Ocean Engineering 146 (2017) 401–410

Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

A combined numerical-experimental approach to analyze cross flow problems in the entrance channel: A case study of Lanshan Port, China

Xing Yang [a,*], Xiangming Wang [b]

[a] Hydraulic Research Institute of Jiangsu, 97# Nanhu Rd., Nanjing, China
[b] Nanjing Hydraulic Research Institute, 223# Guangzhou Rd., Nanjing, China

ARTICLE INFO

Keywords:
Cross flow
Entrance channel
Breakwater
Numerical approach
Physical model experiment

ABSTRACT

The present study combines numerical and experimental approaches to investigate cross flow problems in the entrance channel of a harbor. The combined numerical-experimental approach is illustrated by an application to Lanshan Port (Shandong, China) where large tidal ranges and a newly planned harbor layout may give rise to strong cross flow in the entrance channel of the planned harbor. The resulting flow field with the numerical prediction shows that the layout of newly planned harbor has a major impact on the cross flow speeds in the entrance channel. This will affect the navigation safety of ship entering or leaving the harbor. A corresponding protective scheme related to breakwaters was put forward. A series of physical model experiments under the given flow provided by the numerical approach, were conducted to design the length and crest elevation of a previously proposed breakwater. Results from a series of laboratory model tests show a significant reduction of flow velocities across the entrance channel with longer breakwater length and higher crest elevation. However, a longer or higher breakwater obviously can expand the cross flow influenced zone into the entrance channel. The acceptable breakwater length and crest elevation were determined by taking account the maximum cross flow speed and cross flow influenced zone in addition to the construction cost.

1. Introduction

When encountering strong cross flow, a ship can be pushed sideway in the channel, which affects navigation safety. This leads to many studies on cross flow problems. Methodologies applied for the cross flow research such as field data collection, numerical models, and physical models have been continuously improved. To obtain accurate velocity measurements, advanced techniques have been introduced such as Laser Doppler Velocimetry (LDV), Particle Image Velocimetry (PIV), Photonic Doppler Velocimetry (PDV), Acoustic Doppler Current Profilers (ADCP) and Acoustic Doppler Velocimeter (ADV) over recent decades. Chen and Chang (2006) used a PIV system to measure the cross flow fields of a vessel model in University of Iowa's towing tank with a length of 10 m, a width of 3 m and a water of 3 m.

In physical model approaches, both prototype and laboratory experiments are the main research tools. Linke and Huesig (2000) carried out physical model experiments in a flume with scale 1: 25 for the study on effects of cross currents induced by hydraulic constructions on ship operation in inland waterways. The results show that the dimensions of the hydraulic constructions can be estimated by the physical model experiments. To analyze the influence of cross flow on ship navigation, Cao et al. (2008) carried out maneuvering tests with scales of 1: 36 and 1: 55 in a flume with a length of 40 m, a width of 5 m and a water of 0.22 m.

Besides the use of physical experiments, logistic considerations, cost constraints, and safety requirements may require the utility of numerical models as an alternative research approach to solve cross flow problems. To investigate the tidal current characteristics around the harbor entrance, Xie and Zhang (2010) established a three-dimensional numerical model by the finite difference method, time-splitting technique, and a high-order turbulence closure model. Atkins and Gaffney (1980) applied a numerical simulator to determine whether there were significant differences in the transit performance of different size ships under the same cross flow environmental conditions. Tanaka and Kijima (1993), Kijima and Takazumi (2000) proposed a prediction method to estimate the hydrodynamic force acting on a ship hull in lateral motion based on the discrete vortex method. By means of simplified vortex systems, Karasuno et al. (2001) developed a numerical model to improve the accuracy of cross flow forces in ship's pivot turning motion.

Numerical methods have relied on laboratory or field observations for validation. Seth and Charles (2010) developed a Reynolds Averaged

* Corresponding author.
E-mail address: ydaxue@163.com (X. Yang).

https://doi.org/10.1016/j.oceaneng.2017.10.006
Received 27 August 2016; Received in revised form 3 October 2017; Accepted 8 October 2017
Available online 13 October 2017

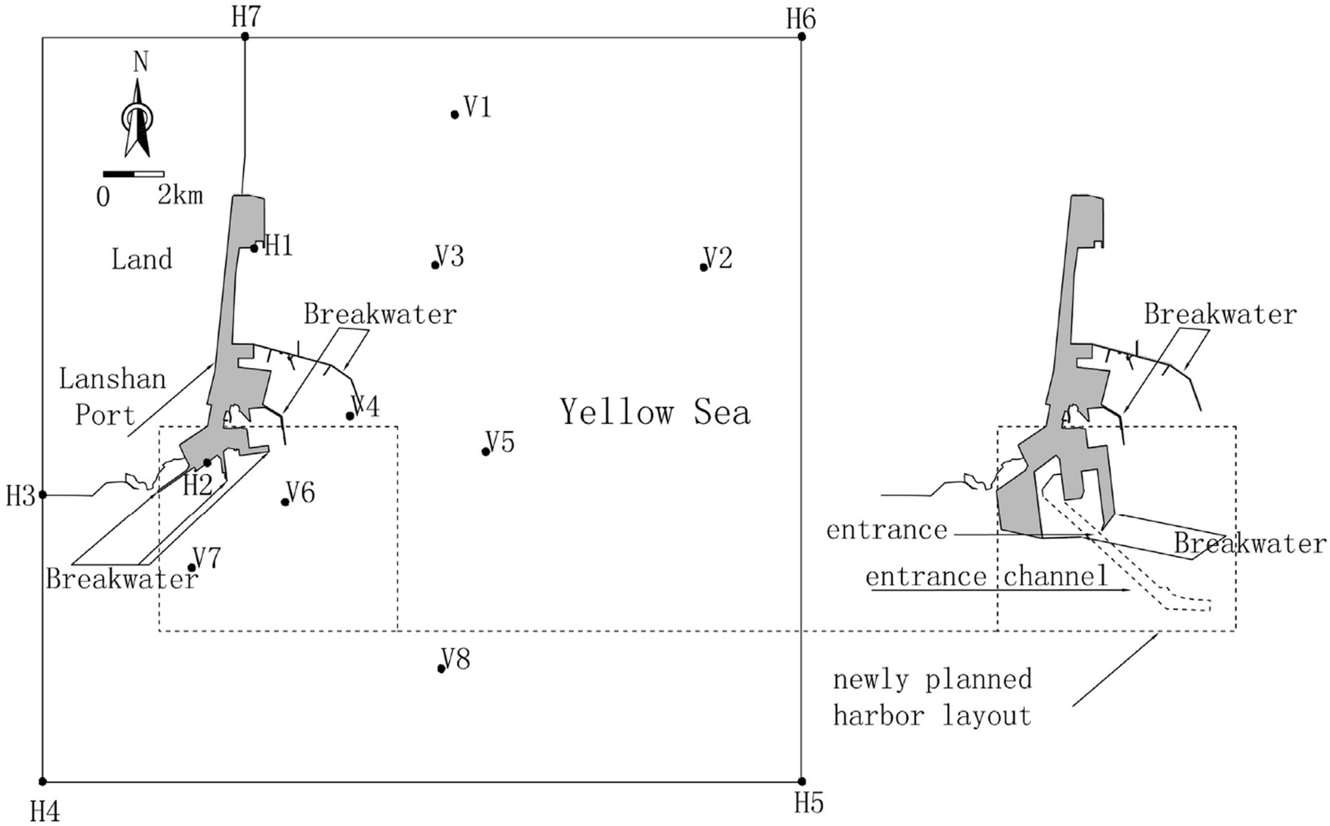

Fig. 1. Study region (left) for numerical simulation; Study region (right) for physical simulation.

Navier-Stokes (RANS) model to simulate a propeller subjected to a cross flow during ship maneuvering. Computational results were compared with the experimental data. Arslan et al. (2016) used computational fluid dynamic software with large-eddy simulation (LES) turbulence model to investigate the three dimensional unsteady flows around different ship sections with an incoming cross flow (90° drift angle). Computed pressure and drag coefficients were compared with experimental measurements.

Laboratory models have played a pivotal role and provided us with reliable design solutions for cross flow problems. However, the design and fabricate of a physical model may encounter many difficult tasks involved in designing and constructing laboratory facilities, minimizing laboratory effects, finding a reasonable model scaling and understanding the scale effects that arise from imperfect similitude. Physical modelling is usually more expensive than numerical modelling.

Numerical modelling has become more attractive because it overcomes the problems of scaling effects from which physical modelling often suffers. Higher cost and longer time involved in constructing and running a physical model can be avoided. Numerical models have many advantages, but several major weaknesses still exist. Numerical modelling can introduce its own inaccuracies and instabilities due to deficiencies in numerical methods. Longer computer run time or large memory ram will require more powerful computing machines. This is particularly true for three-dimensional (3D) numerical modelling as it requires longer computer time as the numerical solution converges more slowly than two-dimensional (2D) numerical solution.

To address the shortcomings described above, a combined numerical-experimental model approach is proposed in the present study to investigate 3D cross flow structures in the entrance channel of a harbor. By combining small-scale laboratory experiment with large-scale two-dimensional numerical model, the objectives are (1) to optimize boundary condition and layout of physical model, and (2) to reduce the experiment costs and increase the efficiency of experimental research work.

The proposed approach is illustrated by an application to Lanshan Port, Shandong Province, China. A potential cross flow problem may influence reconstruction of Lanshan Port. Accordingly, a protective scheme related to breakwaters was put forward, and physical modelling was proposed to evaluate the influence of breakwaters on the cross flow. With this aim, the combined numerical-experimental model approach is presented first, followed by the analysis of the case study is described in detail, and, finally, results are summarized and discussed.

2. Numerical and physical modelling approach description

This section provides a brief description of the proposed method. The present study investigates cross flow behaviour in entrance channel of a harbor by using small-scale physical model and prototype 2D numerical model. Physical model has been used as a reliable research tool in the growth of coastal engineering as a profession. Researchers can use it to deal with 3D complicated flow problems around coastal structures. In this proposed method, the small-scale physical model plays a central role in laboratory measurements of 3D cross flow while the prototype 2D numerical model plays a supporting role. The resulting flow field from the numerical model can provide researchers insight into the hydrodynamic regime of the near harbor region, and to optimize the physical model layout (i.e., model scope, boundary conditions). The use of numerical model can effectively reduce the cost and task of the physical model study.

3. Application example

Based on MIKE21 software (Sharbati, 2011), a 2D hydrodynamic model verified by field observations was established. The resulting flow field with the numerical prediction was used to optimize the physical model layout. A series of physical experiments, corresponding to given