AIN SHAMS UNIVERSITY
FACULTY OF ENGINEERING
Irrigation and Hydraulics Department

Flood Routing Through Lake Nasser Using 2-D Model

A Thesis Submitted for Partial Fulfillment of the Requirements
for the Degree of Doctorate of Philosophy in Civil Engineering
(Irrigation and Hydraulics)

By
ASHRAF MOHAMED EL MOUSTAFA ABDEL BADIE

M.Sc. of Civil Engineering
(Irrigation and Hydraulics)
Ain Shams University, 2002

Supervised By

Prof. Dr. MOUSTAF M. SOLIMAN
Professor of Irrigation Engineering
Irrigation and Hydraulics Department
Faculty of Engineering
Ain Shams University

Prof. Dr. Mohamed Bahaa El Deen
Head of the Irrigation Sector,
Irrigation and Water Resources Vise Minister,
Ministry of Irrigation and Water Resources

Dr. Mohamed Abd Elhamid Gad
Lecturer, Irrigation and Hydraulics Department
Faculty of Engineering
Ain Shams University

Cairo 2007

CHAPTER ONE

Introduction

1.1 General

High Aswan Dam (HAD) construction was completed in 1968 providing full control on the discharges released to the Egyptian irrigation network system. This resulted in the formation of a reservoir that trapped almost all of the inflow and hence formed a large reservoir. The High Aswan Dam Reservoir (HADR) extends for 500 km along the Nile River and covers an area of 6,000 km^2 at water level equal 182.00 m above mean sea level.

An uncontrolled spillway was constructed at the end of Khor Tushka (on the western side of Lake Nasser at about 256 km upstream the dam). This spillway is connected to Tushka depression by a canal by which the excess flood can be turned to the depression.

1.2 Scope of Work

The question of operation of the HAD was neglected for a long time after the dam has been constructed as it seemed simple as the inflows were large enough so the water could always be released to meet downstream requirements and any water remaining was simply used to fill the reservoir behind the dam.

The ministry of irrigation currently decides the daily release from the HAD, and the ministry of electricity determines the distribution of discharges over the 24 hour period in order to effectively integrate the hydroelectric generation into the daily requirements of the national grid.

The development of river basin policy and management plans involves a spectrum of concerned parties and organizations, only a small fraction of which are presented by technical professionals. Easily-used and highly-interactive computer simulations provide one means by which these individuals can develop a conceptual and intuitive understanding for the complex physical behavior of river systems.

The thesis presents the development of a 2D simulation model of HADR. The modeling study aims to determine the optimal management policy for the reservoir in flood conditions by modeling the physical problem via a 2D routing model. The hydrodynamic model simulated the flow fields – steady and unsteady- of the High Aswan Dam Reservoir.

During the work progress a hydrological model, HADR Simulator, was developed for dam operation as the hydrodynamic model generated was useful in future studies of sedimentation or water quality but found to be time consuming if it to be used for dam operation.

1.3 Organization of Work

This thesis is organized in eight chapters as follows to study to what extend the spillway will protect HAD against floods with levels more than the maximum design levels.

Chapter one: gives an introduction about the subject and the organization of the work and objectives.

Chapter two: presents brief notes and literature review about routing techniques, hydrodynamic and hydrological methods, which were used in the development of thesis models.

Chapter three: presents the problem definition, description of High Aswan Dam Reservoir and scope of the thesis.

Chapter four: presents the data preprocessing and the simulation of a of the reservoir bed level with a three dimensional surface.

Chapter five**:** presents the model selection, description and limitations. The development of the two-dimensional hydrodynamic model, calibration and verification process, steady state simulations are also presented.

Chapter six**:** presents the study of various flood waves and the real flood wave on the developed model and the modifications made to the hydrodynamic model to be used in the dam operation.

Chapter seven**:** presents the development of hydrological model for the purpose of dam operation along with the results of testing the effect of varying the model parameters.

Chapter eight**:** presents the main conclusion of the research and also states the recommendations to be taken into consideration in the future work.

1.1 General 3
1.2 Scope of Work 3
1.3 Organization of Work 4

CHAPTER TWO

Literature Review

General

This chapter describes several different hydraulic and hydrologic routing techniques. Assumptions, limitations, data requirements for each and the basis for selection of a particular routing technique are reviewed.

Routing is a process used to predict the temporal and spatial variations of a flood hydrograph as it moves through a river reach or reservoir. The effects of storage and flow resistance within a river reach are reflected by changes in hydrograph shape and timing, attenuation and travel time, as the floodwave moves from upstream to downstream. In general, routing techniques may be classified into two categories: hydraulic routing, and hydrologic routing.

Hydrologic Versus Hydraulic Routing Techniques

The hydrologic method is in general simpler but fails to give entirely satisfactory results in problems other than of determining the progress of a flood down a long river. For example when a flood comes through a junction, backwater is usually produced and this can only be accurately evaluated by the basic hydraulic equations.

Hydraulic routing techniques are based on the solution of the partial differential equations of unsteady flow. Hydrologic routing employs the continuity equation and an analytical or an empirical relationship between storage within the reach and discharge at the outlet.

The hydrologic approaches, which are simpler to use but harder to defend theoretically, on the other hand the hydraulic approaches, which are better grounded in basic theory are relatively difficult to apply. It is the relationship between the geometrical quantity, storage, and the kinematic quantities, discharge hydrographs, which makes the hydrologic and hydraulic approaches differ.

In channel design, floodplain studies, and watershed simulations, generally, hydrologic routing is utilized on a reach-by-reach basis from upstream to downstream. This type of approach is adequate as long as there are no significant backwater effects or discontinuities in the water surface because of jumps or bores.

When there are downstream controls that will have an effect on the routing process through an upstream reach, the channel configuration should be treated as one continuous system. This can only be accomplished with a hydraulic routing technique that can incorporate backwater effects as well as internal boundary conditions, such as those associated with culverts, bridges and weirs.

It is possible that the transverse variations will be of greater importance than the stream wise values. This is why we used a 2D model in our study due to the natural sophisticated lay out of the HAD reservoir.

Hydraulic Routing Technique

The equations that describe the unsteady flow in the three dimensions, the Saint Venant equations, consist of the conservation of mass equation, Equation 2-1, and the conservation of momentum equation, Equation 2-2. The solution of these equations defines the propagation of a flood wave with respect to distance along the channel and time.

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{2-1}$$

$$\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}+\rho w\frac{\partial u}{\partial z}-\frac{\partial}{\partial x}(\varepsilon_{xx}\frac{\partial u}{\partial x})-\frac{\partial}{\partial y}(\varepsilon_{xy}\frac{\partial u}{\partial y})-\frac{\partial}{\partial z}(\varepsilon_{xz}\frac{\partial u}{\partial z})-\frac{\partial p}{\partial x}-\tau_x=0$$

$$\rho\frac{\partial v}{\partial t}+\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}+\rho w\frac{\partial v}{\partial z}-\frac{\partial}{\partial x}(\varepsilon_{yx}\frac{\partial v}{\partial x})-\frac{\partial}{\partial y}(\varepsilon_{yy}\frac{\partial v}{\partial y})-\frac{\partial}{\partial z}(\varepsilon_{yz}\frac{\partial v}{\partial z})-\frac{\partial p}{\partial y}-\tau_y=0$$

$$\rho\frac{\partial w}{\partial t}+\rho u\frac{\partial w}{\partial x}+\rho v\frac{\partial w}{\partial y}+\rho w\frac{\partial w}{\partial z}-\frac{\partial}{\partial x}(\varepsilon_{zx}\frac{\partial w}{\partial x})-\frac{\partial}{\partial y}(\varepsilon_{zy}\frac{\partial w}{\partial y})-\frac{\partial}{\partial z}(\varepsilon_{zz}\frac{\partial w}{\partial z})-\frac{\partial p}{\partial z}-\tau_z=0$$

(2-2)

where

u = average velocity of water in x direction;
x = distance along channel in x direction;
v = average velocity of water in y direction;
y = distance along channel in y direction;
w = average velocity of water in z direction;
z = distance along channel in z direction;
ρ = water density;
τ = shear stress;
t = time;

ε = Eddy viscosity;
Solved together with the proper boundary conditions, Equations 2-1 and 2-2 are the complete dynamic wave equations. The dynamic wave equations are considered to be the most accurate and comprehensive solution to 1-D unsteady flow problems in open channels.

The full unsteady flow equations have the capability to simulate the widest range of flow situations and channel characteristics. Hydraulic models, in general, are more physically based since they only have one parameter (the roughness coefficient) to estimate or calibrate. Roughness coefficients can be estimated with some degree of accuracy from inspection of the waterway, which makes the hydraulic methods more applicable to ungauged situations.

Hydrologic Routing Technique

Hydrologic routing employs the use of the continuity equation and either an analytical or an empirical relation ship between storage within the reach and discharge at the outlet. In its simplest form, the continuity equation can be written as inflow minus outflow equals the rate of change of storage within the reach:

$$I - O = \frac{\Delta S}{\Delta t} \tag{2-3}$$

Where
I = the average inflow to the reach during Δt,
O = the average outflow from the reach during Δt and
S = storage within the reach [L3].

Additional terms can be added to this formula according to its importance and the avaliable data for calculation. One of those important terms in the study of great reservoirs with large surface area is the evaporation factor which acts as an outflow from the reservoirs but in an upword direction, also precipitation can be considered as the evaporation with a negative sign. Bank storage and groundwater-lake interaction are also two important factors that may be considered specially for reservoirs and big rivers.

Some of the famous hydrological routing techniques are summarized below.

1. **Modified Puls Reservoir Routing.**

The equation defining storage routing, based on the principle of conservation of mass, can be written in approximate form for a routing interval Δt . Assuming the subscripts “1” and “2” denote the beginning and end of the routing interval, the equation is written as follows:

$$\frac{O_1+O_2}{2}=\frac{I_1+I_2}{2}-\frac{S_2-S_1}{\Delta t} \qquad (2\text{-}4)$$

2. **Modified Puls Channel Routing.**

Routing in natural rivers is complicated by the fact that storage in a river reach is not a function of outflow alone. During the passing of a flood wave, the water surface in a channel is not uniform. The storage and water surface slope within a river reach, for a given outflow, is greater during the rising stage of a flood wave than during the falling stage. Therefore, the relationship between storage and discharge at the outlet of a channel is not a unique relationship, rather it is a looped relationship.

3. **Muskingum Method.**

The Muskingum method was developed to directly accommodate the looped relation-ship between storage and outflow that exists in rivers.

With the Muskingum method, storage within a reach is visualized in two parts: prism storage and wedge storage. Prism storage is essentially the storage under the steady-flow water surface profile. Wedge storage is the additional storage under the actual water surface profile. During the rising stages of the flood wave, the wedge storage is positive and added to the prism storage. During the falling stages of a flood wave, the wedge storage is negative and subtracted from the prism storage.

$S =$ prism storage + wedge storage

$S = KO + KX(I - O)$

$$S = K[XI + (1 - X)O] \quad (2\text{-}5)$$

Where

S = total storage in the routing reach;

O = rate of outflow from the routing reach;

I = rate of inflow to the routing reach;

K = travel time of the flood wave through the reach;and

X = dimensionless weighting factor, ranging from 0.0 to 0.5

4. Working R & D Routing Method.

This method is also useful in situations wherein the horizontal reservoir surface assumption of the modified puls procedure is not applicable, such as normally occurs in natural channels.

The working R&D procedure could be termed “Muskingum with a variable *K*” or “modified puls with wedge storage.” For a straight line storage-discharge (weighted discharge) relation, the procedure is the same solution as the Muskingum method. For X = 0, the procedure is identical to Modified Puls.

Typically, in rainfall-runoff analysis, hydrologic routing procedures are utilized on a reach-by-reach basis from upstream to downstream. In general, the main goal of the rainfall-runoff study is to calculate discharge hydrographs at several locations in the watershed. In the absence of significant back water effects, the hydrologic routing models offer the advantages of simplicity, ease of use, and computational efficiency .

Also, the accuracy of hydrologic methods in calculating discharge hydrographs is normally well within the range of acceptable values. It should be remembered, however, that insignificant backwater effects alone do not always justify the use of a hydrologic method. There are many other factors that must be considered when deciding if a hydrologic model will be appropriate, or if it is necessary to use a more detailed hydraulic model.

Evaluating The Routing Method

With such a wide range of hydraulic and hydrologic routing techniques, selecting the appropriate routing method for each specific problem is not clearly defined. However, certain thought processes and some general guidelines can be used to narrow the choices, and ultimately the selection of an appropriate method can be made.

There are several factors that should be considered when evaluating which routing method is the most appropriate for a given situation. The following is a list of the major factors that should be considered in this selection process:

1. Backwater effect.
2. Flood plains.
3. Channel slope and hydrograph characteristics.
4. Flow networks.
5. Flow regiem.
6. Data availability.

Steady versus Unsteady Flow Models

The traditional approach to river modeling has been the use of hydrologic routing to determine discharge and steady flow analysis to compute water surface profiles. This method is a simplification of true river hydraulics, which is more correctly represented by unsteady flow. Nevertheless, the traditional analysis provides adequate answers in many cases.

Steady flow analysis is defined as a combination of a hydrologic technique to identify the maximum flows at locations of interest in a study reach (termed a "flow profile") and a steady flow analysis to compute the (assumed) associated maximum water surface profile.

Steady flow analysis assumes that, although the flow is steady, it can vary in space. In contrast, unsteady flow analysis assumes that flow can change with both time and space.

Unsteady flow analysis should be used for all streams where the slope is less than 2 feet per mile. On these streams, the loop effect is predominant and peak stage does not coincide with peak flow. Backwater affects the outflow from tributaries and storage or flow dynamics may strongly attenuate flow; thus, the profile of maximum flow may be difficult to determine.

Previous Studies

Several studies were done on the flood routing technequies, flood routing modelling and dam operation models some of these studies are presented as follows.

M. S.K. Chowdhury and F. C. Bell (1980) developed a new runoff routing model that combines realistic allowances for the spatial distribution of storage with the theoretically satisfying features of the kinematic wave approximation. Appropriate boundary conditions enabled replacement of the partial differential equations describing the flow by tractable total differential equations.

In addition, similar forms of equations have been adopted to describe both overland and channel flow. All these features resulted in a relatively simple model with a small number of physically relevant parameters that are not difficult to evaluate.

The required model input is a temporal pattern of rainfall excess from a runoff generation model. A number of quite different runoff generating models may all be used for this purpose. However, the estimation of rainfall excess will not be considered here, since the focus of this paper is on the development of the new routing procedure.

The use of the current discharge as a state variable enables the model to be automatically tuned to the current conditions and is particularly suitable for short-term flood forecasting.

Bernard L. Golding (1981) developed a Basic language program for routing floods through storage reservoirs or detention basins by the storage-indication working curve method (Modified Plus Method). A sample program was included and explained step-by-step. Standard flood routing equations were included. Many municipalities require that post-development runoff cannot exceed pre-development runoff in their subdivision regulations. Building a retention basin that acts as a small flood control reservoir normally did this.

Stanley S. Butler (1982) presented an alternate reservoir flood routing approach applicable for routing design floods determined from statistically derived design storms. The approach treated routing as an instantaneous discharge point-function process instead of an average discharge incremental time procedure, avoiding some of the difficulties and errors in the traditional methods.

The point-slope method of routing floods through reservoirs used in his work can be described as follows. He used instantaneous-time functions in the form of equations for determining the outflow hydrograph slope and the inflection point of the rising limb of the outflow hydrograph based on the inflow hydrograph, the topography of the reservoir, and the hydraulic characteristics of the outflow structure.

The point-slope method is less broadly applicable than the traditional incremental-time methods, but within its limitations (simple singlepeaked inflow hydrographs), it is fast, accurate, and adaptable for investigating alternate designs. A check and adjustment procedure provides assurance that the result is valid. This procedure and the criteria for determining the inflection point of the outflow hydrograph are of general applicability with reservoir flood routing methods.

Richard J. Heggen (1983) developed a Basic program to route flood discharge through a system of river channels and reservoirs. The brevity of this program and the use of an overlaid computational matrix make it suitable for minicomputer execution.

Channel hydrographs are computed by the Muskingum method; reservoir hydrographs, by the Puls method. Data input consists of routing coefficients for channels, reservoir state-discharge-storage curves, and description of network configuration. The program is suited for analysis of open channels conveyance systems, flood detention reservoirs and combination of two.

Tawatchai Tingsanchali and Shyam K. Manandhar (1985) developed an analytical diffusion model for flood routing, the basic diffusion equation is linearized about an average depth and takes into account backwater effect and lateral flows. The model was applied to route the floods in a hypothetical rectangular channel with different upstream, downstream, and lateral boundary conditions.

The applicability of the model is limited to slow rising floods in which the effects of flow acceleration can be neglected. The channel characteristics were assumed and the results obtained were compared with those obtained by the finite difference method of implicit scheme based on the complete Saint-Venant equations for unsteady open channel flow and were found to have a standard deviation of about 0.035.

The model showed good results when applied to simulate flood flow conditions in 1980 and 1981 in the Lower Mun River, in Northeast Thailand. The model cannot be incorporated with detailed data of cross sections or riverbed geometry. It only requires only their average values.

The Chézy, C and the diffusivity, k due to channel irregularities were used in the model and were determined by trial and error during model calibration. The model provides an excellent means to analyze individual or overall effects of the boundary conditions and requires much less effort and time for computation at a particular station.

D.L. Fread, National weather service (*NWS*) (1988) developed the Hydrologic Research Laboratory (HRL) of the NWS Office of Hydrology dynamic wave routing models suitable for efficient operational use in a wide variety of applications involving the prediction of unsteady flows in rivers, reservoirs, and estuaries. These models are based on an implicit (four-point, nonlinear) finite-difference solution of the complete one-dimensional Saint-Venant equations of unsteady flow.

Because, fixed arrays within the computer program for the number of time steps and number of cross sections severely limit the size of the river systems that can be modeled without breaking up the application into several datasets. Since the mid-1980's, a comprehensive Flood Wave routing model (FLDWAV) has been undergoing development and testing. This state of the art model combines the capabilities of DWOPER and DAMBRK, and provides features not contained in either of these models.

FLDWAV has undergone extensive testing (over 160 datasets) to ensure The same level of accuracy and stability as the DAMBRK and DWOPER models. It has also gone through two years of beta testing. The FLDWAV model will continue to undergo development improvements and testing by the NWS to increase its range of applicability and numerical robustness for more convenient usage.

FLDWAV is a generalized flood routing (unsteady flow simulation) model. The governing equations of the model are the complete one-dimensional Saint-Venant equations of unsteady flow, which are coupled with internal boundary equations representing the rapidly varied (broad-crested weir) flow through structures such as dams, and bridge embankments, which can develop a user, specified time-dependent breach. In addition, appropriate external boundary equations at the upstream and downstream ends of the routing reach are utilized.

The system of equations is solved by an iterative, nonlinear, weighted four-point implicit finite-difference method. The flow may be either subcritical or supercritical or a combination of each varying in space and time from one to the other; fluid properties may obey either the principles of Newtonian (water) flow or non-Newtonian (mud/debris flows or the contents of a mine-tailings dam) flow. The hydrograph to be routed may be user-specified as an input time series, or it can be developed by the model via user-specified breach parameters (size, shape, time of development).

The possible presence of downstream dams, which control the flow and may be breached by the flood, bridge/embankment flow constrictions, tributary inflows, river sinuosity, levees located along the tributaries and/or downstream river, and tidal effects are each properly considered during the downstream propagation of the flood.

H. A. Basha (1995) developed a routing equation for detention reservoir systems from an approximate analytical solution of the nonlinear storage differential equation. The approximate solution was obtained by a two-term perturbation expansion whereby the zeroth-order term is the linear solution and the first-order term is the correction. The first-order approximation, which allows for arbitrary multi-event inflow hydrographs, irregular reservoir configuration, and various types and sizes of outlets, is found to be accurate for all practical purposes. The asymptotic solution allowed the derivation of design equations that can apply for arbitrary reservoir configuration and for various types and sizes of outlet.

Tefaruk Haktanir and Hatice Ozmen (1997) computed, using the computer program DUFLOW package, the Outflow hydrographs for three dams with long lakes in narrow valleys using both hydrologic routing (level-pool routing) and hydraulic routing. These hydrographs were then compared with three inflow hydrographs of different peaks.
The DUFLOW package is based on the one-dimensional partial differential equations to describe unsteady flow in open channels, the continuity and momentum equations.

In all these cases, the difference between outflow hydrographs was greatest at the peak value relative to the magnitude of the inflow hydrograph. The peak outflow by hydraulic routing was smaller than that by hydrologic routing for all the routing combinations, the difference varying between 2 and 11%.

Francisco Nunes Correia et al (1998) presented results based on the use of Intergraph GIS coupled with Idrisi GIS. Using these two systems substantially increased the flexibility of using GIS as a tool for flood studies.
A lumped (XSRAIN) and a distributed (OMEGA) hydrologic model were used to simulate flood hydrographs. The well known HEC-2 Hydraulic model was used to compute flooded areas. These models were applied in the Livramento catchment with very good results. The computation of flooded areas for different flood scenarios, and its

representation in GIS, can be used in the assessment of affected property and associated damages.

This is a very useful GIS-based approach to floodplain management. Geographic Information Systems (GIS) have been recognized as a powerful means to integrate and analyze data from various sources in the context of comprehensive floodplain management. As part of this comprehensive approach to floodplain management, it is very important to be able to predict the consequences of different scenarios in terms of flooded areas and associated risk.

Hydrologic and hydraulic modeling plays a crucial role and there is much to gain in incorporating these modeling capabilities in GIS. This is still a rather complex task and research to be made on the full integration of these models. Interfacing between these models and GIS may be a very efficient way of overcoming the difficulties and getting very good results in terms of engineering practice.

T. R. Neelakantan and N. V. Pundarikanthan (2000) trained a back propagation neural network to approximate the simulation model developed for the Chennai city water supply problem. The neural network was used as a sub-model in a Hooke and Jeeves nonlinear programming model to find ‘‘near optimal policies.’’

The results were further refined using the conventional simulation-optimization model. There have been several attempts to use combined simulation-optimization models to solve reservoir operation problems efficiently. In many cases, complex simulation models are available, but direct incorporation of them into an optimization framework is computationally prohibitive, this is why this model was developed.

Abbas Seifi and Keith W. Hipel (2001) proposed a new method for long-term reservoir operation planning with stochastic inflows. In particular, the problem was formulated as a two-stage stochastic linear program with simple recourse. Multiple inflow scenarios, leading to a very large deterministic model that is hard to solve using conventional optimization methods, approximated the stochastic inflows. An efficient interior-point optimization algorithm was presented for solving the resulting deterministic problem. It was also shown how exploiting the problem structure enhances the performance of the algorithm. Application to regulation of the Great Lakes system shows that the proposed approach could handle the stochastic of the inflows as well as the nonlinearity of the operating conditions in a real-world reservoir system.

Mohamed M. Hantush, Morihiro Harada and Miguel A. Marin (2002) Analytical solutions were developed for routing stream flow, lateral stream-aquifer interactions, and aquifer storage. In effect, the stream- flow routing Muskingum method was modified for bank storage. The analysis was based on one-dimensional lateral groundwater flow in semi-infinite homogeneous unconfined aquifers, which are in contact with streams through semi-pervious bed sediments.

Impulse response and unit step response functions were derived for the stream-aquifer system, using Laplace transformations. These response functions relate stream outflow, stream-aquifer flow, bank storage, and cumulative reach discharge volume to discrete-time inflow hydrographs through convolution integrals.

The impulse response function decreases with increasing aquifer hydraulic conductivity at earlier times, but increases with the conductivity and persists at later times. The unit step response function decreases with aquifer conductivity uniformly in time. The dependence of stream flow and bank storage on aquifer hydraulic conductivity, streambed leakage, stream width, and aquifer diffusivity was investigated. The analysis was extended to discrete input data, and modification of the methodology to route discrete-time inflow hydrographs of general form was achieved, using discrete-time kernels. The presented analysis establishes the time domain for validity of the analytical solutions in terms of the Muskingum parameters and an aquifer-related parameter.

Francisco J. Rueda and S. Geoffrey Schladow (2003) examined the internal dynamics of Clear Lake, California-a large, multibasin and polymictic lake-were using simulations conducted with a three-dimensional hydrodynamic model. The model was based on an accurate and efficient semi-implicit finite difference algorithm for the hydrodynamic equations, which has been previously subject to extensive verification with analytical test cases.

The high level of agreement-without extensive calibration-between the model results and the observations at several locations in the lake is comparable with previously published 3D modeling results. The model results confirmed the baroclinic-pumping model of circulation proposed for the Oaks Arm of Clear Lake.

The simulations showed that the interaction of stratification, periodic wind forcing, and Coriolis effects drive this circulation. The diurnal readjustment of the circulation from being wind driven to baroclinically driven was examined and shown to vary spatially. This transition in circulation-type has a wavelike nature, with a distinct frontal structure and converging currents at the surface. Asymmetries in the forcing and response, combined with rotational effects, impart a cyclonic residual circulation on the flow.

M. El-Moattassem et al (2003) developed an approach for simulating the flow of water in High Aswan Dam Reservoir (HADR) and for predicting the sediment transport and determine solutions to overcome the problems associated with the sediment deposition.

The Surface Water Modeling System (SMS) were used for the study. During a simulated flood event, RMA2 computes a time-history of water depth and depth-averaged velocity at each node in the mesh. These data are used by SED2D to compute a bed shear stress. SED2D compares the bed shear stress with the measured or estimated physical properties of the bed material to compute the sediment transport rate and cumulative scour or deposition at each node of the mesh in the study area.

Luis F. León et al (2004) evaluated the predictive capability of the 3-D Estuary and Lake Computer Model (ELCOM) using relatively high quality data collected on Great Slave Lake - one of the largest lakes of the world in Canada’s northern climatic system. This assessment is an important step in the ongoing research to develop a coupled lake-atmosphere model - a major consideration in the development and testing of the lake model.

A validation run was performed with 2003 data in the Great Slave Lake. Vertical thermistor chain data was compared against model calculations and mean circulation patterns were presented. Comparison runs were made with meteorological field data and with output from a Regional Climate Model (RCM) as input to the hydrodynamic model to determine the differences in forcing data affecting simulations of surface temperature and circulation.

Richard O. Anyah, Fredrick Semazzi and Lian Xie (2006) Thermodynamic and hydrodynamic characteristics of Lake Victoria are investigated based on idealized simulations using a 3D-lake model. A suite of simulations with an elliptic (oval) geometry and prescribed wind speed (surface wind stress), lake-atmosphere temperature difference and vertical temperature profile were performed.

The time evolutions of lake temperature as well as the currents (circulation characteristics) at different depths and/or points are analyzed in order to understand the lake's response to certain aspects of surface forcing conditions. Similarities and differences between the features simulated in a typical tropical lake (Lake Victoria) and typical mid-latitude lake(s) based on the effects of the Coriolis force are also examined.

The simulations revealed a number of unique features in the temperature. Considered at different points on the lake surface, the temperature of both runs with or without effect of Coriolis force equilibrates after almost the same time (between 30-40 days). However, there is a conspicuous difference in the vertical temperature profiles of the two runs (cases). For example, the MIDLAT run is characterized by a 'dome-shaped' profile in the bottom layers (40m and deeper) after 30 days of model integration, in contrast to the VICTORIA case which is nearly isothermal over the full water column.

Perhaps one of the most significant outcomes of the presented study is that the two-gyre circulation pattern shown in the VICTORIA case after 30 days of model integration is also present in the simulations with observed lake bathymetry.

Conclusion

From the previous studies it was found that it is necessary to develop a two dimensional model to simulate the various flow fields, steady and unsteady of the High Aswan Dam Reservoir. Then use this model in the reservoir management and dam operation. Taking into account the water strategy followed by the Egyptian ministry of Irrigation. Which states that when handling the flood discharges the flood year starts with water level of 175.00 m upstream HAD and considering the presence of Tushka side spillway which begins to discharge water to Tushka depression through Tushka canal when water level reaches (178.00) which was built for the safety of the HAD. In addition, it is important to put different scenarios when dealing with flood or drought years.

2.1 General 7
2.2 Hydrologic Versus Hydraulic Routing Techniques 7
2.3 Hydraulic Routing Technique 9
2.4 Hydrologic Routing Technique 10
2.5 Evaluating The Routing Method 14
2.6 Steady versus Unsteady Flow Models 14
2.7 Previous Studies 15
2.8 Conclusion 26

CHAPTER THREE

The Problem Definition

3.1 River Nile

The Nile has played an important part in the cultural history of the world and in the development of great civilizations. While still in the areas from which its waters flow, the water runs unregulated and uncontrolled putting people at the mercy of floods and slack periods.

This is not a matter which affects one country or two but the whole Nile countries are concerned and involved. If all the countries affected by the Nile waters cooperate to control this great force and use it for the benefit of all their peoples, they can make great advances. (Fahmy and El Shibini, 1979).

3.2 High Aswan Dam (HAD)

The HAD is located 6.500 km upstream the old Aswan dam, about 950 km South of Cairo. It is 3600 meters long along the top and has a maximum height of 111 meters. The width of the base is 980 meters. 43 million cubic meter of materials were used in its construction, the structure is seventeen times larger than the Great Pyramid of Giza.

The operation of the HAD is the responsibility of the High Aswan Dam Authority (HADA), whose chairman reports directly to the minister of Irrigation.

The High Aswan Dam is the most famous dam in the world, and the management of such dam is not a problem which will be finally solved and then left to technocrats to implement. The policy for managing this dam will evolve over time as new water resources problems emerge.

3.3 High Aswan Dam Reservoir (HADR)

The construction of High Aswan Dam (HAD) in Upper Egypt resulted in the formation of a reservoir that trapped almost all of the inflow and hence forms a large reservoir.

The length of HAD reservoir is about 500 km with an average width of about 12 km and a surface area of 6540 km^2 at its maximum storage level. Which is (182.00) m. This reservoir is considered to be the second largest man-made lake in the world, where the storage capacity of the reservoir has a volume of 162 billion m^3 allocated as follows:

(1) 31 billion m^3 for dead storage (which corresponds to a lake level of 147 meters above sea level).
(2) 90 billion m^3 for live storage (which corresponds to a lake level up to 175 meters above sea level).
(3) 41 billion m^3 for flood protection (which corresponds to a lake level of 182 meters above sea level).

3.4 Tushka Canal Project

As Lake Nasser reached its operating range, the ministry of irrigation became concerned about the ability of the reservoir to handle a high flood without causing scouring damages of the main channel, barrages and other structures across the Nile main stream.

Tushka project was proposed by the ministry of irrigation to effectively create a safety valve to remove excess water from the HADR by cutting a canal from the western edge of the reservoir north of Abo Smpel through the Western desert to the Tushka depression, where it would empty harmlessly into barren desert.

The canal was initially designed to be 350 meters wide with a capacity of 365 million m^3 per day and to run for 22 km until it empties into Tushka depression. The first phase of the project called for an unregulated spillway at 178 meters above sea level at the canal entrance. (Several studies were carried out to replace such spillway with a controlled structure).

3.5 Problem Identification

The question of operation of the HAD was neglected for a long time after the dam has been constructed as it seemed simple as the inflows were large enough so the water could always be released to meet downstream requirements and any water remaining was simply used to fill the reservoir behind the dam. In the last decay, many researches related to the management of the dam were carried out within the HADA and in some of the ministry's water research centers.

The purpose of any reservoir is to regulate the fluctuations in a river's discharge in order to obtain a more desirable pattern of flows. However, the operating policies differ between reservoirs for three reasons:

(1) The structure of the dam and the physical characteristics of the reservoir's behavior vary in different locations.
(2) The statistical characteristics of flows vary in different rivers.
(3) The value of different levels of achievements of the objectives to the economic and political systems varies in different locations.

The ministry of irrigation currently decides the daily release from the HAD, and the ministry of electricity determines the distribution of discharges over the 24 hour period in order to effectively integrate the hydroelectric generation into the daily requirements of the national grid.

The release of great amount of water from the HAD may result in some degradation. So one of the objectives of the reservoir management is find a pattern of releases which would result in an acceptable level of degradation. The ministry of irrigation also requires that the daily discharges be limited to 250 million m^3 per day in order to avoid damages to the river channel and barrages.

One of the most important projects which are carried out by the ministry of irrigation is the replacement of the great barrages on the Nile in order to increase the maximum daily release of the HAD to help in the agriculture area extending projects in Egypt.

Currently, the management policy of the reservoir, to handle high floods, is to lower the water level by the end of July, that is the beginning time of the year flood, to at least (175.00) m. In order for the reservoir to have the capacity to store the peak of a high flood, the entire incoming flood and all the subsequent inflows of the water year minus the evaporation and seepage losses must be released over a twelve month period in order to bring the water level back to 175 meters by the following end of July.

If a higher level than 175 meters is used, more water is stored as insurance against a series of low years, the head on the turbines is higher, evaporation losses are greater, but the risk of damages from a high flood is increased.

3.6 Scope of the Thesis

The Scope of the thesis can be summarized into the following points:

1. Development of a bathymetry (3 dimensioned bed level profile) of the High Aswan Dam Reservoir from the upstream about 460 kilometers HAD and ending just upstream the dam. This bathymetry will be used to interpolate the levels of mesh points used during the modeling process.

2. Development of a 2 dimensional hydrodynamic model as the first hydrodynamic model for the reservoir. This model is the first step for sediment transport studies and water quality studies of the lake.

3. Study of the water surface profiles variation with respect to different flow rates, flood wave slopes and initial water level in the reservoir.

4. Study of real flood wave movement through the reservoir.

5. Development of an operating model for the High Aswan Dam releases.

3.1 River Nile..28
3.2 High Aswan Dam (HAD) 28
3.4 Tushka Canal Project..30
3.5 Problem Identification 31
3.6 Scope of the Thesis...33

CHAPTER FOUR

Data Preprocessing

4.1 Data Presentation

4.1.1 Introduction:

The collection of data - before the construction of HAD- was made at several control stations such as Donqola (777 km upstream HAD) and Kajnrity (399 km upstream HAD). After the construction of the HAD, Regular trips took place once a year for the measurement of cross sections, velocities, suspended sediment concentration and water levels at fixed locations along the HAD reservoir.

Obtaining an accurate representation of bed topography is likely the most critical, difficult, and time consuming aspect of the 2D modeling exercise. Simple cross-section surveys are generally inadequate. Combined GPS and depth sounding systems for large rivers and distributed total station surveys for smaller streams have been found to be effective. In either event, you should expect to spend a minimum of one week of field data collection per study site. The field data should be processed and checked through a quality digital terrain model before being used as input for the 2D model.

In addition to topographic data, the model requires hydrologic and hydraulic data such as stage and flow hydrographs, measurement velocities, and rating curves to establish initial and boundary conditions and for model calibration and verification.

The discharge boundary condition at the upstream end of the modeled reach will be represented using the hydrograph recorded at Donqola measuring station about 777 km upstream HAD, water surface elevation at the downstream end was determined using data from upstream High Aswan Dam station.

The data used in this study were gathered from the files of the High Aswan Dam Authority (HADA), the Nile Research Institute (NRI) and the Water Resources Research Institute (WRRI).

4.1.2 The Inflow Data:

There is not only a substantial monthly variation in the annual flow of the Nile, but also substantial variation in the annual totals from one year to another. The pattern of the discharge at HADR is (150 billion m^3 high floods, 42 billion m^3 low flood).

The continuous record of discharge at Donqola station shows that there are two stages for the Nile River the rising stage and the falling stage:

(1) The rising stage starts by the end of July and reaches its peak around the middle of September and is distinguished by the sharp increase in the discharge, and an increase in the river levels.

(2) The falling stage where the discharge starts to have lower values during the months October to June.

The measured discharges during the period (1964-2005) at Donqola were collected and maximum recorded inflow could be shown in Figure 4 - 1 starting from the first of May **(Source, HADA and NRI)**. It was noticed that, In general most of the measured discharges range between 900 and 2000 m^3/sec, the maximum discharge recorded was 13,577.80 m^3/sec in September 1998 and the minimum discharge was about 492.0 m^3/sec recorded in February 1991.

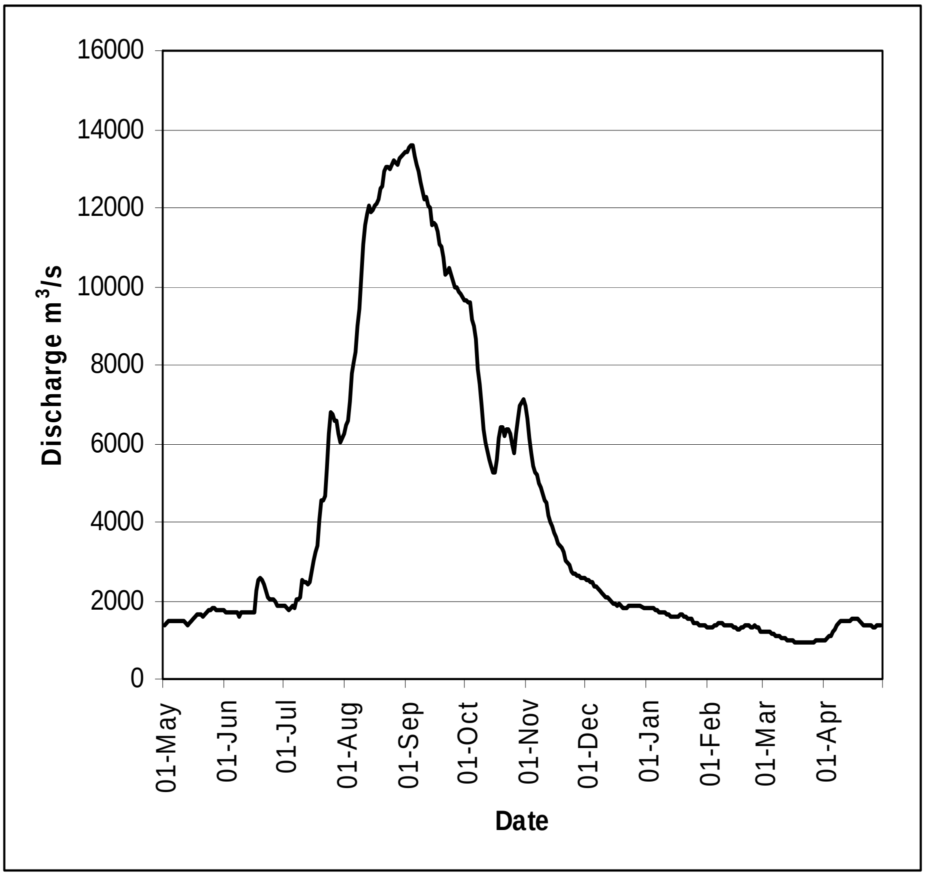

Figure 4 - 1: Maximum Recorded hydrograph at Donqola station (1965-2005)

4.1.3 The Outflow Data:

The water is discharged downstream the dam through 6 tunnels located at (147.00) m above sea level. Therefore, this level was considered as the critical water level for the turbines. On the western side, there is a spillway to release the water that exceeds the maximum storage capacity the crest level of this weir is (178.00) m and it is provided with a redial controlled gate over its crest level.

Another, uncontrolled, spillway was constructed at the end of Khor Tushka (on the western side of Lake Nasser at about 256 km upstream the dam). This spillway is connected to Tushka depression by a canal through which the excess flood can be turned to the depression.

A certain part of the outflow, before the construction of HAD, was used for land irrigation and for domestic purposes and the rest was discharged to the Mediterranean Sea. Agriculture in Egypt depended almost entirely on the natural supply of the river. A short distance downstream Cairo, the river bifurcates into two branches: Damietta and Rosetta. These branches are the main source of water feeding the irrigation canals in Lower Egypt. They were also used before the construction of High Aswan Dam to convey the excess flood water to the Mediterranean Sea. After the construction of HAD a full control of the Nile water is now present.

4.1.4 The Cross Sections Geometric Data:

The field survey of the cross sections was carried out after the construction of HAD and upstream the dam.

Table 4 - 1 shows the names and location of some of the sections that were used in this study **(Source, HADA and NRI)**.

Table 4 - 1: Distances of cross sections upstream HAD

Cross section name	Distance in km Upstream HAD
Malek El Nasser	448.00
Ateere	415.50
Semna	403.50
Morshed	378.00
Gomai	372.00
Amka	364.00
El gandal El thany	357.00
Agreen	331.00
Sarra	325.00
Adindan	307.00
Abo Smpel	282.00
Tushka	256.00
Masmas	237.00
Ebrem	228.00
Krosko	182.00
Elmadik	130.00
Khor Manam	28.00

Other sections at km 487, 466, 431, 394, 368, 352, 347 and 337 were also used in the analysis.

The water depth was measured using echo-sound devices at irregular distances at each section and it was noticed that:

- The cross sections between km 325 and km 368 upstream HAD are very wide where the width varies between (2500 – 8500) m.
- Between km 368 and km 405 the width ranges between (1000 – 2500) m.

- And between km 405 and km 490 the sections are relatively narrower and the width ranges between (500 – 1000) m.

The available data were not consistent as they were collected from several sources as mentioned in the previous chapter. So they had to be put together in the same format and projection.

The cross sections profiles show the bed level measured from the left bank so in order to obtain the longitude and latitude of these points some calculations should be done first. A sample of those sections is shown in Figure 4 - 2.

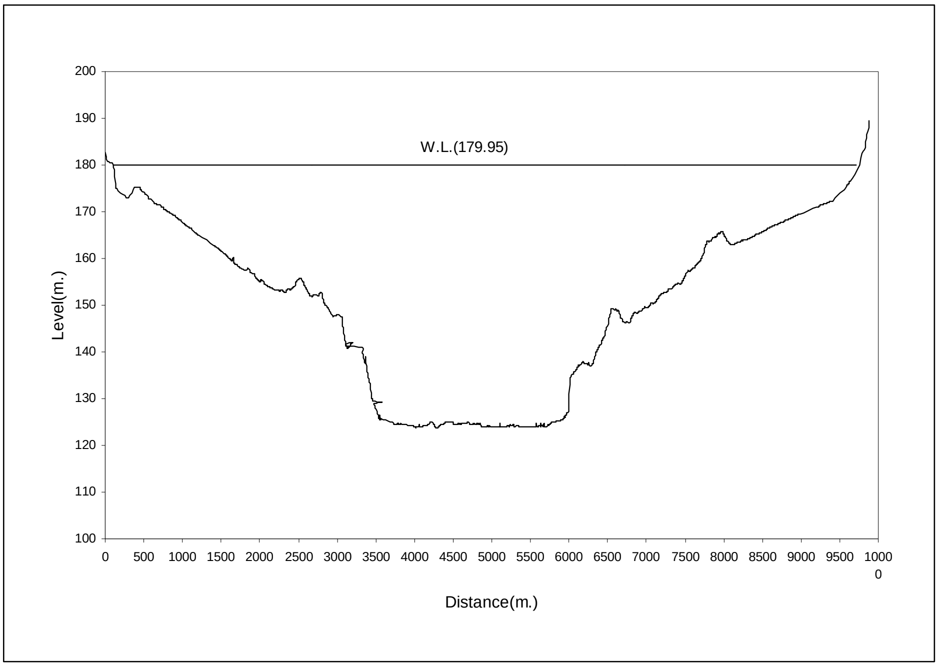

Figure 4 - 2: Cross section at Adenean in the year 2000
(307 km US HAD)

By knowing the coordinates of the sections head points the azimuth angle of each section (α) could be calculated.

$$\tan(\alpha) = \frac{E_L - E_R}{N_L - N_R} \quad 4.1$$

where:

E_L	left bank longitude
E_R	right bank longitude
N_L	left bank latitude
N_R	right bank latitude

The coordinates of all section points were calculated using the azimuth angle and length between each point and the left bank head point.

$$E_X = E_L + L \sin(\alpha) \quad 4.2$$

$$N_X = N_L + L \cos(\alpha) \quad 4.3$$

Where:

E_X	longitude of point X on the section bed level
N_X	latitude of point X on the section bed level
L	distance measured from the left bank to point X

The coordinates of the sections head points were used, along with the sections bed levels, in a Microsoft Excel spreadsheet to transform all the points into (E,N) coordinates with a known elevation Z using equations 4.1,4.2 and 4.3.

4.1.5 <u>Longitudinal Section</u>

The longitudinal section, (Figure 4 - 3), profile was used to enhance increase the accuracy of the interpolation of the bed level in the area where no cross section data are available **(Source, HADA).**

.

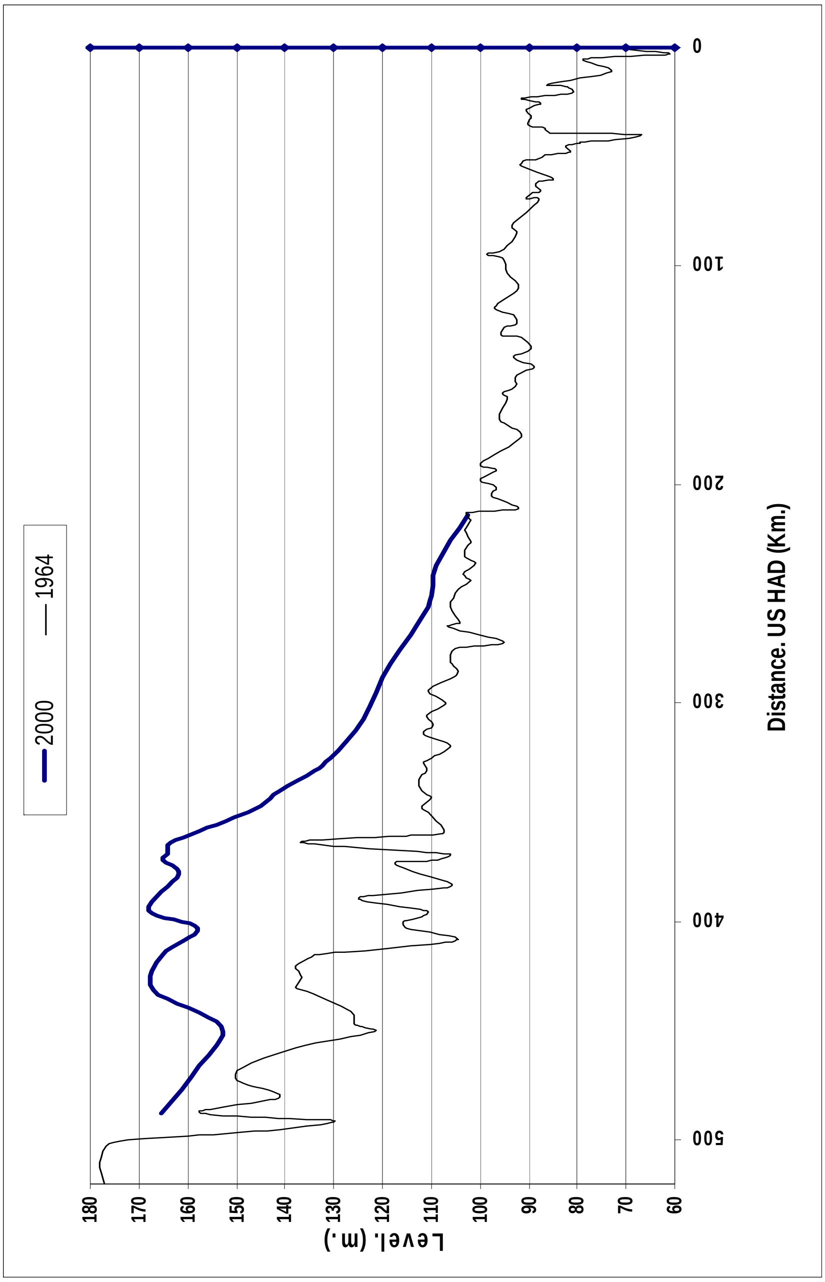

Figure 4 - 3: Longitudinal Section of the HAD Reservoir (1964-2000)

4.1.6 Satellite Images

World geode (WGS84) geographical maps for the area of Nasser Lake were digitized from a satellite images to obtain a digital format of the lake **(Source, WRRI)**. These images were taken in;

- November 1987 (Figure 4 - 4) where the water level in the reservoir was (158.44),
- November 1998 (Figure 4 - 5) where the water level was (181.21),
- And January 2001 (Figure 4 - 6) where the water level was (180.15).

These Images shows only the perimeter of the reservoir, it is when the water in the reservoir touches the land. As the water bodies appear in the satellite images as black bodies.

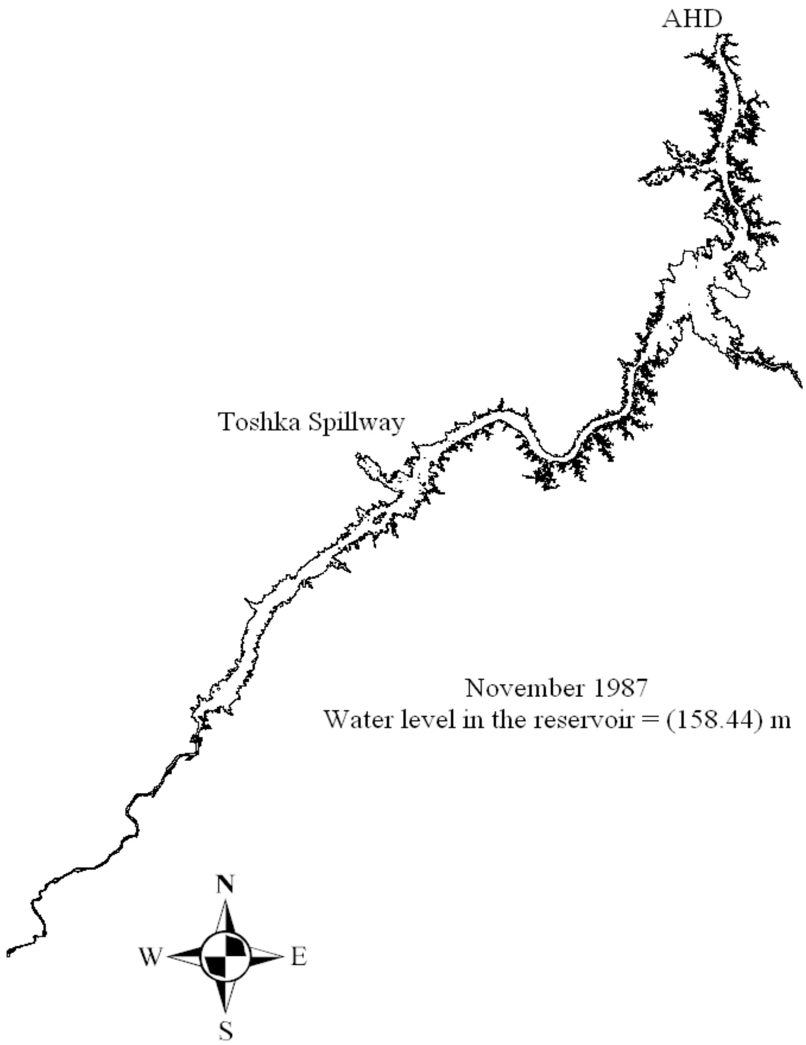

Figure 4 - 4: Boundaries of the lake obtained using a LANDSAT image acquired November 1987

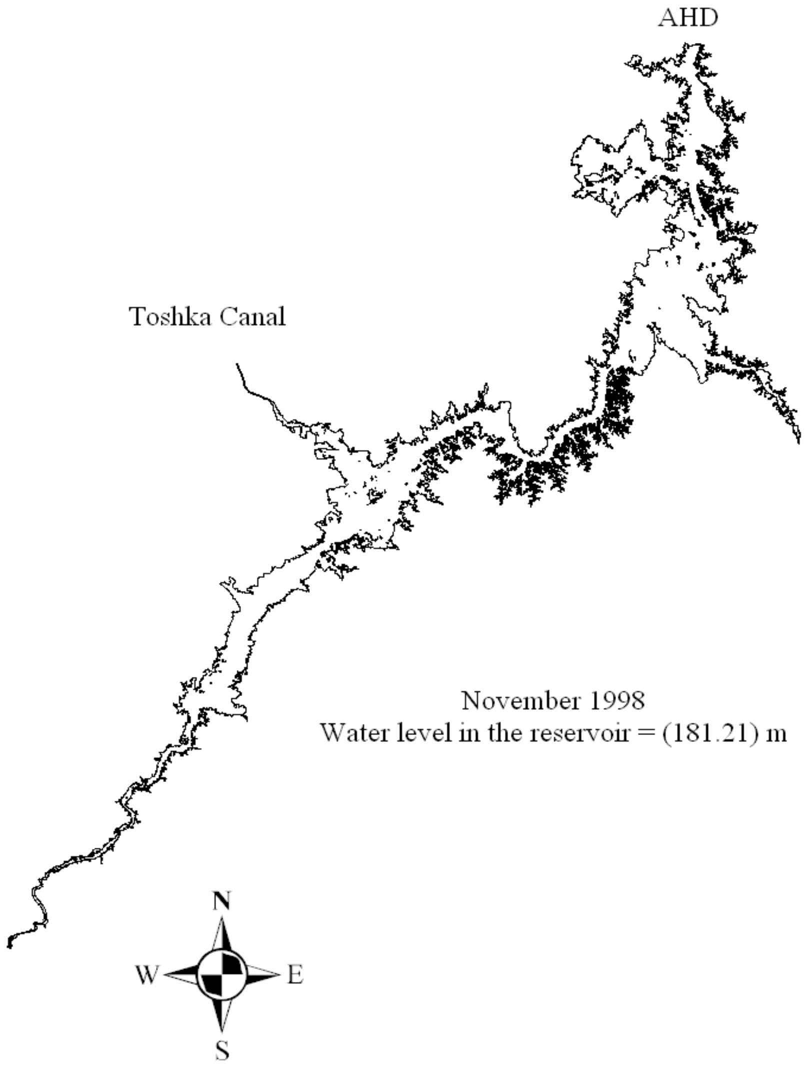

Figure 4 - 5: Boundaries of the lake obtained using a LANDSAT image acquired November 1998

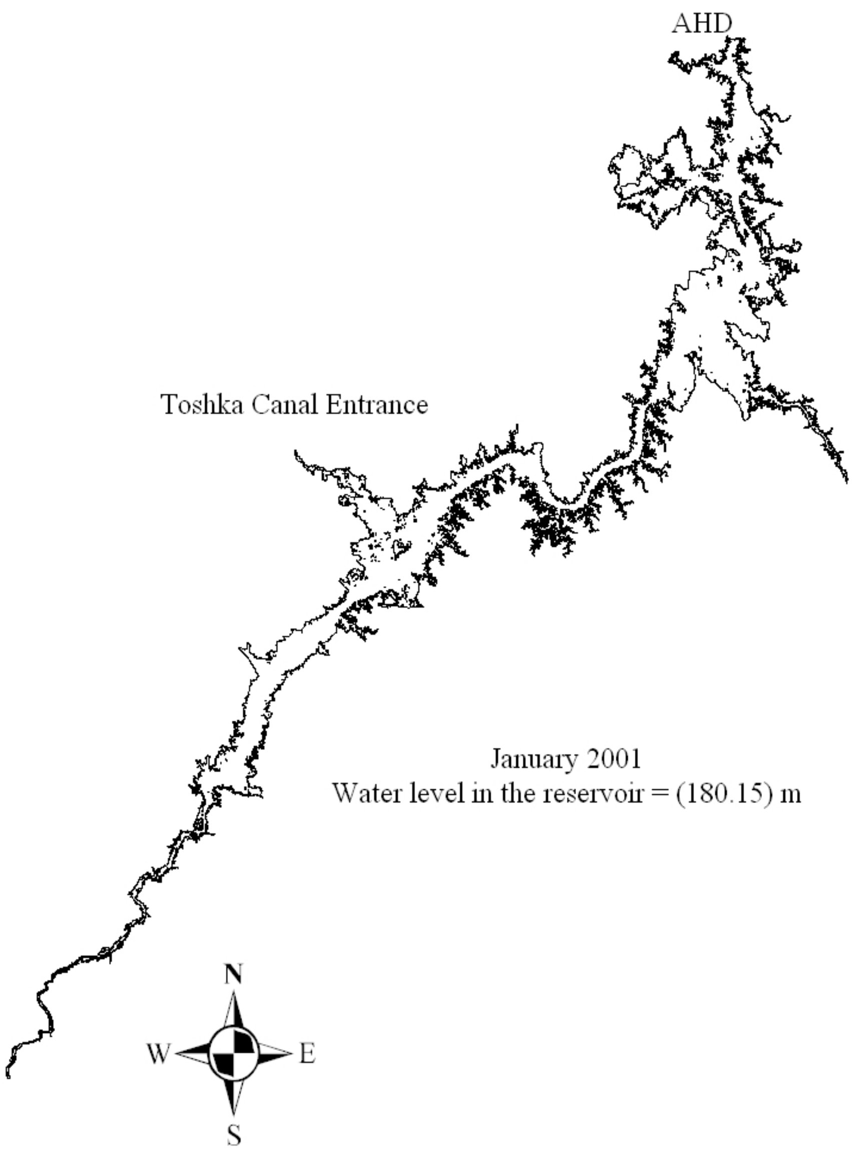

Figure 4 - 6: Boundaries of the lake obtained using a LANDSAT image acquired January 2001

It can be noticed the change in shape of the reservoir from level to another. In 1987 image when the water level is lower than the two other images we can just see the Tushka spillway but in 1998 when the water level is higher the spillway becomes submerged under the water and the water can be seen in Tushka canal down stream the spillway.

4.2 Geo-Referencing the Data

The Geographic Information System (GIS) application has been recognised as a powerful mean to integrate and analyse data from various sources. It was used to project both the High Aswan Dam Reservoir maps and the section points into the Universal Transverse Mercator grid Zone 36 (UTM36).

The cross section points along with the 3 contours of the reservoir perimeter at levels (158.44), (181.21) and (180.15), obtained from the satellite images, were used to form a group of scatter points (x,y,z) with the (UTM36) as a defined projected coordinate system. These points will be used later in the generation of the bathymetry mesh.

4.3 Mesh Generation

The Surface water Modeling System (SMS) interface was used during this stage as it is a graphical aided interface of creating the mesh, and it also can support the file format used by many of hydrodynamic modeling programs.

After creating a polygon which encloses the study area a mesh was generated using the adaptive tessellation technique which is a mesh generation technique used to fill the interior of a polygon.

The technique uses the existing spacing on the polygons to determine the element sizes on the interior. Any interior arcs and refine points are forced into the new mesh. If the input polygon has varying node densities along its perimeter, SMS attempts to create a smooth element size transition between these areas of differing densities. By altering the size bias, the user can indicate whether SMS should favor the creation of large or small elements. Decreasing the bias will result in smaller elements; increasing the bias will result in larger elements (in this study the bias was set to a value of 1.0 as the mesh size was already very large and a small elements will cause a long time in calculations).

In either case, the elements in the interior of the mesh will honor the arc edges and the element sizes specified at nodes. The bias simply controls the element sizes in the transition region.

The mesh size was determined based on the capability of the available hardware as increasing the mesh density will cause the solution time to increase.

The spacing between nodes was chosen to be at an average of 250 meters. This resulted in a mesh of about (92,590) nodes and (43,540) elements. Better results could be obtained with advanced hardware capability.

To maintain good mesh properties the following points should be reviewed after the mesh generation;

- •A well constructed mesh must first have good element properties.
- •The overall bathymetric contours should be smooth.
- •Wetting and drying studies work best when the element edges to lie on bathymetric contours.
- •Ideally, any boundary break angle should not exceed 10 degrees.

•Neighboring elements should not differ in size by more then 50%.
•Use adequate resolution to model the features of the prototype plan.
•Maintain a length to width ratio of less than one to ten.
•Restrict element shapes to avoid highly distorted triangles or rectangles.
•Create elements with corner angles greater than 10 degrees.
•Bathymetric elevations should lie almost in a plane.
•Maintain longitudinal element edge depth changes of less than 20%.

It should be noticed that most of the produced meshes will violate one or all of these properties, but not excessively. They still are goals to strive for. So, after the mesh was generated it was then checked out against those errors as the SMS interface provided an automatic representation of the mesh quality. Yet, it has to be corrected manually.

The errors (around 200 errors) were found in the mesh geometry and were corrected manually by modifying the mesh elements to solve the error according to its type. Some times an element was to be split into more than one, another time it was to be merged or even deleted.

4.4 Bathymetry Interpolation

After the mesh generation, the scatter points obtained from the GIS were used to form a triangles irregular net (TIN) which will be used in the interpolation of the bed levels.

One of the most commonly used techniques for interpolation of scatter points is inverse distance weighted (IDW) interpolation. Inverse distance weighted methods are based on the assumption that the interpolating surface should be influenced most by the nearby points and less by the more distant points.

The interpolating surface is a weighted average of the scatter points and the weight assigned to each scatter point diminishes as the distance from the interpolation point to the scatter point increases.

The simplest form of inverse distance weighted interpolation is sometimes called "Shepard's method" (Shepard 1968). The equation used is as follows:

$$F(x,y) = \sum_{i=1}^{n} w_i f_i \qquad 4.4$$

where n is the number of scatter points in the set, f_i are the prescribed function values at the scatter points (e.g. the data set values), and w_i are the weight functions assigned to each scatter point. The weight function varies from a value of unity at the scatter point to a value approaching zero as the distance from the scatter point increases, and they are normalized so that the weights sum to unity.

The effect of the weight function is that the surface interpolates each scatter point and is influenced most strongly between scatter points by the points closest to the point being interpolated. The following weight function is used in SMS:

$$W_i = \frac{\left[\dfrac{R - h_i}{Rh_i}\right]^2}{\sum_{j=1}^{n} Rh_j} \qquad 4.5$$

Where h_i is the distance from the interpolation point to scatter point i, R is the distance from the interpolation point to the most distant scatter point, and n is the total number of scatter points. This equation has been found to give superior results to the classical equation (Franke & Nielson, 1980).

The weight function is a function of Euclidean distance and is radically symmetric about each scatter point. As a result, the interpolating surface is somewhat symmetric about each point and tends toward the mean value of the scatter points between the scatter points. Shepard's method has been used extensively because of its simplicity.

Using of a subset of the scatter points in the computation of the nodal function coefficients and in the computation of the interpolation weights drops distant points from consideration since they are unlikely to have a large influence on the nodal function or on the interpolation weights. In addition, using a subset can speed up the computations since fewer points are involved.

Two options are available for defining which points are included in the subset. In one case, only the nearest N points are used. In the other case, only the nearest N points in each quadrant are used.

If a subset of the scatter point set is being used for interpolation, a scheme must be used to find the nearest N points. The scatter points are triangulated to form a temporary TIN before the interpolation process begins. To compute the nearest N points, the triangle containing the interpolation point is found and the triangle topology is then used to sweep out from the interpolation point in a systematic fashion until the N nearest points is found. This scheme is fast for large scatter point sets.

Natural neighbor interpolation is also supported in SMS. It was first introduced by Sibson (1981). A more detailed description of natural neighbor interpolation in multiple dimensions can be found in Owen (1992). The basic equation used in natural neighbor interpolation is identical to the one used in IDW interpolation.

The difference between IDW interpolation and natural neighbor interpolation is the method used to compute the weights and the method used to select the subset of scatter points used for interpolation.

Natural neighbor interpolation is based on the Thiessen polygon network of the scatter point set. The Thiessen polygon network can be constructed from the Delauney triangulation of a scatter point set. A Delauney triangulation is a TIN that has been constructed so that the Delauney criterion has been satisfied.

There is one Thiessen polygon in the network for each scatter point. The polygon encloses the area that is closer to the enclosed scatter point than any other scatter point. The polygons in the interior of the scatter point set are closed polygons and the polygons on the convex hull of the set are open polygons.

Each Thiessen polygon is constructed using the circumcircles of the triangles resulting from a Delauney triangulation of the scatter points.

The vertices of the Thiessen polygons correspond to the centroids of the circum circles of the triangles.

To obtain good results from the interpolation of the bed, the Natural neighbor technique was used for the interpolation and the inverse distance technique for the extrapolation.

4.5 Enhancing the Resulted Bathymetry

After interpreting the first resulted bathymetric contours it was found that this bathymetry should be enhanced because some parts were not covered by the hydrographical survey may cause errors both in the Sudanese part where the stream is narrower and in the Egyptian part.

Therefore a Digital Elevation Model (DEM) of the surrounding area (Figure 4-7) was used to generate a Triangulated Irregular Network (TIN) using the GIS software. The DEM is a raster image divided into a group of cells and each cell has a value the represent the elevation of its location.

The resulted TIN (Figure 4 - 8) was then used in the interpolation process to enhance the bathymetric contours. The Enhanced Bathymetry is shown in Figure 4-9.

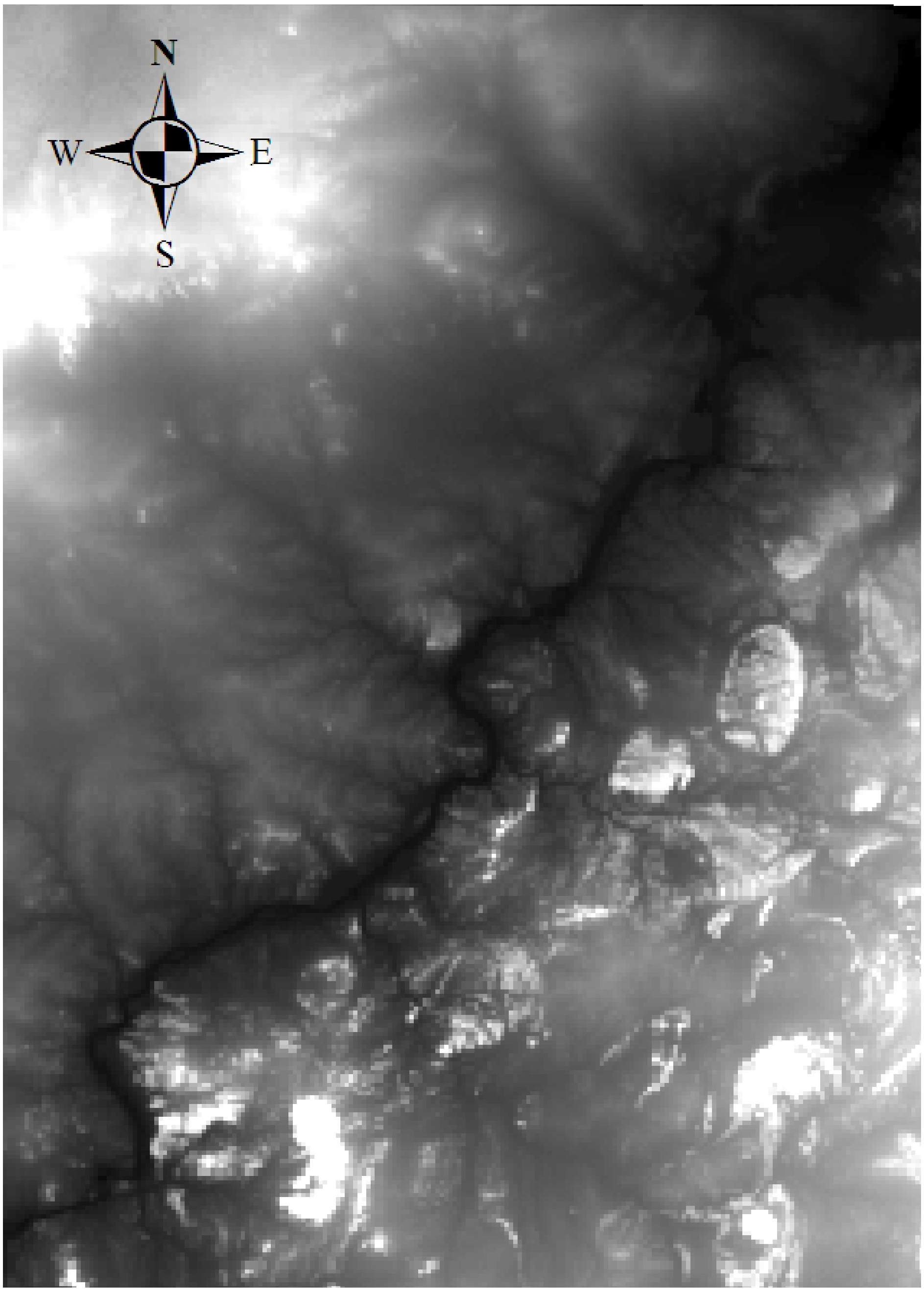

Figure 4 - 7: A 90 Meters Cell Size SRTM Digital Elevation Model

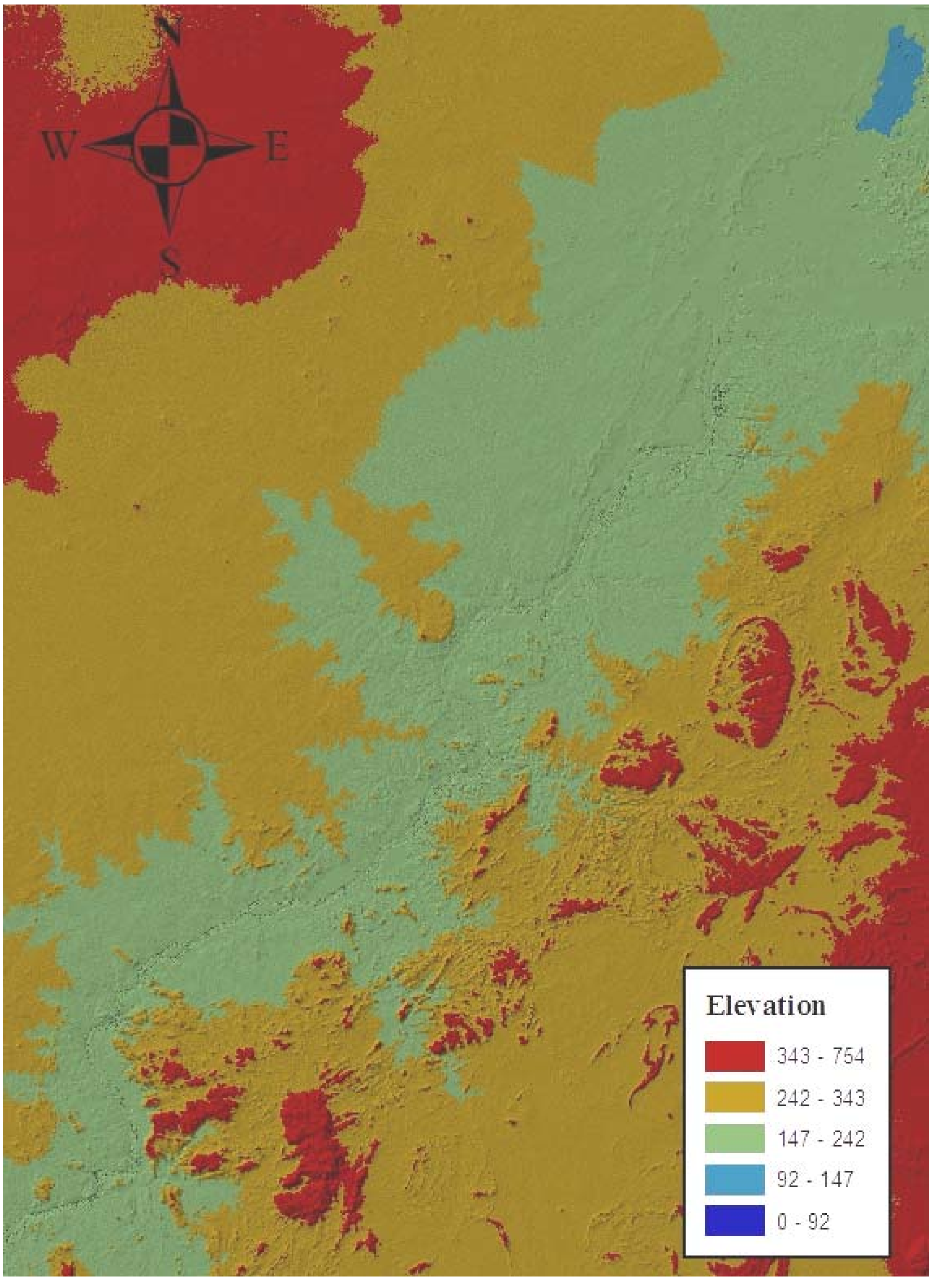

Figure 4 - 8: The Resulted TIN

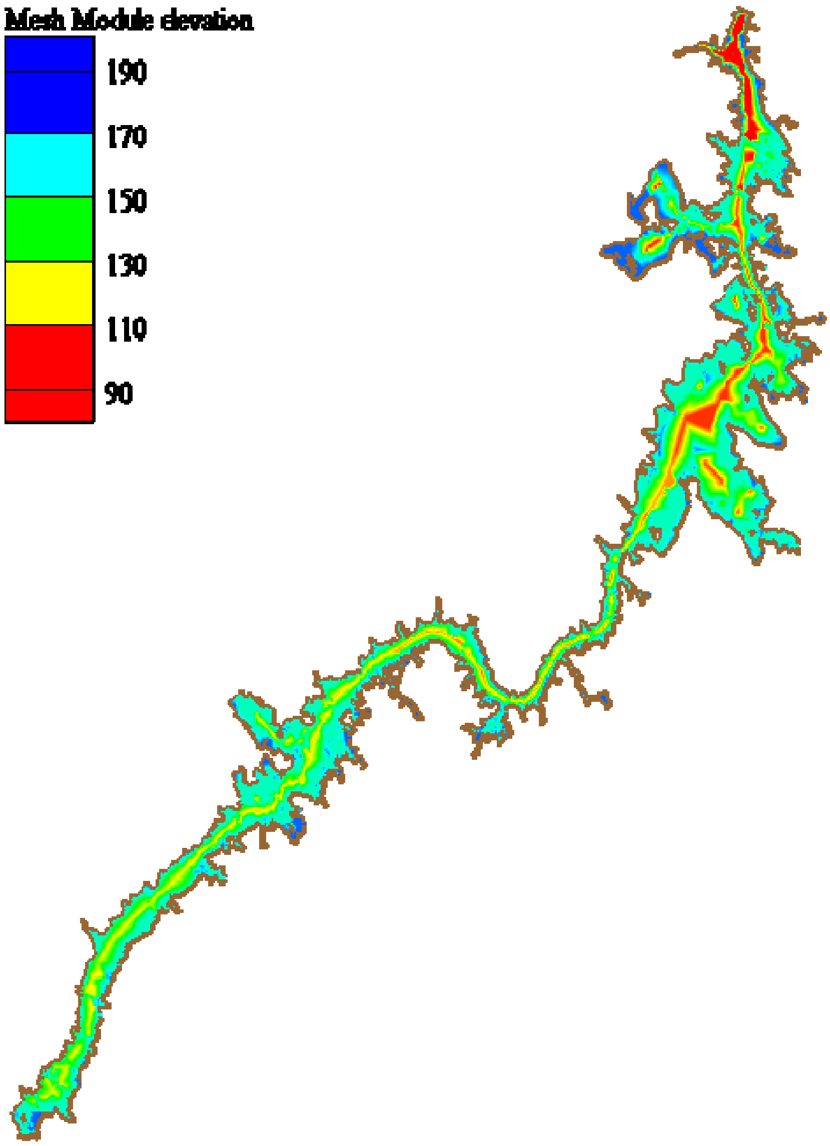

Figure 4 - 9: The Resulted Bathymetry

4.1 Data Presentation 35
4.1.1 Introduction: 35
4.1.2 The Inflow Data: 36
4.1.3 The Outflow Data: 38
4.1.4 The Cross Sections Geometric Data: 38
4.1.5 Longitudinal Section 41
4.1.6 Satellite Images 43
4.2 Geo-Referencing the Data 47
4.3 Mesh Generation 47
4.4 Bathymetry Interpolation 50
4.5 Enhancing the Resulted Bathymetry 53

Figure 4 - 1: Maximum Recorded hydrograph at Donqola station 37
Figure 4 - 2: Cross section at Adenean in the year 2000 40
Figure 4 - 3: Longitudinal Section of the HAD Reservoir (1964-2000) .42
Figure 4 - 4: Boundaries of the lake obtained using a LANDSAT image acquired November 1987 44
Figure 4 - 5: Boundaries of the lake obtained using a LANDSAT image acquired November 1998 45
Figure 4 - 6: Boundaries of the lake obtained using a LANDSAT image acquired January 2001 46
Figure 4 - 7: A 90 Meters Cell Size SRTM Digital Elevation Model 54
Figure 4 - 8: The Resulted TIN 55
Figure 4 - 9: The Resulted Bathymetry 56

Table 4 - 1: Distances of cross sections upstream HAD 39

CHAPTER FIVE

The Hydrodynamic Model

5.1 Model selection

Reliable assessment and resolution of river hydraulics issues depend on the engineer's ability to understand and describe, in both written and mathematical forms, the physical processes that govern a river system. Three categories of methods for predicting river hydraulic conditions were identified by Rouse (1959).

The first and oldest uses engineering experience acquired from previous practice by an individual. The second utilizes laboratory scale models (physical models) to replicate river hydraulic situations at a specific site or for general types of structures. The third category is application of analytical (mathematical) procedures and numerical modeling.

Recent use of physical and numerical modeling in combination, guided by engineering experience, is termed "hybrid modeling" and has been very successful.

To decide if a multidimensional study is needed, or a one-dimensional approach is sufficient, a number of questions must be answered. Is there a specific interest in the variation of some quantity in more than one of the possible directions? If only one principal direction can be identified, there is a good possibility that a one-dimensional study will suffice.

One dimensional analysis implies that the variation of relevant quantities in directions perpendicular to the main axis is either assumed or neglected, not computed. Common assumptions are the hydrostatic pressure distribution, well-mixed fluid properties in the vertical, uniform velocity distribution in a cross section, zero velocity components transverse to the main axis, and so on.

It is possible that actual transverse variations will differ so greatly from the assumed variation that streamwise values, determined from a one-dimensional study, will be in significant error. If flow velocities in floodplains are much less than that in the main channel, actual depths everywhere will be greater than those computed on the basis of uniform velocity distribution in the entire cross section. It is possible that the transverse variations will be of greater importance than the streamwise values. This is of particular importance when maximum values of water surface elevation or current velocity are sought.

The choice of appropriate analytical methods to use during a river hydraulics study is predicated on many factors including the objectives, the level of detail being called for, the regime of flow expected, the availability of necessary data, and the availability of time and resources to properly address all essential issues.

A Survey was made on the 2D Hydrodynamic models. The selection and the comparison between models were based on the following points:

1- The availability.
2- The ability to be applied to calculate water levels and flow distribution in rivers and reservoirs.
3- Easiness to learn and use.
4- The Availability of the documentation of the model.
5- Very good visual representation of the inputs and outputs which helps in the interpretation of the results.

The **DYNHYD** a Hydrodynamic Program is a simple link-node hydrodynamic program capable of simulating variable tidal cycles, wind, and unsteady flows. It produces an output file that supplies flows, volumes, velocities, and depths (time averaged) for the WASP modeling system.

The hydrodynamics model DYNHYD is an enhancement of the Potomac Estuary hydrodynamic model which was a component of the Dynamic Estuary Model. DYNHYD solves the one-dimensional equations of continuity and momentum for a branching or channel-junction (link-node), computational network. Driven by variable and downstream heads, simulations typically proceed at one- to five-minute intervals. The resulting unsteady hydrodynamics are averaged over larger time intervals and stored for later use by the water quality program.

The hydrodynamic model solves one-dimensional equations describing the propagation of a long wave through a shallow water system while conserving both momentum (energy) and volume (mass). The equation of motion, based on the conservation of momentum, predicts water velocities and flows.

The equation of continuity, based on the conservation of volume, predicts water heights (heads) and volumes. This approach assumes that flow is predominantly one-dimensional, Coriolis and other accelerations normal to the direction of flow are negligible, channels can be adequately represented by a constant top width with a variable hydraulic depth, i.e., rectangular, the wave length is significantly greater than the depth, and bottom slopes are moderate.

Although no strict criteria are available for the latter two assumptions, most natural flow conditions in large rivers and estuaries would be acceptable. Dam-break situations could not be simulated with DYNHYD nor could small mountain streams.

TUFLOW (Two-dimensional Unsteady FLOW) is a two-dimensional (2D) and one dimensional (1D) flood and tide simulation software. It simulates the hydrodynamics of water bodies using 2D and 1D free-surface flow equations. TUFLOW is specifically orientated towards establishing flow patterns in coastal waters, estuaries, rivers and floodplains where the flow patterns are essentially 2D in nature and cannot or would be awkward to represent using a 1D network model.

A powerful feature of TUFLOW is its incorporation of the 1D hydrodynamic network software, ESTRY (i.e. 2D and 1D domains are linked to form one integrated model). TUFLOW continues to develop and evolve to meet the challenges of hydrodynamic modeling. Its strengths include rapid wetting and drying, powerful 1D and 2D linking options, multiple 2D domains, 1D and 2D representation of hydraulic structures, automatic flow regime switching over levees and embankments, 1D and 2D supercritical flow, effective data handling and quality control outputs. It is suited to modeling flooding in major rivers through to complex overland and piped urban flows, and estuarine and coastal hydraulics. TUFLOW uses GIS to manage, manipulate and present data, and third-party software such as SMS to view and animate results.

The TUFLOW software continues to develop and evolve to meet the challenges of hydraulic modeling. Its strengths are rapid wetting and drying, powerful 1D and 2D linking options, multiple 2D domains, 1D and 2D modeling of hydraulic structures, treatment of levees and embankments, effective data handling, 1D and 2D supercritical flow and quality control outputs. TUFLOW is applicable for modeling flooding in major rivers through to complex overland and piped urban flows, and estuarine and coastal hydraulics. TUFLOW uses GIS as its primary method of data management, manipulation and presentation. The use of GIS allows easy inclusion of model topography changes to assess floodplain impacts.

AquaDyn is a powerful and easy-to-use hydrodynamic simulation model essential for water resources engineering studies, risk assessment, and impact studies. AquaDyn allows the complete description and analysis of hydrodynamic conditions (e.g., flow rates and water levels) of open channels such as rivers, lakes, or estuaries. Engineers, specialists, and decision-makers can use the specialized modules of the simulation package to predict impacts on water flow conditions. For instance, AquaDyn provides a reliable way to forecast the consequences of different activities such as dredging or building dikes, bridges piers, and embankments. AquaDyn can be used to model steady and unsteady flows in supercritical as well as subcritical conditions and therefore permits the user to take into account and study the effects of weirs, contractions, and tidal waves.

The **RMA2** model under the SMS interface was selected in the case study and will be described in the following sections.

5.1.1 Origin of the Program

The original RMA2 was developed by Norton, King and Orlob (1973), of Water Resources Engineers, for the Walla Walla District, Corps of Engineers, and delivered in 1973. Further development, particularly of the marsh porosity option, was carried out by King and Roig at the University of California, Davis. Subsequent enhancements have been made by King and Norton, of Resource Management Associates (RMA), and by the Waterways Experiment Station (WES) Coastal and Hydraulics Laboratory-USA, culminating in the current version of the code.

5.1.2 Model Description

RMA2 is a two-dimensional depth averaged finite element hydrodynamic numerical model. It computes water surface elevations and horizontal velocity components for subcritical, free-surface two-dimensional flow fields.

RMA2 computes a finite element solution of the Reynolds form of the Navier-Stokes equations for turbulent flows. Friction is calculated with the Manning's or Chezy equation, and eddy viscosity coefficients are used to define turbulence characteristics. Both steady and unsteady (dynamic) problems can be analyzed.

The program has been applied to calculate water levels and flow distribution around islands, flow at bridges having one or more relief openings, in contracting and expanding reaches, into and out of off-channel hydropower plants, at river junctions, and into and out of pumping plant channels, circulation and transport in water bodies with wetlands, and general water levels and flow patterns in rivers, reservoirs, and estuaries.

5.1.3 Limitations of RMA2

RMA2 operates under the hydrostatic assumption; meaning ***accelerations in the vertical direction are negligible***. It is two-dimensional in the horizontal plane. It is not intended to be used for near field problems where vortices, vibrations, or vertical accelerations are of primary interest. Vertically stratified flow effects are beyond the capabilities of RMA2 and velocity vectors generally point in the same direction over the entire depth of the water column at any instant of time. It expects a vertically homogeneous fluid with a free surface.

RMA2 is a free-surface calculation model for subcritical flow problems. More complex flows where vertical variations of variables are important should be evaluated using a three-dimensional model, such as RMA10.

The generalized computer program RMA2 solves the ***depth-integrated equations of fluid mass*** and ***momentum conservation*** in two horizontal directions by the finite element method using the Galerkin method of weighted residuals.

$$h\frac{\partial u}{\partial t}+hu\frac{\partial u}{\partial x}+hv\frac{\partial u}{\partial y}-\frac{h}{\rho}\left[E_{xx}\frac{\partial^2 u}{\partial x^2}+E_{xy}\frac{\partial^2 u}{\partial y^2}\right]+gh\left[\frac{\partial a}{\partial x}+\frac{\partial h}{\partial x}\right]+\frac{gun^2}{h^{1/3}}$$
$$+\sqrt{(u^2+v^2)}=0$$

$$h\frac{\partial v}{\partial t}+hu\frac{\partial v}{\partial x}+hv\frac{\partial v}{\partial y}-\frac{h}{\rho}\left[E_{yx}\frac{\partial^2 v}{\partial x^2}+E_{yy}\frac{\partial^2 v}{\partial y^2}\right]+gh\left[\frac{\partial a}{\partial y}+\frac{\partial h}{\partial y}\right]+\frac{gvn^2}{h^{1/3}}$$
$$+\sqrt{(u^2+v^2)}=0$$

Where;

h	=	Depth
u, v	=	Velocities in the Cartesian directions
x, y, t	=	Cartesian coordinates and time
ρ	=	Density of fluid
E	=	Eddy viscosity coefficient,

for *xx* = normal direction on x axis surface
for *yy* = normal direction on y axis surface
for *xy* and *yx* = shear direction on each surface

g	=	Acceleration due to gravity
a	=	Elevation of bottom

n = Manning's roughness n-value

The shape functions are quadratic for velocity and linear for depth. Integration in space is performed by Gaussian integration. Derivatives in time are replaced by a nonlinear finite difference approximation.

The solution is fully implicit and the set of simultaneous equations is solved by Newton-Raphson non linear iteration. The computer code executes the solution by means of a front-type solver, which assembles a portion of the matrix and solves it before assembling the next portion of the matrix.

5.2 The Modeling Process

The following flow chart illustrates the RMA2 modeling process.

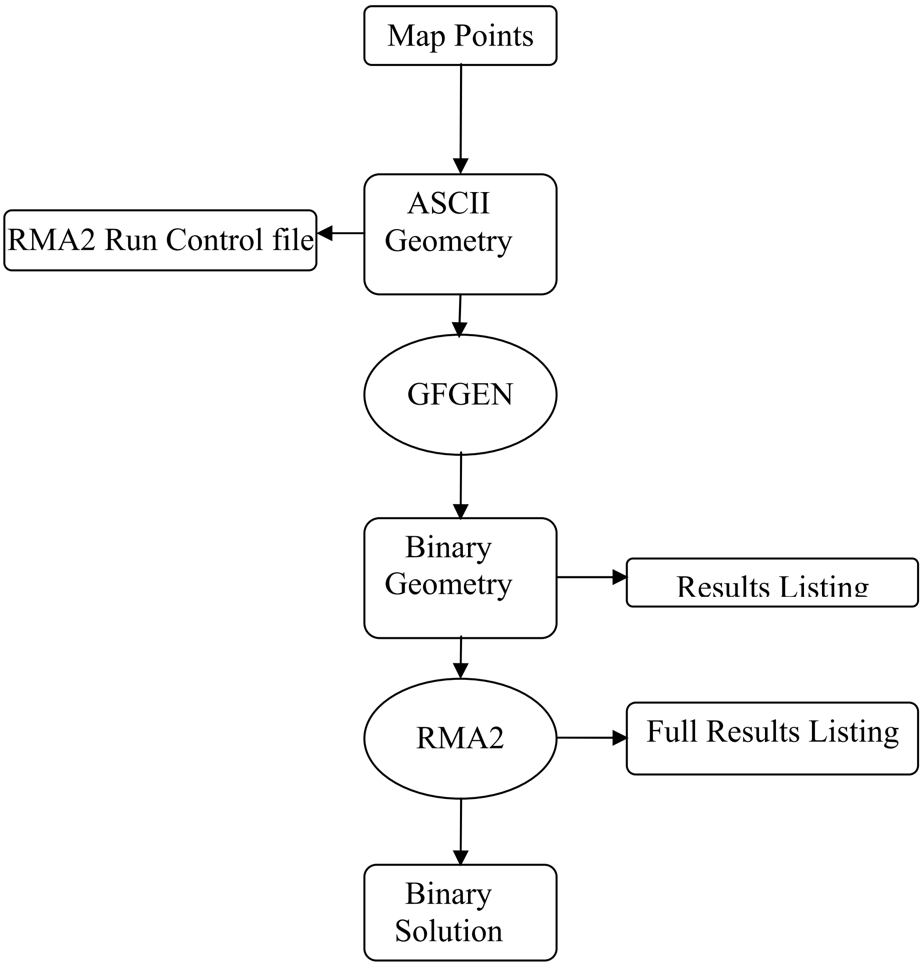

Figure 5 - 1: Flow chart for the RMA2 modeling process

The Geometry File GENeration program (GFGEN) creates geometry and finite element mesh files for input to the modeling system programs.

When RMA2 is used for a dynamic (unsteady state) simulation, it first must solve a steady state problem. The results from this steady state case are used to start the dynamic simulation. Unless specified otherwise, RMA2 will continue directly from the steady state solution and begin the dynamic simulation.

5.3 Guidelines for Obtaining a Good Solution

All aspects of the geometry and the numerical model simulation must run in harmony. In addition to the geometry, the RMA2 run control (boundary condition) file must contain the proper information if the simulation is to be successful. A graphical user interface will help in building a run control file, but it is a must to examine that file and double check the run control selections.

The primary requirement for a successful numerical model is preparing a good mesh; developing it with the following recommended guidelines in mind. These guidelines are briefly described below.

Maintain Good Element Properties

- Maintain a length to width ratio of less than one to ten.
- Restrict element shapes to undistorted triangles or rectangles.
- Create elements with corner angles greater than 10 degrees.
- All bathymetric elevations should lie in one plane.
- Maintain longitudinal element edge depth changes of less than 20%.

Maintain Good Mesh Properties

- A well constructed mesh must first have good element properties.
- The overall bathymetric contours should be smooth.

● Wetting and drying studies need the element edges to lie on bathymetric contours.
● Ideally, any boundary break angle should not exceed 10 degrees.
● Neighboring elements should not differ in size by more then 50%.
● Use adequate resolution to model the features of the prototype plan field.

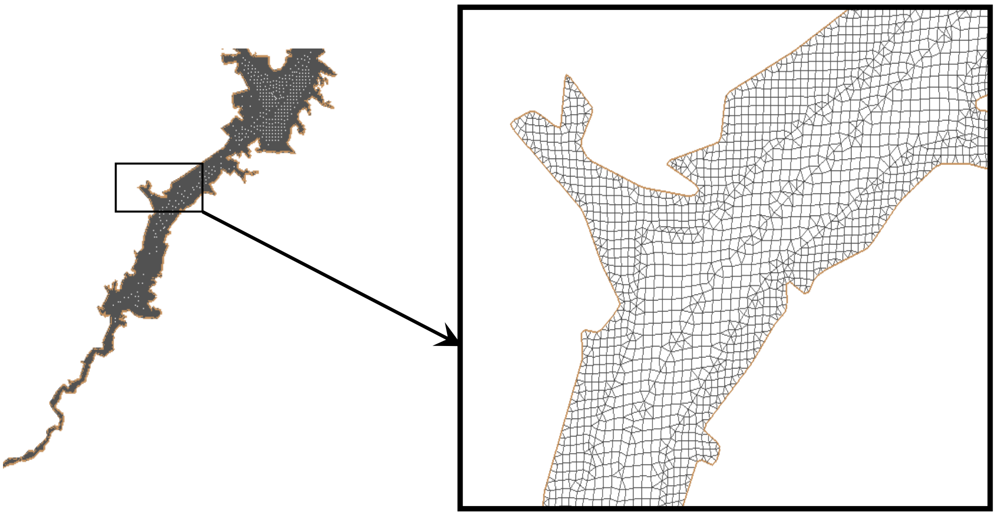

Figure 5 - 2: Mesh generation

As described in the previous chapter the mesh was chosen to be dense near the boundary were the banks have a sophisticated shape and around the islands.

In the middle part of the channel as can be seen in figure 5-2 the mesh become less dense to reduce the number of nodes and elements and hence reduce the calculation time.

5.4 Specifying Boundary Conditions

Boundary conditions usually take the form of a specified total discharge at inflow sections and fixed water surface elevations or rating curves at outflow sections. Since 2D models make no implicit assumptions about flow direction or magnitude, discharge divisions in splitting channels and the discharge given inflow and outflow elevations can be calculated directly.

Locating flow boundaries some distance from areas of interest is important to minimize the effect of boundary condition uncertainties. Initial conditions are important, even for steady flow, since they are usually used as the initial guess in the iterative solution procedure. A good guess will significantly reduce the total run time and may make the difference between a stable run and an unstable one.

Boundary conditions are required to drive RMA2 throughout a simulation. They are constraints which are applied along the flow boundaries of the solution domain, and required to eliminate the constants of integration that arise when numerically integrating the "Boundary conditions" is a mathematical term which specifies the loading for a particular solution to a set of partial differential equations. In more practical terms, boundary conditions for an unsteady flow model are the combination of flow and stage time series, which when applied to the exterior of the model either duplicates an observed event or generates a hypothetical event such as a design flood. For an observed event, the accuracy of the boundary conditions affects the quality of the reproduction.

In a similar but less detectable manner the reasonableness of the boundary conditions for a hypothetical event (because accuracy can seldom be established) limits the quality of the conclusions. Furthermore, the way that the boundary conditions are applied can control the overall accuracy and consistency of the model.

External boundary nodes along the downstream end of the network are typically assigned a water-level (head) boundary condition. Also, boundary nodes along the upstream end of the network are typically assigned an exact flow or discharge boundary condition.

Each side wall of the network is automatically assigned a parallel flow boundary condition (i.e., slip flow) which allows the program to calculate the velocity adjacent and parallel to the side wall as well as the flow depth there.

Boundary conditions may be specified on a nodal basis, along the edge of an element, or across a continuity check line. No special equations are required for boundary nodes. The use of a boundary condition specification removes either the depth, or one or both of the velocity components from the computations, and the program expects those values to be entered as boundary input data.

All boundary conditions hold from one time step to the next unless they are specifically modified. RMA2 does not permit a new boundary condition location to be specified in mid-run, nor does it allow a change in the type of boundary condition at a previously specified boundary location.

5.4.1 Upstream Boundary Condition

The recorded hydrographs at Donqola station 777 km upstream the HAD forms the upstream boundary of the model and the maximum recorded flood hydrograph (Figure 5-20) was used during the simulations.

5.4.2 Downstream Boundary Condition

The water level at the HAD was used as the downstream boundary condition during the simulations. During the calibration and verification processes the recorded water levels at the HAD station were used, Figure 5 - 3 shows the daily recorded levels from 1998 till 2002.

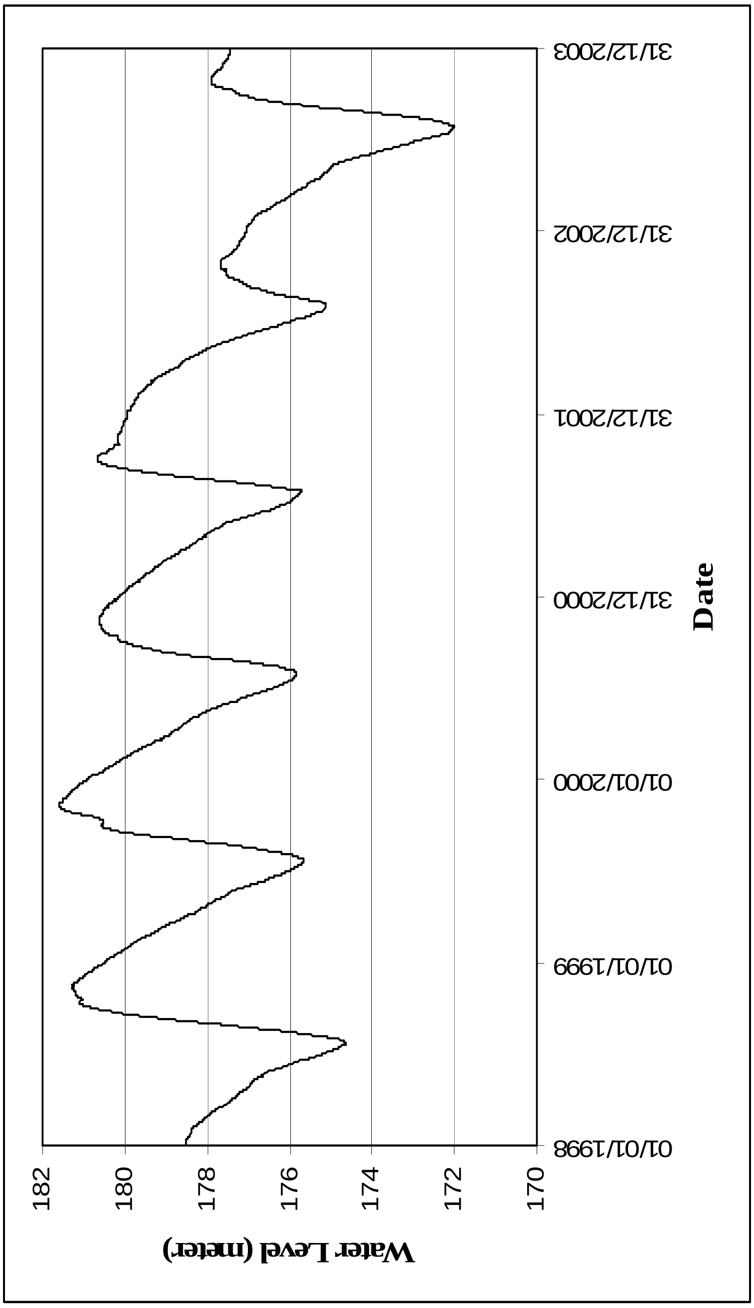

Figure 5 - 3: Water Level Recorded Upstream HAD (1996-2003)

5.5 Model Checking For Continuity

Continuity refers to checking the water mass flux. The objective when simulating is to retain the correct amount of fluid flow from one point to the next, within a tolerance of about plus or minus 3%. Continuity check lines provide a means to determine if your steady state simulation is locally maintaining mass conservation at a given location.

Continuity check lines are typically used to estimate the flow rates at cross-sections perpendicular to the flow path and serve as an error indicator. The RMA2 model globally maintains mass conservation in a weighted residual manner. Locally, continuity check lines can be used to check for apparent mass changes in a different way, by direct integration. Large discrepancies between the results of these two methods indicate probable oscillations and a need to improve model resolution and/or to correct large boundary break angles.

Although continuity checks are optional, they are a valuable tool for diagnosing a converged steady state solution. For steady state, the continuity check lines should represent ***total inflow equals total outflow***. However, if the continuity checks indicate a mass conservation discrepancy of ±3%, you may want to address the resolution in the geometry.

5.6 Parameters Estimation

This pie chart (Figure 5 - 4) illustrates the approximate relative importance to the simulation of the different aspects of an RMA2 simulation study. As it can be seen, the structure of the geometry and overall study design are the most significant, followed by the boundary condition assignments. The "other" category includes field data issues, amount of time devoted to the effort, approach chosen to analyze data. Study design includes model choice and boundary placement.

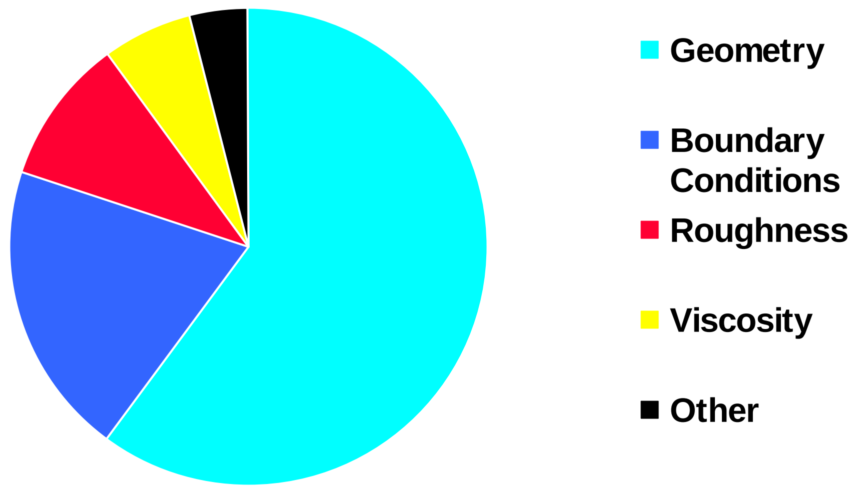

Figure 5 - 4: Relative Importance to Calibration

It may be fair enough to say that a model is only as good as its input data, but it is true. As input data, 2D hydrodynamic models require channel bed topography, roughness and transverse eddy viscosity distributions, boundary conditions, and initial flow conditions. In addition, some kind of discrete mesh or grid must be designed to capture the flow variations.

5.6.1 Bed Friction Computation

The bed friction energy transfer computation, or bottom roughness, is one of the primary verification tools for RMA2. Changing the bed friction provides some control over the fluid velocity magnitude and direction.

Bed friction is typically calculated with Manning's equation if the input roughness value is < 3.0; otherwise a Chezy equation is used. By far, the popular choice is Manning's roughness coefficient (n) value, and these roughness values may be assigned globally throughout the mesh by material type, or on the elemental level.

RMA2 provides the means to input only the bottom roughness, not the side wall roughness. Because there is no wall roughness in RMA2, it was exaggerated on the elements forming the edge of the waterway in order to approximate the wall roughness.

The roughness is a function of many variables such as type of bed and bank materials, river geometry and irregularities. The accurate prediction of the roughness in such case is a rather complicated task. The values of Chezy and Manning's roughness coefficient can be estimated using references such as Chow (1959), Barnes (1967) and Krony (1992).

In this study, the Manning's coefficient was taken as a constant value over the river part and another constant value on the reservoir part to make the simulation more easily those values were to be changed during the simulation as when the water level rises the water covers more area and the perimeter gets more irregular and this affects the roughness values.

5.6.2 Specifying Turbulence

Turbulent exchanges are sensitive to changes in the direction of the velocity vector. Conversely, small values of the turbulent exchange coefficients allow the velocity vectors too much freedom to change magnitude and direction in the iterative solution.

The result is a numerically unstable problem for which the program will diverge rather than converge to a solution. One recourse is to continue increasing the Eddy viscosity (E), until a stable solution is achieved.

There are many ways to control the turbulent exchange coefficient; (E) one of them is the direct assignment method

The first and direct way to assign the turbulent exchange coefficient, E, is to assign a particular value for each individual material type. As a guideline for selecting reasonable values for the turbulent exchange coefficients for a given material type the following algorithm were followed:

1. A representative length of the elements within the material type was determined (250 meter in average).
2. The dominant stream wise velocity for the given geometry estimated ranged was in average of 0.05 m/s according to the flow rate of about 2000 m^3/s.
3. Then the Peclet number (P) equation was been solved for E (Equation 5.1)

$$P = \frac{\rho v dl}{E} \qquad 5.1$$

Where;

ρ = fluid density
v = average elemental velocity
dl = length of element in stream wise direction
E = Eddy Viscosity

Transverse eddy viscosity distributions are important for stability in some finite difference and finite element models and are often assigned unrealistically large values. They can be also be used as calibration factors for measured flow distributions. Stable shock capturing and high resolution numerical schemes are not sensitive to these values. In cases where accurate determination of eddy viscosity is required, a coupled turbulence model should be considered.

The Eddy viscosity used during the modeling process was found to be equal to 1500 kg/m.s. and that gave an average value of Peclet No. equal to 5.

5.7 Model Calibration

Some numerical modelers refer to calibration and verification as a two-step process. Using this terminology, adjustments are made to model coefficients and inputs so as to optimize agreement between model and observed prototype data during the calibration step.

Then the model is run to attempt reproduction of a different set of prototype data without further model adjustment. If the second run is satisfactory then the model may be considered as verified.

This procedure sounds imminently reasonable, but experience suggests that it is a naïve approach and it faces some problems such as:

1. All field data include errors, and sometimes dramatic errors. Thus, they are not an absolute standard.
2. Field measurements include a variety of effects that may not be reproduced in the model (for example, groundwater flow into the model).
3. Conditions often change between field surveys, implying that coefficients should also change (for example, differences in bed forms at different flows may dictate a change in bed roughness coefficients).

4. Most natural waterways cannot be adequately characterized by two field data sets. Five or ten may be needed, but available resources usually limit the field data.
5. If the model reproduces one field data set adequately, but not the second, it should be decided whether to:

   - Proceed with modeling, conceding an incomplete verification.
   - Continue adjusting/revising to obtain a balanced quality of reproduction.
   - Conduct a re-analysis/re-collection of field data.

The objective of the calibration process is to match the output of the model with observed data. This process is performed by adjusting one or more parameters, such as Manning's n, until a satisfactory match of model results with known data is achieved.

When a set of known conditions has been approximately matched by the model, one can apply the model to unknown conditions with more confidence that the model output is reasonably representative of the physical processes associated with that event. However, to be confident, the observed data for calibration should be obtained from an event that is near the scale of the events to be modeled.

When a model is calibrated, the parameters which control the model's performance, primarily Manning's n and reach storage, are determined. The key to a successful calibration is to identify the true values of the parameters which control the system and not to use values that compensate for shortcomings in the geometry and/or the boundary conditions.

The High Aswan Dam Reservoir can be divided into two parts or segments during the modeling process according to stream width, these parts are the river with a width ranging from 950 meters to about 2000 meters and the reservoir with a width more than 2000 meters and up to about 15000 meters.

The river parts starts from the reservoir inlet in the Sudanese side of the lake and ends 360 kilometers upstream HAD where the reservoir part starts. During the calibration process each part was assumed to have a different roughness value as the flow behaves differently when passing through each segment, Figure 5 - 5 shows those two parts.

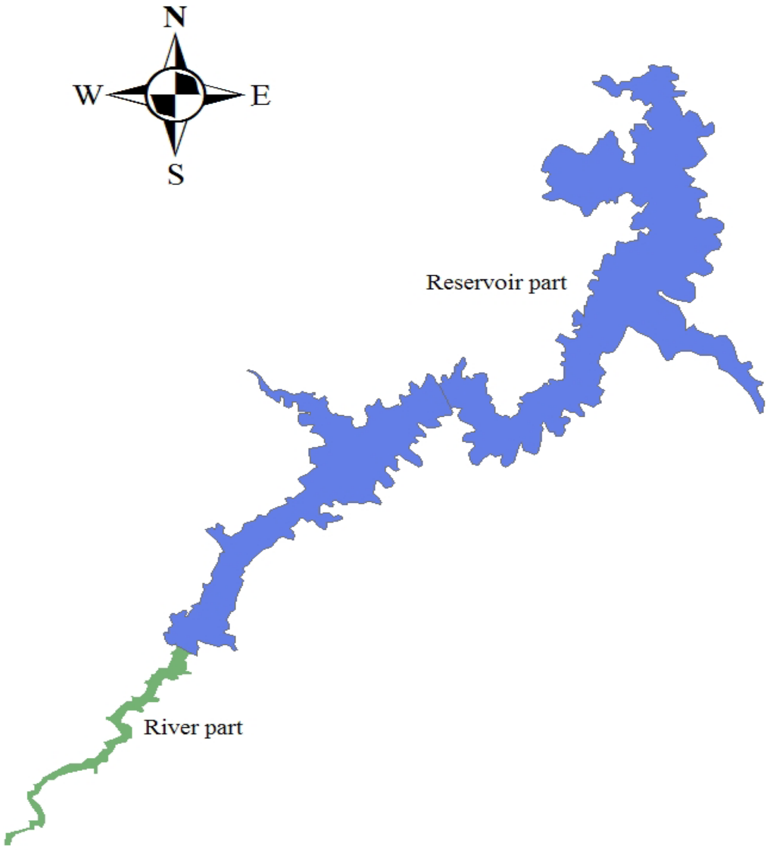

Figure 5 - 5: River part and Reservoir part

In the beginning with low flow rate the calibrated roughness was found to be equal to 0.04 through out the whole reservoir but it increased to about 0.09 in the river part at flow rate of about 7500 m3/sec and remained unchanged in the reservoir part. It then increased again to about 0.13 at flow rate of about 10,000 m3/sec then again to 0.15 at the peak discharge.

Figure 5 - 6 shows the values of roughness used in the calibration process, those values were calculated when the downstream boundary (water level upstream HAD) was assumed 175 m.

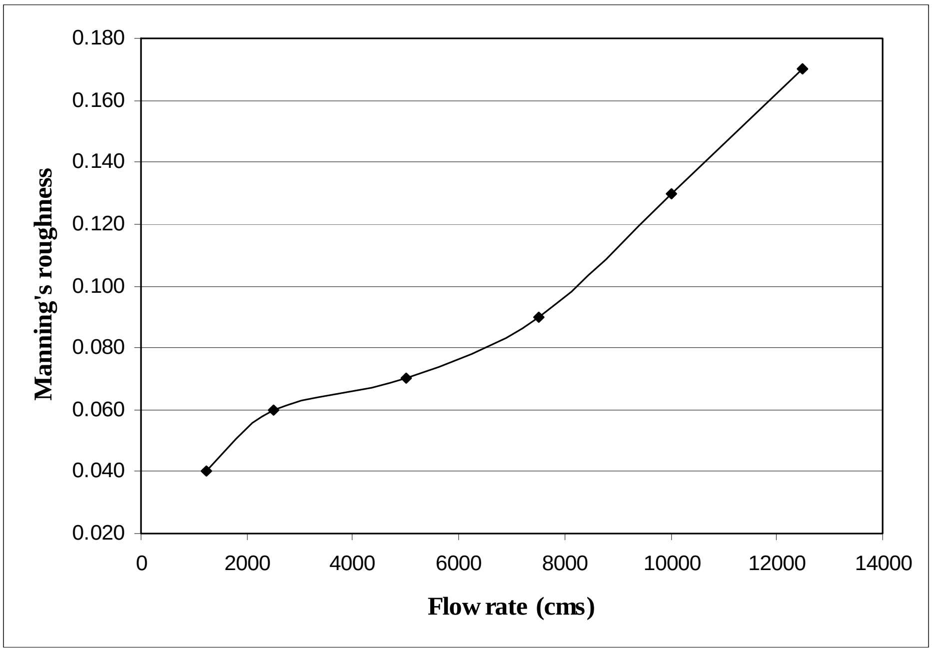

Figure 5 - 6: Values of calibration roughness coefficient

The calibration process where carried out using the recorded water levels at various sections upstream High Aswan Dam during the mission of the year 2000 of the High Aswan Dam Authority in the Sudanese part of the reservoir. The water levels recorded at those sections are presented in Figure 5 - 7 along with the date they were recorded on.

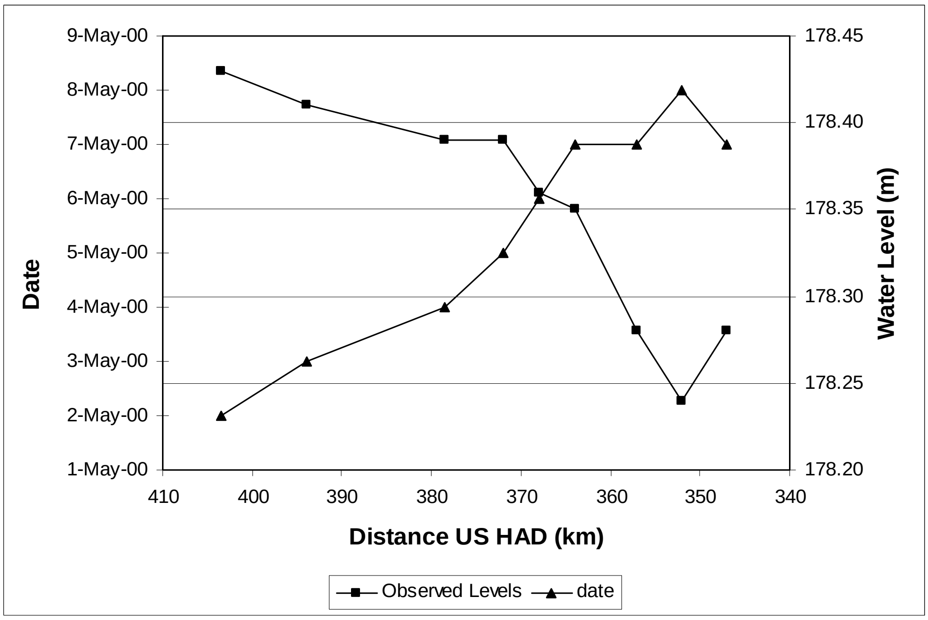

Figure 5 - 7: The measured water levels used in calibration process

Figure 5 - 8 shows the recorded water levels upstream HAD during the simulation period which begins by the 20th of April 2000 and ends at the 15th of May 2000, and Figure 5 - 9 shows the corresponding recorded discharges at Donqola station during the same period.

The travel time was calculated assuming that the flood wave celerity is as 1.5 times as the flow velocity that has an average of 0.25 m/sec in the Sudanese sector of the lake as observed by the HADA. The travel time was calculated from the upstream Station (Donqola) to the entrance of the study reach and was found to be 9 days. So the boundary interval was chosen more than 9 days later than the first recorded level.

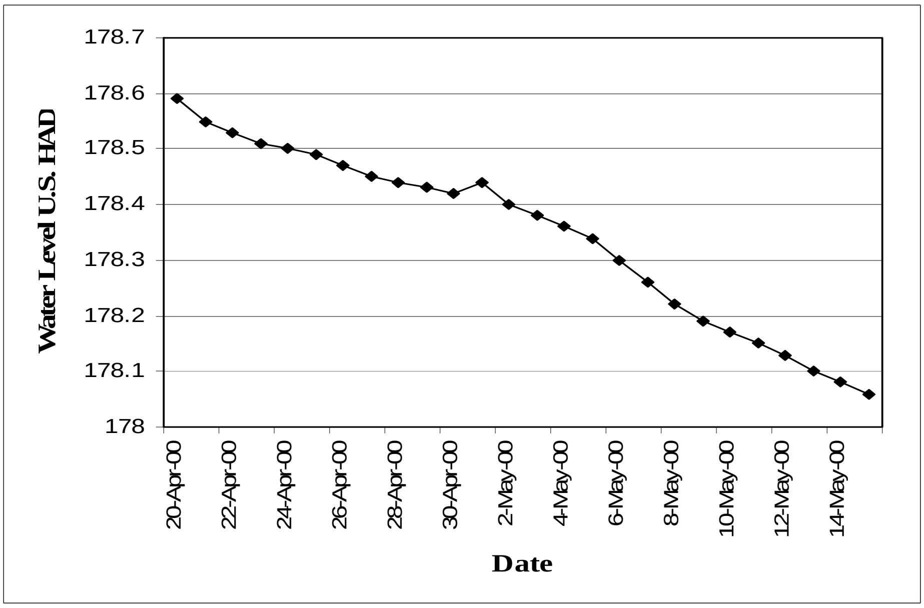

Figure 5 - 8: Water level U.S. HAD

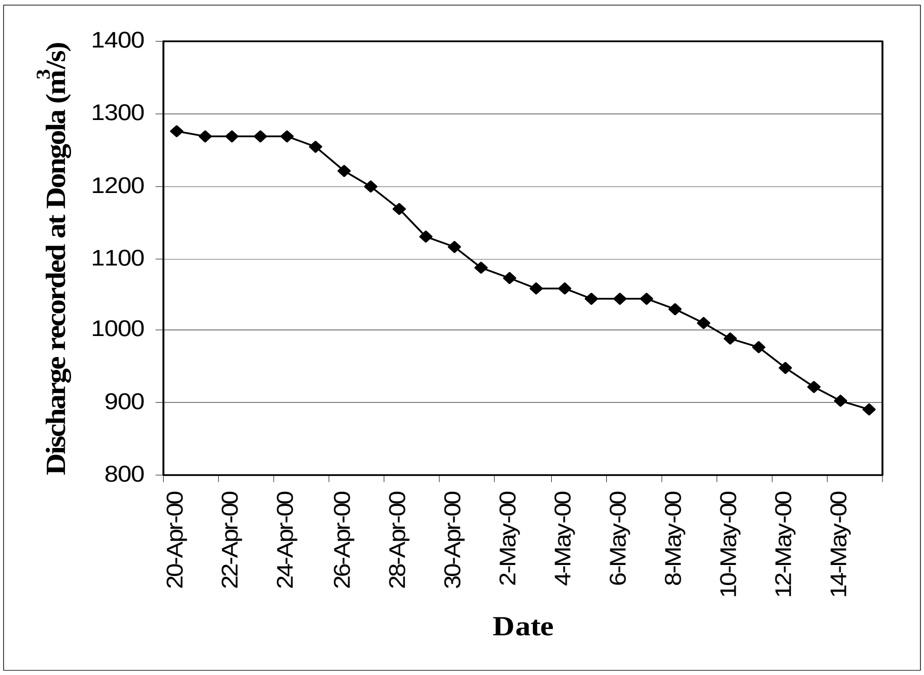

Figure 5 - 9: Discharge recorded at Donqola station

The measured levels during the simulation interval were compared with the levels calculated from the model and the results were plotted as shown in Figure 5 - 10. The results show an absolute error value ranging from 0.8 cm to 8.01 cm, which is a good result.

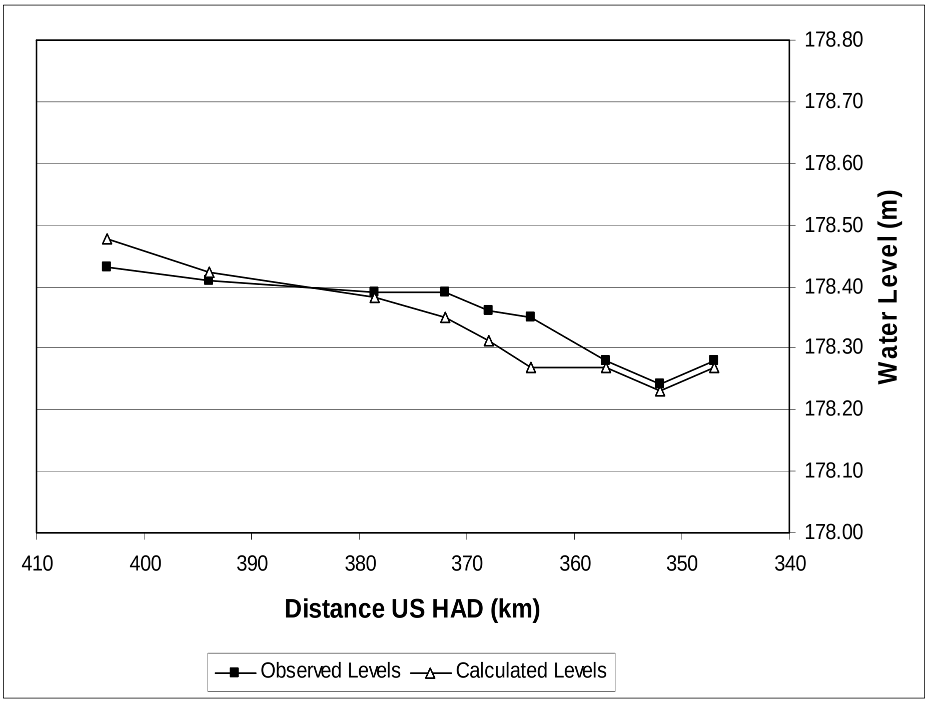

Figure 5 - 10: Observed Vs calculated water levels

The model results can be shown in the following figures, which present the resulted water levels and velocity magnitudes at the end of the simulated interval.

The difference between the observed and calculated water levels at km 365 is due to uncertainty of geometric data at this zone during data filtering process.

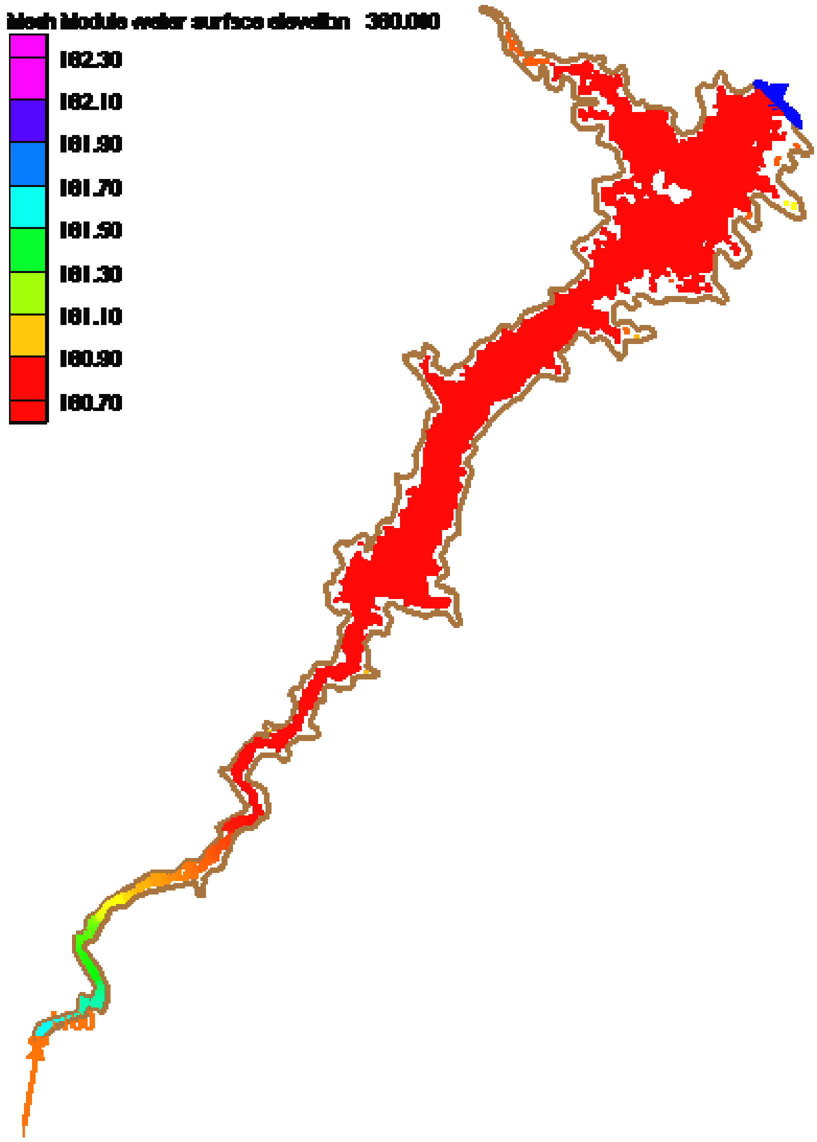

Figure 5 - 11: Simulated Water Surface Elevations on 9 May 2000

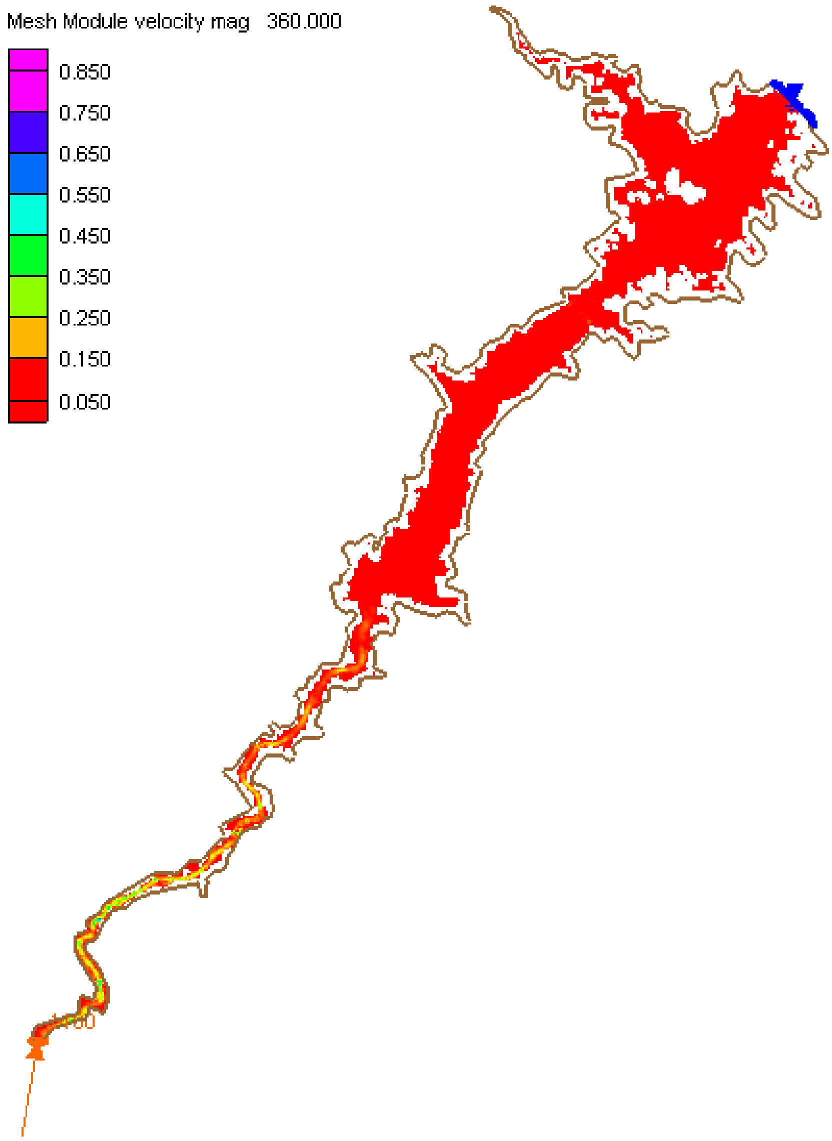

Figure 5 - 12: Simulated Velocity Magnitudes on 9 May 2000

5.8 Model Parameters Verification

Verification is a multi-step process of model adjustments and comparisons, leavened with careful consideration of both the model and the data. It is not a simple two-step (calibration – verification) procedure. The purpose of numerical modeling, as stated by W. A. Thomas, a retired research hydraulic engineer from WES, is to "*gain insight, not answers*".

The verification process where carried out using the recorded water levels at various sections upstream High Aswan Dam during the mission of the year 2000 of the High Aswan Dam Authority in the Egyptian part of the reservoir. The water levels recorded at those sections are presented in Figure 5-13 along with the date they were recorded on.

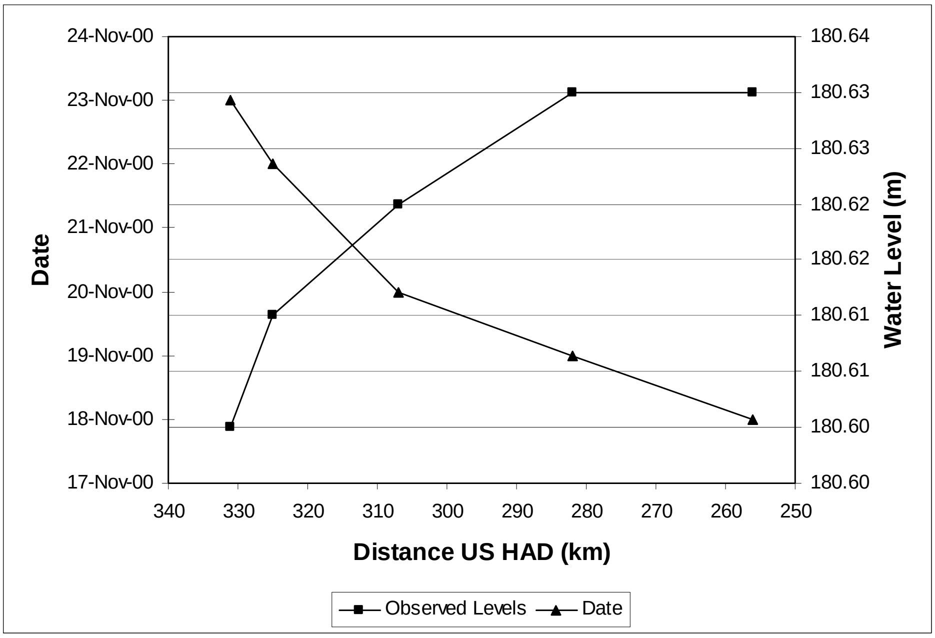

Figure 5 - 13: The measured water levels used in verification process

Figure 5-14 shows the recorded water levels upstream HAD during the simulation period which begins by the 10th of November 2000 and ends at the 25th, and Figure 5-15 shows the corresponding recorded discharges at Donqola station during the same period.

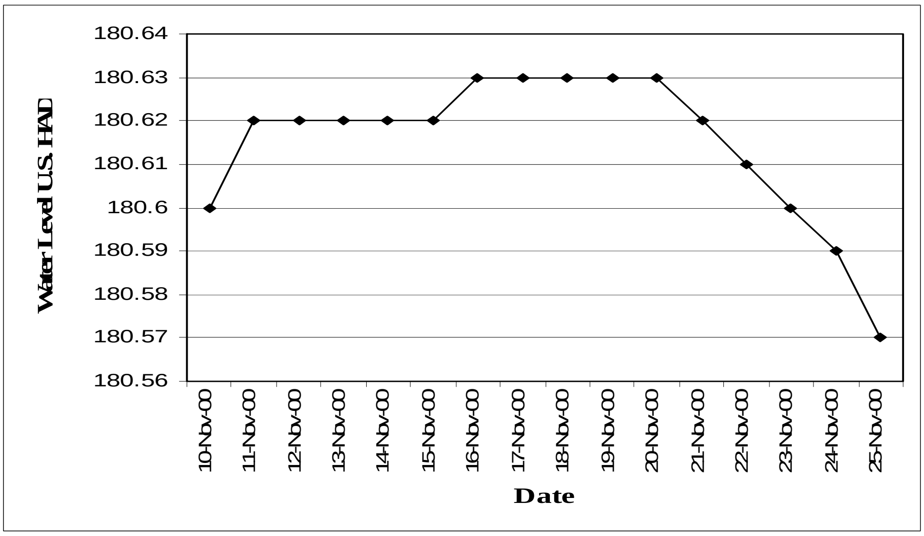

Figure 5 - 14: Water level U.S. HAD

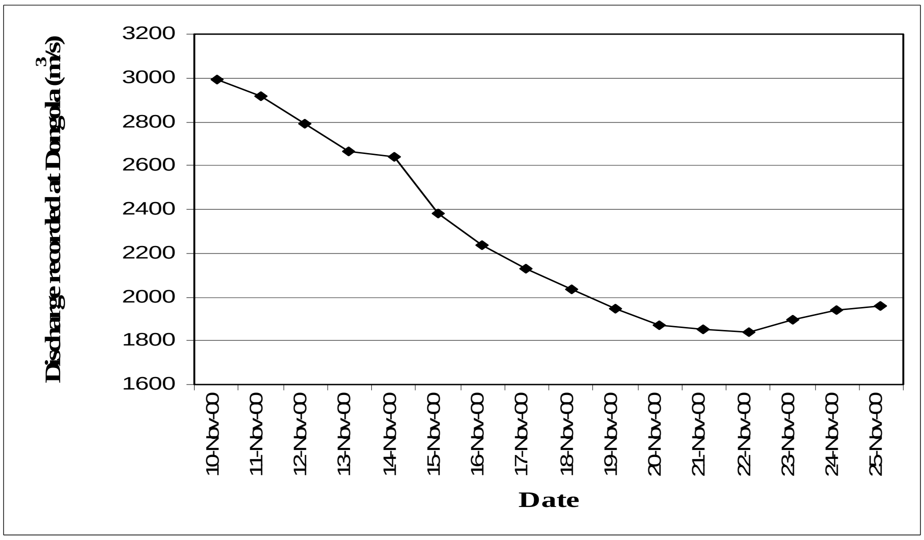

Figure 5 - 15: Discharge recorded at Donqola station

Those measured levels where compared with the levels calculated from the model and the results were plotted as shown in Figure 5-16. The results show an absolute error value ranging from 0.02 cm to 1.28 cm which is a good result.

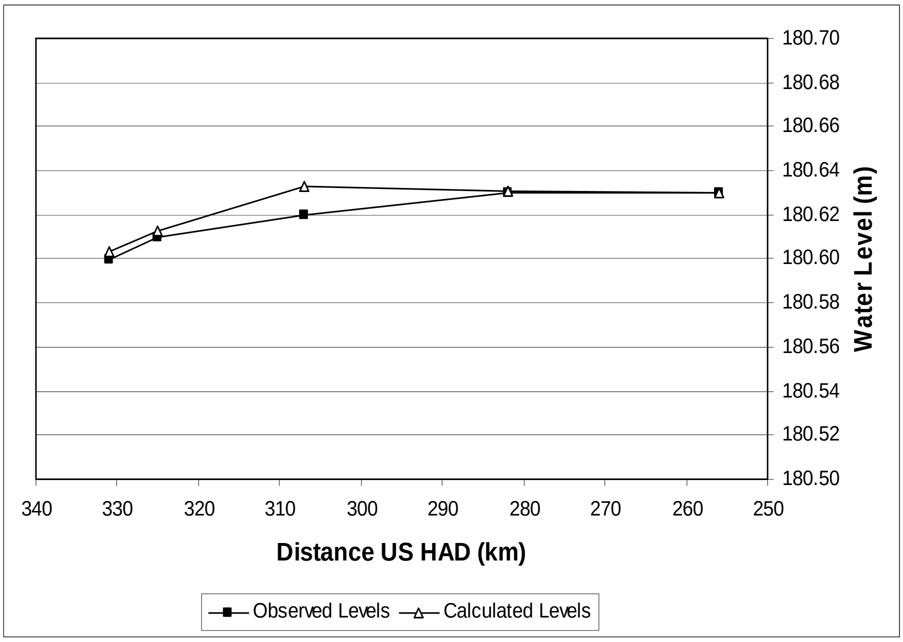

Figure 5 - 16: Observed Vs calculated water levels

The model results can be shown in the following figures, which present the resulted water levels and velocity magnitudes at the end of the simulated interval.

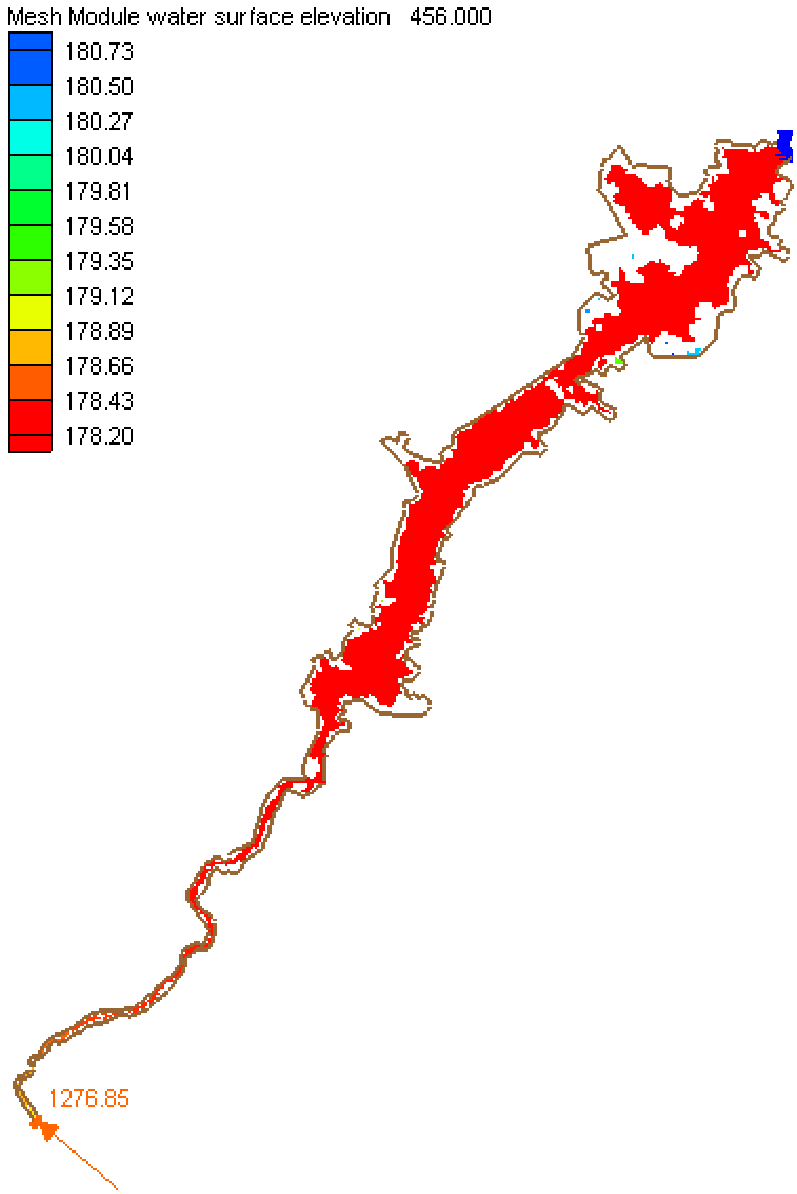

Figure 5 - 17: Simulated Water Surface Elevations on 23 Nov. 2000

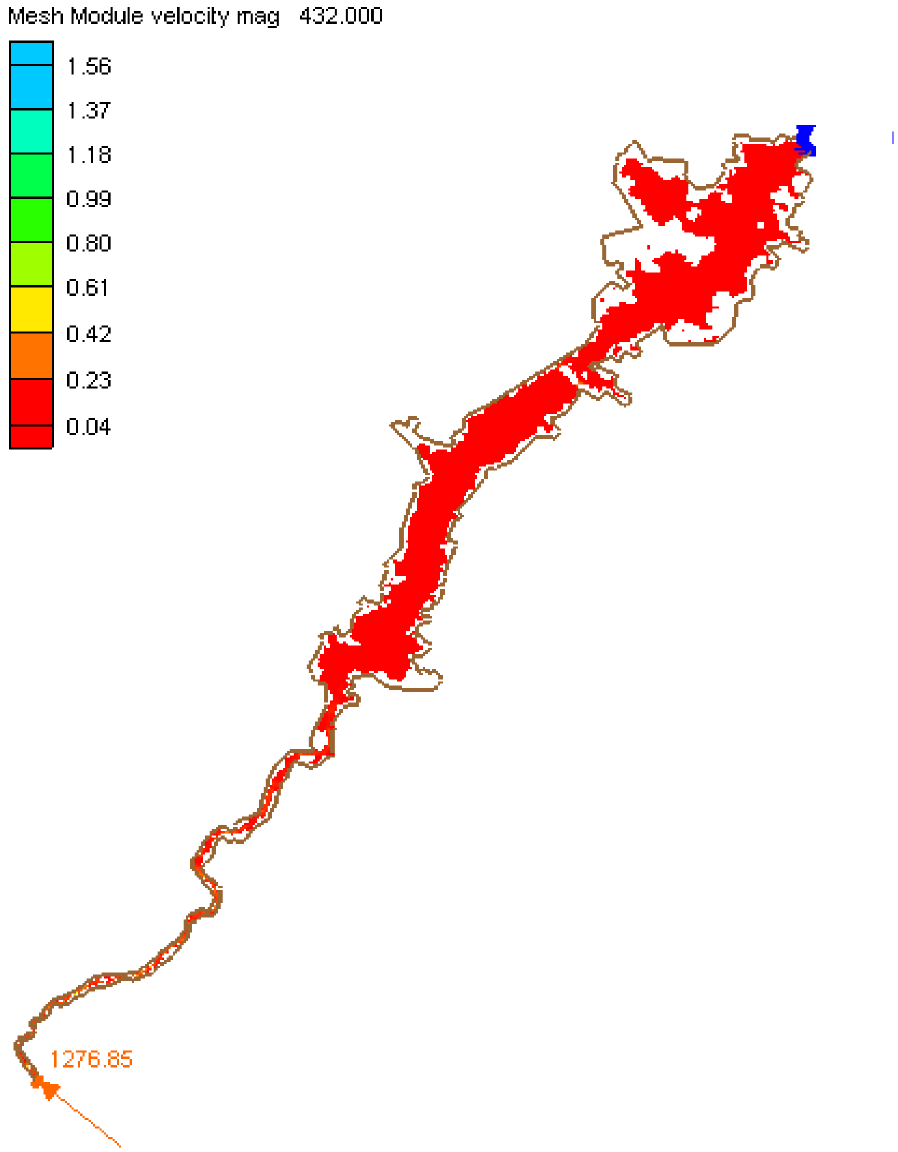

Figure 5 - 18: Simulated Velocity Magnitudes on 23 Nov. 2000

5.9 The Steady State Simulations

During the steady state simulations analysis the Manning's roughness coefficient (n) needed to be calibrated as the inflow rate increased from about 1250 m^3/sec at the beginning of the flood to more than 12,500 m^3/sec at its peak value which is a quite big range.

Steady state simulations for a flow rate of (1250, 2500, 5000, 7500 and 10,000 m^3/sec) were carried out to study the hydraulic gradient variation in the reservoir corresponding to various flood events, the water surface profiles resulted from this study are shown in Figure 5-19 and the difference of water level from the water level upstream the HAD is shown in Figure 5-20.

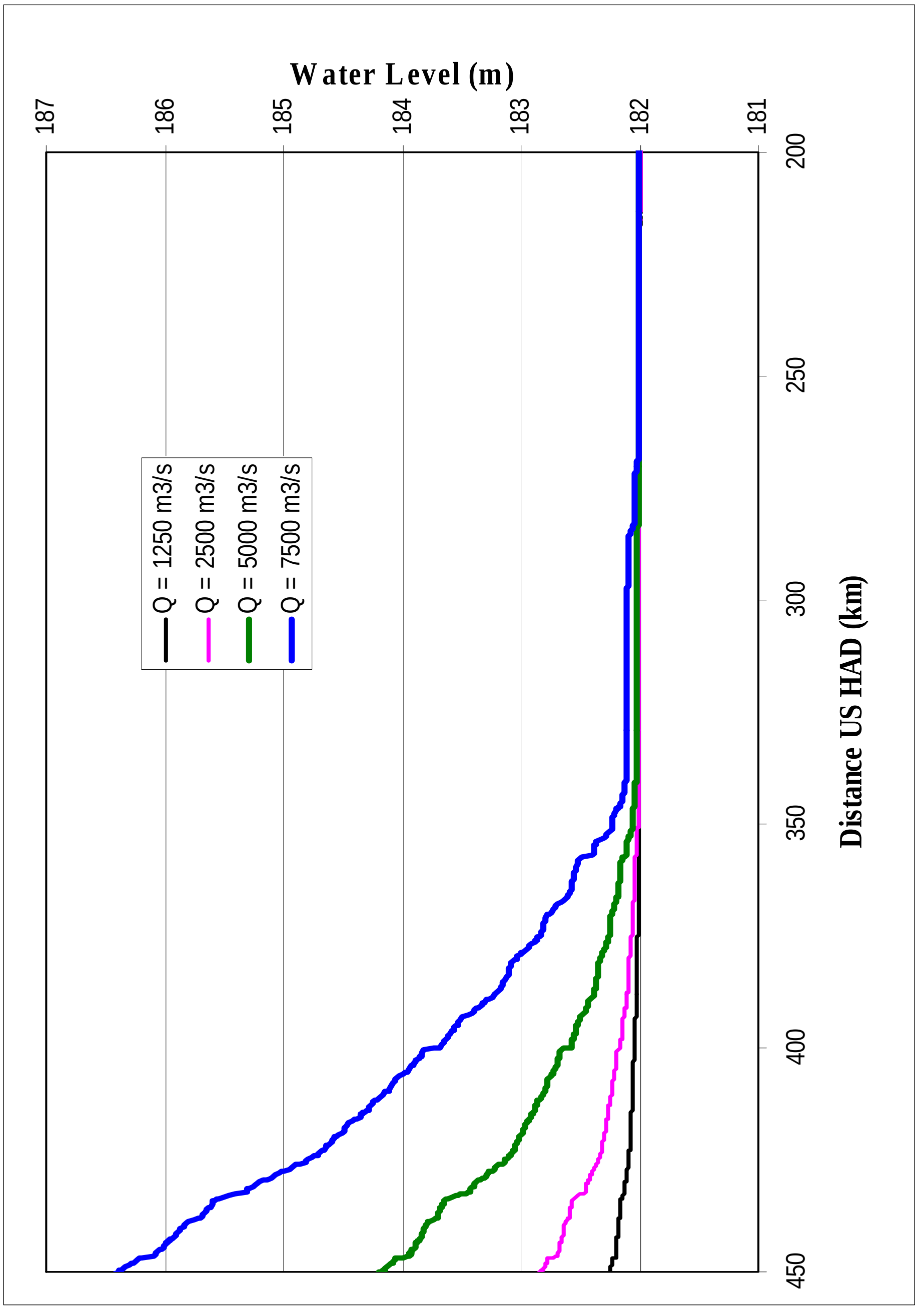

Figure 5 - 19: The water surface profiles in HADR

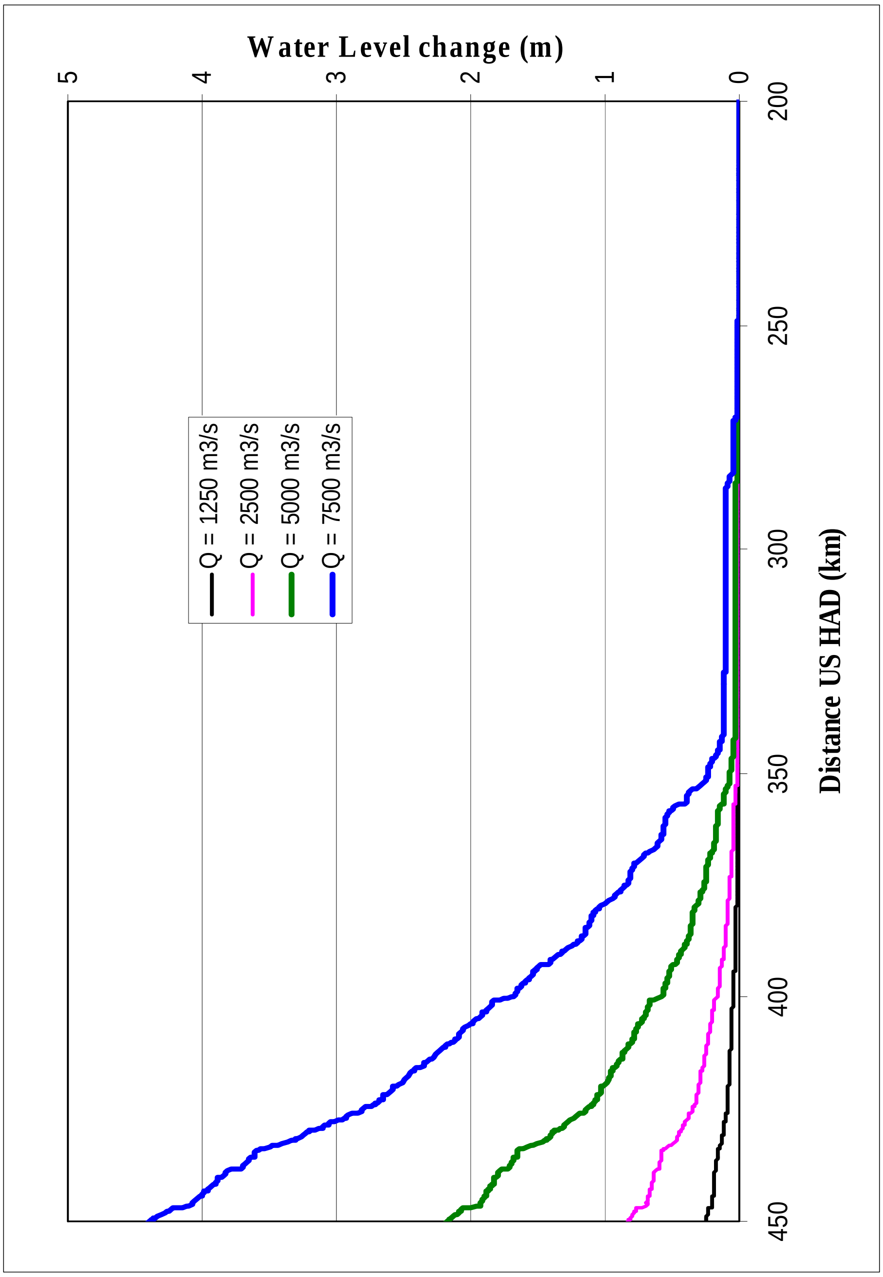

Figure 5 - 20: The water level change in HADR

5.9.1 Interpretation of the Results

From the pervious figures it was concluded that the water level is almost constant up to 250 km upstream the HAD due to the backwater effect of the dam then it began to change slightly up to 340 km upstream the HAD with hydraulic gradient of about 4.2 x 10^{-6} that causes a rise in the water surface elevation of less than 0.15 m as shown in Figure 5 - 20.

Then due to the sophisticated shape of the lake and the irregularity of the cross section along the rest of the lake, this part is known as Elshalalat which means the small waterfalls, the water surface gradient increases to an average of 2.5 x 10^{-5} and causes a rapid change in the water level with a rise of about 4.3 meters corresponding to a flow rate of 7500 m3/sec in an average distance of 100 kilometers as shown in Figure 5 - 20 .

The resulted figures assure the difficulty of taking measurements of the cross sections geometry or the flow characteristics in this part of the reservoir (upstream station 350) during high floods. So the High Aswan Dam Authority (HADA) send the missions during the month of April and November when the flow rate is about 2000 m^3/sec so they can manage to take measurements during low velocity currents.

5.1 Model selection 58
5.1.1 Origin of the Program 62
5.1.2 Model Description 63
5.1.3 Limitations of RMA2 63
5.2 The Modeling Process 66
5.3 Guidelines for Obtaining a Good Solution 67
5.4 Specifying Boundary Conditions 69
5.4.1 Upstream Boundary Condition 71
5.4.2 Downstream Boundary Condition 71
5.5 Model Checking For Continuity 73
5.6 Parameters Estimation 73
5.6.1 Bed Friction Computation 74
5.6.2 Specifying Turbulence 75
5.7 Model Calibration 77
5.8 Model Parameters Verification 86
5.9 The Steady State Simulations 91
5.9.1 Interpretation of the Results 94

Figure 5 - 1: Flow chart for the RMA2 modeling process.... 66
Figure 5 - 2: Mesh generation 68
Figure 5 - 3: Water Level Recorded Upstream HAD (1996-2003).... 72
Figure 5 - 4: Relative Importance to Calibration 74
Figure 5 - 5: River part and Reservoir part 79
Figure 5 - 6: Values of calibration roughness coefficient.... 80
Figure 5 - 7: The measured water levels used in calibration process 81
Figure 5 - 8: Water level U.S. HAD.... 82
Figure 5 - 9: Discharge recorded at Donqola station 82
Figure 5 - 10: Observed Vs calculated water levels 83
Figure 5 - 11: Simulated Water Surface Elevations on 9 May 2000.... 84
Figure 5 - 12: Simulated Velocity Magnitudes on 9 May 2000 85
Figure 5 - 13: The measured water levels used in verification process .. 86
Figure 5 - 14: Water level U.S. HAD.... 87
Figure 5 - 15: Discharge recorded at Donqola station 87
Figure 5 - 16: Observed Vs calculated water levels 88
Figure 5 - 17: Simulated Water Surface Elevations on 23 Nov. 2000 ... 89
Figure 5 - 18: Simulated Velocity Magnitudes on 23 Nov. 2000.... 90
Figure 5 - 19: The water surface profiles in HADR 92
Figure 5 - 20: The water level change in HADR 93

CHAPTER SIX

Unsteady State Simulation

Time Step Control

Timing is introduced into the simulation when the boundary conditions (head, velocity, discharge) vary in time. This is known as a dynamic or unsteady state simulation. The time step used depends upon several factors.

Although RMA2 uses an implicit solution scheme, some experimentation is usually required when establishing the delta time step for dynamic simulations. The modeling process started with a value appropriate for the type of computations, and then the delta time step was increased to the largest value that is numerically stable and physically representative of the problem.

The time step is dependent on a dimensionless flow parameter called the Courant number. Some hydrodynamic models developers suggest that to maintain numerical stability and produce accurate results, the Courant number should not exceed 1.0 (Westerink, Blain, Luettich, & Scheffner, 1994) while others (like the developers of MIKE21 in the DHI) say that it should be kept under 5.0.The Courant number is defined as:

$$C_N = \frac{\sqrt{gh}\,\Delta t}{\Delta l} \qquad 6.1$$

Where:

g	= acceleration due to gravity	
h	= nodal depth	
Δt	= time step (seconds)	
Δl	= nodal spacing	

With each flow scenario, the increase (or decrease) in the flow rate will affect the total volume of water in the system. Following the continuity equation, higher flow rates will make the water surface elevation increase due the larger volume of water in the river, causing numerical instability if the same time step is used for all scenarios. Therefore, each flow scenario uses a different Δt value.

In this case study the nodal depth varies along the reach an average of 5 meters depth in the river part to almost 60 meters in the reservoir part and the average nodal spacing is equal to 250 meter and also varies in the river part than the reservoir part.

One approach to select this interval in this case, is to employ a large time step size and to run a test case in which the time step size is reduced until the solution does not change, and use this interval as the time step size during the simulation. Using such technique the step size during the modeling process was taken equal to 12 hrs.

For dynamic simulation runs, the computational time interval should be as small as necessary to capture the extremes of the dynamic boundary conditions and maintain numerical stability. Yet, to reduce the computational time taken to complete a simulation, the time step Interval should be as large as possible, while small enough to still accurately simulate the hydrodynamics of the modeled area.

The interval should be small enough to:

- Capture the extremes (highest and lowest peaks) of the boundary condition signal. If the interval is too large, the peaks of the signal may be missed.
- Accommodate rapid changes in water surface elevation.

Because unsteady flow models reproduce the entire range of flows, they should be calibrated to reproduce both low and high flows.

A Factitious Flood Pulse Simulation

The first step taken was to simulate a factitious flood wave with a rapid rising limb to study the wave propagation through the lake and test the stability of the model with different unsteady flow rates.

The flood pulse shown in Figure 6 - 1 was used at the inlet and during this simulation the water level at the downstream boundary was assumed to be 180 m. The results of the 2D hydrodynamic model (Figures 6.1 to 6.6) showed the flood wave propagation in the reservoir, and the changes to such wave in the downstream sections up till the High Aswan Dam.

From those resulted hydrographs it could be noticed that a hydrograph of a 10,000 m^3/s as a peak value and with a rising limb slope of 2000 $m^3/s/hr$ will travel trough the reservoir in about 16 hours and the peak will fall to about 3500 m^3/s which is, fortunately, dose not happen as the real flood rising limb slope is much more milder than that. It could also be noticed that most of the attenuation occurred in the reservoir part when the width increases to more than 2500 m

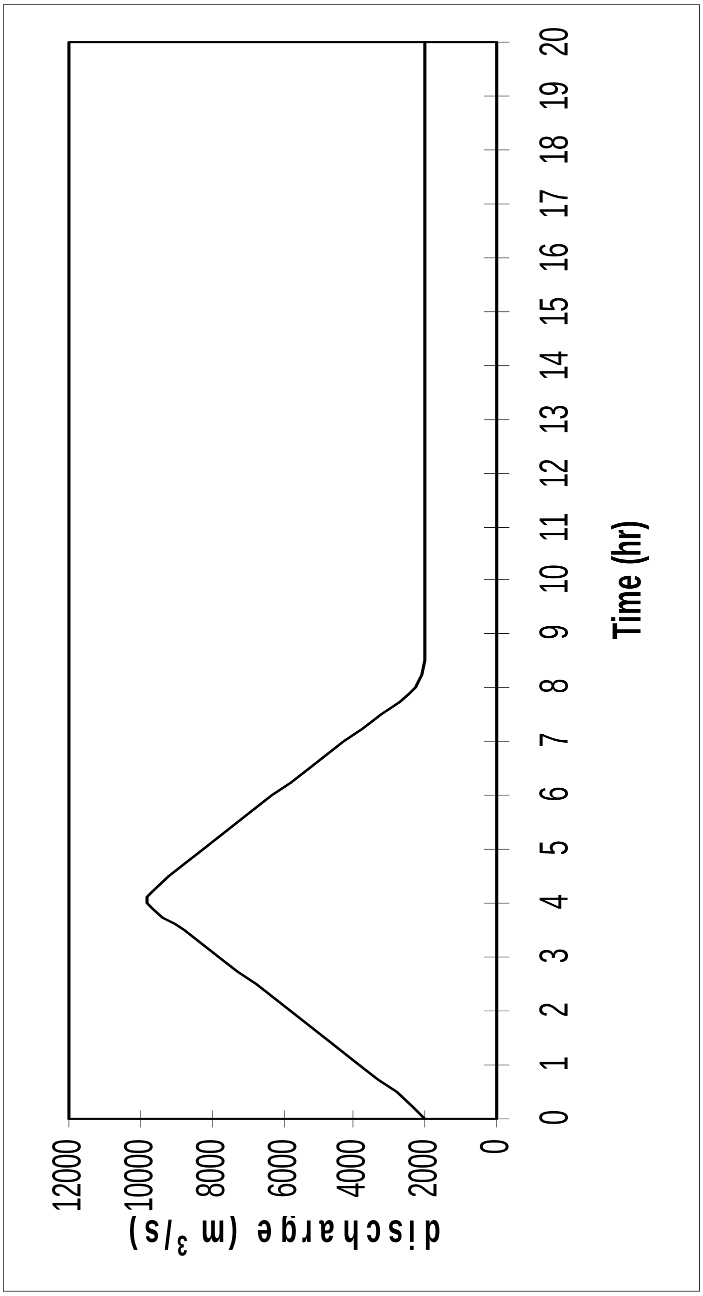

Figure 6 - 1: An Input factitious Flood wave Pulse

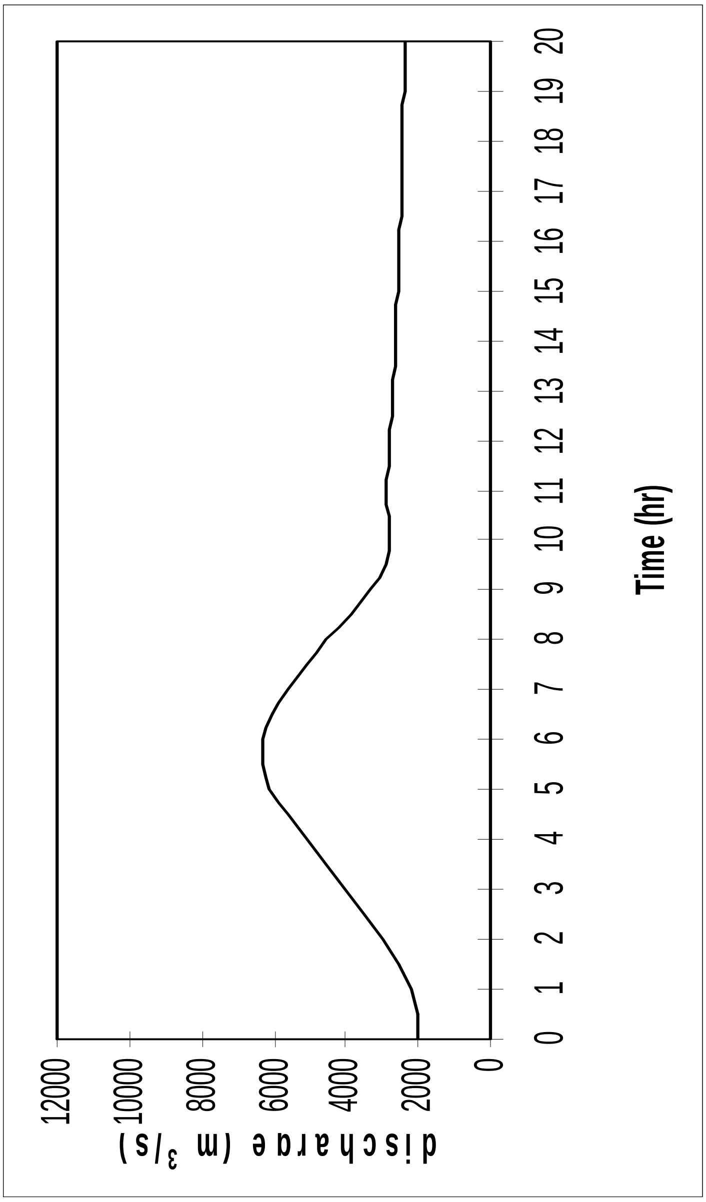

Figure 6 - 2: Hydrograph at section 5, 378 km upstream HAD

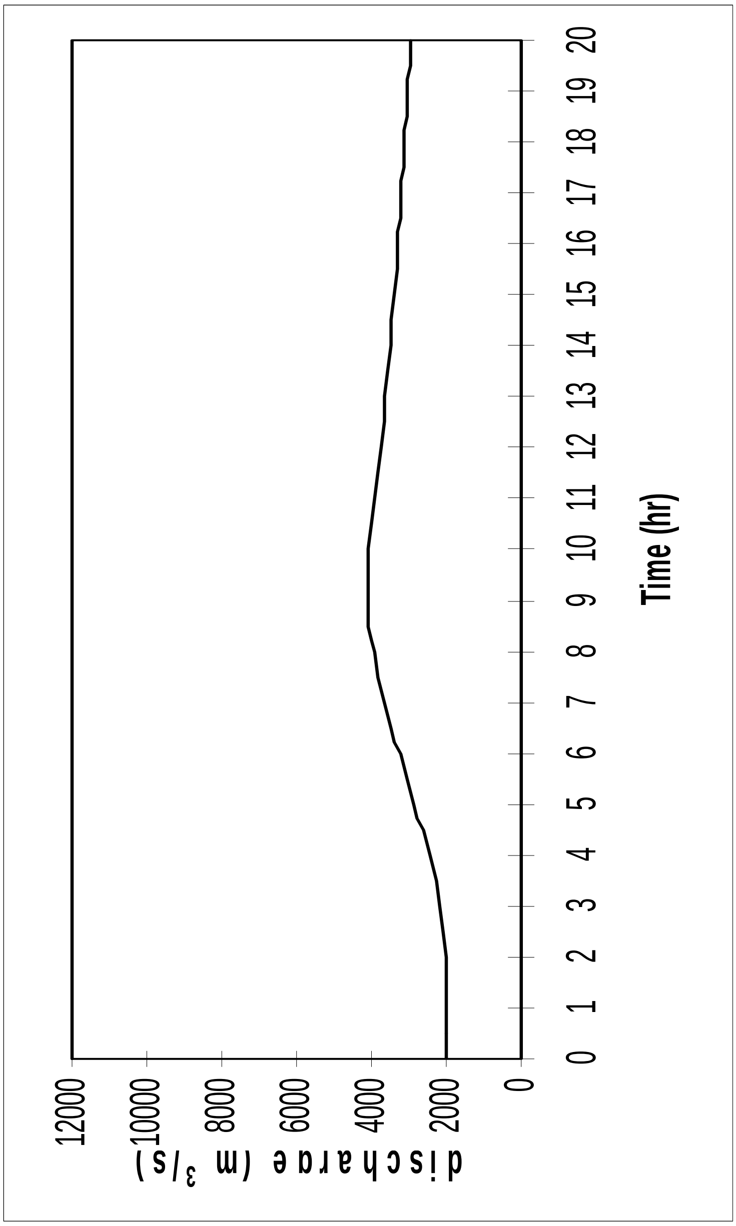

Figure 6 - 3: Hydrograph at section 8, 357 km upstream HAD

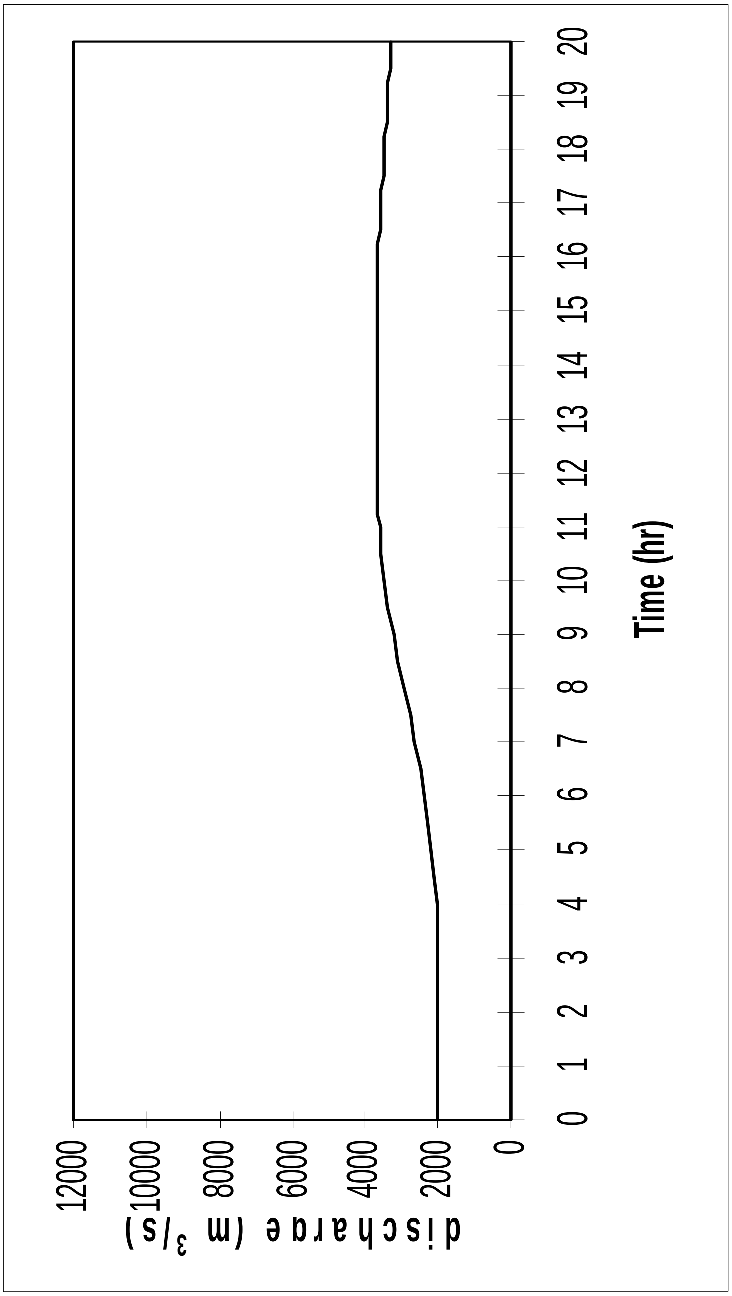

Figure 6 - 4: Hydrograph at section 10, 221 km upstream HAD

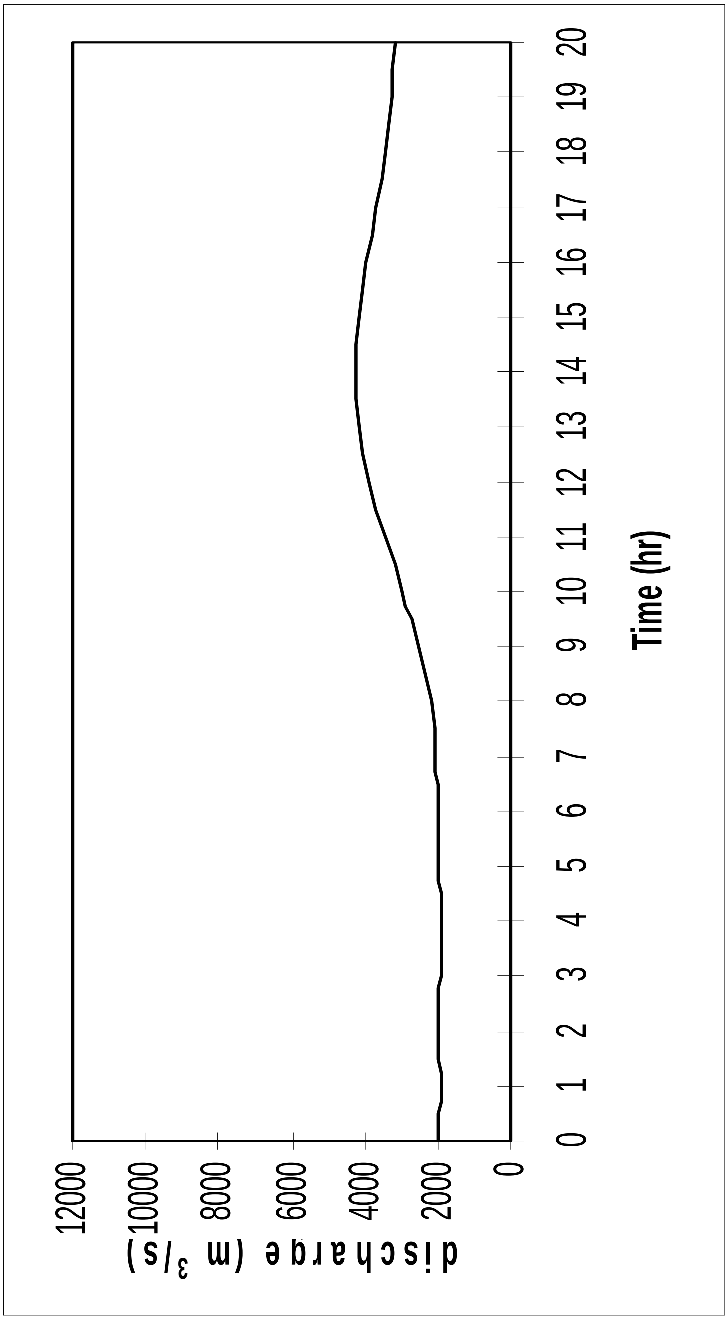

Figure 6 - 5: Hydrograph at section 11, 135 km upstream HAD

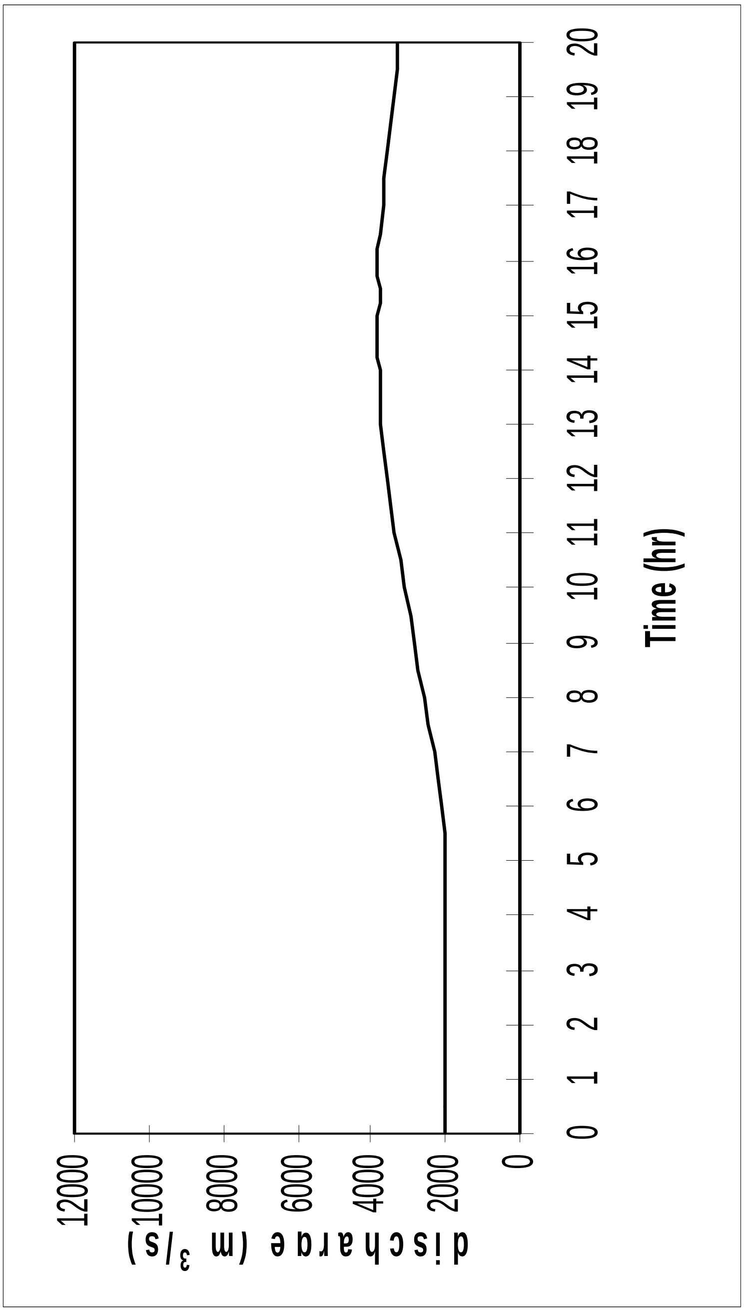

Figure 6 - 6: Hydrograph at HAD

Study of Various Flood Waves

In order to study the flood wave propagation through the reservoir, various simulations were carried out with different water levels in the downstream of reservoir (182, 180, 178 and 175) meters and varying the inflow rising and falling flood wave limbs slope (10, 20, 50) $m^3/s/hr$.

First rising limb waves were applied to the model and the flood wave movement was monitored through out the High Aswan Dam Reservoir and the lag time, the outflow flood wave slope and the attenuation were recorded.

These Results were summarized in the following figures.

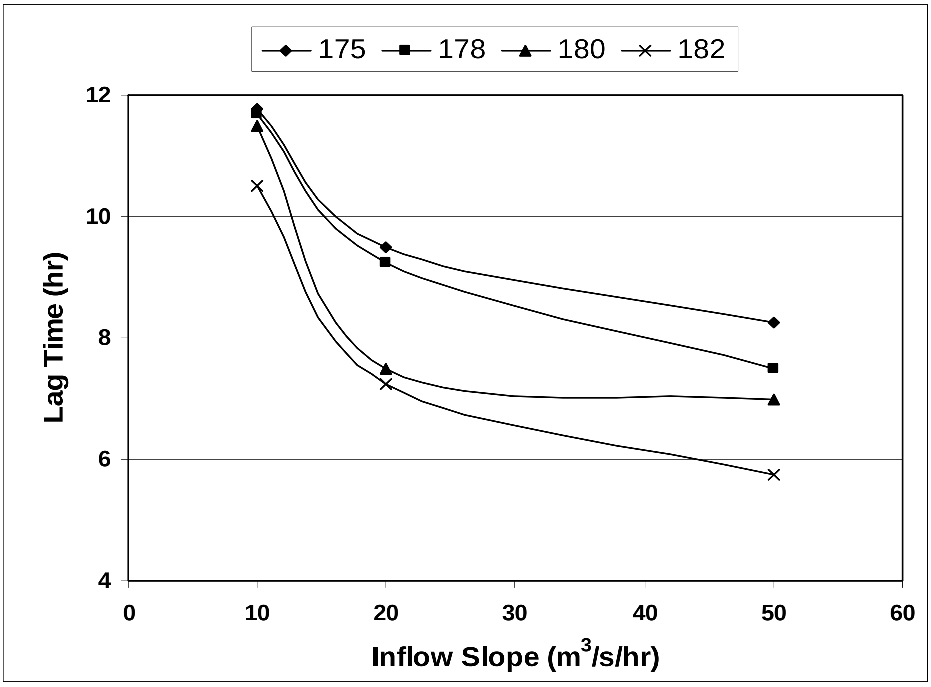

Figure 6 - 7: Inflow rising flood wave slope Vs Lag time

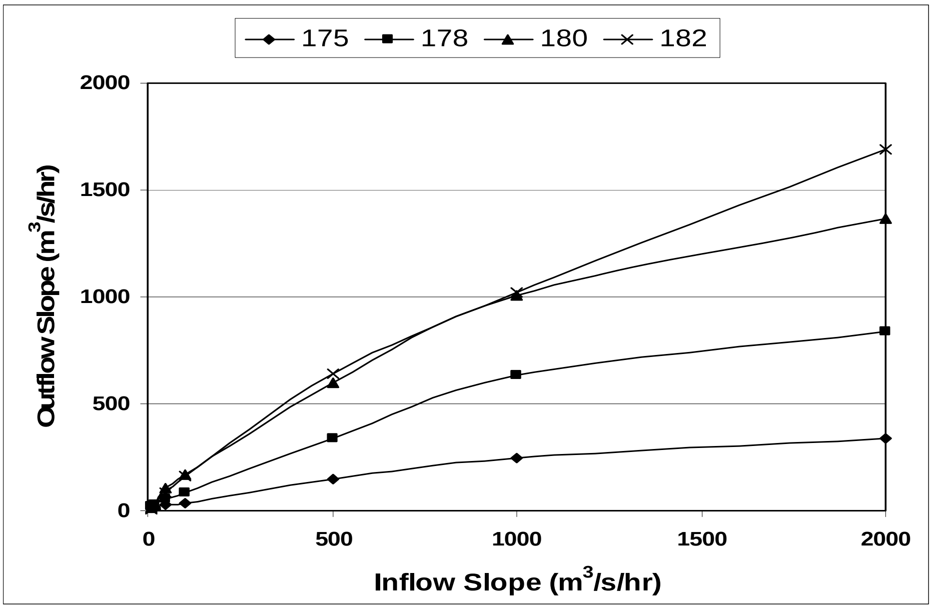

Figure 6 - 8: Inflow rising flood wave slope Vs Outflow flood wave slope

The interpretation of the previous curves yields to the derivation of these relations,

$$L_t = 13.321(IS)^{-0.1457} \quad (6.2)$$

With a least squared value = 0.8907

$$OS = 3.3496(IS)^{0.7365} \quad (6.3)$$

With a least squared value = 0.9943

Where,

L_t = Lag time in hours

IS = Inflow slope in $m^3/s/hr$

OS = Outflow slope in $m^3/s/hr$

Falling limb waves were applied to the model and the flood wave movement was monitored through out the High Aswan Dam Reservoir and the lag time and the outflow flood wave slope were recorded, these Results were summarized in the following two figures.

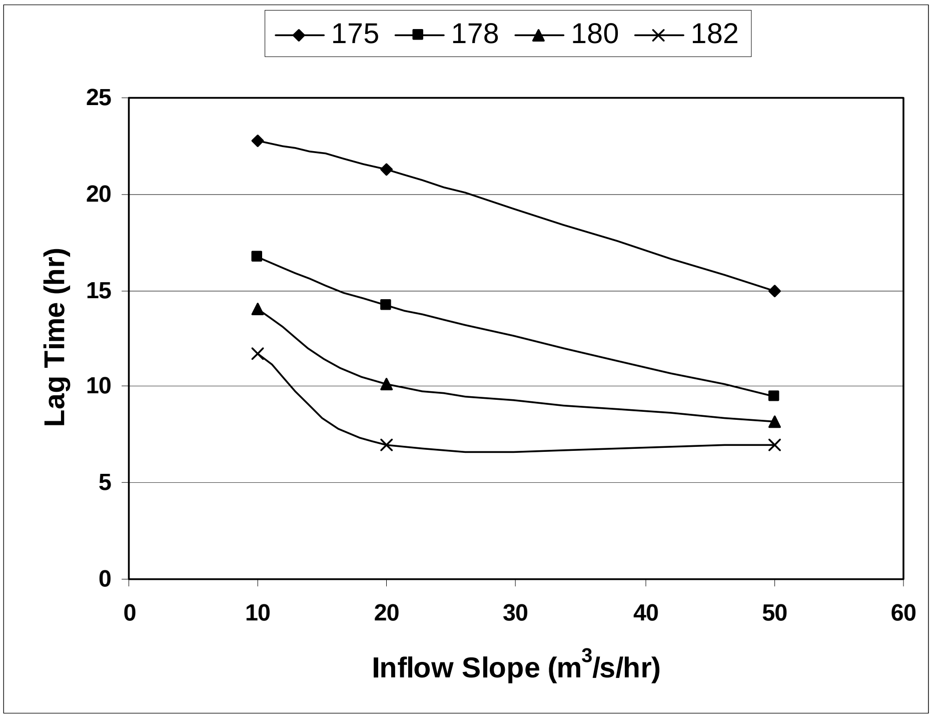

Figure 6 - 9: Inflow falling flood wave slope Vs Lag time

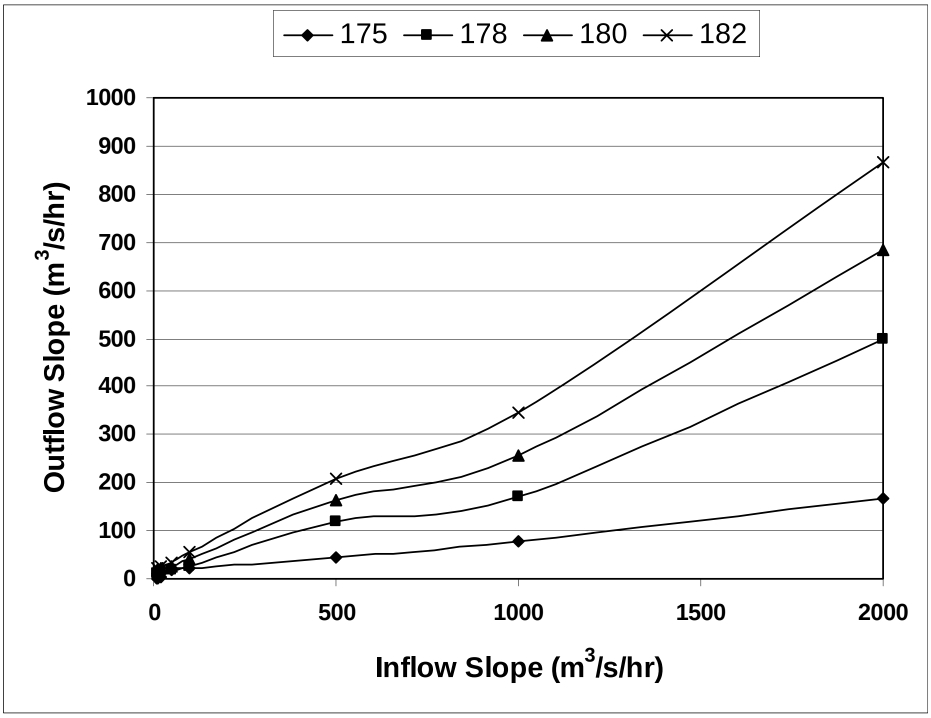

Figure 6 - 10: Inflow falling flood wave slope Vs Outflow wave slope

The interpretation of the previous two curves yields to the derivation of these relations,

$L_t = 22.99(IS)^{-0.1964}$ 6.4

With a least squared value = 0.8786

$OS = 1.8132(IS)^{0.6793}$ 6.5

With a least squared value = 0.9355

Where,

L_t = Lag time in hours

IS = Inflow slope in $m^3/s/hr$

OS = Outflow slope in $m^3/s/hr$

It must be noticed that these charts and equations were developed assuming that the downstream water level is kept to a pre-specified values (175, 178, 180 and 182 m in our case) as a downstream boundary condition.

There are three types of unsteady open channel waves commonly used in civil engineering to study the behavior and progression of flood waves (Ponce, 1989). The most complicated and generally applicable form of flood wave is the dynamic wave, based on the St. Venant equations.

Kinematic waves are based solely on the principle of conservation of mass within a control volume. This means that the difference between inflow and outflow is equal to the change in storage volume, based on a balance between friction and gravity.

Diffusion wave is more widely used because it applies to a wider range of real flood waves. Most flood waves have some degree of physical diffusion and this makes the applicability of this equation much wider.

From the resulted figures 6-7 and 6-9, it was concluded that the wave travels through the reservoir in a kinematic nature that is the flood wave speed is directly proportional with the depth of water.

If a downstream control structure such as the HAD exists at the downstream this will cause the attenuation and the lag time to increase, the following section will discuss the simulation of a real flood wave with a control structure at the downstream end.

The Real Hydrograph Simulation

In this section a subroutine is developed to apply the real maximum recorded hydrograph at Donqola station on the model with a controlled downstream boundary condition and taking into consideration the evaporation losses as estimated by the ministry of Irrigation (Figure 6 - 11).

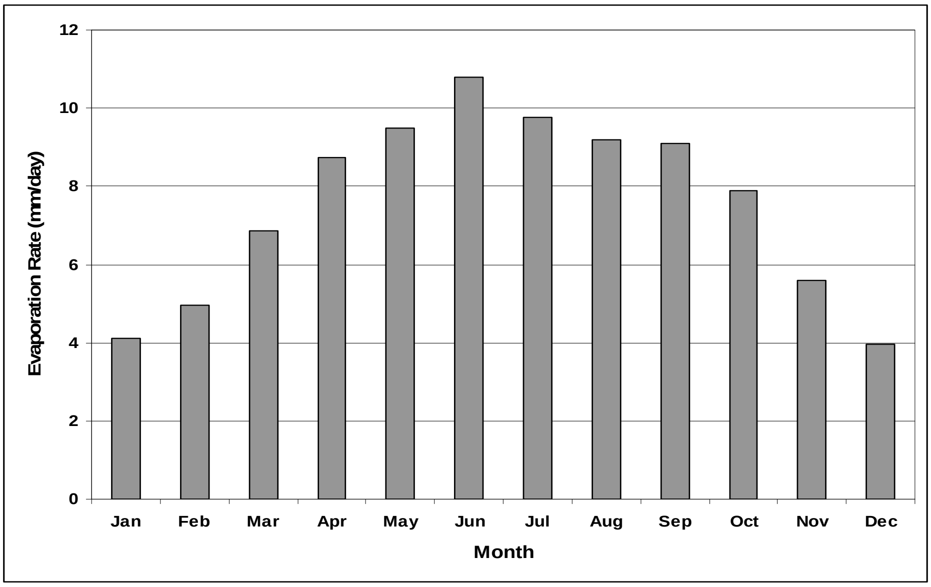

Figure 6 - 11**:** Estimates of Monthly Evaporation Rates From (HADR) Source: Omar, M.H. and El-Bakry (1970)

The hydraulic routing of the flood started at the beginning of August with an initial water level in the reservoir of 175.00 meters above mean sea level, as it is recommended by the ministry of irrigation to start the flood year by the water level not more than this value.

Hydrodynamic Reservoir Operation Model

The developed 2D hydrodynamic model was to be used in dam operation, to help predicting the water levels in the reservoir. Unfortunately, this model can not simulate a controlled structure in the as a downstream boundary condition.

Some modification had to be made to the model so it can be used in the dam operation. The following section describes such procedure.

The storage in the reservoir was calculated from the model results using the following equation:

ΔStorage = (Inflow – Demand – Tushka spillway overflow – Evaporation Losses) Δt

Figure 6 - 12 illustrate the storage volume in the reservoir versus the water level (Egyptian Ministry of Irrigation, High Aswan Dam Authority). From which a relation between the storage volume of the lake and the water level could be derived (equation 6.6).

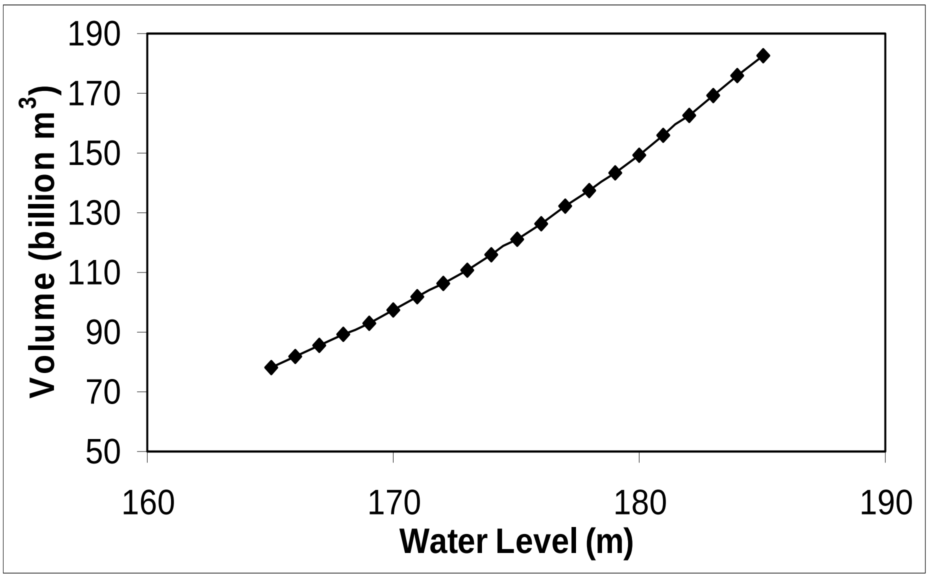

Figure 6 - 12**:** Storage Vs Water Level in the Reservoir

$S = 2 \times 10^{-15} \times (WL)^{7.4647}$ 6.6

Where:

S is the stored water volume in billion cubic meters.

WL is the water level in the reservoir.

Tushka spillway overflow can be calculated from the water level using the following equation (M.Abdel Motaled and A.Allaithy) Hydraulic Research Institute (HRI) 1998:

$Q_{Tushka} = 2.012\ B\ (WL - \text{Crest Level})^{1.569}$ 6.7

Where:

B is the spillway crest width in meters.

WL is the water level in the reservoir.

It should be noticed from the equation that the spillway starting to release water when the water level is above its crest level.

Figure 6 - 13 illustrate the reservoir surface area versus the water level (Egyptian Ministry of Irrigation, High Aswan Dam Authority). From which a relation between the surface area of the lake and the water level could be derived (equation 6.8). The evaporation losses were then calculated in m^3/s by multiplying the surface area by the evaporation rate Figure 6 - 11.

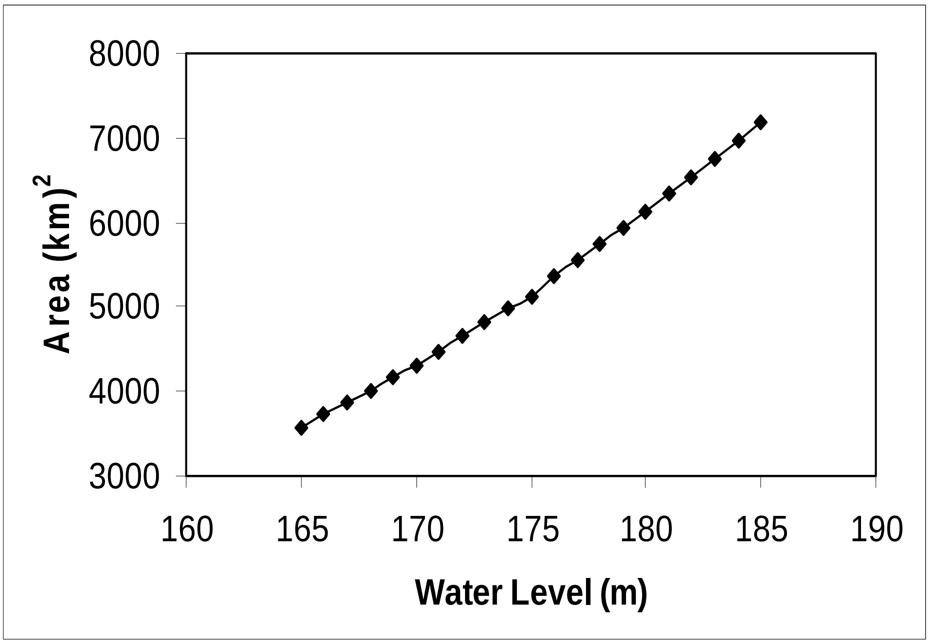

Figure 6 - 13**:** Surface area Vs Water Level in the Reservoir

$A = 1 \times 10^{-10} \times (WL)^{6.0889}$ 6.8

Where:

A is surface area in square kilometers.

WL is the water level in the reservoir.

<u>The Model Modification Process</u>

The Hydrodynamic model was modified to be used in the routing process of the real flood by following this algorithm:

1. The routing starts by the 1^{st} of August with the water level in the reservoir equals to (175.00) meter above the mean sea level.

2. A 10 days simulation interval was then carried out assuming the downstream boundary condition equal to (175.00) meters.

3. The arrived flow at HAD was obtained, after running the hydrodynamic model, and then compared with the water releases.

4. A Subroutine uses the Hydrodynamic model to evaluate the downstream water surface boundary condition as will be described in the following steps.

5. The difference between the arrived flow and the releases in this interval was used to calculate the change in the storage using equation (6.9) and (6.10):

   $$\Delta Storage = \Delta t \times \Sigma(Q_{Arrived} - Q_{Demand}) - \Delta Q_{Evaporation} \quad 6.9$$

   $$\Delta Q_{Evaporation} = \text{Evaporation Rate} \times \text{Surface Area} \quad 6.10$$

6. The new water volume in the reservoir was used to calculate the new water level to be used as the downstream boundary condition for the next 10 days interval using equation (6.6).

7. Another routing, 10 days interval, is to be started with the water level calculated from step 6.

8. If the water level exceeded 178 meters the Tushka spillway will begin to release flow and equation (6.11) should be modified.
   ΔStorage =
   $$\Delta t \times \Sigma(Q_{Arrived} - (Q_{Demand} + Q_{Tushka})) - \Delta Q_{Evaporation} \qquad 6.11$$

Figure 6 - 14 shows the flow chart of the modification made to the RMA2 model to simulate the real case study and the results of this simulation are shown in Figure 6 - 15.

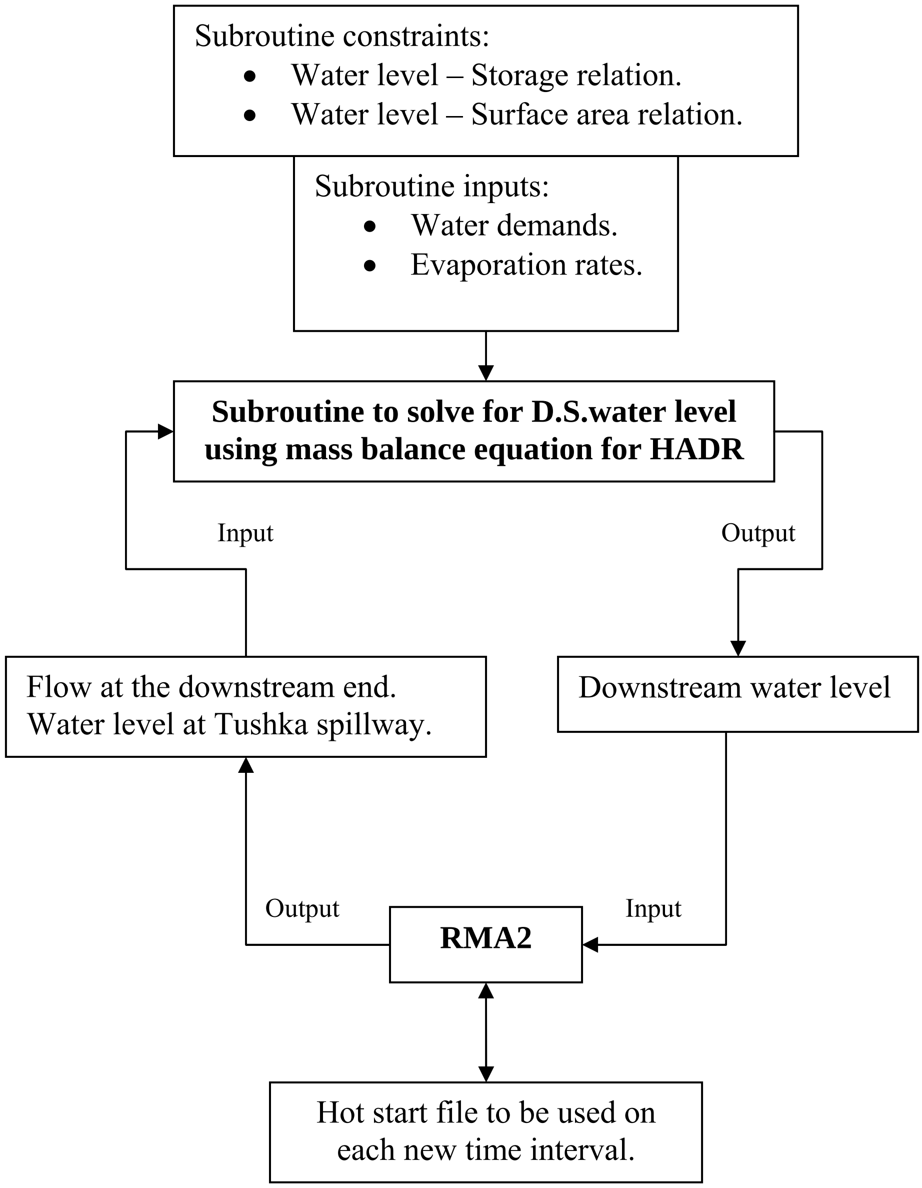

Figure 6 - 14: Flow chart of the modification made to RMA2

The Modified Model Results

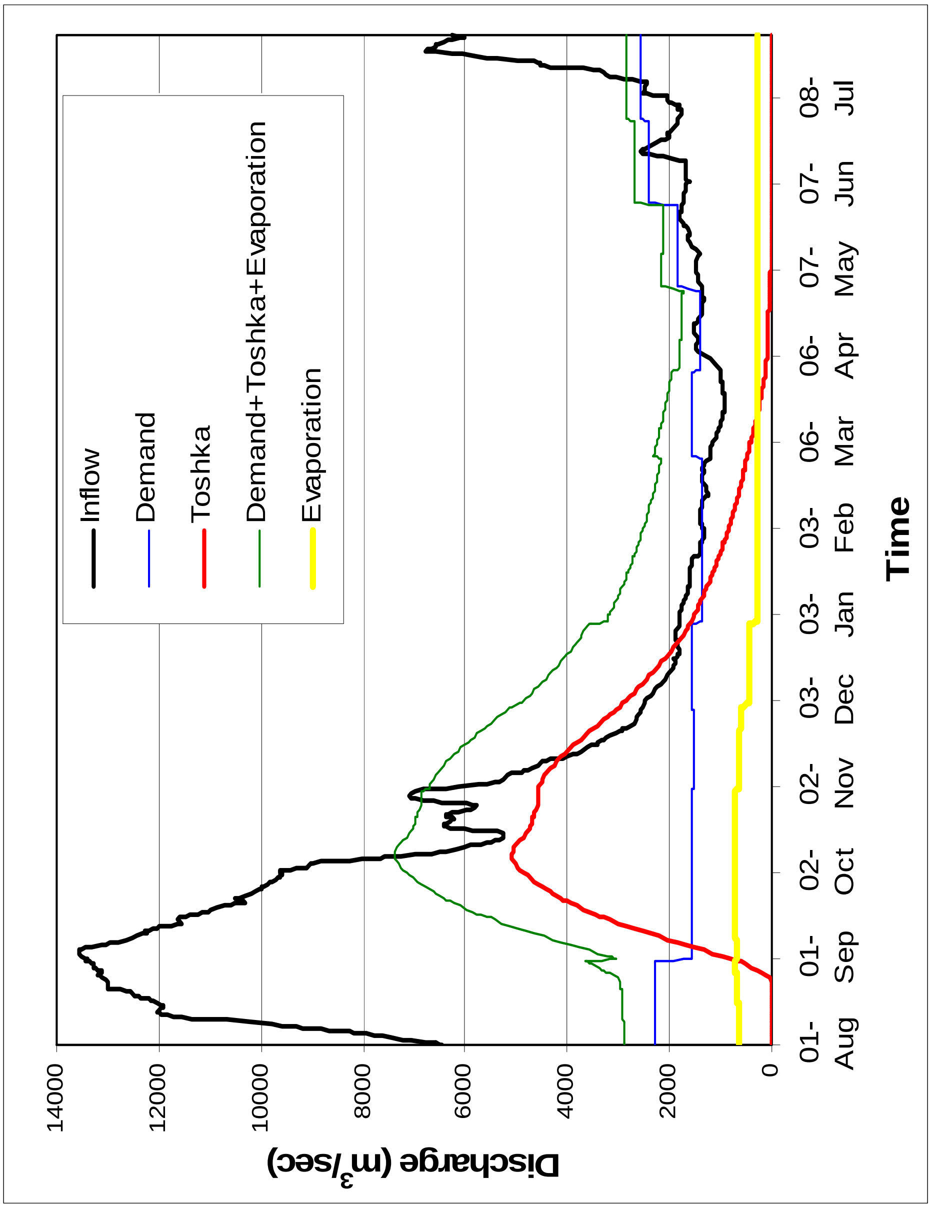

Figure 6 - 15: Inflow, Demand withdrawals, Tushka spillway outflow and Evaporation losses Vs. Time

The resulted storage and water level upstream the HAD during 1 year simulation period are shown in Figure 6 - 16.

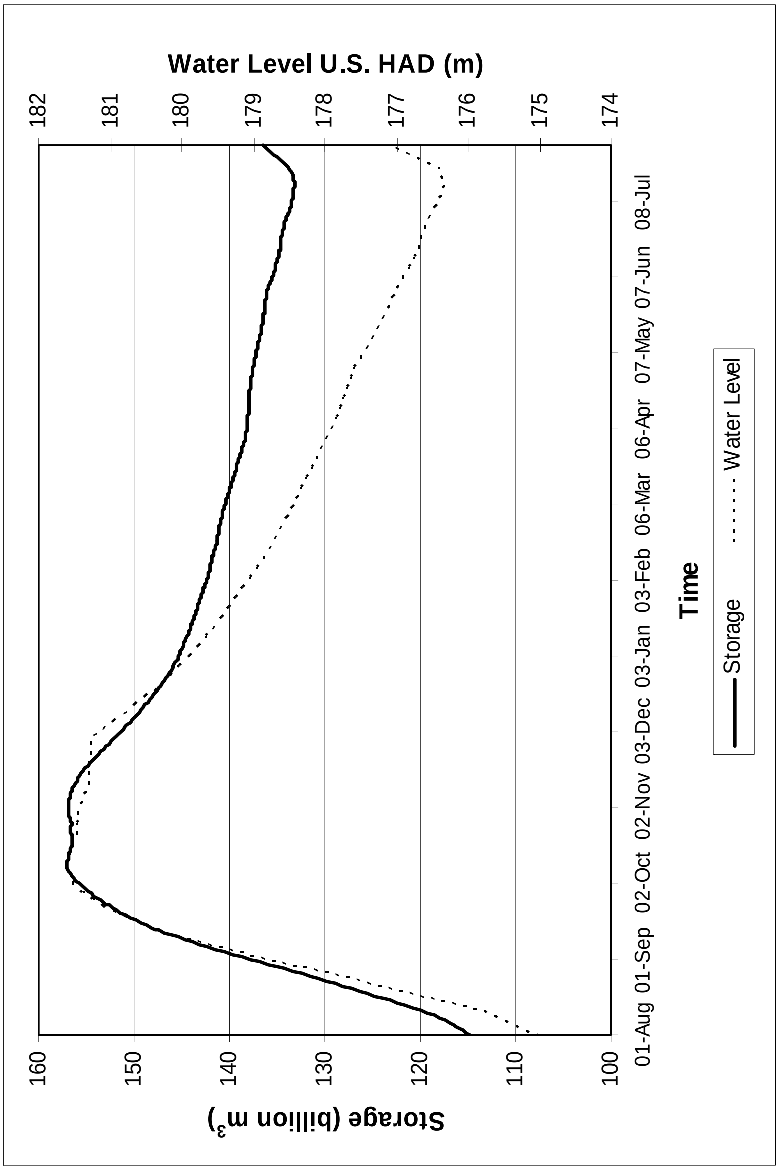

Figure 6 - 16: Storage and Water level U.S. HAD Vs. Time

Interpretation of the Results

The decision makers put a scenario to insure that the water level by the first of August is not more than (175.00) m. By applying this scenario the maximum water level that would reached is (181.55) m by the mid of October which is less than the maximum designed level.

It is proposed to start by the first of July with a water level in the reservoir equal to (174.85) m when applying this scenario the water level just reached the (182.00) m level by the mid of October, and that causes increase of the storage in the reservoir by 2.898 billion cubic meters than the currently proposed scenario.

The configuration of Tushka channel has to be modified to be able to convey a maximum expected flow of about 5071 m^3/s or the water level may rise more than the calculated value as the downstream end of the Tushka channel will control the flow diverted to the depression.

6.1 Time Step Control 97
6.2 A Factitious Flood Pulse Simulation 99
6.3 Study of Various Flood Waves 106
6.4 The Real Hydrograph Simulation 111
6.5 Hydrodynamic Reservoir Operation Model...................................112
6.5.1 The Model Modification Process 115
6.5.2 The Modified Model Results 118
6.5.3 Interpretation of the Results 120

Figure 6 - 1: An Input factitious Flood wave Pulse..............................100
Figure 6 - 2: Hydrograph at section 5, 378 km upstream HAD 101
Figure 6 - 3: Hydrograph at section 8, 357 km upstream HAD 102
Figure 6 - 4: Hydrograph at section 10, 221 km upstream HAD 103
Figure 6 - 5: Hydrograph at section 11, 135 km upstream HAD 104
Figure 6 - 6: Hydrograph at HAD...105
Figure 6 - 7: Inflow rising flood wave slope Vs Lag time....................106
Figure 6 - 8: Inflow rising flood wave slope Vs Outflow flood wave slope 107
Figure 6 - 9: Inflow falling flood wave slope Vs Lag time 108
Figure 6 - 10: Inflow falling flood wave slope Vs Outflow wave slope109
Figure 6 - 11**:** Estimates of Monthly Evaporation Rates From (HADR) 111
Figure 6 - 12**:** Storage Vs Water Level in the Reservoir 113
Figure 6 - 13**:** Surface area Vs Water Level in the Reservoir................114
Figure 6 - 14: Flow chart of the modification made to RMA2.............117
Figure 6 - 15: Inflow, Demand withdrawals, Tushka spillway outflow and Evaporation losses Vs. Time..118
Figure 6 - 16: Storage and Water level U.S. HAD Vs. Time 119

CHAPTER SEVEN

High Aswan Dam Reservoir Simulator

- **General**

The Previous chapters discussed the procedure followed to develop a two dimensional hydrodynamic model for High Aswan Dam Reservoir. The objective was to use such model for reservoir operation.

First the HAD reservoir bed surface was simulated then the hydrodynamic model was developed, calibrated and verified resulting in simulating the flow fields, steady and unsteady, of the reservoir.

The developed model was then applied on a real flood wave to test its ability to be used for reservoir operation. But, the effort and time consumed during such a process proved that a hydrodynamic model for HADR is not a good tool to be used for dam operation. Yet, it can be used in sedimentation or water quality studies.

The following sections describe the development of a hydrological model (HADR Simulator) for dam operation along with its results.

Model Development

The model was based on the mass balance equation of the reservoir, which can be described as the summation of the inflow equal the summation of the out flow and can be represented in the following form:

The change in storage =
(Inflow – Releases – Tushka spillway overflow – Evaporation losses) Δt

As the results of the hydrodynamic model show, the water level in the reservoir is almost constant from the HAD upstream to Tushka spill way the release over the weir can be directly calculated from the water level using the weir equation presented in chapter six, equation 6.7.

The evaporation losses, Tushka overflow and storage can be calculated as a function in water level as described in chapter six, equations 6.6 to 6.8.

The model developed interface have options to run for predefined releases or it may be used to maximize the water level or, and the hydropower. The flowchart, Figure 7 - 1, describes the model parts.

HADR Simulator Interface

The HADR Simulator interface was designed using the Visual Basic programming language. It was designed to be used interactively through out message boxes that pop out according to the user clicks.

It is easy to use as the input and output files are written and read in text format, which can be exported to any data processing program that can display them in charts.

Figure 7 - 2 shows the HADR Simulator starting window as you initiate the program.

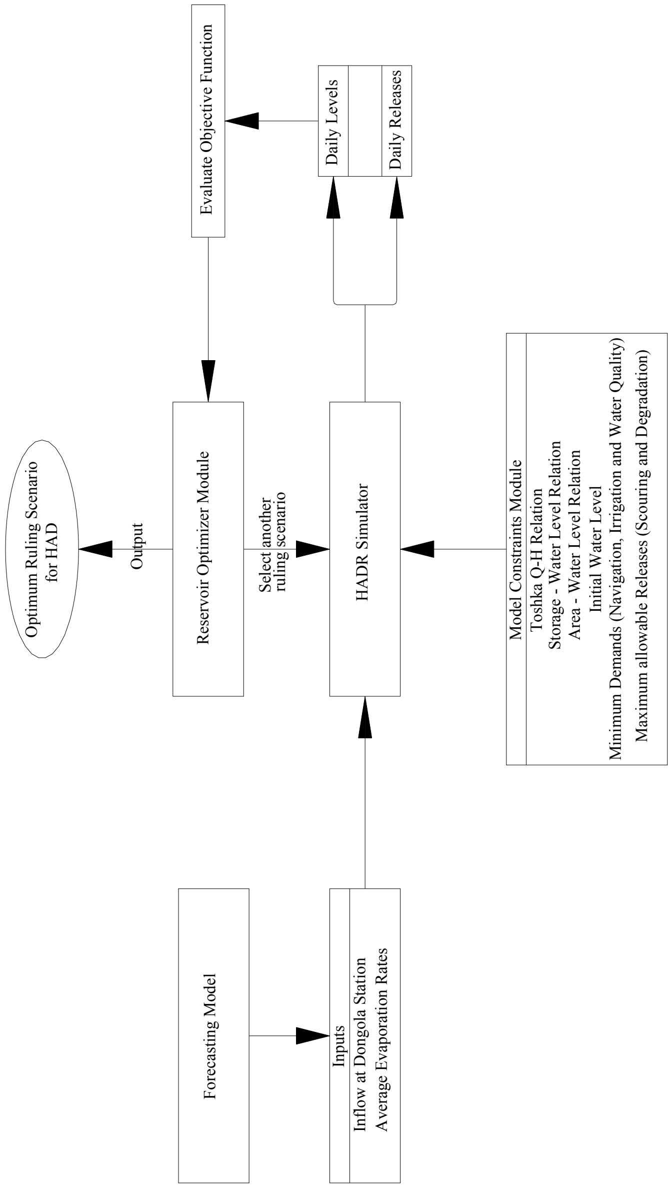

Figure 7 - 1: Flow chart of HADR Simulator Model

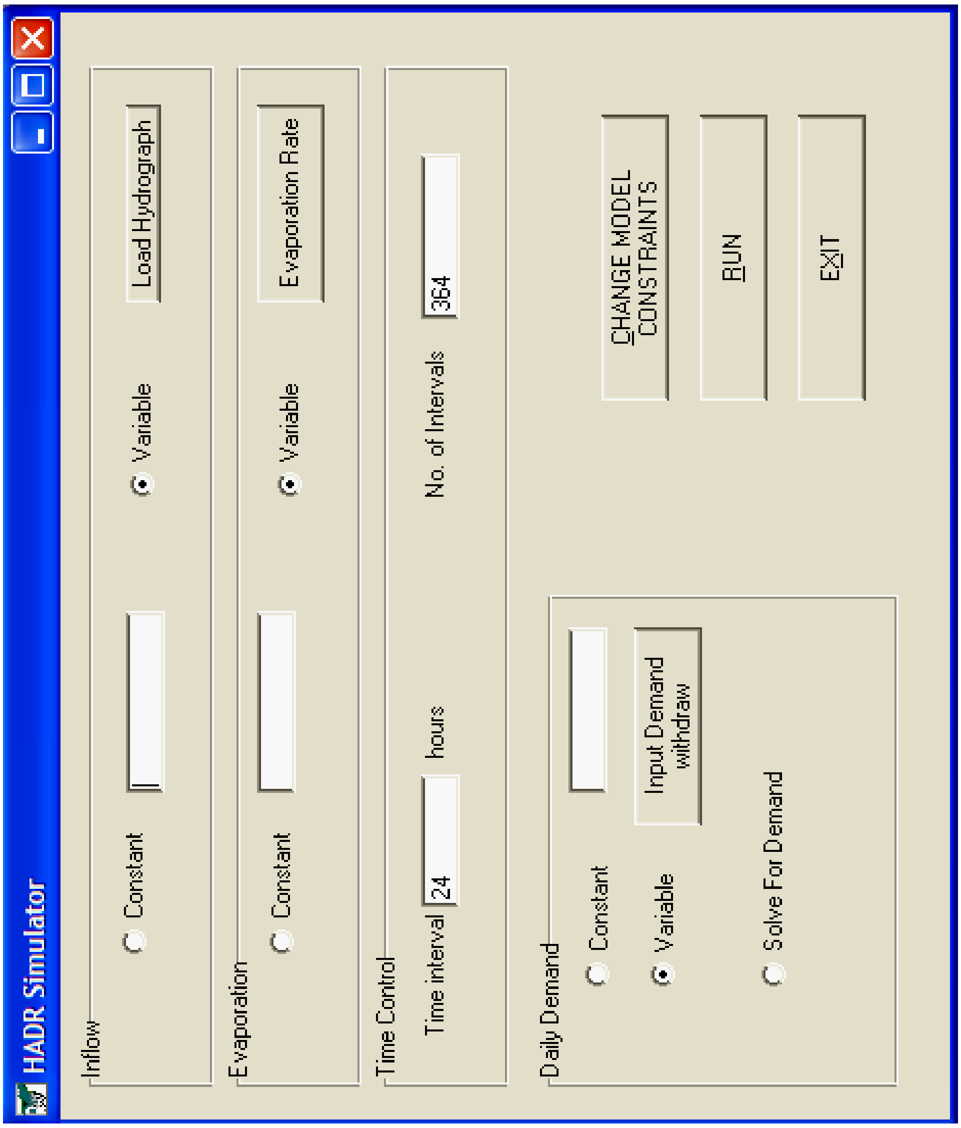

Figure 7 - 2: HADR Simulator main window

Model Constraints

The model constraints can be summarized in the following points;

- Maximum releases from HAD was assumed to be 280 million m^3/day as the maximum capacity of the river cross sections downstream to prevent scouring of the bed downstream the Nile Barrages.

- Minimum releases from HAD was assumed to be 60 million m^3/day for navigation, hydropower and water quality reasons.

- The released volume should not exceed Egypt share of Nile water, which is 55.5 billion m^3/year, according to the Nile Basin Countries Agreement.

- The starting water level upstream HAD before the beginning of the year, and the target ending water level obtained from a long term forecasting model.

- Water level – Area and Water level – Storage relations.

- Tushka spillway boundary conditions, as Water level – Q relation.

Figure 7 - 3 shows the window used to change the model constraints interactively during the model run, the present values are those driven from equations presented in the previous chapter.

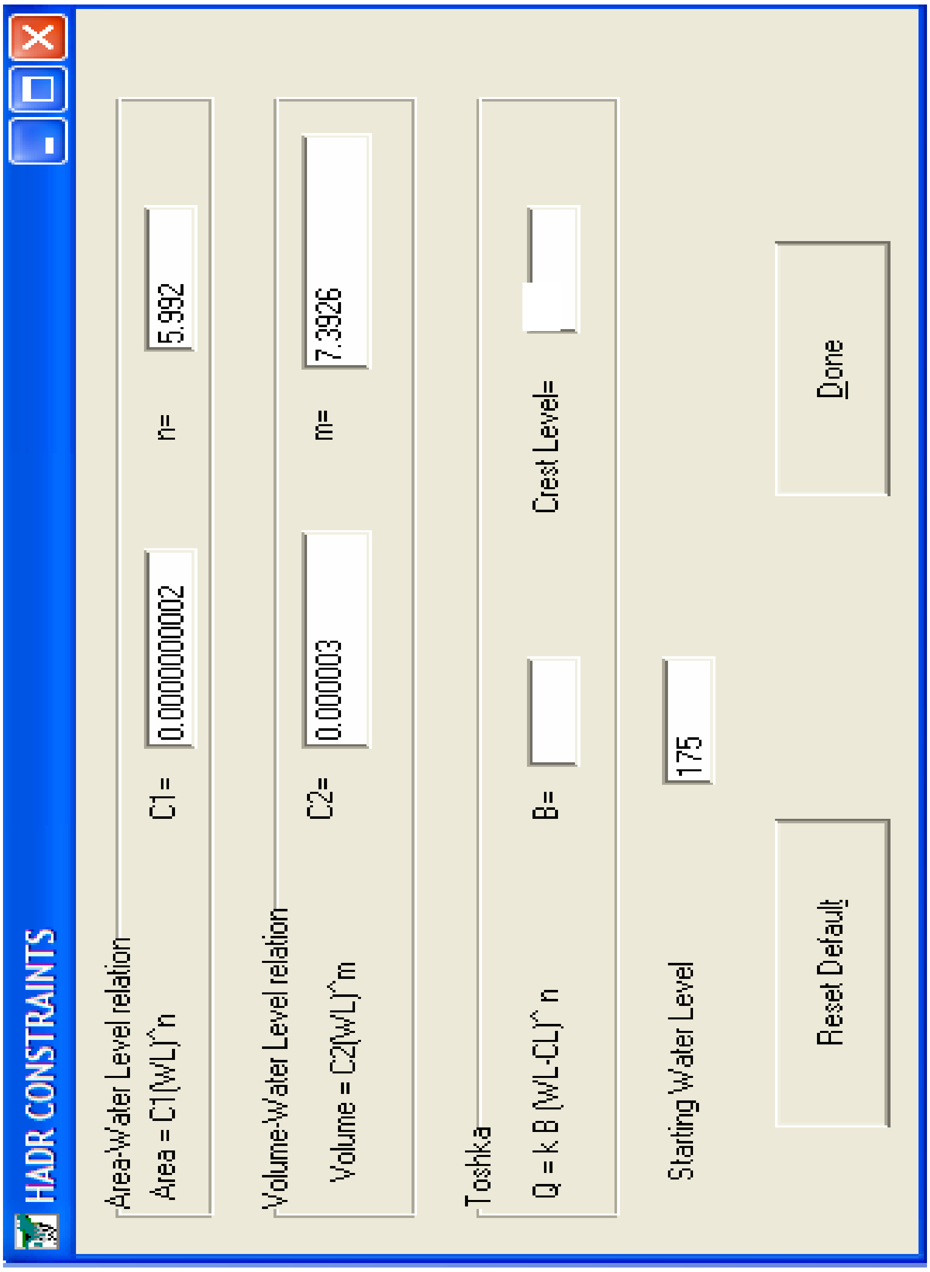

Figure 7 - 3: HADR Simulator, constraints window

Model Results

Hydrodynamic model Versus HADR Simulator

After the model was developed, a comparison between its results and those of the hydrodynamic model had to be made. The following figure shows that both results almost match. The HADR Simulator proved to be an easy and fast tool to be used in the reservoir management.

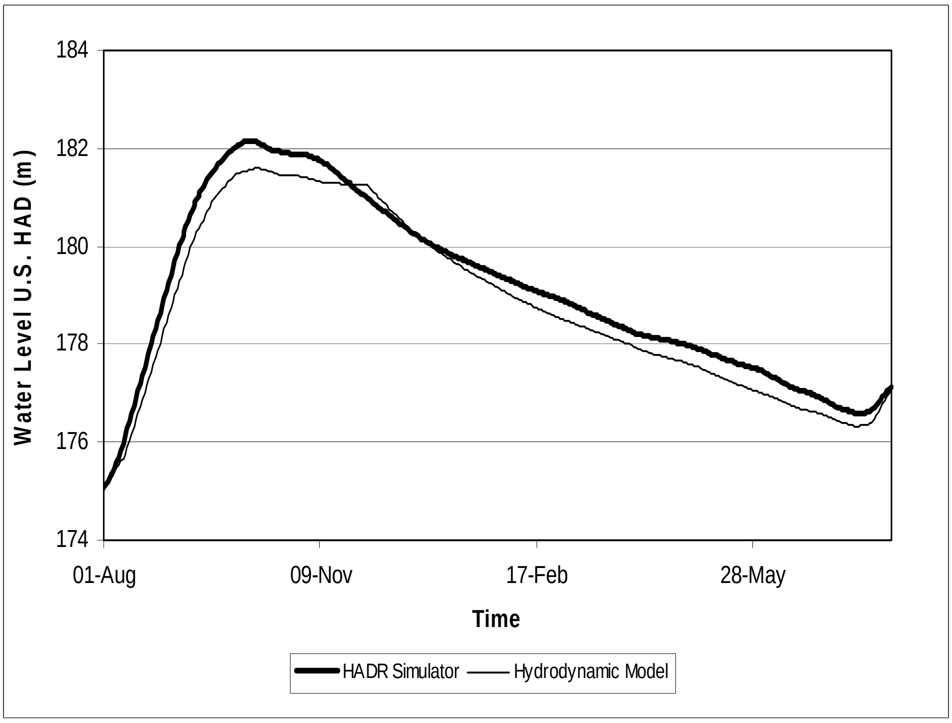

Figure 7 - 4: HADR Simulator results Vs Hydrodynamic model results

Effect of Varying the HAD Releases

The management of the reservoir flood concerns the control of the HAD releases in such a way that maximize the objective function of the model such as the water level and, or the hydropower. To test this effect, the developed model was used to calculate the water level with a two release scenarios, maximum and minimum, previously proposed by the High Aswan Dam Authority (HADA) and assuming the maximum recorded hydrograph at Donqola gagging station as the inflow at the upstream.

Figure 7 - 5 shows the results of those two scenarios from which it could be noticed that the proposed release scenario affect the water level during the period from the beginning of November to the end of April. During this period the incoming flood is in its peak while the demand is in its lowest values, as shown in Figure 7 - 6.

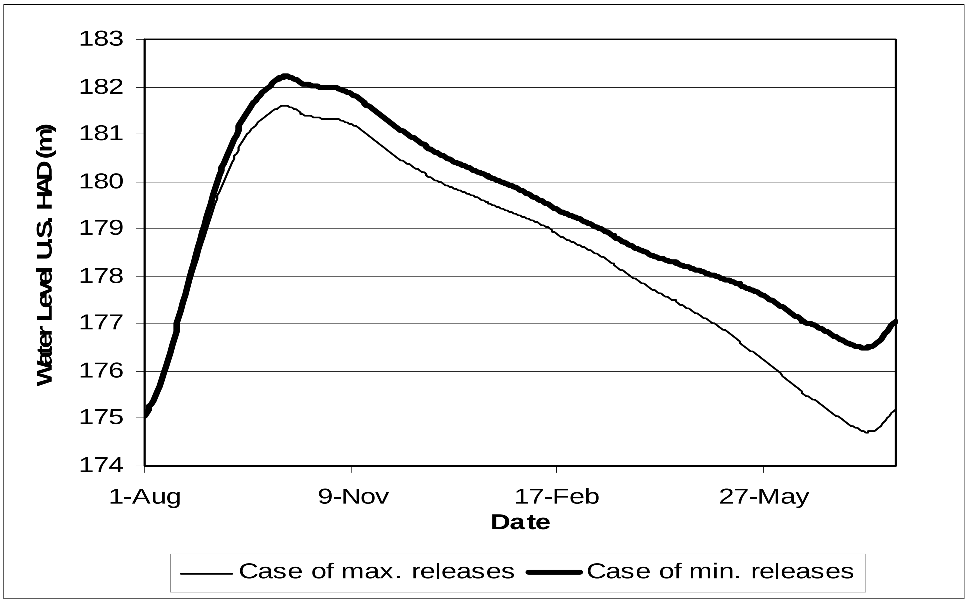

Figure 7 - 5: Water levels resulted during maximum and minimum proposed release scenarios

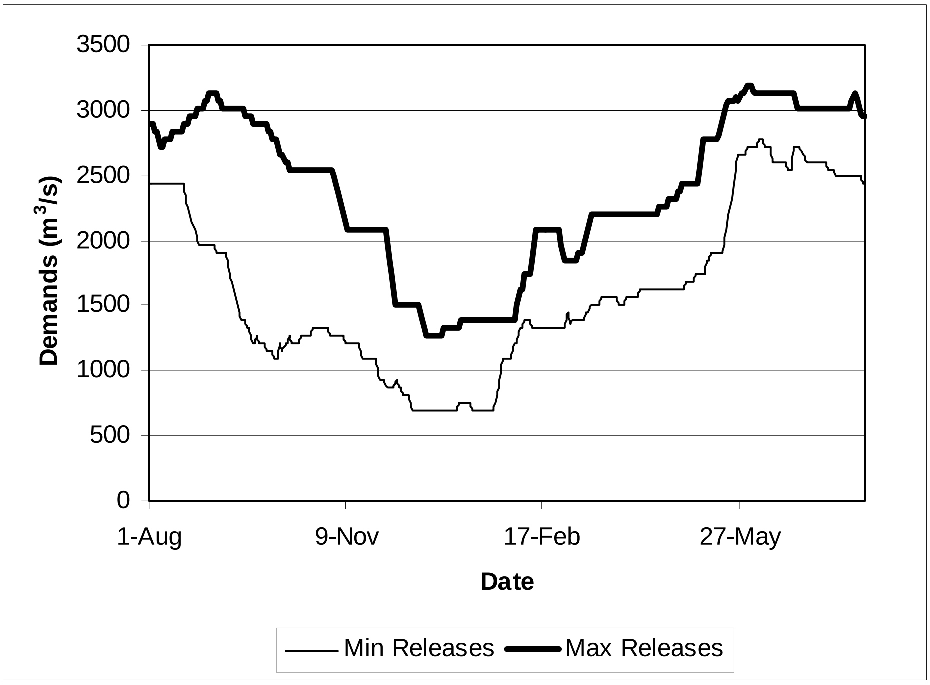

Figure 7 - 6: Maximum and minimum demands scenarios, previously proposed by the HADA (1990-2003).

Effect of Varying the Initial Water Level

The water level in the reservoir by the 1st of August, which is defined in the model as the initial water level, is the level to start the simulation with.

Various simulations were carried out to test the affect of varying this level with maximum releases to the Nile River downstream HAD and during a flood year, maximum expected flood recorded at Donqula station. The resulted water levels upstream HAD during those simulations are presented in Figure 7-7.

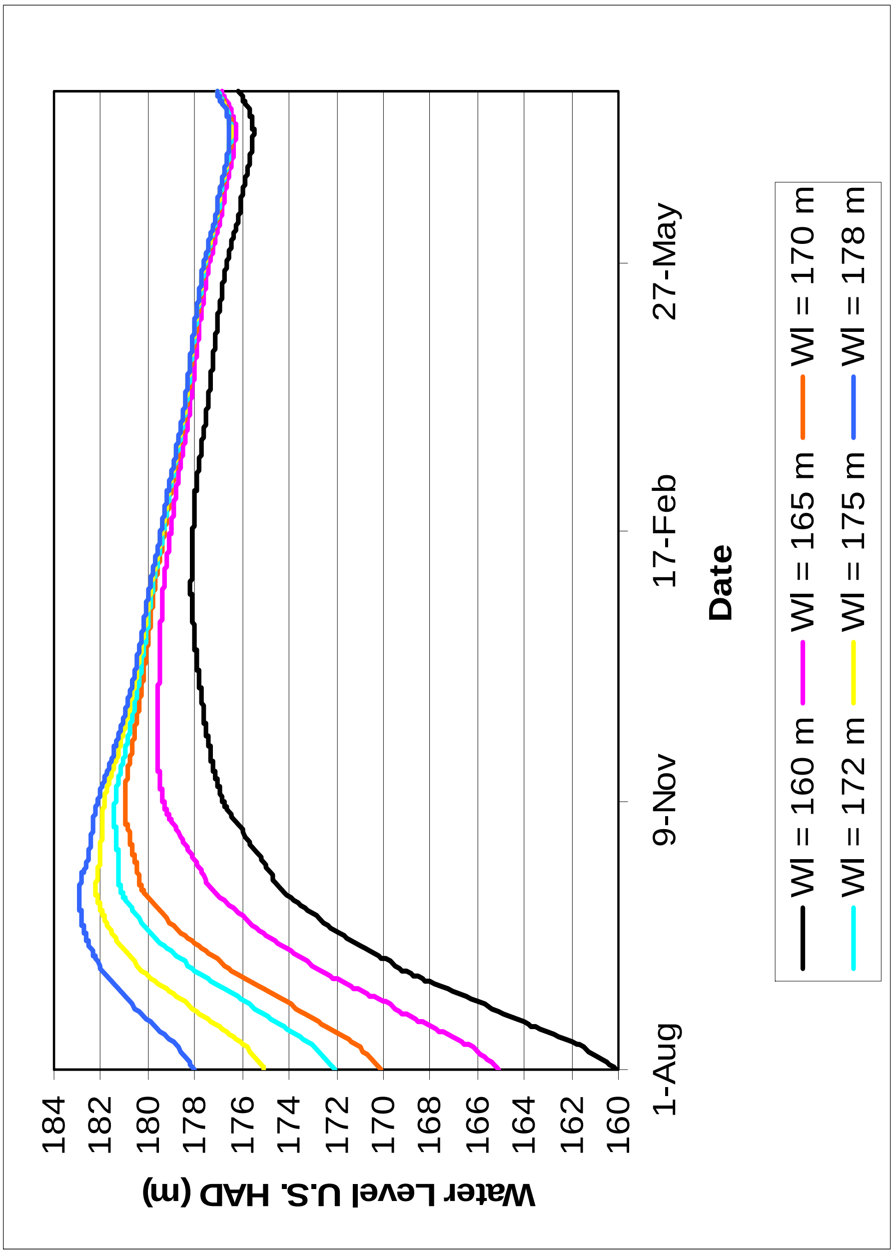

Figure 7 - 7: Water level U.S. HAD during a flood year and with minimum releases scenario

From this figure, it could be noticed that, in case of a flood year the initial starting water level will not affect the ending water level. As the lake keep filling and the change in water level due to change in storage will become slightly noticeable because the top width of the reservoir cross sections become very wide, any small change in water level will cause great variation in the total storage of the reservoir.

Almost all the proposed scenarios did not exceeded the maximum designed level of the HAD, which is 182 meter above mean sea level. Except, if we started with an initial water level of 178 m.

The ending water level, almost in all cases, will reach some where between level 176 and level 177 meter which will cause threats to the dam if two successive extreme floods is expected in two successive years.

Another set of simulations were carried out to test the affect of varying the initial water level in the reservoir with minimum releases to the Nile River downstream HAD and during a drought year. The resulted water levels upstream HAD during those simulations are presented in Figure 7-8.

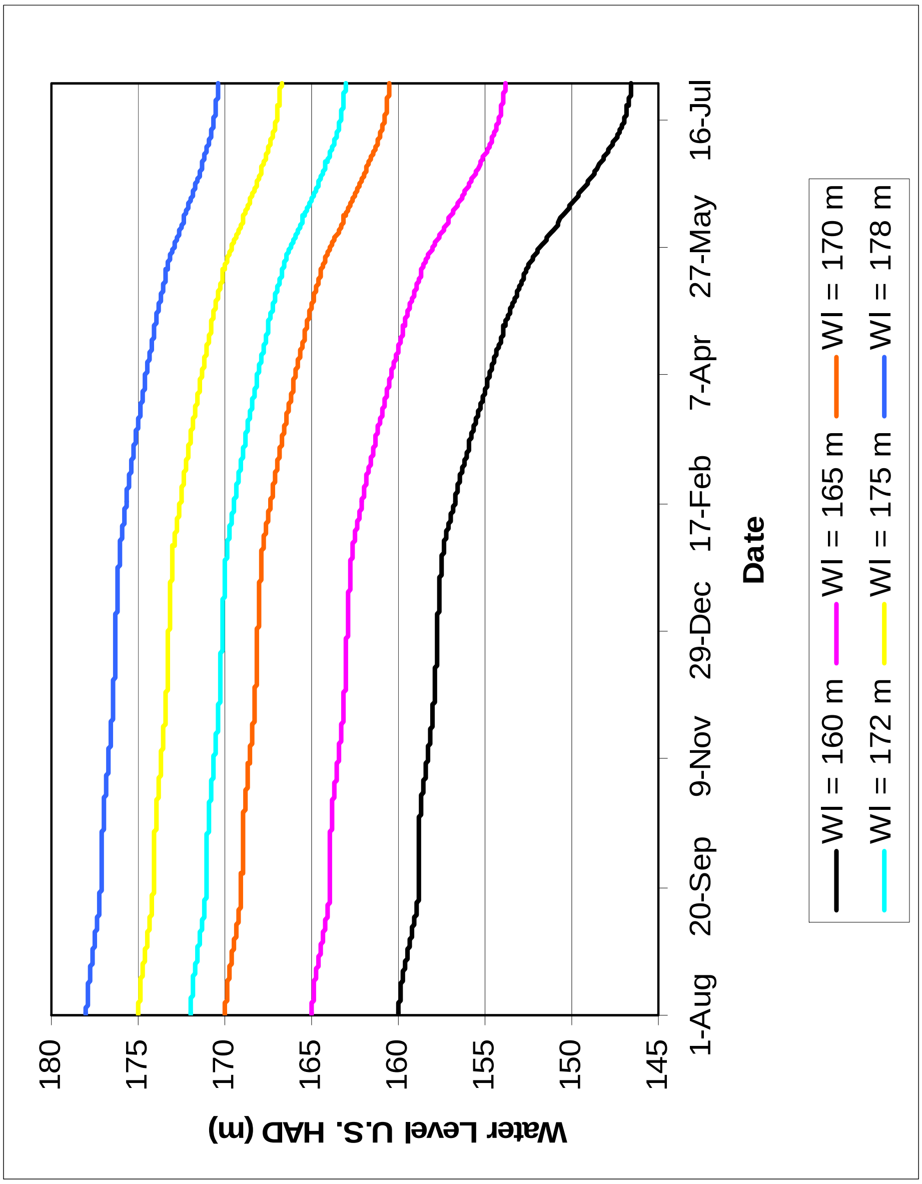

Figure 7 - 8: Water level U.S. HAD during a drought year and with minimum releases scenario

From this figure, it could be noticed that, in case of a drought year the initial starting water level will affect the ending water level noticeably. As the HAD Reservoir is releasing minimum demands to the downstream, the change in water level due to change in storage becomes noticeable, any small change in the storage will cause great variation in the water level upstream the dam.

If the proposed initial level in the reservoir is less than 170 meter this will cause great threat on the strategic storage of the water and may reduce the power generated from the turbines to a very law levels. If we started with an initial water level of 160 m or less this may cause to completely empty the live storage from the reservoirs and cause the turbines to shut down.

If two successive draught years followed each other this will threat the country economy as the water level in the reservoir will fall below the dead storage water level unless a release operation rule is suggested in such case to reduce the release flow to the downstream as much as possible.

Long Term Forecasting Model

The previous studies and figures could be used to simulate the water level variation in the reservoir. Yet, a long term forecasting model is very essential to propose a scenario for operation of the dam, from which a target ending water level will be defined according the long term forecasting.

Model Optimization

The developed model was designed to be used in an optimization process with an objective functions to maximize the water level in the reservoir and, or maximize the power output.

The Tushka spillway constraint can be modified in the calculations to change its configurations, rise and lower its crest level or increase and decrease its width. This will help in the long-term simulations and to reduce the losses of water to Tushka depression during the draught years and to release more water to it if there is a threat on the HAD.

The model begins the solution with an assumed releases derived from the pattern of the previous recorded releases from the dam with constraints of the minimum release required for navigation and water quality purposes.

The resulted water levels are stored and then another releases scenario is assumed and the resulted water levels is then used along with the previous results in the model optimization process, as described in Figure 7 - 1.

7.1 General..122
7.2 Model Development 122
7.2.1 HADR Simulator Interface 123
7.2.2 Model Constraints 126
7.3 Model Results...128
7.3.1 Hydrodynamic Versus HADR Simulator 128
7.3.2 Effect of varying the HAD releases 129
7.3.3 Effect of varying the initial water level 130
7.4 Long Term Forecasting Model...134
7.5 Model Optimization..135

Figure 7 - 1: Flow chart of HADR Simulator Model 124
Figure 7 - 2: HADR Simulator main window 125
Figure 7 - 3: HADR Simulator, constraints window 127
Figure 7 - 4: HADR Simulator results Vs Hydrodynamic model results 128
Figure 7 - 5: Water levels resulted during maximum and minimum proposed release scenarios..129
Figure 7 - 6: Maximum and minimum demands scenarios, previously proposed by the HADA. 130
Figure 7 - 7: Water level U.S. HAD during a flood year and with minimum releases scenario..131
Figure 7 - 8: Water level U.S. HAD during a drought year and with minimum releases 133

CHAPTER EIGHT

Conclusions and Recommendations

General

The study was carried out in order to develop a hydrodynamic two-dimensional model for the High Aswan Dam Reservoir. This model is to be used in several tasks such as:

- Dam operation.
- Sediment transport studies
- Water quality studies.

As the work proceeded, it was found that, using the hydrodynamic model for dam operation is a time consuming process and a hydrological model will be sufficient for this task.

Conclusions

Based on the research results the following conclusions were obtained:

1. A Geo-database for the reservoir was built containing geometric data, evaporation data with spatial variation, recorded inflow hydrographs, recorded water levels at various, sediment data, etc.

2. A three dimensional surface of the HADR bed was generated as it is a very important step for the hydrodynamic modeling.

3. A two-dimensional hydrodynamic model was developed to simulate the flow fields (steady and unsteady) of the HADR as the first 2D hydrodynamic model for the lake.

4. The hydraulic gradient in the High Aswan Dam Reservoir was simulated and it was found that it varies between $4x10^{-6}$ during low flood events (about 1000 m^3/s) and more than $2.5x10^{-5}$ during high flood events (about 7500 m^3/s).

5. Various flow rates during different water levels were applied to the model to study the wave movement through out the HADR. And it was found that the flood wave travels through the reservoir in a kinematic nature and the flood wave speed is directly proportional with the depth of water.

6. Modifications were made to the hydrodynamic model so it can be used in reservoir operation.

7. The decision makers put an operation rule to insure that the water level by the first of August will not exceed (175.00) m. By applying this rule, the maximum water level that would be reached is (181.55) m by the mid of October which is less than the maximum designed level.

8. Another proposed operation rule is to start by the first of July with a water level in the reservoir of (174.85) m. By applying this rule to the model, the maximum water level that was just reached was (182.00) m level by the mid of October.
   This will increase the strategic storage in the reservoir by 2.898 billion cubic meters than the currently proposed operation rule.

9. A modification has to be made to Tushka channel to convey a discharge of about 5071 m^3/s during high water levels.

10. A hydrological model for dam operation, HADR Simulator, was developed for the purpose of dam operation.

11. The initial water level in the reservoir before the start of the flood is very important factor to be considered during the management process, Also is the target ending water level.

12. During flood years there is almost no danger on the HAD except if we started at the 1st of August by a level more than 176 m.

13. If two successive high floods is expected the release scenario is to be modified to insure that the water level upstream the HAD will not exceed 182m.

14. During drought years the release scenario is to be modified to insure that the water level upstream the HAD will not drop in a dramatic way that threat the strategic storage in the reservoir.

15. For the purpose of dam operation a hydrological model is sufficient with a reliable long term forecasting model of the Nile basin.

Recommendations

For future studies,

1. The HAD reservoir Geo-database should be updated as this may cause some changes in the constraints of any reservoir model.

2. Further work should be done to update the bathymetry map obtained specially on the Sudanese side because of sedimentation and scouring process.

3. The data used in the calibration and verification (the velocity fields) should have a spatial representation on the reservoir map to help in the calibration process more effectively.

4. A short term forecasting model is very important for the modeling of the reservoir.

5. More accurate results will be obtained if there is a long term forecasting model to feed the operation model with the initial and target ending water levels.

6. The results of the hydrodynamic model could be used in the sediment transport and water quality studies of the reservoir.

7. Equipment for measuring spatial evaporation rates are needed to fully evaluate the evaporation over the reservoir surface.

8. More effort should be made for the study of the ground water lake interaction.

9. An operation rule for the releases of the HADR to the downstream during successive drought year should be studied.

10. Flow automatic gagging stations are required along the reservoir as the current stations which record water levels and discharge about 700 km apart, HAD station and Donqola Station.
    If intermediate stations records are available, the reservoir could be divided in the simulation process into sub reaches using those records. This will help in;
    - Reduce the processing time.
    - Reduce the time and effort taken in the calibration process.

8.1 General 137
8.2 Conclusions 137
8.3 Recommendations 140