Applied Mathematical Modelling 37 (2013) 8652–8672

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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

Inter-comparison and validation of computational fluid dynamics codes in two-stage meandering channel flows

Ponnambalam Rameshwaran [a,*], Pamela Naden [a], Catherine A.M.E. Wilson [b], Rami Malki [b], Deepak R. Shukla [c], Koji Shiono [c]

[a] Centre for Ecology and Hydrology, Wallingford OX10 8BB, UK
[b] Hydro-environmental Research Centre, Cardiff School of Engineering, Cardiff University, Queen's Buildings, Cardiff CF24 3AA, UK
[c] School of Civil and Building Engineering, Loughborough University, Loughborough LE11 3TU, UK

ARTICLE INFO

Article history:
Received 1 October 2012
Received in revised form 19 June 2013
Accepted 8 July 2013
Available online 16 July 2013

Keywords:
Computational fluid dynamics (CFD)
Floodplain
Hydraulic models
Model uncertainty
River engineering
Two-stage meandering channel

ABSTRACT

This paper presents a study in the inter-comparison and validation of three-dimensional computational fluid dynamics codes which are currently used in river engineering. Finite volume codes PHOENICS, FLUENT and SSIIM; and finite element code TELEMAC3D are considered in this study. The work has been carried out by competent hydraulic modellers who are users of the codes and not involved in their development. This paper is therefore written from the perspective of independent practitioners of the techniques. In all codes, the flow calculations are performed by solving the three-dimensional continuity and Reynolds-averaged Navier–Stokes equations with the k–ε turbulence model. The application of each code was carried out independently and this led to slightly different, but nonetheless valid, models. This is particularly seen in the different boundary conditions which have been applied and which arise in part from differences in the modelling approaches and methodology adopted by the different research groups and in part from the different assumptions and formulations implemented in the different codes. Similar finite volume meshes are used in the simulations with PHOENICS, FLUENT and SSIIM while in TELEMAC3D, a triangular finite element mesh is used. The ASME Journal of Fluids Engineering editorial policy is taken as a minimum framework for the control of numerical accuracy. In all cases, grid convergence is demonstrated and conventional criteria, such as Y^+, are satisfied. A rigorous inter-comparison of the codes is performed using large-scale experimental data from the UK Flood Channel Facility for a two-stage meandering channel. This example data set shows complex hydraulic behaviour without the additional complications found in natural rivers. Standardised methods are used to compare each model with the available experimental data. Results are shown for the streamwise and transverse velocities, secondary flow, turbulent kinetic energy, bed shear stress and free surface elevation. They demonstrate that the models produce similar results overall, although there are some differences in the predicted flow field and greater differences in turbulent kinetic energy and bed shear stress. This study is seen as an essential first step in the inter-comparison of some of the computational fluid dynamics codes used in the field of river engineering.

---

* Corresponding author. Tel.: +44 1491692526.
E-mail address: ponr@ceh.ac.uk (P. Rameshwaran).

0307-904X/$ - see front matter © 2013 Published by Elsevier Inc.
http://dx.doi.org/10.1016/j.apm.2013.07.016

1. Introduction

In recent years, three-dimensional (3D) computational fluid dynamics (CFD) codes have been increasingly used in a number of river engineering applications, notably those which need distributed output from a complex flow field. There are a number of general purpose and free-surface flow 3D CFD codes available commercially and academically which can be used in river engineering. They all provide a numerical solution of the continuity and Reynolds-averaged Navier–Stokes (RANS) equations with a turbulence closure model but each code incorporates slightly different assumptions and formulations, offers different options for the numerical solution of the equations and puts different constraints on boundary conditions such as the roughness function. Despite the recent applications of CFD codes in the complex natural environment such as a meander channel, river confluences and flood flows (e.g. Bradbrook et al. [1]; Hodskinson and Ferguson [2]; Lane et al. [3] and Nicholas and McLelland [4]; Rameshwaran and Naden [5]), there has been very little effort made in inter-comparison and validation of these codes. Indeed, Rameshwaran and Naden [6] and Wilson et al. [7] compared the performance of a 2D depth-averaged code and a 3D code in the numerical simulation of flows in a meandering compound channel however, given the numerous CFD codes available and the importance of river and flood modelling, there is a growing demand for more comparative studies to be conducted.

The objective of this paper is to provide a quantitative evaluation of CFD codes by performing benchmark testing against a complex turbulent flow case. As a first step, this paper uses four of the available CFD codes – PHOENICS, FLUENT, SSIIM and TELEMAC3D. A steady state turbulent flow in a two-stage meandering channel is considered because it produces a more complex three-dimensional flow behaviour, resulting from the interaction between the floodplain flow and the main channel flow, than that in simple open channels [8,9]. All simulations were performed by different research groups who are competent hydraulic modellers and users of CFD but not involved in the development of the codes. Although, each group has tried to use a similar modelling approach, this was not always possible because of constraints embedded within each code. The performance of the 3D codes is evaluated by a rigorous comparison of results generated by each group and with the detailed experimental data obtained from the UK Flood Channel Facility (UK-FCF). The simulated results are compared in terms of streamwise transverse velocities, secondary flow, turbulent kinetic energy, bed shear stress and free surface elevation. An overall assessment of model uncertainty is also provided.

The accuracy of a CFD model of the physical system is governed by the numerical technique used to solve the governing equations and the initial and boundary conditions used to specify the problem. In recent years, several journals have adopted an editorial policy statement on numerical accuracy to improve the quality of publications (e.g. American Society of Mechanical Engineers (ASME) Journals and American Institute of Aeronautics and Astronautics (AIAA) Journals [10]). The ASME Journal of Fluids Engineering editorial policy [10] statement is considered as a minimum framework for this model inter-comparison and validation study. For natural open channel flows, Lane et al. [11] made some additional comments on these policy statements which are also considered.

2. CFD codes

The CFD codes considered in this study are PHOENICS (Version 3.5), FLUENT (Version 6.1), SSIIM (Version 1) and TELEMAC3D (Version V5P4). PHOENICS and FLUENT are commercially available general purpose CFD codes which are developed by Concentration Heat and Momentum Limited (CHAM) and Fluent Inc, respectively. SSIIM is an academic code which is developed by Professor. Nils Reidar B. Olsen and is freely available and specifically geared to river channel applications. TELEMAC3D is an open source code for free-surface flow developed by the Laboratoire National d'Hydraulique, Electricité de France (EDF). Although the non-hydrostatic version of TELEMAC3D is used in this study, it does differ from the other codes in that it solves the RANS equations for velocity and depth, rather than velocity and pressure. It is also a finite element code whereas the other three are finite volume codes. Whichever numerical code is used, a suitable mesh has to be chosen and additional assumptions have to be made regarding the boundary conditions, turbulence model and the numerical scheme used to solve the equations.

3. Experimental data

A brief description of the UK Flood Channel Facility Series B experimental set up is given below since the data were used in this investigation. The Series B programme has been described by Ervine et al. [8] and Sellin et al. [12]. Series B experiments were for the study of meandering channels with non-mobile channel beds (Fig. 1). The UK Flood Channel Facility flume is 60 m long and 10 m wide, with a maximum discharge of 1.1 $m^3 s^{-1}$. Experiments were performed in two-stage meandering channels consisting of flat floodplains with straight floodplain walls and a sinuous main channel, as shown in Fig. 2 and Table 1. The top width of the main channel was 1.2 m and the bank slopes were 45° with a bank-full depth of 0.15 m (Fig. 2b). The sinuosity of the channel was 1.374 and the longitudinal channel slope was 0.996×10^{-3}. The flow rate was measured using calibrated orifice plates. The water surface elevations were measured using digital point gauges. Detailed free-surface elevation and measurements of horizontal velocity were made in a series of cross-sections spaced along the channel under steady flow. The discharge was 0.25 $m^3 s^{-1}$ and the water depth in the main channel was 0.2 m. The flow angle was recorded by a vane connected to a rotary potentiometer and the horizontal velocity was measured using a