3 (75,276 lb). Thus for this
m at the top of the pier at a lower

ytical solution buckling loads.

ad	Analytical Solution Buckling Load
	4.02 kN *(904 lb)*
)	164.2 kN *(36,924 lb)*
)	655.1 - 1658.5 kN *(147,278 - 372,856 lb)*

rated ultimate capacity of the
y is 89 kN (20,000 lb). Thus,
high factor of safety for all
that the pier system would have
well.

that for typical installations
working capacities of the
cceptable factors of safety for

for piles." *Proceeding of the*
nd Foundation Engineering,

to the linear elastic buckling
ng," thesis presented to the
in partial fulfillment of the

ersity of Missouri Research
O for their contributions to this

Proceedings of
Engr. Mechanics Div 10th annual specalty conf.
May 1995 Boulder Co.

Proceedings of
"Engineering Mechanics Division
10th Annual Specality Conference
May 1995, Boulder Co."

Simulation of the Flow Field at An Interconnected Riverine System

K.-H. Wang[1] and T.G. Cleveland[1]

Abstract

A three-dimensional, time-dependent hydrodynamic flow model was developed to simulate flood flows at the confluence of Buffalo and White Oak Bayous in Houston. This model solves full Navier-Stokes equations in a curvilinear coordinate system. A simplified moving boundary algorithm was also incorporated in the model to determine the moving fluid domain. The three-dimensional velocity field and free-surface elevation at this interconnected bayou system were computed. Numerical simulations were conducted for both existing and proposed channel geometries. Evaluation of the velocity change due to the proposed channel improvements was studied. The effects of the bayou bathymetry and existing structures on the flow field were also discussed.

Introduction

The physical behavior of an interconnected riverine system is a complex, dynamic process. It involves the three-dimensional interaction of the inflows at the confluence. The river geometries, bank roughness, and existing structures all play important roles to influence the velocity field. The flow modeling effort can provide important hydraulic information to assist in evaluation of the design of the channel improvements.

Figure 1 shows the confluence of Buffalo Bayou and White Oak Bayou relative to downtown Houston. The Harris County Flood Control District (HCFCD) is planning White Oak Bayou channel improvements by widening the base channel and smoothing the channel bank with concrete pavement. The change of flow field and the associated bank erosion impact after the channel improvements need to be examined.

In the present study, a three-dimensional, time-dependent hydrodynamic model with a moving boundary algorithm was developed to simulate flood flows at the confluence of Buffalo Bayou and White Oak Bayou. This model solves full Navier-Stokes equations in a curvilinear-grid system. The important factors, such as geometry of confluence, variable depths, bottom stresses, stream inflows and physiographic features are included. A simplified moving boundary algorithm was also incorporated in the model to update the computational domain.

The numerical simulations were conducted for both existing and proposed channel geometries. The design hydrographs of 100-year floods were used as model inputs for production runs. The fluid velocities and water surface elevations in the existing channel and proposed channel, under different inflow conditions, were generated for comparisons. Water velocities at sensitive cross sections of the bayous and the location of turbulent eddy within the confluence can be determined.

---

[1] Assistant Professor, Department of Civil and Environmental Engineering, University of Houston, Houston, TX 77204-4791

Basic equations and boundary conditions

The curvilinear three-dimensional Navier-Stokes equations were solved numerically to determine the three-dimensional velocity field and the corresponding water surface elevations. The detailed equations were given in Wang et al. (1991).

The boundary conditions used for the model are: prescribed shear stress at the fluid-solid interface (bayou bottom and banks) and prescribed flux at the inflow and outflow boundaries. The bottom stress can be computed based on a quadratic stress law. Along the flux (inflow/outflow) boundaries, the normal velocity component is prescribed at the upstream inflow boundaries, and the normal velocity component is assumed to be of zero slope at the downstream outflow boundary.

Numerical Method

The numerical approach is based on a three-time-level implicit finite-difference scheme. A staggered grid is used in both the horizontal and vertical directions of the computation domain. Mode splitting technique is applied to permit that the computation of the free-surface elevation is decoupled from the three-dimensional equations. The external mode of the flow, which is described by the free-surface elevation and the vertically integrated velocities, can be calculated from the vertically integrated governing equations by using the ADI (alternating-direction-implicit) method.

The internal mode of the three-dimensional velocity components is then evaluated by solving the full three-dimensional equations. The present code allows the vertical turbulent eddy coefficients determined from a simplified second order closure model. A system of algebraic, equilibrium equations reduced from the turbulent second-order correlations are solved to obtain the eddy viscosity for the flow simulation. The finite-difference formulations and numerical scheme were described in Wang et al. (1991).

Moving Boundary Algorithm

To study the three-dimensional flow interaction and turbulent recirculation at the confluence, a moving boundary algorithm was developed and included in the hydrodynamic model to simulate the moving fluid domain.

In order to accomplish the simulation of moving flow region, the maximal region which is expected to be wetted by the design flood is first specified. In this study, expected flow field using 100-year flood hydrographs were selected for this purpose. A physical boundary based on the existing structures and bottom elevation of 36 ft (relative to the mean-sea-water level) was defined as the maximal extent of flow. The domain inside this area is considered as possible wet area. The domain outside this boundary is a no-flow zone for the entire simulation.

The bayou bottom elevations are input in the model to evaluate the water depths. If the channel bottom is below the free- surface elevation, the water depth in the bayou domain is obtained by subtracting the elevation of the channel bottom from the surface elevation. This free-surface elevation is then used as a reference to determine whether an adjacent grid point need to be modeled as a dry cell or as a wet cell. Once the free-surface elevation rises above the bottom elevation of an adjacent dry cell along the banks, the dry cell is re-defined as a wet cell. The flow simulation then continues using the newly defined computational domain. The procedure to test the wet area is repeated and next flow boundary is generated. This moving fluid domain can be clearly observed in the velocity vector plots.

Results and Discussion

This modeling effort is to evaluate the hydraulic efficiency and flow condition for the proposed channel configuration. First, the velocity field under the inflow conditions of 100-year floods from both bayous is presented. The predicted velocity vectors in the existing channel are compared with those in the proposed channel. The contour maps

---

and geographic fea
vector plots to assist
The velocity dis
is shown in Fig. 2.
the flow is basicall
velocity magnitud
especially around th
formation of turbul
approaches the conf
Buffalo Bayou. In
similar trends as the
The surface vel
Fig. 3. From Fig.
smoother flow fiel
Bayou is no longer
uniformly. The slig
because the banks a
by the fact that the
will have greater re
The velocity v
presented in Fig.
channels, reflectin
flow at these disch
along the propose
formed right behin
(See the enlarged l
of peak flow from
the proposed Whit
conveys the flood
Two more cas
the inputs of 100-
2000 cfs from Bu
the inputs of 100-
cfs from White O
in Wang et al. (19
in the wake of the
White Oak Bayou
formed at the sou
simulations. This
Bayou plays an in
south pier of the M

Conclusions

A three-dimen
time varying velo
of Buffalo Bayo
developed and in
fluid velocities in
conditions, were
that the proposed
more smoothly.
engineering tool t

ns were solved numerically
rresponding water surface
al. (1991).
rescribed shear stress at the
bed flux at the inflow and
ased on a quadratic stress
nal velocity component is
ormal velocity component is
undary.

vel implicit finite-difference
and vertical directions of the
ermit that the computation
mensional equations. The
surface elevation and the
ertically integrated governing
method.
nponents is then evaluated
t code allows the vertical
nd order closure model. A
n the turbulent second-order
flow simulation. The finite-
Wang et al. (1991).

turbulent recirculation at the
oped and included in the

gion, the maximal region
irst specified. In this study,
e selected for this purpose. A
bottom elevation of 36 ft
ximal extent of flow. The
The domain outside this

evaluate the water depths. If
water depth in the bayou
l bottom from the surface
ence to determine whether
as a wet cell. Once the free-
adjacent dry cell along the
ation then continues using
st the wet area is repeated
d domain can be clearly

cy and flow condition for
under the inflow conditions
dicted velocity vectors in the
annel. The contour maps

---

and geographic features of streets, bridges and buildings, are also included on the vector plots to assist in physical interpretation.

The velocity distribution near the water surface of the existing channel at t = 6 hr. is shown in Fig. 2. At this early stage of simulation (rising limb of input hydrograph), the flow is basically contained in the normal stream channels. Abrupt changes in velocity magnitude and flow direction can be observed in the existing channel, especially around the bend of White Oak Bayou, where the model predicts the potential formation of turbulence recirculating eddy. The flow path of White Oak Bayou as it approaches the confluence is nearly perpendicular to the downstream flow direction of Buffalo Bayou. In general, the fluid velocities along the bayou bottom are shown similar trends as the surface fluid velocities but with smaller magnitude.

The surface velocity vector plot for the proposed channel at t = 6 hr. is presented in Fig. 3. From Fig. 3, it is found that the proposed channel configuration leads to a smoother flow field, and the potential turbulent eddy around the bend of White Oak Bayou is no longer apparent. Over an entire section, the velocity is redistributed more uniformly. The slightly larger velocities are shown along the improved channel bank because the banks are relatively smoother. The potential for increased scour is mitigated by the fact that these hydraulically smoother channels are paved with concrete, which will have greater resistance to erosion than the existing natural channel material.

The velocity vector plot in the proposed channel at t = 18 hours (high flow) is presented in Fig. 4. The model boundary is extended beyond the normal bayou channels, reflecting the increased water surface area and depth associated with flood flow at these discharges. The uniformity of the velocity field patterns is still evident along the proposed White Oak Bayou. It is interesting to note a recirculation flow is formed right behind the south pier of the main street bridge in the proposed condition (See the enlarged local velocity vector plot). It is also found that, even during the stage of peak flow from White Oak Bayou, a smoother flow pattern is still observed along the proposed White Oak Bayou. The velocity field suggests that the improved channel conveys the flood discharge more efficiently and smoothly.

Two more cases with different inflow conditions were also simulated. One is for the inputs of 100- year flood flows from White Oak Bayou and a steady base flow of 2000 cfs from Buffalo Bayou (White Oak Bayou dominant flow) and the other is for the inputs of 100- year flood flows from Buffalo Bayou and a steady base flow of 2500 cfs from White Oak Bayou (Buffalo Bayou dominant flow). The results were presented in Wang et al. (1991). It is interesting to point out that a distinct recirculation is depicted in the wake of the south pier of the Main Street Bridge at high flow in the case of the White Oak Bayou dominant flow. However, It is noted that no recirculation current is formed at the south pier of the Main Street Bridge in the Buffalo Bayou dominant simulations. This prediction indicates that the contribution of flow from White Oak Bayou plays an important role in generating recirculating currents in the wake of the south pier of the Main Street Bridge.

Conclusions

A three-dimensional hydrodynamic model that uses bayou inflows to compute the time varying velocity field and the free-surface elevation was applied to the confluence of Buffalo Bayou and White Oak Bayou. A moving boundary algorithm was also developed and included in the flow model to simulate the moving fluid domain. The fluid velocities in the existing channel and proposed channel, under different inflow conditions, were generated for comparisons. Interpretation of model results indicate that the proposed bayou modifications studied will convey White Oak flood waters more smoothly. This hydrodynamic flow model is demonstrated to be a useful engineering tool to predict the velocity field in a riverine system.

Acknowledgments

This work was sponsored by the Harris County Flood Control District under University of Houston contract 0279154.

References

Wang, K.H., Cleveland, T.G.,Rogers, J.R. and Ren, X. 1991 " A Three-Dimensional Hydrodynamic Flow and Transport Model of the Confluence of Buffalo Bayou and White Oak Bayou in Houston, Texas - Theory, Numerical Implementation, and Results" Final Report to Harris County Flood Control District. UH-CEE 91-7.

Fig. 1. Study Area - Confluence of Buffalo Bayou and White Oak Bayou.

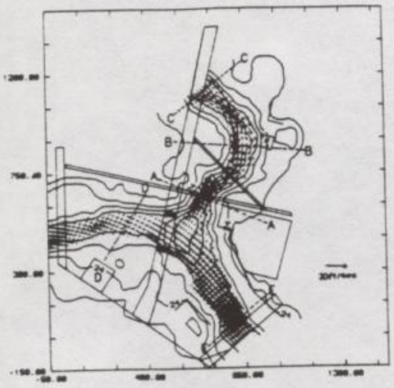

Fig. 2. Surface Velocity Field for the Existing Channel at t = 6 hours under Combined Inflows.

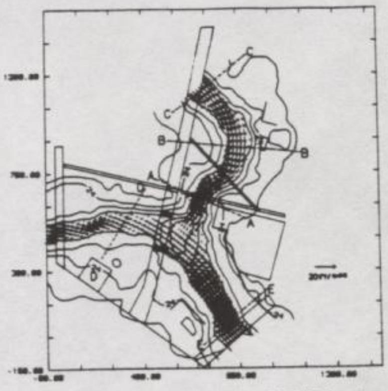

Fig. 3. Surface Velocity Field for the Proposed Channel at t=6 hours under Combined Inflows.

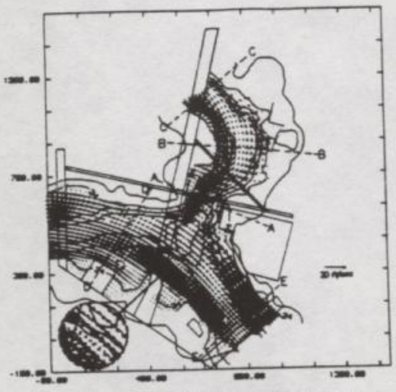

Fig. 4. Surface Velocity Field for the Proposed Channel at t = 18 hours under Combined Inflows.

Applicat

Nel

ABSTRACT

Experiments ar
beach at one end. I
measured under the
of turbulence on the
energy equations. B
the different terms i
to determine their
investigation into su

EXPERIMEN

Irregular surfac
ure 1). The horizo
were measured in th
Using the techniqu
aged energy equatio
of twelve realizatio
realization the wav
locity time series ar
to settle. Then the
realization. In this
rameters can be cal
Peak period is 1.5

---

[1] Graduate Stud
neering, Texas A&
[2] Assistant Profe
Texas A&M Unive