



The University of Bradford Institutional Repository

<http://bradscholars.brad.ac.uk>

This work is made available online in accordance with publisher policies. Please refer to the repository record for this item and our Policy Document available from the repository home page for further information.

To see the final version of this work please visit the publisher's website. Access to the published online version may require a subscription.

Link to publisher's version: <http://dx.doi.org/10.1016/j.compfluid.2015.03.010>

Citation: Pu JH (2015) Turbulence modelling of shallow water flows using Kolmogorov approach. *Computers and Fluids*. 115: 66-74.

Copyright statement: © 2015 Elsevier. Reproduced in accordance with the publisher's self-archiving policy. This manuscript version is made available under the [CC-BY-NC-ND 4.0 license](https://creativecommons.org/licenses/by-nc-nd/4.0/).

1 Turbulence Modelling of Shallow Water Flows Using Kolmogorov Approach

2

3 JAAN HUI PU

4 *Lecturer, School of Engineering, Faculty of Engineering and Informatics, University of Bradford, Bradford*
5 *DB7 1DP, UK.*

6 *(Visiting Scientist, State Key Laboratory of Hydro-Science and Engineering, Tsinghua University, Beijing*
7 *100084, China)*

8 *Email: j.h.pu1@bradford.ac.uk*

9

10 **ABSTRACT**

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

26

27 *Keywords:* Kolmogorov K41 scaling law; SGUM model; SWE k- ϵ model; non-uniform flow
28 experiment; Reynolds stress; turbulence structures

29

30 **1 Introduction**

31 In order to simulate the turbulence structures in various flow conditions, the full 3D Navier Stokes
32 (NS) numerical models are usually used (e.g., in Liu and Garcia [1] and Bihs and Olsen [2]). There are
33 several 3D NS modelling numerical methods discovered in the recent decades that can be used to
34 capture the free surface flow characteristics, namely the Marker and Cell (by Harlow and Welch [3]),
35 the Volume of Fluid (by Lin and Liu [4]), the Arbitrary Lagrangian Eulerian (by Zhou and Stansby
36 [5]) and the Level-Set methods (by Iafrazi et al. [6]). However, the numerical simulation of the 3D NS
37 equations to resolve the flow turbulence characteristics usually demands high computational cost,
38 which strongly restricts its application in practical engineering aspects. There are two main reasons for
39 that: (1) turbulent flows usually involve extensive and complex spatial domain evolution with very
40 fine numerical meshes needed, and (2) those flows usually have very unsteady numerical wave speeds
41 and that couple with small meshing areas will limit the maximum computational time step that can be
42 employed to achieve accurate turbulent flow results. In the view of these reasons, the search for more
43 computationally efficient model is crucial to achieve practical turbulent characteristics representation
44 in various water engineering applications.

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

55 recently by Cea [11]; Jiang et al. [12] and Pu et al. [13]), and by employing the RSM model might
56 defeat the purpose to create a computationally practical model.

57 Compared to the usual way of turbulence modelling by 3D NS model, the RANS type of Shallow
58 Water Equations (SWE) model is more numerically efficient. However, there is a challenge to
59 implement the numerical calculation of the turbulent intensity and Reynolds stress into the SWE
60 model due to its Reynolds decomposed feature that discounts the velocity fluctuation terms in all
61 directions. The comparative study on various numerical models conducted by Cea et al. [14,15] has
62 proven that the 2D depth-averaged turbulence models can combine with the SWE model to give
63 reasonable representation to flow turbulence structures in shallow flow condition. Inspired by them,
64 this study implemented the Kolmogorov K41 scaling model (originally suggested by Kolmogorov [16-
65 18] and normally used to represent flow power spectrum such as in Pu et al. [13]; Nezu and Nakagawa
66 [19]; and Hunt et al. [20]) into the k- ϵ equations to describe a new model that can be combined with
67 the SWE model to give efficient turbulent structures simulation. The new model is efficient because:
68 (1) its SWE simulates only 2D flow conditions, and (2) its velocity fluctuations are not represented by
69 any complicated strain rate or vorticity tensor (as in the normal RSM and Boussinesq approaches), but
70 instead are computed by the Kolmogorov method.

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

79 the simulation accuracy of the flow turbulence structures. To this end, the newly proposed numerical
80 model has been improved in both its modelling approach and numerical method.

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

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

96

97 **2 Shallow Water Equations (SWE) Model**

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

$$101 \quad \frac{\partial \phi}{\partial t} + \frac{\partial \phi u}{\partial x} + \frac{\partial \phi v}{\partial y} = 0 \quad (1)$$

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

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

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

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

$$112 \quad S_{fx} = \frac{n^2 u \sqrt{u^2 + v^2}}{h^{4/3}}, \text{ and } S_{fy} = \frac{n^2 v \sqrt{u^2 + v^2}}{h^{4/3}} \tag{4}$$

113 where n is the Manning's friction coefficient.

114

115 3 Turbulence Model Implementation

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

$$117 \quad \frac{\partial \phi k}{\partial t} + \frac{\partial \phi uk}{\partial x} + \frac{\partial \phi vk}{\partial y} - \frac{\partial}{\partial x} \left[\frac{\nu_t}{\sigma_k} \cdot \frac{\partial (\phi k)}{\partial x} \right] - \frac{\partial}{\partial y} \left[\frac{\nu_t}{\sigma_k} \cdot \frac{\partial (\phi k)}{\partial y} \right] = g \cdot R_h + g \cdot R_k - \phi \varepsilon \tag{5}$$

$$118 \quad \frac{\partial \phi \varepsilon}{\partial t} + \frac{\partial \phi u \varepsilon}{\partial x} + \frac{\partial \phi v \varepsilon}{\partial y} - \frac{\partial}{\partial x} \left[\frac{\nu_t}{\sigma_\varepsilon} \cdot \frac{\partial (\phi \varepsilon)}{\partial x} \right] - \frac{\partial}{\partial y} \left[\frac{\nu_t}{\sigma_\varepsilon} \cdot \frac{\partial (\phi \varepsilon)}{\partial y} \right] = \frac{\varepsilon}{k} (g \cdot C_1 \cdot R_h - C_2 \cdot \phi \varepsilon) + g \cdot R_\varepsilon \tag{6}$$

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

$$120 \quad R_h = \frac{v_t}{z} \left\{ 2 \left[\frac{\partial(hu)}{\partial x} \right]^2 + 2 \left[\frac{\partial(hv)}{\partial y} \right]^2 + \left[\frac{\partial(hu)}{\partial y} + \frac{\partial(hv)}{\partial x} \right]^2 \right\}, R_k = \frac{n^2 g}{h^{\frac{1}{3}}} (u^2 + v^2)^{\frac{3}{2}}, \text{ and}$$

$$121 \quad R_\varepsilon = \frac{C_2 C_\mu n^{\frac{5}{2}} g^{\frac{5}{4}} (u^2 + v^2)^2}{h^{\frac{17}{12}}} \quad (7)$$

122 The turbulence parameters used in equations (5) – (7) are $C_1 = 1.432$, $C_2 = 1.913$, $\sigma_k = 0.990$, and
 123 $\sigma_\varepsilon = 1.290$ (refer to the study by Pu et al. [13]). By using the combination of the above k- ε turbulence
 124 model with the Kolmogorov's [16-18] law in estimating velocity fluctuations, we can compute the
 125 turbulence structures, including turbulence intensity and Reynolds stress, for the proposed model.

126 Adapting the derived equation from the Kolmogorov K41 scaling law used in Nezu and Nakagawa
 127 [19] and Hunt et al. [20], the streamwise velocity fluctuation can be described in our numerical
 128 computation as

$$129 \quad (u')_i^N = \sqrt[3]{\frac{(\varepsilon)_i^N \cdot (L_x)_i^N}{k_L}} \quad (8)$$

130 where i and N represent the numerical simulation space and time steps, respectively. u' is the
 131 fluctuation of the streamwise velocity (which represents turbulence intensity in x-direction), L_x is the
 132 macroscale of turbulence, and k_L is the turbulence coefficient that can be calculated by equation (9).

133 In equation (8), the numerical main-stream (streamwise) velocity fluctuation was calculated by a
 134 relationship between the numerically calculated ε and comparative parameter L_x .

$$135 \quad k_L = \left[2 / (\pi C_k) \right]^{3/2} \alpha^{5/2} \quad (9)$$

136 In equation (9), C_k is a universal constant varies from 0.45 to 0.55. α is a dimensionless parameter
 137 defined as $\alpha = L_x k_0$ in which the reciprocal k_0^{-1} is also a macroscale of turbulence, as the same as L_x .

138 Hence, this study adopted the suggestion of Nezu and Nakagawa [19] in its numerical calculations,
 139 where α was estimated to be in the order of unity. The macroscale of turbulence L_x in equation (8)
 140 can be calculated by the Kolmogorov microscale of turbulence η_k as

$$141 \quad (L_x)_i^N = 0.91 Re^{3/4} (\eta_k)_i^N \quad (10)$$

142 where

$$143 \quad (\eta_k)_i^N = \left[\frac{\nu_k^3}{(\varepsilon)_i^N} \right]^{1/4} \quad (11)$$

144 in which, Re is the Reynolds number; and ν_k is the kinematic viscosity of flow.

145 As an extension towards the mixing length theory proposed by Prandtl in 1945 to describe the
 146 dominant eddies mixing process, the turbulent kinetic energy (TKE) can be usually represented by the
 147 velocity fluctuations in different directions [19]. In our SWE model, due to the absent of vertical
 148 velocity fluctuation through depth-averaging, the numerically computed k could be estimated as

$$149 \quad (k)_i^N = \frac{1}{2} \left[(u'^2)_i^N + (v'^2)_i^N \right] \quad (12)$$

150 where v' is the lateral velocity fluctuation (turbulence intensity in y-direction). Combining the
 151 streamwise turbulent intensity calculated using Kolmogorov approach in equation (8) and the
 152 computed k from k - ε equations (5) and (6) into equation (12), we can estimate the lateral turbulent
 153 intensity as well as u-v Reynolds stress to fully study the flow turbulence structures. The Kolmogorov
 154 model used in this study computes separate streamwise and lateral turbulent intensities before
 155 combining them to calculate the u-v Reynolds stress; and by this way it provides addition turbulent
 156 intensity information to investigate the flow turbulence structures. Since the Kolmogorov model
 157 calculates separate turbulent intensities for different directions, it can also be easily converted to use in
 158 1D model or to extend to consider flow turbulence structures in 3D model with appropriate
 159 assumptions. However, the current Kolmogorov approach extension to 3D model should be done in

160 caution as the 3D consideration will increase the computational cost as compared to the current 2D
161 SWE approach, and this will hinder the practicality of the model.

162 Also worth noting that the utilised k- ϵ model in equations (5) – (6) is valid under turbulent flow
163 Reynolds number as suggested by Rodi [10] and Cea [11]. Its applications into various shallow flow
164 cases have been tested by Cea [11]; Jiang et al. [12] and Younus and Chaudhry [27] for its validity on
165 flows under different velocity and uniformity. For the Kolmogorov method used in this study, it is
166 valid to represent turbulent structures in wide open channel flows as suggested by Nezu and Nakagawa
167 [19]. Besides the method also showed stable characteristics towards the turbulent Reynolds number
168 flow case, which this stable results was showing when k_L in equation (9) was tested against Reynolds
169 number [19]. Thus from these two separate findings on the SWE k- ϵ model and Kolmogorov method,
170 this study suggests their combination to calculate the shallow water turbulence structures, which has
171 been lost through the Reynolds decomposition process.

172

173 **4 Numerical Schemes**

174 In this study, the numerical flux terms were discretized using a Godunov-type Hancock scheme. The
175 Hancock scheme was upgraded by a two-stage predictor-corrector time-stepping approach. A standard
176 Harten Lax van Leer-contact (HLLC) approximate Riemann solver was used to couple with the
177 Hancock scheme for the Riemann data reconstruction process. The slope limiter method was used in
178 the HLLC solver to ensure the space discretization scheme satisfies the flux-limiting property [28-30].
179 The source term of the proposed numerical scheme was modelled using a surface gradient upwind
180 method (SGUM) proposed by Pu et al. [24].

181 The Godunov-type scheme was used in this study as it has been proven in Toro [28] and Toro and
182 Garcia-Navarro [31] to show good capability to resolve the discontinuous condition in various shallow
183 flow applications. This criterion is important for us to model the flow turbulence with discontinuities

184 and shocks under the shallow flow condition. Besides, the combination of HLLC approximate
 185 Riemann solver with Godunov-type scheme has been proven by Toro [28] to be numerically well-
 186 fitted together by causing least spurious oscillations when compared to Reo or HLL solvers. Due to all
 187 these reasons, the full Godunov-type HLLC approximate Riemann numerical formulation is used in
 188 this study to simulate the flow turbulence in shallow water applications.

189 To ease the numerical representation, equations (1) – (3) and (5) – (6) are combined into a single
 190 vector operation as follows

$$191 \quad \frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot \mathbf{F} = \mathbf{S}_T \quad (13)$$

192 where

$$193 \quad \mathbf{U} = \begin{bmatrix} \phi \\ \phi u \\ \phi v \\ \phi k \\ \phi \varepsilon \end{bmatrix}, \mathbf{F} = \begin{bmatrix} \phi u_t \\ \phi u u_t + \frac{\phi}{2} \left(\phi + \frac{4}{3} k \right) - 2v_t \cdot \left(\nabla_x \cdot (\phi u_t) + \nabla_y \cdot \left(\frac{\phi u}{2} \right) \right) \\ \phi v u_t + \frac{\phi}{2} \left(\phi + \frac{4}{3} k \right) - 2v_t \cdot \left(\nabla_y \cdot (\phi u_t) + \nabla_x \cdot \left(\frac{\phi v}{2} \right) \right) \\ \frac{v_t}{\sigma_k} \cdot \nabla \cdot (\phi k) \\ \frac{v_t}{\sigma_\varepsilon} \cdot \nabla \cdot (\phi \varepsilon) \end{bmatrix}, \mathbf{S}_T = \begin{bmatrix} 0 \\ g\phi(S_{ox} - S_{fx}) \\ g\phi(S_{oy} - S_{fy}) \\ g \cdot P_h + g \cdot P_k - \phi \varepsilon \\ \frac{\varepsilon}{k} (g \cdot C_1 \cdot P_h - C_2 \cdot \phi \varepsilon) + g \cdot P_\varepsilon \end{bmatrix} \quad (14)$$

194
 195 \mathbf{U} , \mathbf{F} and \mathbf{S}_T in equations (13) and (14) represent the matrices for the flow conserved variables,
 196 numerical flux and source terms, respectively; u_t is the resultant velocity defined by $u_t = \sqrt{u^2 + v^2}$;
 197 and ∇ is the gradient operator that can be expressed as $\nabla = \nabla_x + \nabla_y$, where $\nabla_x = \mathbf{i} \cdot \partial / \partial x$ and
 198 $\nabla_y = \mathbf{j} \cdot \partial / \partial y$. \mathbf{i} and \mathbf{j} are the unit vectors in x- and y-directions, respectively.

199

200 4.1 Monotone Upwind–Hancock Scheme

201 In our employed monotonic upwind scheme for conservative laws (MUSCL), the data reconstruction
 202 process of the flow conserved variables vector gives [28]

$$203 \quad \mathbf{U}_{i+1/2}^L = \mathbf{U}_i - \frac{\Pi(q_i) \cdot \Delta \mathbf{U}_{i-1/2}}{2}, \text{ and, } \mathbf{U}_{i+1/2}^R = \mathbf{U}_{i+1} + \frac{\Pi(q_{i+1}) \cdot \Delta \mathbf{U}_{i+1/2}}{2} \quad (15)$$

$$204 \quad \text{where } q_i = \frac{\Delta \mathbf{U}_{i+1/2}}{\Delta \mathbf{U}_{i-1/2}}, q_{i+1} = \frac{\Delta \mathbf{U}_{i+3/2}}{\Delta \mathbf{U}_{i+1/2}}, \Delta \mathbf{U}_{i+1/2} = \mathbf{U}_{i+1} - \mathbf{U}_i, \text{ and } \Delta \mathbf{U}_{i-1/2} = \mathbf{U}_i - \mathbf{U}_{i-1} \quad (16)$$

205 where Π is the slope limiter; and q is the gradient of successive $\Delta \mathbf{U}$. The van Leer's limiter, which
 206 proposed and tested by van Leer [32] into the MUSCL scheme, has been used in equation (15). As
 207 suggested by the numerical tests of Toro [28] and Hu et al. [30], the van Leer's limiter gives the best
 208 converged result to the analytical solution as compared with various other slope limiters; hence it is
 209 employed in this study.

210 A Hancock two-stage predictor-corrector scheme was utilised to update \mathbf{U} across the time domain in
 211 the proposed explicit model. Unlike some common numerical schemes, e.g. the weighted average flux
 212 (WAF) and first-order centred (FORCE) schemes that use the Lax-Wendroff (LW) and slope limiter
 213 centred (SLIC) methods, respectively, the MUSCL-Hancock scheme reconstructs its solution through
 214 the piece-wise linear functions that depend on values extrapolated from time-evolving boundary
 215 conditions [28]. In this way, the MUSCL-Hancock scheme can achieve second order accuracy in the
 216 spatial and temporal domains while maintains its stability.

217 The predictor-corrector steps are given as [29,30]

$$218 \quad \text{Predictor Step: } \mathbf{U}_i^{N+1/2} = \mathbf{U}_i^N - \frac{\Delta t}{2\Omega_i} (\mathbf{F}_{i+1/2}^N - \mathbf{F}_{i-1/2}^N) \quad (17)$$

$$219 \quad \text{Corrector Step: } \mathbf{U}_i^{N+1} = \mathbf{U}_i^N - \frac{\Delta t}{\Omega_i} (\mathbf{F}_{i+1/2}^{N+1/2} - \mathbf{F}_{i-1/2}^{N+1/2}) \quad (18)$$

220 where Ω_i is the cell area at i step for the SWE model (Ω will be the cell volume for a 3D model).

221 The Courant-Friedrichs-Lewy stability criterion was used to ensure Δt does not exceed its maximum
 222 allowable limit, as represented below

$$223 \quad \Delta t \leq C_{FL} \left[\frac{\Omega}{|u_t \cdot \mathbf{s}| + c \cdot |\mathbf{s}|} \right] \quad (19)$$

224 where $\mathbf{s} = \mathbf{i} + \mathbf{j}$ represents the resultant normal unit vector; c is the wave celerity ($c = \sqrt{gh}$); and C_{FL}
 225 is the Courant number, which is limited by $0 < C_{FL} \leq 1$. By using smaller values of C_{FL} , the simulation
 226 accuracy will be improved; however the computational cost will increase. The combination of CFL
 227 criterion with MUSCL-Hancock scheme has been well-tested by Toro [28], hence it is adopted here. In
 228 this study, a smaller C_{FL} number of 0.6 was used for the complex multiple obstructions induced flow
 229 simulations in Section 6.1; whereas a larger C_{FL} number of 0.8 was found to give stable simulations of
 230 the non-uniform flow experiments tested in Sections 6.2 and 6.3.

231

232 4.2 Source Terms Scheme

233 An original SGUM source terms treatment scheme proposed by Pu et al. [24] was integrated into this
 234 study to simulate the combined operation of \mathbf{F} and \mathbf{S}_T in equation (13). This combination of \mathbf{F} and
 235 \mathbf{S}_T in the numerical iterations can improve the numerical accuracy to predict the flow under different
 236 turbulence conditions and it has been fully tested on various numerical benchmark problems and
 237 experimental data in Pu and Lim [33] and Pu et al. [24] under conditions with and without coupling to
 238 the k- ϵ model, respectively. In this work, the SGUM is used to improve the numerical scheme to treat
 239 the combined source terms from SWE and k- ϵ equations. By applying the SGUM approach, the
 240 MUSCL-Hancock scheme in equations (17) – (18) will be altered to

$$241 \quad \text{Predictor Step: } \mathbf{U}_i^{N+1/2} = \mathbf{U}_i^N - \frac{\Delta t}{2\Omega_i} (\mathbf{f}_{i+1/2}^N - \mathbf{f}_{i-1/2}^N) \quad (20)$$

242 *Corrector Step:* $\mathbf{U}_i^{N+1} = \mathbf{U}_i^N - \frac{\Delta t}{\Omega_i} (\mathbf{f}_{i+1/2}^{N+1/2} - \mathbf{f}_{i-1/2}^{N+1/2})$ (21)

243 where $\mathbf{f} = \mathbf{F} - \Omega \cdot \mathbf{S}_T$.

244

245 4.3 Boundary Conditions

246 The double boundary condition, tested in Hu et al. [30], is used for the proposed model, where two
 247 extra ghost-cells are utilised outside the computational space domain. There are two kinds of boundary
 248 considered, the transmissive and repulsive boundaries. Each of their corresponding boundary vectors

249 \mathbf{U}^B can be represented as

250 *Transmissive Boundary:* $\mathbf{U}^B = [\phi \ \phi u \ \phi v \ \phi k \ \phi \varepsilon]^T$ (22)

251 *Repulsive Boundary:* $\mathbf{U}^B = [\phi \ -\phi u \ -\phi v \ \phi k \ \phi \varepsilon]^T$ (23)

252 The afore-mentioned boundary conditions are updated by using

253 $\mathbf{U}_{m+1}^B = \mathbf{U}_m^B$ (24)

254 $\mathbf{U}_{m+2}^B = \mathbf{U}_{m-1}^B$ (25)

255 where m is the last space cell in the computational boundary excluding the ghost cells.

256

257 5 Experimental Model

258 An experimental study was carried out and its measured data were used to validate the proposed
 259 numerical model. The descriptions of the experimental instruments and conditions are discussed in the
 260 following sub-sections.

261

262 5.1 Experimental Instruments

263 A rectangular tilting flume, which has dimensions of length 12m, width 0.45m and height 0.50m, was
264 used in this study. The physical flume was located in the Hydraulic Laboratory at the University of
265 Bradford, where all our experimental tests had been carried out. The upstream end of the flume is
266 connected to the outlet pipe of a water pump, and its downstream end runs into a water tank. The water
267 tank collects the water at downstream end before directing it to the pump to be recirculated into the
268 flume. The flume has glasswalls and a painted steel base. An adjustable gate is located at the
269 downstream end of the flume to control the flow elevation. The flume is also equipped with a track
270 parallel to the flume base for attaching the measurement trolleys, which can be used as the Acoustic
271 Doppler Velocimeter (ADV) or vernier water gauge holder. The flume slope is controlled by a
272 mechanical screw located at the downstream side of the flume, and is equipped with a calibrated scale
273 that indicates the vertical movement of the flume. This calibrated scale allows the tilted vertical
274 distance to be determined up to an accuracy of one millimeter. For detailed experimental descriptions
275 refer to Pu et al. [13] and Pu [25].

276 The ADV used in this study is equipped with the four-probe-receiver to reduce the noise signal of the
277 measurements as compared to the conventional three-probe-receiver ADV, as the fourth probe
278 provides direct estimation to the instrumentation noise level [34]. It was suggested by Lemmin and
279 Rolland [35] using the investigation on the time-averaged flow field data that the error sources of
280 ADV measurements are generally contributed less than 4% relative error to the velocity
281 measurements. Besides, Hurther and Lemmin [36] also suggested using the investigation on turbulent
282 kinetic energy (TKE), turbulent intensity and Reynolds stress that the four-probe-receiver ADV allows
283 measurements with relative error of less than 10%. Conclusively, these studies constantly suggested
284 the reliability of the four-probe ADV in velocity and turbulence measurements.

285

286 5.2 *Experimental Conditions*

287 A summary of all the hydraulic conditions in different non-uniform flow experiments is presented in
288 Table 1. The velocity measurements were made at four separate streamwise locations (at 3m, 5m, 6m,
289 and 7m from the flume inlet). At each streamwise location, the velocity measurements were made at
290 several vertical positions. Fifteen to twenty-five vertical measurement points were used in a single
291 location depending on the flow condition (presented in Table 1). Each sampling point can have a
292 minimum sampling volume of 1mm^3 ; however for the measurement points that have low signal-to-
293 noise ratio (SNR), the sampling volume will be increased. In all Test 1 – 4, the velocity measurements
294 were conducted at the ADV sampling frequency of 100Hz for 5 minutes of the sampling time, which
295 this sampling frequency was suggested by Hurther and Lemmin [36] to be sufficient ADV output rate
296 for turbulence measurements.

297

298 **6 Results and Discussions**

299 The presented numerical model was applied to various flow applications to investigate: (1) its k- ϵ and
300 SGUM models, and (2) the proposed Kolmogorov k- ϵ model to reproduce flow turbulence. For
301 validation of the k- ϵ and SGUM models, a multiple obstructions induced flow outlined in the literature
302 was tested and compared with the proposed model simulation. Then, the non-uniform flow
303 experiments discussed before was used to investigate the combined Kolmogorov k- ϵ model and its
304 ability to reproduce the flow turbulence structures.

305

306 *6.1 Multiple Obstructions Induced Flows*

307 A multiple obstructions induced flow is used in this section to produce a combination of different
308 constructive and diffusive turbulent eddy effects to test the proposed k- ϵ SWE model. The
309 experimental measurements of this flow test were conducted by Kabir et al. [37], where the
310 dimensions and sizes of the tested obstructions in the flow are presented in Fig 1. In Kabir et al. test,

311 the full channel had a dimension of 3m length and 0.9m width. However, the channel was bounded by
312 guide-walls to constraint the flow region to 2m length and 0.3m width, where the basic schematic
313 diagram of this bounded region is presented in Fig 1. Three rectangular obstacles (one $l \times b$ and two
314 parallel $L \times c$ blocks) were stationed in the channel against the flow. By referring to Fig 1, some
315 dimensions of the channel and obstacles were fixed as: $w = b = 25\text{mm}$, $L = 200\text{mm}$, and $c = 2.5\text{mm}$.
316 In the experiment, different dimensions of g (50.0, 75.0, 100.0, 125.0, 150.0, 175.0 and 200.0mm)
317 were tested against the $l \times b$ block size with ratio of $l/b = 1.5$. The initial conditions of water depth,
318 streamwise velocity, transverse velocity were set as $h_i = 0.125\text{m}$, $u_i = 0.24\text{m/s}$, and $v_i = 0$,
319 respectively. Using the each settings of the experiment, separate simulations were run until the steady
320 state was reached and the numerical results were compared with the experimental findings.

321 The simulated flow fields of the two most extreme cases in the range of g , i.e. when $g = 50.0\text{mm}$ and
322 $g = 200.0\text{mm}$, are presented at Figs 2 and 3, respectively. In Fig 2 when $g = 50.0\text{mm}$, one can
323 observe that the turbulent eddies created at the back of $l \times b$ block causing a chaotic flow vorticity
324 behaviour as compared with Fig 3 for $g = 200.0\text{mm}$, where the turbulent eddies occur more
325 tranquilly. When the numerical tests were further run for different ratios of l/b , namely
326 $l/b = 1.0$, 0.5 , and 0.3 , we obtained different trend of change for velocity ratio at the flow inlet and
327 outlet (u_i and u_o - the points are presented in Fig 1). Figs 4(a) - (d) show the simulated results of
328 u_i/u_o at different l/b ratios with three different sets of mesh as compared to the experimental
329 measurements by Kabir et al. [37]. From this mesh refinement test, the reasonably converged results
330 has been obtained from coarse mesh (200×20) to fine mesh (800×20) simulation for all l/b ratio
331 tests.

332 In Figs 4(a) – (d), the most obvious difference between the numerical predictions and measurements
333 occurs when g/w ratio is small. When the gap g is small (since w fixed as constant), the secondary

334 currents are expected to be more significant in the space between the obstructions, i.e. at the back of
335 $l \times b$ block, compared to the flow through bigger g space. At small g , the flow features a stronger 3D
336 characteristic due to the existing of strong secondary currents, and hence the 2D SGUM-SWE model
337 could not capture it satisfactorily. Furthermore, the disagreement between the 2D SGUM-SWE model
338 with measurements are also expected to be progressively more severe if gap g becomes even smaller.
339 Apart from that, Figs 4(a) – (d) show convincingly that the presented 2D SGUM-SWE model can be
340 utilised as a reasonable tool to simulate the obstructed flow applications, due to its much lower
341 computational cost than the 3D flow models. Moreover, due to the fact that the numerical iteration to
342 resolve the turbulent eddies is usually a very time-consuming process, the SWE-type approach should
343 be seriously considered as to achieve practical engineering simulations.

344

345 6.2 *Model Validation on Flow Velocity and Turbulent Kinetic Energy (TKE)*

346 In this section, the numerical simulations are completed for the physical experiments explained in
347 Section 4, where the mesh size of 480×45 (excluding ghost cells) are found to give the most optimum
348 results. Fig 5 shows the depth-averaged streamwise velocity comparisons between the numerical
349 simulations and experimental measurements (for all Test 1 – 4 in Table 1). In the figure, one can
350 observe that Test 1 – 3 had the accelerating characteristics across the streamwise direction from 3m to
351 7m location, whereas Test 4 had the decelerating characteristics; hence, they are classified as the
352 spatial-accelerating and spatial-decelerating flows, respectively. In detail comparison from the non-
353 uniform flow depth-averaged velocity variation across the streamwise locations, Test 1 and 3 match
354 the measurements better than Test 2 and 4. However, all numerical simulations show reasonably low
355 discrepancies of about 2% with respect to the experimental data. This comparison shows that the
356 proposed simulated velocity represents the measured data well.

357 After gaining the insight of velocity profile discrepancy, in Fig 6 the depth-averaged numerical
358 simulated TKE is presented and compared with the experimental data for the tests. The plot are
359 produced against a dimensionless water height ratio h_D , which it is defined as below

$$360 \quad h_D = \frac{h - h_{MI}}{h_{MA} - h_{MI}} \quad (26)$$

361 where h_{MA} and h_{MI} are the maximum and minimum water depths across the channel, respectively. h_D
362 is a ratio to define the water-head difference in a flow system. In a spatial-accelerating flow, h_D is
363 decreasing from the upstream to downstream in the flow streamwise direction, whereas for a spatial-
364 decelerating flow h_D is increasing.

365 In Fig 6, the numerical simulations of the non-uniform flow depth-averaged TKEs at different
366 streamwise locations compare well with the experimental measurements. In comparison, the spatial-
367 decelerating flow in Test 4 shows greater energy gradient variations than all other spatial-accelerating
368 flows. This higher energy gradient is caused by the larger bed slope used in the spatial-decelerating
369 flow that creates a bigger flow pressure gradient. Also, due to the steeper bed slope used in Test 4, a
370 higher numerical discrepancy in the simulated TKEs can also be observed. In numerical terms, this
371 larger discrepancy is caused by the increased S_o source terms used in the numerical simulation.

372

373 6.3 Comparison with Boussinesq Model

374 In Figs 7 – 8, the numerical predictions of the streamwise and lateral depth-averaged turbulence
375 intensities, $\overline{u'}$ and $\overline{v'}$, respectively, are compared with the measured data for Test 1 – 4. There is no
376 comparison presented involving the vertical turbulence intensity due to the depth-averaged nature of
377 SWE. The numerical predictions of all depth-averaged streamwise and lateral turbulence intensities
378 match our experimental data reasonably. Using an eddy viscosity assumption, the Reynolds stress τ
379 can be modelled by the Boussinesq hypothesis on the diffusive momentum transport. In the well-

380 known Boussinesq model, suggested by Launder and Spalding [38] as a combination with the k-ε
381 model, each element of Reynolds stress τ_{ij} is related to the TKE and strain rate of mean flow as
382 follows

$$383 \quad \frac{\tau_{ij}}{\rho} = 2\nu_t S_{ij} - \frac{2}{3} \delta_{ij} k \quad (27)$$

384 where ρ is the flow density; $S_{ij} = 0.5(\partial u_i / \partial x_j + \partial u_j / \partial x_i)$ is the element of mean strain rate; and δ_{ij} is
385 the Kronecker delta. Numerical simulations using equation (27) is employed here to compare with the
386 proposed Kolmogorov k-ε model for reproducing the measured flow Reynolds stress. In Fig 9, we can
387 observe that both proposed and Boussinesq models simulate the measured Reynolds stress with
388 satisfactory correspondence.

389 Table 2 presents the proposed model numerical discrepancies of Test 1 – 4 by benchmarking using the
390 experimental data. The table shows that $\overline{u'}$, $\overline{v'}$ and $\tau = -\overline{u'v'}$ have the averaged discrepancies of
391 2.33%, 2.23% and 3.68%, respectively, across all tests, which they outline the accuracy of the
392 proposed model to reproduce the flow turbulence structures. In comparison, the streamwise and lateral
393 turbulence intensities show similar averaged discrepancy, which they are both lower than that of their
394 correlative Reynolds stress. However, all their discrepancies are significantly low, and that shows the
395 proposed model capability. These comparisons further strengthen the idea of combining the
396 Kolmogorov approach into the k-ε SWE model to propose a computationally cost-saving method for
397 the practical simulation of RANS turbulence structures.

398

399 **7 Conclusions**

400 A numerical model has been proposed to combine the shallow water model with k-ε equations to study
401 the flow turbulence structures. The Kolmogorov K41 scaling law was utilised to compute the flow
402 velocity fluctuations in the combined k-ε SWE model, in order to determine the flow turbulence

403 intensities in different directions and Reynolds stress. The model was also further improved in its
404 source terms numerical representation by using a SGUM approach. Literature studies and laboratory
405 flow experiments, which were performed under the non-uniform flow conditions, were used to
406 validate the proposed numerical model.

407 The comparison with literature showed that the proposed k- ϵ and SGUM models were well-combined
408 to reproduce the flow characteristics of the investigated multiple-obstructions induced flow. In our
409 experiments, the numerical and experimental comparisons were accomplished in the flow velocity,
410 TKE, streamwise and lateral turbulence intensities as well as Reynolds stress to fully investigate the
411 proposed model representation of the flow turbulence structures. Besides, a standard Boussinesq
412 model was also utilised to compare with our numerical and experimental Reynolds stress results,
413 where good agreement was observed in between one another.

414 All the comparison results showed that the presented model captured the experimental flow
415 characteristics reasonably well in all the considered flows. All of these comparisons proved that the
416 proposed k- ϵ SWE numerical model was capable to represent the actual flow turbulence structures
417 after combining with the Kolmogorov K41 scaling model to perform the computationally efficient
418 calculation in 2D. The finding of this study also proves that the Kolmogorov model should be given
419 attention by future researches as an achievable approach to resolve the computationally demanding
420 flow turbulence.

421

422 **Acknowledgements**

423 The author acknowledges the support of the Major State Basic Research Development Program (973
424 program) of China (No. 2013CB036402). The author would also like to thank the associate editor and
425 four anonymous reviewers for their insightful comments to improve the paper.

426

427 **References**

- 428 1. Liu X, Garcia MH. Three-dimensional numerical model with free water surface and mesh
429 deformation for local sediment scour. *J Hydraul Eng* 2008;134(4):203-217.
- 430 2. Bihs H, Olsen NRB. Numerical modeling of abutment scour with the focus on the incipient motion
431 on sloping beds. *J Hydraul Eng* 2011;137(10):1287-1292.
- 432 3. Harlow FH, Welch JE. Numerical calculation of time-dependent viscous incompressible flow.
433 *Phys Fluids* 1965;8:2182-2189.
- 434 4. Lin P, Liu PFL. A numerical study of breaking waves in the surf zone. *J Fluid Mech*
435 1998;359:239-264.
- 436 5. Zhou JG, Stansby PK. An arbitrary Lagrangian-Eulerian σ (ALES) model with non-hydrostatic
437 pressure for shallow water flows. *Comput Meth Appl Mech Eng* 1999;178:199-214.
- 438 6. Iafrati A, Di Mascio A, Campana EF. A level set technique applied to unsteady free surface flows.
439 *Int J Numer Meth Fluids* 2001;35:281-297.
- 440 7. Chu V. Shear instability, wave and turbulence simulations using the shallow-water equations. *J*
441 *Hydro-environ Res* 2010;3(4):173-178.
- 442 8. Talstra H. Large-scale turbulence structures in shallow separating flows. [PhD Thesis], Delft
443 University of Technology, the Netherlands; 2011.
- 444 9. Shih TH, Zhu J, Lumley JL. Calculation of wall-bounded complex flows and free shear flows. *Int J*
445 *Numer Methods in Fluids* 1996;23(11):1133–1144.
- 446 10. Rodi W. Turbulence models and their application in hydraulics. IAHR Monograph, Taylor and
447 Francis, New York; 1993.
- 448 11. Cea L. An unstructured finite volume for unsteady turbulent shallow water flow with wet-dry
449 fronts: numerical solver and experimental validation. [PhD Thesis], Universidad De La Coruna,
450 Spain; 2005.
- 451 12. Jiang CB, Yang C, Liang DF. Computation of shallow wakes with the fractional step finite
452 element method. *J Hydraul Res* 2009;47(1):127-136.

- 453 13. Pu JH, Shao S, Huang Y. Numerical and experimental turbulence studies on shallow open channel
454 flows. *J Hydro-environ Res* 2014;8:9-19.
- 455 14. Cea L, Puertas J, Vazquez-Cendon ME. Depth average modelling of turbulent shallow water flow
456 with wet-dry fronts. *Arch Comput Meth Eng* 2007;14(3):303-341.
- 457 15. Cea L, Pena L, Puertas J, Vazquez-Cendon ME, Pena E. Applications of several depth-averaged
458 turbulence models to simulate flow in vertical slot fishways. *J Hydraul Eng* 2007;133(2):160-172.
- 459 16. Kolmogorov AN. The local structure of turbulence in incompressible viscous fluid for very large
460 Reynolds number. *C R Acad Sci URSS* 1941;30:301-305.
- 461 17. Kolmogorov AN. On the degeneration of isotropic turbulence in an incompressible viscous fluid.
462 *C R Acad Sci URSS* 1941;31:538-541.
- 463 18. Kolmogorov AN. Dissipation of energy in isotropic turbulence. *C R Acad Sci URSS* 1941;32:19-
464 21.
- 465 19. Nezu I, Nakagawa H. Turbulent open-channel flows. IAHR Monograph, A. A. Balkema, the
466 Netherlands; 1993.
- 467 20. Hunt JCR, Eames I, Westerweel J, Davidson PA, Voropayev S, Fernando J, Braza M. Thin shear
468 layers – the key to turbulence structure? *J Hydro-environ Res* 2010;4(2):75-82.
- 469 21. Caselles V, Donat R, Haro G. Flux-gradient and source-term balancing for certain high resolution
470 shock-capturing schemes. *Comput Fluids* 2009;38:16-36.
- 471 22. Fernandez-Nieto ED, Marin J, Monnier J. Coupling superposed 1D and 2D shallow-water models:
472 source terms in finite volume schemes. *Comput Fluids* 2010;39:1070-1082.
- 473 23. Xia X, Liang Q, Pastor M, Zou W, Zhuang YF. Balancing the source terms in a SPH model for
474 solving the shallow water equations. *Adv Water Resour* 2013;59:25-38.
- 475 24. Pu JH, Cheng NS, Tan SK, Shao S. Source terms treatment of SWEs using surface gradient
476 upwind method. *J Hydraul Res* 2012;50(2):145-153.

- 477 25. Pu JH. Efficient finite volume numerical modelling and experimental study of 2D shallow water
478 free surface turbulent flows. [PhD Thesis], University of Bradford, UK; 2008.
- 479 26. Cea L, Puertas J, Vazquez-Cendon ME. Depth average modelling of turbulent shallow water flow
480 with wet-dry fronts. *Arch Comput Meth Eng* 2007;14(3):303-341.
- 481 27. Younus M, Chaudhry MH. A depth-averaged k- ϵ turbulent model for the computation of free-
482 surface flow. *J Hydraul Res* 1994;32(3):415-444.
- 483 28. Toro EF. Riemann solvers and numerical methods for fluid dynamics: a practical introduction.
484 Springer-Verlag, Germany; 1999.
- 485 29. Mingham CG, Causon DM. Calculation of unsteady bore diffraction using a high resolution finite
486 volume method. *J Hydraul Res* 2000;38(1):49-56.
- 487 30. Hu K, Mingham CG, Causon DM. A mesh patching method for finite volume modelling of
488 shallow water flow. *Int J Numer Meth Fluids* 2006;50(12):1381-1404.
- 489 31. Toro EF, Garcia-Navarro P. Godunov-type methods for free-surface shallow flows: a review. *J*
490 *Hydraul Res* 2007;45(6):736-751.
- 491 32. van Leer B. Towards the ultimate conservative difference scheme II. Monotonicity and
492 conservation combined in a second order scheme. *J Comput Phys* 1974;14:361-370.
- 493 33. Pu JH, Lim SY. Efficient numerical computation and experimental study of temporally long
494 equilibrium scour development around abutment. *Environ Fluid Mech* 2014;14:69-86.
- 495 34. Lohrmann A, Malzone C. Vectrino – Micro ADV comparison. Nortek Technical Note No. TN-
496 022; 2004:1-8.
- 497 35. Lemmin U, Rolland T. Acoustic velocity profiler for laboratory and field studies. *J Hydraul Eng*
498 1997;123(12):1089-1098.
- 499 36. Hurther D, Lemmin U. A correction method for turbulence measurements with a 3D acoustic
500 Doppler velocity profiler. *J Atmos Oceanic Tech* 2001;18:446-458.

- 501 37. Kabir MA, Khan MMK, Bhuiyan MA. Flow phenomena in a channel with different shaped
502 obstructions at the entrance. Fluid Dyn Res 2004;35:391-408.
- 503 38. Launder BE, Spalding DB. The numerical computation of turbulent flows. Comput Method Appl
504 Mech Eng 1974;3:269-289.
- 505

506

507

508

Table 1. Summary of experimental measurement conditions

Test No.	Slope ($\times 10^{-3}$)	Discharge (m^3/s)	Flow Characteristics	No. of Measuring Points in a Single Location
1	Flat	0.0270	Spatial-Accelerating	20
2	Flat	0.0315	Spatial-Accelerating	22
3	Flat	0.0360	Spatial-Accelerating	25
4	2.50	0.0315	Spatial-Decelerating	15–19

509

510

511

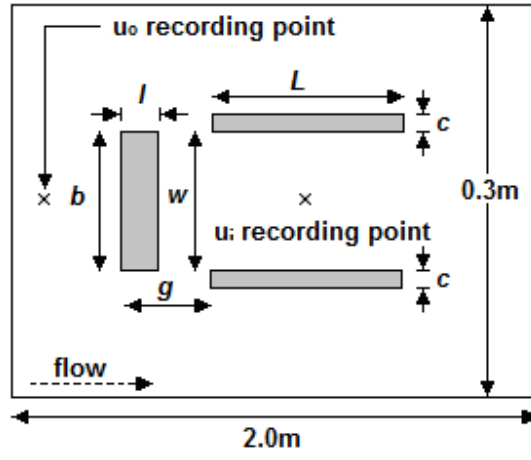
512 Table 2. Average numerical simulation discrepancies of the depth-averaged $\overline{u'}$, $\overline{v'}$ and τ (benchd by
513 experimental measurements) in Figures 7 – 9 (for Test 1 – 4)

514

Turbulence Structures	Averaged Numerical Discrepancies (in %)			
	Test 1	Test 2	Test 3	Test 4
$\overline{u'}$	2.3	2.4	2.1	2.5
$\overline{v'}$	2.6	2.5	2.0	1.9
$\tau = -\overline{u'v'}$	2.9	4.1	4.0	3.7

515

516



517

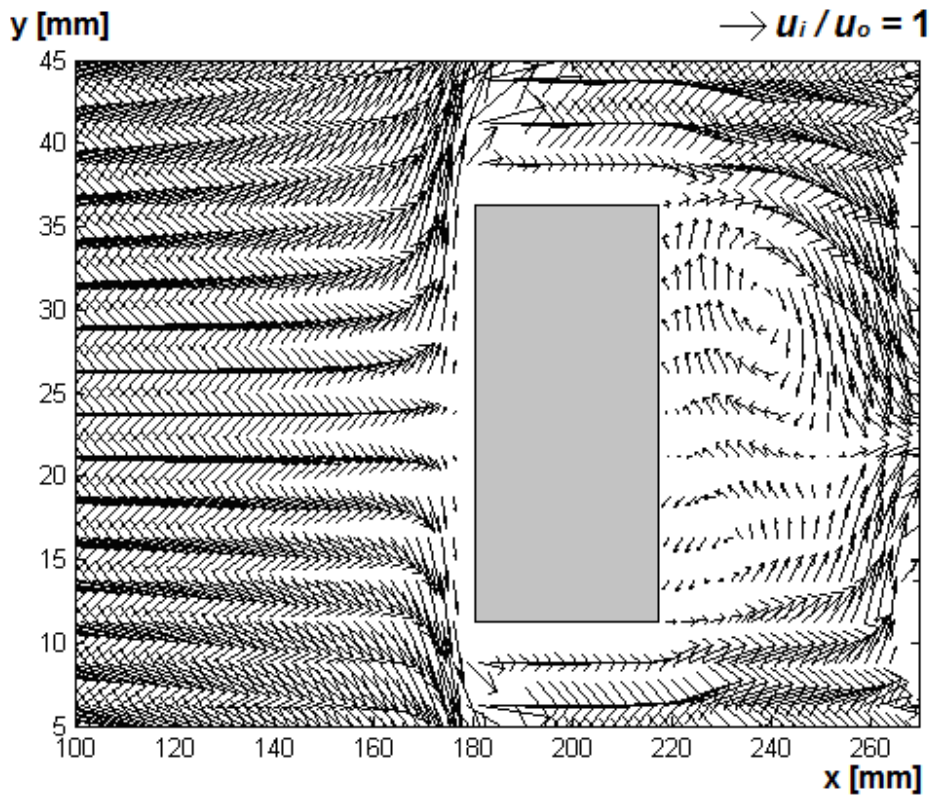
518

Figure 1. Layout of multiple obstructions induced flow

519

520

521



522

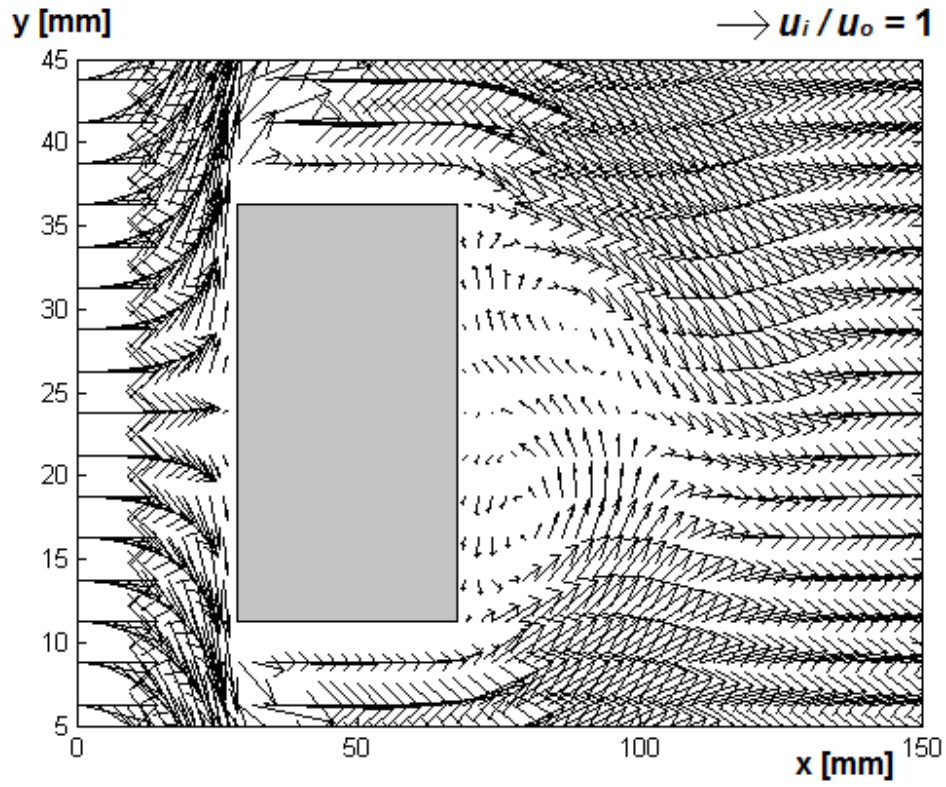
523

Figure 2. Flow field around $l \times b$ block at dimensions $g = 50.0\text{mm}$ and $l/b = 1.5$

524

525

526



527

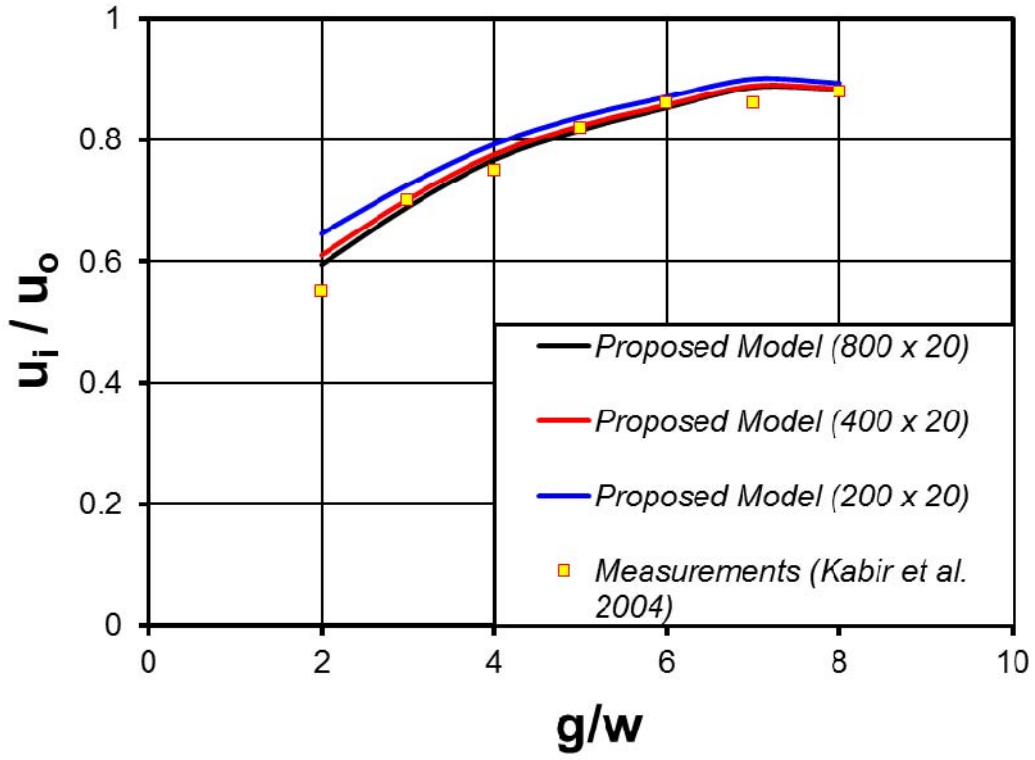
528

Figure 3. Flow field around $l \times b$ block at dimensions $g = 200.0\text{mm}$ and $l/b = 1.5$

529

530

531

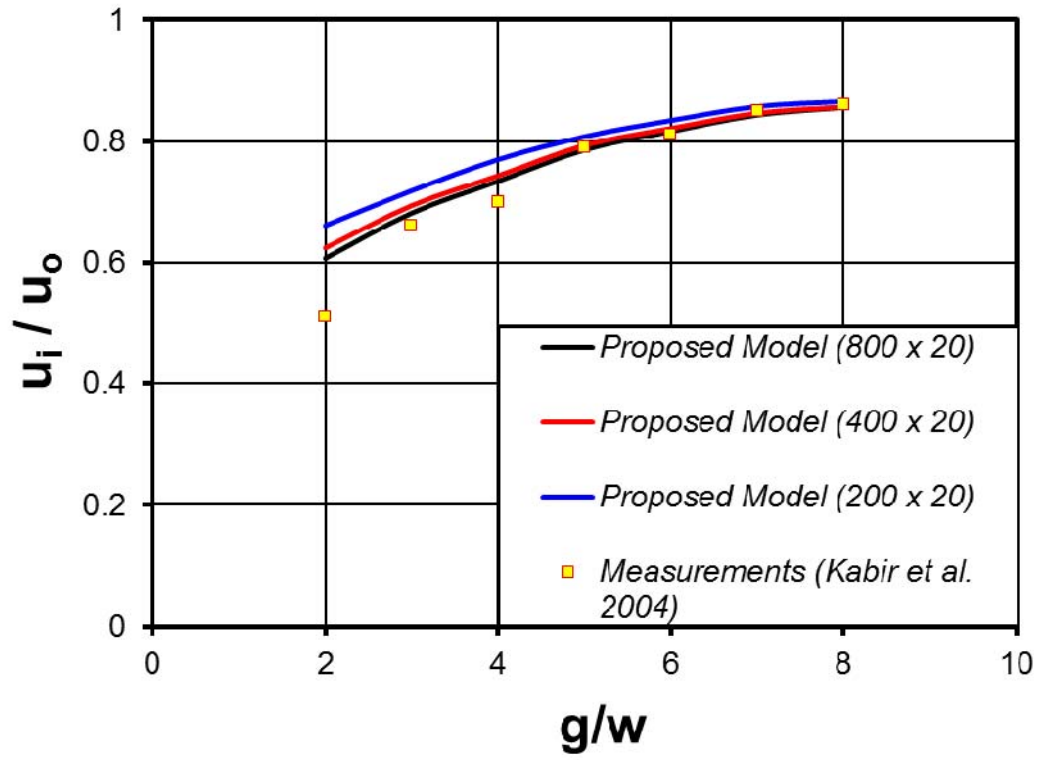


532

533

Figure 4(a). u_i / u_o comparison when $l/b = 1.5$

534



536

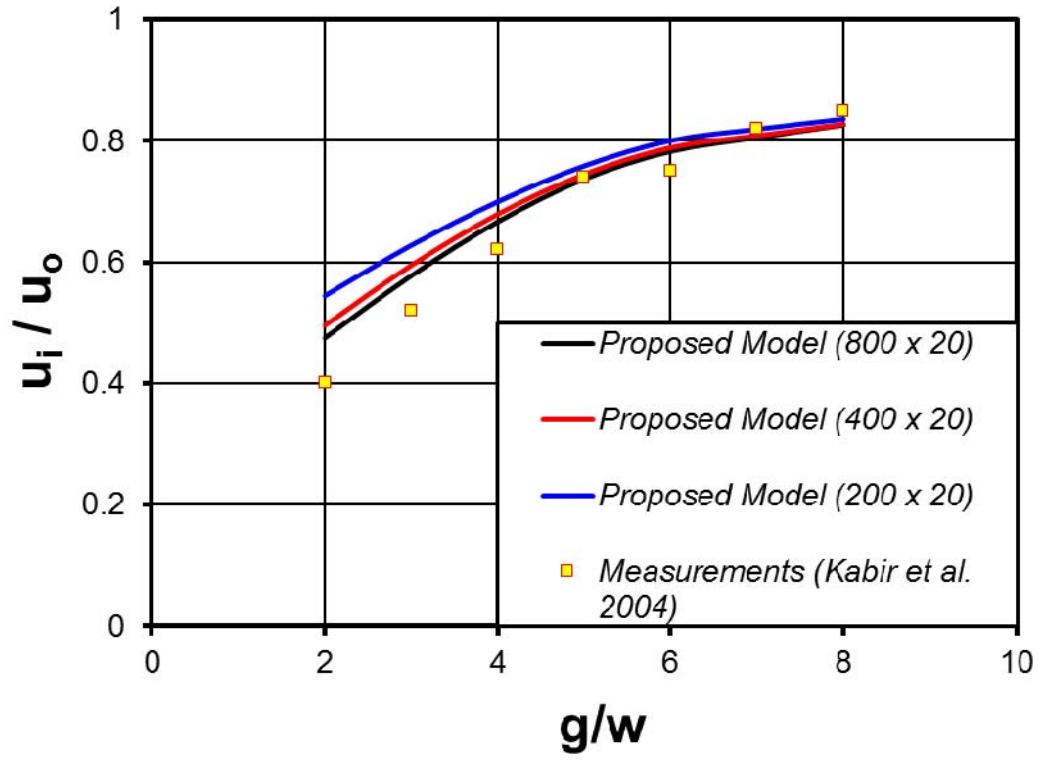
537

Figure 4(b). u_i/u_o comparison when $l/b=1.0$

538

539

540



541

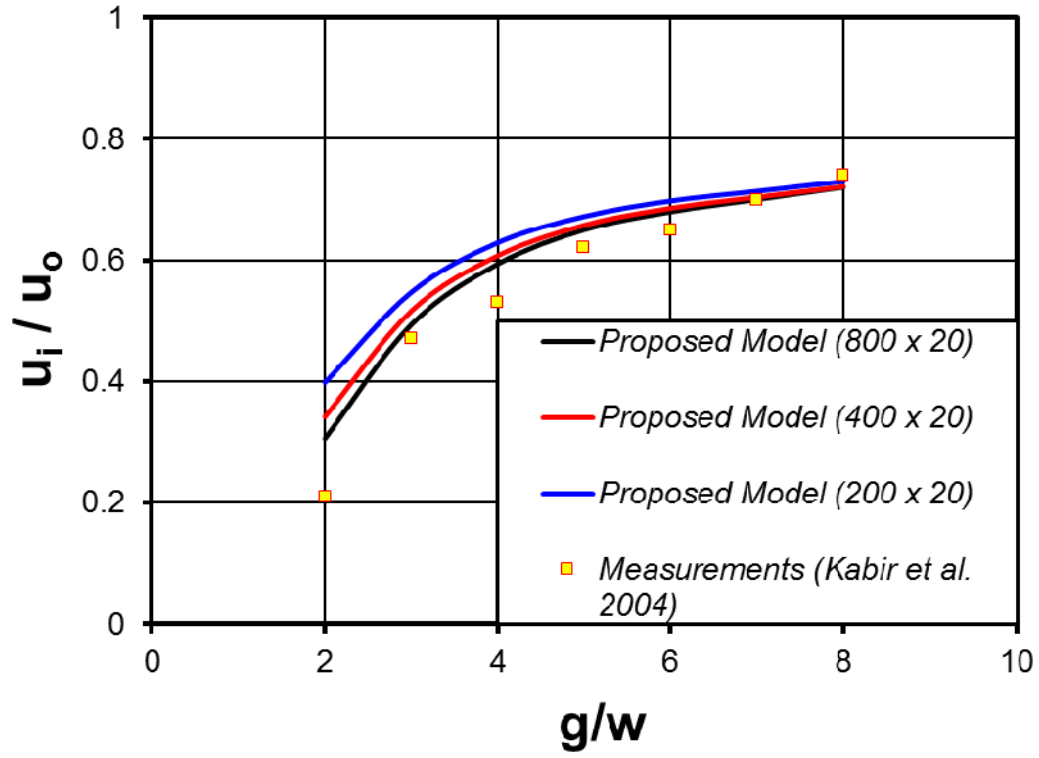
542

Figure 4(c). u_i / u_o comparison when $l/b = 0.5$

543

544

545



546

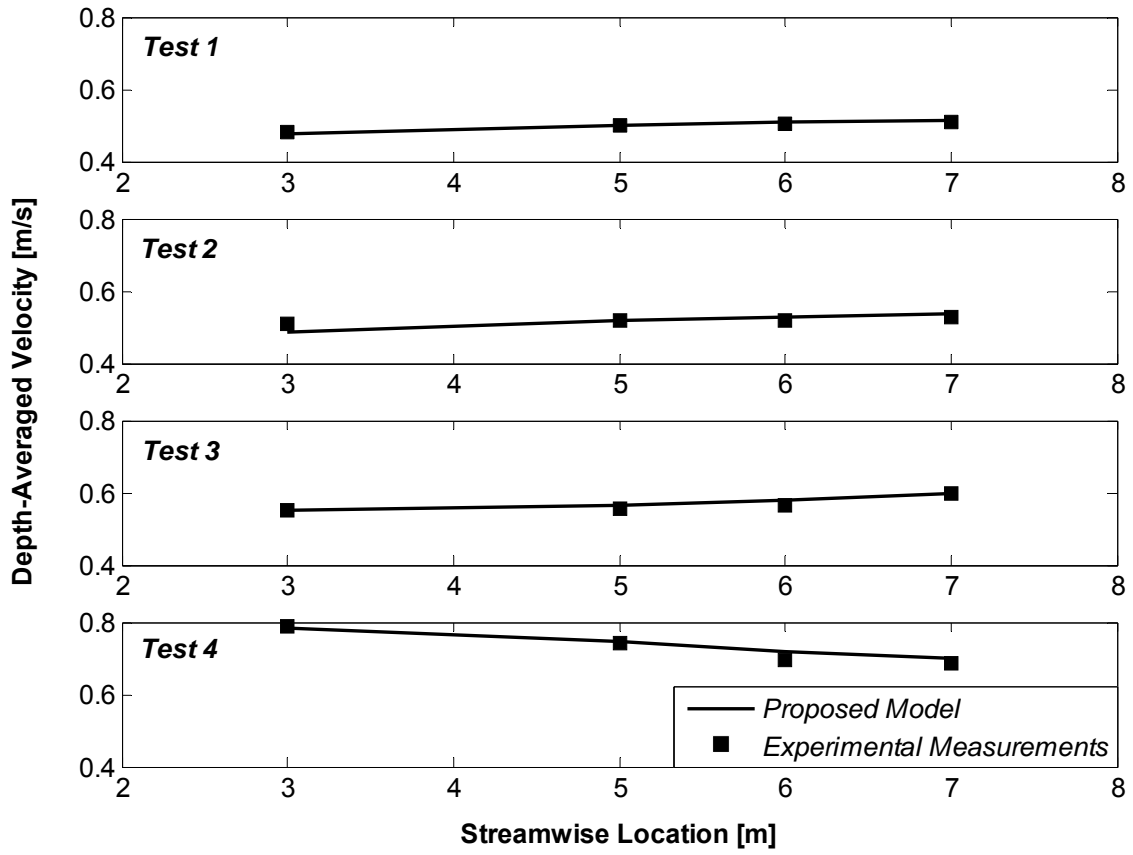
547

Figure 4(d). u_i / u_o comparison when $l/b = 0.3$

548

549

550



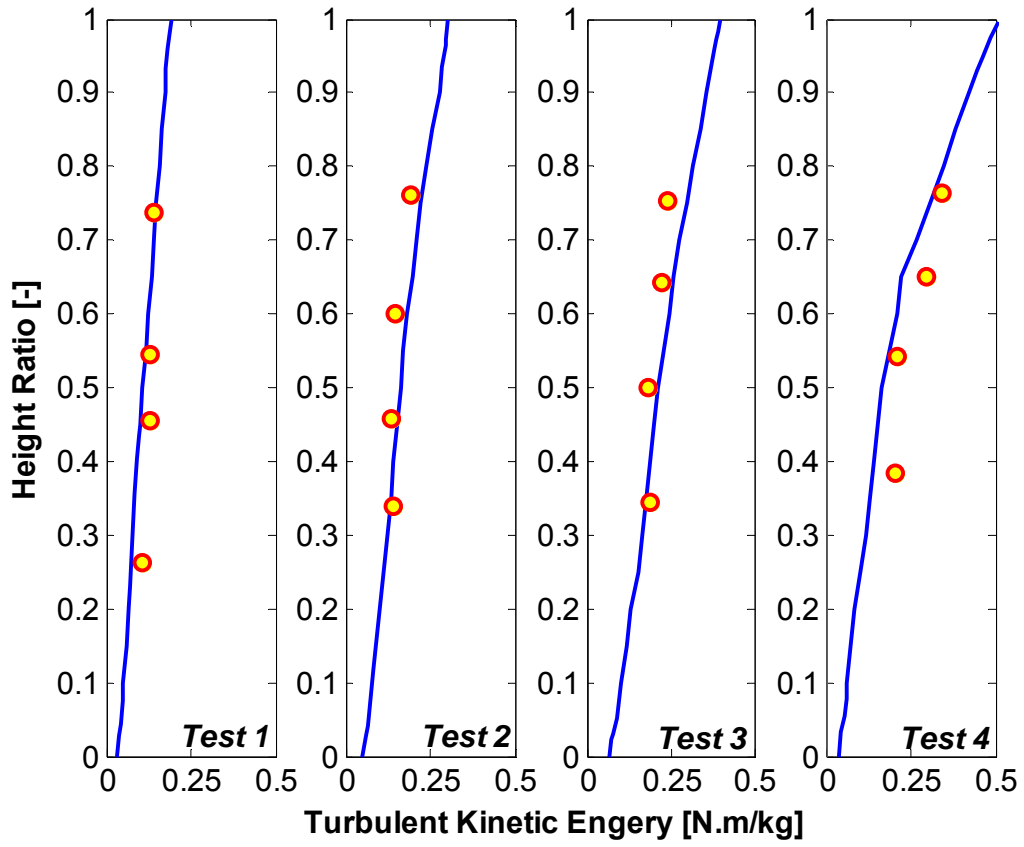
551

552

553 Figure 5. Comparison of numerical simulated and experimental measured depth-averaged velocities for
554 different tests

555

556



558

559

560

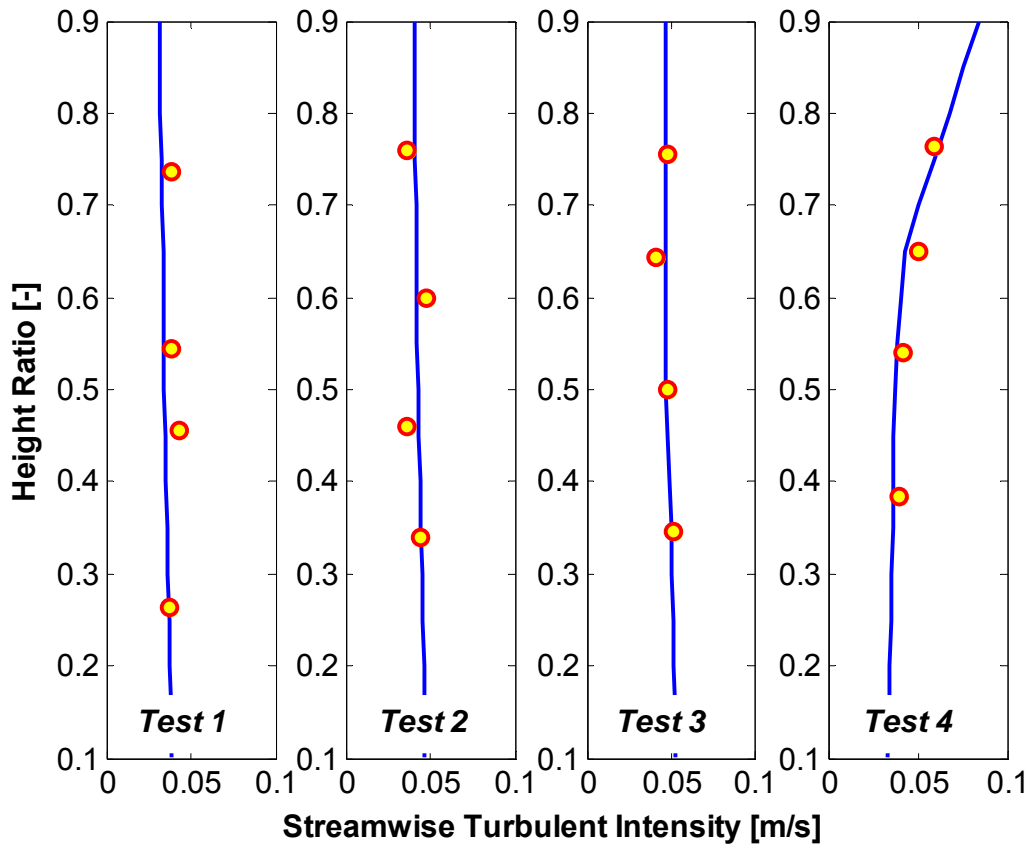
Figure 6. TKE against the water height ratio h_D (symbol – experimental data, blue solid line –

561

proposed numerical simulations)

562

563



565

566

567 Figure 7. Streamwise turbulence intensity against the water height ratio h_D (symbol – experimental

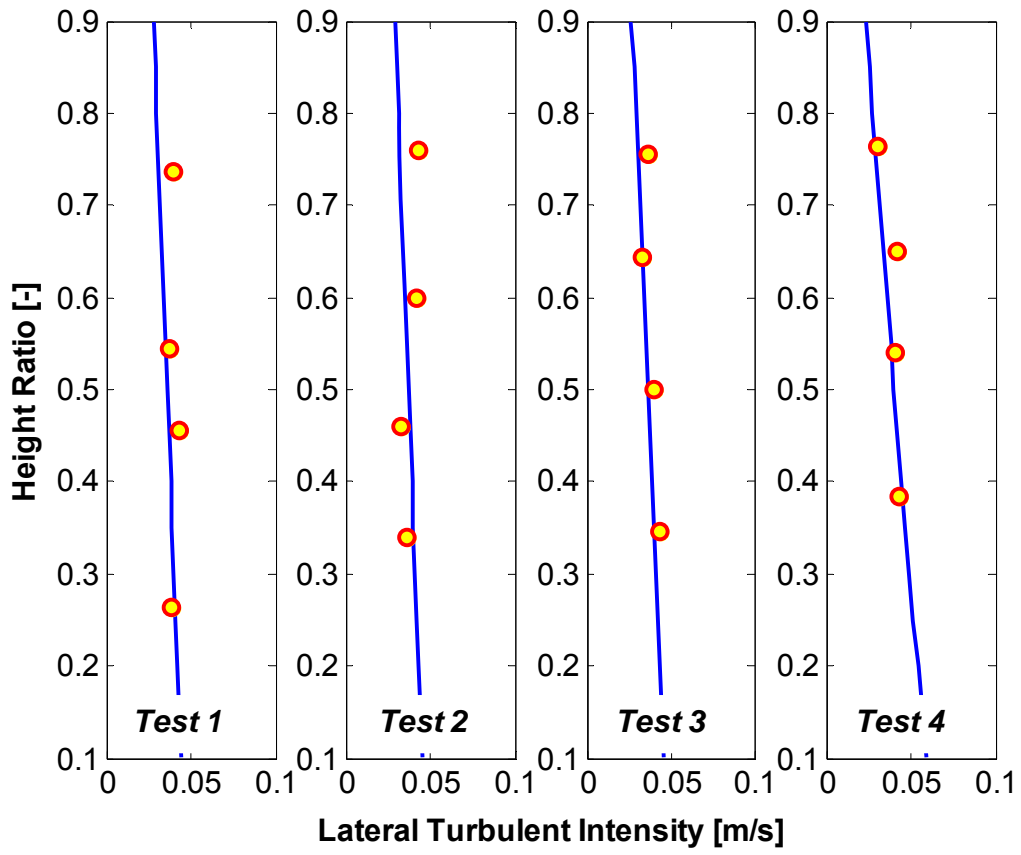
568

data, blue solid line – proposed numerical simulations)

569

570

571



572

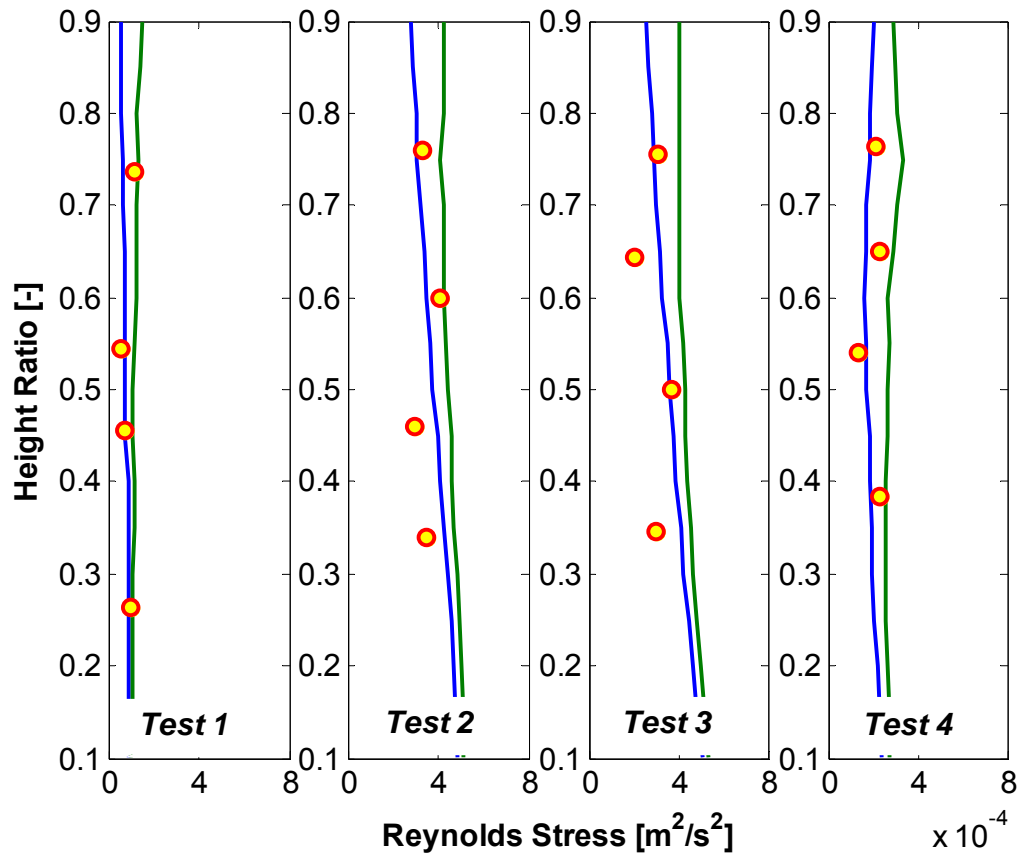
573

574 Figure 8. Lateral turbulence intensity against the water height ratio h_D (symbol – experimental data,

575 blue solid line – proposed numerical simulations)

576

577



579

580

581 Figure 9. Reynolds stress against the water height ratio h_D (symbol – experimental data, green solid

582 line – Boussinesq model simulations, blue solid line – proposed numerical simulations)

583