} = 0
\]
Where \(S_{ui}\) is a source term. The time-averaged form of these equations for turbulent flows can be derived by substituting the mean and fluctuating component of flow variables, e.g. \(U = \bar{U} + \tilde{u}\), and \(P = \bar{P} + \tilde{p}\). Eq. (5) can rewritten in terms of the time averaged terms as
\[
\frac{\partial}{\partial x_i} (\rho \bar{U}_i) = 0
\]
and the time-averaged momentum equation can be derived as
\[
\frac{\partial}{\partial x_j} (\rho \bar{U}_i \bar{U}_j - \bar{T}_{ij}) + \frac{\partial \bar{p}}{\partial x_i} - \bar{S}_{ui} = \frac{\partial}{\partial x_j} (-\rho \bar{u}_i \bar{u}_j)
\]
3.1. The General Form of the Navier Stokes Equations
Turbulent stresses in Reynolds-averaged Navier Stokes equations can be closed using any of several exiting turbulence models. The simple and most widely used two-equation turbulent model is (k-\(\varepsilon\)) model that solves two separate equations to allow the turbulent kinetic energy and dissipation rate to be independently determined, (Ferzieger and Peric, 1996).
1. The turbulent kinetic energy, \(k\), is modeled as:
\[
\frac{\partial k}{\partial x} + \frac{\partial v_i k}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \nu \frac{\partial k}{\partial x_i} \right) + \frac{\partial}{\partial y} \left( \nu \frac{\partial k}{\partial y} \right) + G - \varepsilon
\]
2. The dissipation rate of \(k\) is denoted, \(\varepsilon\), and modeled as
\[
\frac{\partial \varepsilon}{\partial x} + \frac{\partial v_i \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \nu \frac{\partial \varepsilon}{\partial x_i} \right) + \frac{\partial}{\partial y} \left( \nu \frac{\partial \varepsilon}{\partial y} \right) + C_1 \frac{\varepsilon}{k} G - C_2 \frac{\varepsilon^2}{k}
\]
The term \(G\), representing the production of turbulent kinetic energy, is modeled identically for the standard, RNG, and realizabile (k-\(\varepsilon\)) models. From the exact equation for the transport of \(k\), this term may be defined as:
\[
G = -\rho \bar{u}_i \bar{v}_i + \frac{\partial}{\partial x_i} \left[ \mu_t \left( \frac{\partial \bar{u}_i}{\partial x_i} + \frac{\partial \bar{u}_j}{\partial x_j} \right) \right]
\]
The "eddy" or turbulent viscosity \(\mu_t\) is computed by combining \(k\) and \(\varepsilon\) as follows
\[
\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}
\]
The two dimensional CFD modeling is used to solve a steady state incompressible Reynolds averaged Navier Stokes equations with (k-\(\varepsilon\)) turbulence closure model which is expressed as : (Douglas and Matthews, 1996)
\[
u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \frac{F_x}{\rho} - \frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{1}{\rho} \left( \nu \frac{\partial u}{\partial x} \right)
\]
\[
u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = \frac{F_y}{\rho} - \frac{1}{\rho} \frac{\partial p}{\partial y} + \frac{1}{\rho} \left( \nu \frac{\partial v}{\partial y} \right)
\]
The 2-D continuity equation for incompressible fluid of steady flow can be expressed as:
\[
u \frac{\partial u}{\partial x} + v \frac{\partial v}{\partial y} = 0
\]
predict the turbulent flow, always providing that the eddy viscosity is evaluated.

3.2 Defining Boundary Conditions

Different types of boundary conditions are used for modeling inlet, outlet and walls. Generally, defining boundary conditions includes identifying the location of the boundaries and providing required information at the boundaries. The information required at any boundary usually depends upon the boundary condition type and the physical models used. A fluid boundary is an external surface of a fluid domain and supports following boundary conditions (ANSYS Help, 2011):

- Inlet: Fluid predominantly flows into the domain.
- Outlet: Fluid predominantly flows out of the domain.
- Wall: Impenetrable boundary to fluid flow.
- Symmetry Plane: A plane of both geometric and flow symmetry.

4. RESULTS AND DISCUSSIONS

4.1. Comparisons between the leaf gate with and without well

Fig. (8). illustrates a slight difference between Bottom downpull coefficient \(k_b\) of gate with and without well that computed by ANSYS FLUENT models.

Different profiles of velocity streamline, velocity vector, and pressure contour estimated by ANSYS FLUENT program for a tunnel with (41m) length, (6m) height, (1m) gate thickness with Well and lip angle \((\theta=45^\circ)\), \((b_2/b_1=0.2)\) with gate opening \((YY_0=0.2)\) are plotted in Fig. (9) and the following conclusions can be obtained:

- Maximum velocity reaches to 9.716 m/s.
- The pressure head distribution appears to be uniform at upstream and non-uniform under the gate and at the gap \((b_2)\).
- The streamline shows contraction and separation for the flow field around and near the gate. Moreover, it appears the shape of submerged hydraulic jump \(D/S\) of the gate.

4.2. The effects of gap widths on bottom downpull coefficients

Fig. (10) illustrates bottom downpull coefficients \((k_b)\) computed by ANSYS FLUENT program for tunnel length (41m), lip angle \((\theta=45^\circ)\), pressure head \((\Delta h/H = 10\%)\), gap width ratio \((b_2/b_1) = 0.2, 0.5, 1)\) and reveals an increase for \(k_b\) values with increase of gap width ratio. The figure also shows different shape for \((k_b)\) distribution for the case of \((b_2/b_1=1)\). This phenomenon appears due to the reattachment of separated water at \(Y/Y_0=0.6\)
Different Layouts of velocity streamline, velocity vector, and pressure contour estimated by ANSYS FLUENT program for different gap width ratio, $\Delta h/H =10\%$, and gate opening ($Y/Y_0=0.2$) in 2D are plotted in Figs. (11) and (12). From these figures, one can obtain the followings:

a. Maximum velocity reached to 9.922 m/s and 9.799 m/s for ($b_2/b_1=0.25$ and 1) respectively.

b. The piezometric head distributed non-uniformly around the gate due to change the pressure head above and contraction of the streamline along the gate lip for both ($b_2/b_1=0.5$ and $b_2/b_1=1$).

c. The streamline separates under the gate and then contracted downstream of the gate. The figure also shows a difference in turbulence shape of hydraulic jump for ($b_2/b_1=0.5$ and $b_2/b_1=1$).

4.3. A Comparisons between the leaf gate with and without well with 3D Simulation by ANSYS FLUENT Program

Bottom downpull coefficient ($k_b$) computed by ANSYS FLUENT program for three-dimensional simulation are shown in figure (13). The figure illustrates that there are some differences between the ($k_b$) values for the cases of gate with and without well computed by 3D ANSYS FLUENT models especially at ($Y/Y_o=0.4$). One can conclude that the three-dimensional analyses are not efficient since it is sensitive and needs a
specific size and shape of the elements to reach the stability especially for the case of welled gate. Different profiles of velocity streamline, velocity vector, and pressure contour estimated by ANSYS FLUENT program for a tunnel with (41m) length, (6m) height, (1m) gate thickness with and without well and with lip angle (θ = 45°), considering three-dimensional model and gate opening (Y/Y₀ = 0.2) with (b₂/b₁=0.2) are plotted in figures (14 and 15), and the following conclusions can be obtained:

a. Maximum velocity reaches to 12.022m/s and 9.897m/s for gate without well and with well respectively.

b. The pressure head distribution appears to be uniform at upstream and non-uniform under the gate and at gab (b₂).

c. The streamlines reveal contraction and separation for the flow field around and near the gate. Moreover, it illustrates the submerged hydraulic jump D/S of the gate for two cases.

5. CONCLUSIONS

The following conclusions can be outlined:
1. A comparison between bottom downpull coefficient (k_b) for the gate with and without
well computed by ANSYS FLUENT Program showed a slight difference between of them.

2. An increase of gab width ratio from 0.2 to 1 showed an increase of bottom downpull coefficient ($k_b$) values from 0.46 to 0.62 at gate opening ratio 0.4.

3. A comparison between bottom downpull coefficient ($k_b$) computed by 3D simulation of ANSYS FLUENT Program showed some difference between them especially at $(Y/Y_o)=0.4$.

REFERENCES

- Al-Kadi, B. 1997. *Numerical Evaluation of Downpull Force in Tunnel Gates*. Ph. D Thesis Submitted to the College of Engineering, University of Baghdad.
- Almaini, R., Al-Kifae, M., and Alhashimi, S. 2010. *Prediction Downpull Force on Tunnel Gate with Different Gate Lip Geometry*. Journal of Kerbala University, Vol. 8, No. 4.
- Ansys Help, 2011.
- Al-Kadi, B. and SaidAli, B. 2015. *Simulation of Downpull Forces of Leaf Gate in High Dams*. Thesis Submitted to Department, dams and water resources Engineering and Salahaddin University -Erbil.
- Douglas, J., and Matthews, R., 1996. *Solving Problems in Fluid Mechanics*. Longman Group Limited.
- Durst, F., and Schmitt, F. 1985. *Experimental Studies of High Reynolds Number Backward-Facing Step Flows*, in Proc. Fifth Symp. Turbulent Shear Flows, Cornell University, Pp.5.19-5.24.
- Ferzieger, J., and Peric, M. 1996. *Computational Methods for Fluid Dynamics*. Springer-Verlag, Heidelberg.
- Naderi, M., and Hadipour, E. 2013. *Numerical Simulation of Flow in Bottom Outlet of Narmashir Dam for Calculating of Hydrodynamic Forces*. European Journal of Applied Engineering and Scientific Research, Vol. 2, No. 2, Pp. 33-40.
- Naudascher, E., Rao, P., Richter, A., Vargas, P., and Wonik, G., 1986. *Prediction and Control of Downpull on Tunnel Gates*. Hydraulic Engineering, Vol. 112, No. 5, Pp. 392-416.
- Naudascher, E., Kobus, H., and Rao, R. 1964. *Hydrodynamic Analysis for High-Head Leaf Gates*. ASCE, Journal of the Hydraulic Division. Vol. 90, No. 3, Pp. 155-192.
- Piradeepan, N. 2002. *An Experimental and Numerical Investigation of a Turbulent Airfoil Wake in 90° Curved Duct*. Ph.D Thesis Submitted to Department, Mechanical Engineering and University of Brunel.
- Shamsai, A., and Soleymanzadeh, R. 2006. *Numerical Simulation of Air-Water Flow in Bottom Outlet*. International Journal of Civil Engineering. Vol. 4, No. 1, March.
- Thang, N. D. and Naudascher, E. 1983. *Approach Flow Effect on Downpull of Gates*. ASCE, Vol. 109, No. 11, November.