MODELLING TIDAL POWER PLANT AT SAPHALE

D. N. Deokar  N. S. Lingayat  S. A. Khaparde  S. P. Sukhatme
Energy Systems Engineering  Electrical Engineering Department  Mechanical engineering Department
esakia@ee.iitb.ernet.in
Indian Institute of Technology, Bombay, Mumbai 400076.

ABSTRACT
A small scale site is available near Saphale village in Thane district of Maharashtra state. The shorter construction times and local consumption of the power generated makes this site economically attractive for establishment of the first pilot tidal power plant in the country.

The use of mathematical modeling in planning and design of tidal power plant is established and accepted practice world over. Predicting the changes in tidal dynamics is a major aspect of tidal power projects. The flat estuary model is used for initial estimation of energy potential in a tidal cycle. The one dimensional model is used during feasibility analysis of the tidal power plant. The fundamental principles used in modeling the dynamical behavior of the various components of the tidal power plant and its application to small scale tidal power generation are discussed in this paper.

1 INTRODUCTION
The tidal power plant aspects that can be studied and evaluated using mathematical modeling include[1]:

- The energy output of the barrage.
- The effects of the barrage on water levels in the basin and to seaward.
- The effects of the part built barrage on water levels and tidal currents.
- The effects of the barrage on water levels during river floods and exceptionally high tides.
- The effects of the barrage on tidal currents across and along the estuary.
- The effect of the barrage on sediments movements.
- The effect of the barrage on waves.
- Comparison of different methods of operating the tidal power plant.
- The effect of the barrage on water quality.

The flat estuary model is used for initial energy estimate. In the one dimensional model the direction of flow is assumed to be one dimensional and in a straight line as in the straight pipe of uniform diameter. The water particles move tangent to the flow axis. It is assumed that tidal current moves in the same direction and variations in the velocity are usually ignored. The model combines the principle gravity, inertia and friction forces with shape of the estuary. The one dimensional model is ideal for studying the hydraulic behavior of a barrage. The flow of water in open channels under the influence of the tides is governed by the two basic physical phenomena:

- The continuity of mass
- The conservation of momentum.

The assumption of one dimensional flow is valid in a long narrow estuary. This implies that at any section across the river or estuary all of the flow properties and applied forces (including acceleration, pressure and momentum forces and friction losses) can be defined by the cross section average velocity and level along with knowledge of cross section geometry. The model does not include explicitly the effects of vertical and lateral velocities and the distribution of the longitudinal velocity over the cross section. The equations representing the continuity of mass, the conservation of momentum and friction losses for gradually varying one dimensional flow are applied to represent estuary hydrodynamics. The barrage is assumed to comprise a solid embankment with a number of openings in it. There are two type of openings; the Sluice gates and Turbines. Each type of opening is modeled with specific equations relating the discharge through the opening to the head difference across the barrage. For turbines the operating conditions are the turbines generating power and when they are not generating power. The basic relationship between the flow and total head loss across the barrage in both the operating conditions is represented. These equations constitutes a system of nonlinear first order partial differential equations of hyperbolic type. The numerical methods are used for the solution of these equations. The partial differential equations are replaced with algebraic equations using a four point implicit finite difference method. The algebraic equations are then solved using Newton Raphson method.

2 FLAT ESTUARY MODEL[2]
The discharge can be estimated as:

\[ Q = \frac{AH_o}{t} \]

From the survey of the site, we found
Area A of the basin = 141,000m²
Tidal range Ho = 3.02 m.
Considering the turbine operation for around 4 hours.
Discharge \( Q = 3.02 \times 141,000/4 \times 3600 \approx 30 \text{ m}^3/\text{sec} \)

Assuming tidal curve variation as cosine, the tidal level \( Y_t = 1.51 \cos \left( \frac{\phi}{120} \right) \)

Let the minimum head required for start of turbine be 1m. so,

\( I = 1.51 \cos \left( \frac{\phi}{120} \right) \)

The starting time of turbine \( t_s = 2.342 \text{ Hrs.} \)

The basin level \( Y_b = 0.21276 \times 10^{-3} \times t + 3.3 \)

Again at 5 Hrs. the head difference \( (Y_b - Y_t) \) is 0.8 m. we are closing the turbine at that moment. \( t_2 = 5 \text{ Hrs.} \)

Energy available at the site :

\[
E = \eta \rho g Q \int_{t_1}^{t_2} H \, dt
\]

\[
E = 0.85 \times 1000 \times 9.81 \times 30 \left( Y_b - Y_t \right) \int_{t_1}^{t_2} \, dt
\]

\[= 0.33 \text{ Mwh.} \]

3 ONE DIMENSIONAL MODEL[3,4]

The flow of water in open channels such as estuaries is governed by two laws of conservation:

- Law of conservation of mass
- Law of conservation of momentum

Two variables such as flow level and velocity or flow level and discharge are sufficient to define flow conditions at an estuary cross section. The continuity and momentum equations derived using the two laws of conservation are:

Continuity of mass can be represented as

\[ F = T \frac{\partial y}{\partial t} + \frac{\partial Q}{\partial x} = 0 \quad \ldots \ldots \quad (2.1) \]

Conservation of momentum can be put down as

\[
G = \left(1 - \frac{Q^2 T}{gA^2} \right) \frac{\partial y}{\partial x} + \left( 2 \frac{Q}{gA^2} \right) \frac{\partial Q}{\partial x} + \left( 1 - \frac{Q}{gA^2} \right) \frac{\partial Q}{\partial t} + S_t - S_0 = 0
\]

\[\ldots \ldots \quad (2.2) \]

The continuity equation (2.1) and momentum equation (2.2) constitutes a system of non linear first order partial differential equations of hyperbolic type. A four point implicit finite difference method is used to simulate the flow dynamics.

3.1 Finite Difference Equations[5,6]

The numerical solution of equations (2.1) and (2.2) is obtained in the x-t plane at discrete rectangular set of points as shown in fig. 1

[Diagram of point mesh on the (x-t) plane]

For the development of necessary set of algebraic finite difference equations all the values at time stage J are assumed to be known. The values are used to advance the solution to time stage J+1, where, \( t_{j+1} = t_j + \Delta t \)

The partial derivative of a function 'f' (where f can be Q, y or other variable) at a point M centered with in four grid points with respect to x and t can be expressed as :

\[
\frac{\partial f}{\partial t} = \frac{1}{2\Delta t} \left[ (f_{i+1} + f_{i-1}) - (f_{i+1} + f_i) \right]
\]

\[\ldots \ldots \quad (2.3) \]

\[
\frac{\partial f}{\partial x} = \frac{1}{\Delta x} \left[ (f_{j+1} - f_{j-1}) + (1 - \Psi)(f_{j+1} - f_j) \right]
\]

\[\ldots \ldots \quad (2.4) \]

and variables other than derivatives are approximated as :

\[ f = \frac{1}{2} \left[ (1 - \Psi)(f_{j+1} - f_j) + (1 - \Psi)(f_{j+1} - f_{j-1}) \right] \]

\[\ldots \ldots \quad (2.5) \]

Writing continuity and momentum equation in finite difference form using equations (2.3) to (2.5) we get,

\[
F = T \frac{\partial y}{\partial t} + \frac{\partial Q}{\partial x} + \left[ (f_{i+1} + f_{i-1}) - (f_{i+1} + f_i) \right]
\]

\[
\ldots \ldots \quad (2.6) \]

\[
G = \left(1 - \frac{Q^2 T}{gA^2} \right) \frac{\partial y}{\partial x} + \left( 2 \frac{Q}{gA^2} \right) \frac{\partial Q}{\partial x} + \left( 1 - \frac{Q}{gA^2} \right) \frac{\partial Q}{\partial t} + S_t - S_0
\]

\[\ldots \ldots \quad (2.7) \]

Here T, A, R, and Q represents average value of the corresponding variable as given by the equation 2.5.
Let,
\[
Q_T = \frac{1}{2\Delta t} \left[ (Q^t_{x+1} + Q^t_x) - (Q^t_{x-1} + Q^t_x) \right]
\]  \hspace{1cm} (2.8)
\[
Q_X = \frac{1}{\Delta x} \left[ \psi(Q^t_{x+1} - Q^t_x) + (1-\psi) \left( Q^t_{x+1} - Q^t_x \right) \right]
\]  \hspace{1cm} (2.9)
\[
Y_T = \frac{1}{2\Delta t} \left[ (y^t_{x+1} + y^t_x) - (y^t_{x-1} + y^t_x) \right]
\]  \hspace{1cm} (2.10)
\[
Y_X = \frac{1}{\Delta x} \left[ \psi(y^t_{x+1} - y^t_x) + (1-\psi) \left( y^t_{x+1} - y^t_x \right) \right]
\]  \hspace{1cm} (2.11)

Substituting the values of QT, QX, YT and YX in equations 2.6 and 2.7 we get continuity and momentum equations in modified finite difference form :
\[
F = T \cdot YT + YX
\]  \hspace{1cm} (2.12)
\[
G = \left( 1 - \frac{Q^T}{\Delta A \cdot \Delta x} \right) \cdot YX + 2 \frac{Q^T}{\Delta A \cdot \Delta x} \cdot QX + \frac{\psi}{\Delta A \cdot \Delta t} \cdot QT
\]  \hspace{1cm} (2.13)

Let the channel be divided in N sections. Applying equations (2.12) and (2.13) to each section there are 2N equations. Using newton raphson method the system of equations can be represented in matrix form as follows[7]. The matrix is solved for unknowns using Gauss elimination method.

\[
\begin{bmatrix}
\partial G_1/\partial Y_1 & \partial G_1/\partial Q_1 \\
\partial F_1/\partial Y_1 & \partial F_1/\partial Q_1 & \partial F_1/\partial Q_2 & \partial F_1/\partial Q_3 \\
\partial G_1/\partial Y_1 & \partial G_1/\partial Q_1 & \partial G_1/\partial Q_2 & \partial G_1/\partial Q_3 \\
\partial F_N/\partial Y_N & \partial F_N/\partial Q_N
\end{bmatrix}
\begin{bmatrix}
\delta Y_1 \\
\delta Q_1 \\
\delta Y_2 \\
\delta Q_N
\end{bmatrix}
= \begin{bmatrix}
-R_{G,0} \\
-R_{F,1} \\
-R_{G,1} \\
-R_{F,N}
\end{bmatrix}
\]  \hspace{1cm} (2.14)

4 APPLICATION TO SAPHALE SITE
The proposed model has been applied to Saphale site. Here tidal data at a point away from the barrage and at the barrage has been collected for fifteen days, also tidal currents at both points during spring and neap tide has been measured.

A computer program based on the equations and method given above is used to simulate flow dynamics in a trapezoidal channel.

4.1 Channel Description
The study area is a part of Vaitrana Creek located near village Saphale in the Thane district of Maharashtra state. The channel is approximately 4.7 km long with an average bottom width of 45 m. Initially it is assumed that water level through the channel length is constant. In our case minimum water level available at the mouth of the channel (node1) is considered constant throughout the channel. It is 2.61m. This is the initial condition. The top width of the channel varies from 60m to 80m. The barrage is approximately at the center of the channel. The other end of the channel is closed hence discharge at the last point of the channel is zero. This is the boundary condition. The change in water level at the mouth of the channel is the forcing function.

4.2 Results
The computer program with several sets of combination of computational parameters viz. m, n, \psi, \Delta x, and discharge coefficient have been tried out. The channel bottom slope is neglected (flat channel). Fig 2 shows variation of water level at node1, node 5 (barrage), and node10. The upper curve, middle curve and lower curve represent water level variation at node1, node 5 and node 10.

5 CONCLUSIONS
The equation of continuity and momentum for gradually varied flow can be used to simulate the flow dynamics in open channel. A four point implicit scheme (Preissman scheme) is the most accurate faster and stable method for solving these equations.

In the present study turbine operation is not considered. The present computer program can be modified by changing the corresponding matrix elements. Depending upon the minimum head required for the operation of the turbine, time of turbine operation can be found out and the preliminary estimates of the energy potential can be done.[7]

Initial estimates of optimum values of gate area and turbine area can be done with the total available area as a constraint. Once the total area available for turbine operation is known runner diameter of the turbine can be fixed based on the economical depth available at the barrage. The width of the turbine area and runner diameter can be used to calculate number of turbines.
NOMENCLATURE

A = Flow area  
F = Continuity equation  
f = Function under consideration  
g = Acceleration due to gravity  
G = Momentum equation  
N = number of sections  
n = Manning's coefficient  
Q = Rate of discharge  
R = Hydraulic radius  
S_f = Frictional slope  
S_o = Channel bottom slope  
T = Top width of the channel  
t = Time  
\Delta t = Grid interval along t-axis  
\nu = Flow velocity  
x = Distance along the channel  
\Delta x = Grid interval along x-axis  
y = Flow level  
\psi = Weighting coefficient  
\rho = Density of water  
\eta = Turbine efficiency

REFERENCES


Acknowledgment

The authors would like to thank All India Council for Technical education for providing research grant No. TMT012 / Rec 379 for this work. We also thank Tata Electric Cos. R&D Department for permitting D.N. Deokar to work on this project.