Computers & Fluids 115 (2015) 66–74

Contents lists available at ScienceDirect

Computers & Fluids

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

Turbulence modelling of shallow water flows using Kolmogorov approach

Jaan Hui Pu *

School of Engineering, Faculty of Engineering and Informatics, University of Bradford, Bradford DB7 1DP, UK
State Key Laboratory of Hydro-Science and Engineering, Tsinghua University, Beijing 100084, China

ARTICLE INFO

Article history:
Received 17 March 2014
Received in revised form 7 March 2015
Accepted 12 March 2015
Available online 20 March 2015

Keywords:
Kolmogorov K41 scaling law
SGUM model
SWE k–ε model
Non-uniform flow experiment
Reynolds stress
Turbulence structures

ABSTRACT

This study uses an improved k–ε coupled shallow water equations (SWE) model that equipped with the numerical computation of the velocity fluctuation terms to investigate the turbulence structures of the open channel flows. We adapted the Kolmogorov K41 scaling model into the k–ε equations to calculate the turbulence intensities and Reynolds stresses of the SWE model. The presented model was also numerically improved by a recently proposed surface gradient upwind method (SGUM) to allow better accuracy in simulating the combined source terms from both the SWE and k–ε equations as proven in the recent studies. The proposed model was first tested using the flows induced by multiple obstructions to investigate the utilised k–ε and SGUM approaches in the model. The laboratory experiments were also conducted under the non-uniform flow conditions, where the simulated velocities, total kinetic energies (TKE) and turbulence intensities by the proposed model were used to compare with the measurements under different flow non-uniformity conditions. Lastly, the proposed numerical simulation was compared with a standard Boussinesq model to investigate its capability to simulate the measured Reynolds stress. The comparison outcomes showed that the proposed Kolmogorov k–ε SWE model can capture the flow turbulence characteristics reasonably well in all the investigated flows.

1. Introduction

In order to simulate the turbulence structures in various flow conditions, the full 3D Navier Stokes (NS) numerical models are usually used (e.g., in Liu and Garcia [1] and Bihs and Olsen [2]). There are several 3D NS modelling numerical methods discovered in the recent decades that can be used to capture the free surface flow characteristics, namely the Marker and Cell (by Harlow and Welch [3]), the Volume of Fluid (by Lin and Liu [4]), the Arbitrary Lagrangian Eulerian (by Zhou and Stansby [5]) and the Level-Set methods (by Iafrati et al. [6]). However, the numerical simulation of the 3D NS equations to resolve the flow turbulence characteristics usually demands high computational cost, which strongly restricts its application in practical engineering aspects. There are two main reasons for that: (1) turbulent flows usually involve extensive and complex spatial domain evolution with very fine numerical meshes needed, and (2) those flows usually have very unsteady numerical wave speeds and that couple with small meshing areas will limit the maximum computational time step that can be employed to achieve accurate turbulent flow results. In the view of these reasons, the search for more computationally efficient model is crucial to achieve practical turbulent characteristics representation in various water engineering applications.

In the more computationally effective 2D turbulence structures representation, some complex numerical models, such as the direct numerical simulation – DNS model [7] and large-eddy simulation – LES model [8], has been studied due to their previous success in simulating the 3D NS flows. Despite their high computational costs, their success has been restrained by the meshing control and tracking of turbulence eddies break-down, which subsequently contributed to their employment of high demanding numerical approaches. Other way to model the flow turbulence intensity or Reynolds stress in Reynolds Averaged Navier Stokes (RANS) model is by using the Reynolds stress-type model (RSM), such as the non-linear RSM model suggested by Shih et al. [9]. From the complex closure formulation of the RSM equations, it can be observed that the model is more computationally expensive than the k-type models, such as the k–ε model (refer to the studies by Rodi [10]; and more recently by Cea [11]; Jiang et al. [12] and Pu et al. [13]), and by employing the RSM model might defeat the purpose to create a computationally practical model.

* Address: School of Engineering, Faculty of Engineering and Informatics, University of Bradford, Bradford DB7 1DP, UK.
E-mail address: j.h.pu1@bradford.ac.uk

http://dx.doi.org/10.1016/j.compfluid.2015.03.010
0045-7930/

Compared to the usual way of turbulence modelling by 3D NS model, the RANS type of Shallow Water Equations (SWE) model is more numerically efficient. However, there is a challenge to implement the numerical calculation of the turbulent intensity and Reynolds stress into the SWE model due to its Reynolds decomposed feature that discounts the velocity fluctuation terms in all directions. The comparative study on various numerical models conducted by Cea et al. [14,15] has proven that the 2D depth-averaged turbulence models can combine with the SWE model to give reasonable representation to flow turbulence structures in shallow flow condition. Inspired by them, this study implemented the Kolmogorov K41 scaling model (originally suggested by Kolmogorov [16–18] and normally used to represent flow power spectrum such as in Pu et al. [13]; Nezu and Nakagawa [19] and Hunt et al. [20]) into the k–ε equations to describe a new model that can be combined with the SWE model to give efficient turbulent structures simulation. The new model is efficient because: (1) its SWE simulates only 2D flow conditions, and (2) its velocity fluctuations are not represented by any complicated strain rate or vorticity tensor (as in the normal RSM and Boussinesq approaches), but instead are computed by the Kolmogorov method.

As highlighted by the studies of Caselles et al. [21], Fernandez-Nieto et al. [22] and Xia et al. [23] using different numerical models and schemes, the numerical source terms are crucial to be treated in well-balanced manner for the complex flow modelling, especially for the turbulent flow induced from obstruction and complex geometry. Hence, a recently proposed surface gradient upwind method (SGUM) by Pu et al. [24] was used in this study to improve the numerical source term simulations by integrating them into the main upwind scheme commonly used to update the numerical flux terms. The utilised SGUM approach integrated the combined source terms from the SWE and k–ε equations into a monotonic upwind-Hancock (MUSCL-Hancock) scheme for the numerical fluxes and improved the simulation accuracy of the flow turbulence structures. To this end, the newly proposed numerical model has been improved in both its modelling approach and numerical method.

In order to verify the proposed model, both the obstruction induced flows and experimentally investigated non-uniform flows were used to compare with the model. In the obstructed flows investigation, a complex multiple-block obstructions induced flow study in literature was used to compare with the proposed model simulations. Furthermore, a laboratory experiment was also conducted under different non-uniform flow conditions to validate the presented model. Four different flow non-uniformity conditions were considered in our experiment, and multiple measurements at separate flow locations were taken for each non-uniform flow. All the flow experiments were conducted using the physical water flume facility located in the Hydraulic Laboratory at the University of Bradford (refer to Pu et al. [13] and Pu [25]).

The comparisons between the numerical, experimental as well as literature studies showed that the proposed model can simulate the flow turbulence structures reasonably well for all the investigated flow conditions. These comparisons showed that the proposed model successfully combines the k–ε and Kolmogorov approaches into the 2D SWE model to efficiently re-generate the flow turbulence structures that are lost during the Reynolds decomposition process, which it represents an important numerical modelling aspect for simulating open channel flow applications in a practical manner.

2. Shallow Water Equations (SWE) model

The SWE model is used to couple with the k–ε turbulence model in this study. Eqs. (1)–(3) present the 2D fully conservative SWE, and it is combined with the numerical flux terms from the k–ε model.

$$\frac{\partial \phi}{\partial t}+\frac{\partial \phi u}{\partial x}+\frac{\partial \phi v}{\partial y}=0 \tag{1}$$

$$\frac{\partial \phi u}{\partial t}+\frac{\partial(\phi u^2+\phi^2/2)}{\partial x}+\frac{\partial \phi uv}{\partial y}-\frac{\partial}{\partial x}\left[2v_t\frac{\partial(\phi u)}{\partial x}-\frac{2}{3}\phi k\right]-\frac{\partial}{\partial y}\left[v_t\left(\frac{\partial(\phi u)}{\partial y}+\frac{\partial(\phi v)}{\partial x}\right)\right]=g\phi(S_{ox}-S_{fx}) \tag{2}$$

$$\frac{\partial \phi v}{\partial t}+\frac{\partial \phi uv}{\partial x}+\frac{\partial(\phi v^2+\phi^2/2)}{\partial y}-\frac{\partial}{\partial x}\left[v_t\left(\frac{\partial(\phi u)}{\partial y}+\frac{\partial(\phi v)}{\partial x}\right)\right]-\frac{\partial}{\partial y}\left[2v_t\frac{\partial(\phi v)}{\partial y}-\frac{2}{3}\phi k\right]=g\phi(S_{oy}-S_{fy}) \tag{3}$$

In the equations above, the variable ϕ refers to geopotential and is given by $\phi = g\cdot h$, where h is the water depth and g is the gravitational acceleration. u and v are the depth averaged flow velocities in streamwise and lateral directions, respectively. k is the flow turbulent kinetic energy (TKE), and the depth-averaged turbulent viscosity v_t is calculated as $v_t = C_\mu k^2/\varepsilon$, where C_μ is the turbulence viscosity coefficient (used in this study as $C_\mu = 0.09$) and ε is the flow TKE dissipation rate. x, y and t denote the spatial-longitudinal, spatial-lateral and temporal domains, respectively.

In Eqs. (2) and (3), S_{ox} and S_{oy} are the bed slopes in the streamwise and lateral directions, respectively. For the friction slope of the channel S_f, they are computed as follows

$$S_{fx}=\frac{n^2u\sqrt{u^2+v^2}}{h^{4/3}},\ \text{and}\ S_{fy}=\frac{n^2v\sqrt{u^2+v^2}}{h^{4/3}} \tag{4}$$

where n is the Manning's friction coefficient.

3. Turbulence model implementation

The 2D k–ε equations coupled with the SWE model are presented below [25,26]

$$\frac{\partial \phi k}{\partial t}+\frac{\partial \phi uk}{\partial x}+\frac{\partial \phi vk}{\partial y}-\frac{\partial}{\partial x}\left[\frac{v_t}{\sigma_k}\cdot\frac{\partial(\phi k)}{\partial x}\right]-\frac{\partial}{\partial y}\left[\frac{v_t}{\sigma_k}\cdot\frac{\partial(\phi k)}{\partial y}\right]=g\cdot R_h+g\cdot R_k-\phi\varepsilon \tag{5}$$

$$\frac{\partial \phi \varepsilon}{\partial t}+\frac{\partial \phi u\varepsilon}{\partial x}+\frac{\partial \phi v\varepsilon}{\partial y}-\frac{\partial}{\partial x}\left[\frac{v_t}{\sigma_\varepsilon}\cdot\frac{\partial(\phi \varepsilon)}{\partial x}\right]-\frac{\partial}{\partial y}\left[\frac{v_t}{\sigma_\varepsilon}\cdot\frac{\partial(\phi \varepsilon)}{\partial y}\right]=\frac{\varepsilon}{k}(g\cdot C_1\cdot R_h-C_2\cdot\phi\varepsilon)+g\cdot R_\varepsilon \tag{6}$$

Each of the parameters R_h, R_k, and R_ε in Eqs. (5) and (6) can be represented as

$$R_h=\frac{v_t}{z}\left\{2\left[\frac{\partial(hu)}{\partial x}\right]^2+2\left[\frac{\partial(hv)}{\partial y}\right]^2+\left[\frac{\partial(hu)}{\partial y}+\frac{\partial(hv)}{\partial x}\right]^2\right\},$$
$$R_k=\frac{n^2g}{h^{\frac{1}{3}}}(u^2+v^2)^{\frac{3}{2}},\ \text{and}\ R_\varepsilon=\frac{C_2C_\mu n^{\frac{5}{2}}g^{\frac{5}{4}}(u^2+v^2)^2}{h^{\frac{17}{12}}} \tag{7}$$

The turbulence parameters used in Eqs. (5)–(7) are $C_1 = 1.432$, $C_2 = 1.913$, $\sigma_k = 0.990$, and $\sigma_\varepsilon = 1.290$ (refer to the study by Pu et al. [13]). By using the combination of the above k–ε turbulence model with the Kolmogorov's [16–18] law in estimating velocity fluctuations, we can compute the turbulence structures, including turbulence intensity and Reynolds stress, for the proposed model.

Adapting the derived equation from the Kolmogorov K41 scaling law used in Nezu and Nakagawa [19] and Hunt et al. [20], the streamwise velocity fluctuation can be described in our numerical computation as

$$(u')_i^N=\sqrt[3]{\frac{(\varepsilon)_i^N\cdot(L_x)_i^N}{k_L}} \tag{8}$$

where i and N represent the numerical simulation space and time steps, respectively. u' is the fluctuation of the streamwise velocity