FINITE ELEMENT HYDRODYNAMIC MODELING USING AN OPTIMIZED GRID FOR HARBOR PLANNING IN COAST AREAS: A CASE STUDY

Abolghasem Kamkar Rouhani
Shahrood University of Technology

kamkarr@yahoo.com

INTRODUCTION

The three finite-element, finite-difference and finite-volume numerical methods for hydrodynamic modeling of different water situations are based on solving the governing differential equations on the modeling domain grids. In general, a grid is a partition of a domain into sub-domains or subdivisions. This process of subdivision is called grid generation. Although generating a denser grid often leads to a higher resolution of the modeling system in expense of higher computational cost, this is not always the case, and thus, we have to use an optimized grid for the domain to ensure the efficiency of the modeling process (Berg et al., 1997). In this research work, hydrodynamic modeling of Weser river estuary, located at German North Sea coast, has been carried out by considering different number of nodes and triangular elements in a finite element approach using the Swevolver and xf4 software packages (2-D hydrodynamics Swevolver manual, 2004; xf4 manual, 1999). Then, the modeling results have been compared together to obtain an optimized grid for the area, in which a new harbor place for container ships should be designed based on the results.

HYDRODYNAMIC MODELING AND GRID GENERATION

Grids may be classified into structured and unstructured grids. In this research work, structured grids or elements of triangular shape in a finite element modeling approach were applied to the study area, which is shown in Figure 1. Hydrodynamic modeling simulations were made using a coarse grid (and a relatively large time step in the modeling process), and then, mesh or element refinement with various degrees was applied to the grid. The modeling simulations were, then, repeated with the new grids and different time steps in the modeling process. For this, the nodal data comprising the locations and water depths at the grid nodes and the geometric specifications of the triangular elements used in the finite element modeling approach was, first, imported in the Swevolver software package. The original data file consisted of 22448 nodes. Different time steps of 5000, 1000, 100 and 20 seconds were used in the modeling simulations. For the boundary conditions, a 26-hour time series of tidal data (2 tidal cycles) at the open sea and at the river was used. Then the modeling was carried out to calculate hydrodynamic quantities such as velocity, total depth, etc. Based on the obtained hydrodynamic quantities, a comparison between the results of various models and also between computation times for different time steps used in the modeling process, and later, the comparison for different time steps and grid nodes together in the models was made. As a general conclusion from the comparison, we have found out that although the models having more nodes and smaller time steps have taken more time for computation, the results of these models appear to be more accurate. However, to optimize the modeling time and cost, we have to consider a model having comparatively less number of nodes but relatively good results. The procedure to achieve this goal and the obtained results have been presented and discussed in the following.

Figures 2, 3 and 6 show the total depth results for the study area obtained from simulations of the original model containing 22448 nodes and time steps 5000, 1000 and 100 seconds, respectively. It seems that the total depth results for these three simulations are more or less similar. However, the results for the simulation with time step 100 seconds are found to be more accurate comparatively. Table 1 presents the computation time for of these three simulations and other models and simulations, obtained by running the simulations on a computer of type Pentium 4, 2.60 GB speed and 512 MB RAM. As indicated in this Table, the computation time for the