Computational Fluid Dynamic (CFD) Study of Scour Gap Effect around a Pipe
Abdul Ali $^1$, P. Ganesan $^1$, Shatirah Akib $^{2,*}$

$^1$ Department of Mechanical Engineering, Faculty of Engineering, University of Malaya Kuala Lumpur, Malaysia-50603.

$^2$ Department of Civil Engineering, Faculty of Engineering, University of Malaya, Kuala Lumpur, Malaysia-50603, Phone: +03-79677651

Abstract

A numerical investigation of incompressible and transient flow around circular pipe has been carried out at different five phases. Flow equations such as Navier-Stokes and continuity equations have been solved using finite volume method. Unsteady horizontal velocity and kinetic energy square root profiles are plotted using standard $k$ – $\varepsilon$ model and Large Eddy Simulation (LES) model and their sensitivity is checked against published experimental results. Flow parameters such as drag coefficient and lift coefficient are studied and presented graphically to investigate the flow behavior around an immovable pipe and scoured bed. The standard $k$-$\varepsilon$ model and LES both can predict accurate results in comparison to other turbulence models.

Keywords: CFD; unsteady flow; flow around circular pipe; turbulence models; vortex shedding; scouring; interfacial forces; vortex induced vibration (VIV).

*Corresponding author: shatirah@um.edu.my
1 Introduction

Scouring can be defined as the erosion of sand bed sediment surrounding the obstruction i.e., bridge piers and abutments when the obstruction is exposed to continuously strong flow field or flood events [1-4]. The sand bed can be undermined due to the normal flow field subjected to the obstruction under flow conditions by which its rate increases with larger flow events. In other words, scouring is basically caused when the foundation of the bed is swept away under flood conditions in which the flow around the obstruction accelerates and induces high shear stress over the seabed surface [5, 6]. The resulted reduction of the sand bed around the pier and abutment below the normal and natural river level is called the scour depth. A scour hole is a pit or void that forms as a result of the sand bed sediment removal from the river bed [7].

Cylindrical bridge piers are the mostly used hydraulic structures in coastal, offshore and river engineering. Thus, scour around bridge piers and submarines prediction is attracted by the hydraulic and ocean engineers. Local scouring surrounding the bridge piers is considered to be one of the mostly common causes of bridge pier failure [8-10]. The local scour around river hydraulic structures is a disaster mitigation of the engineering structure [11, 12]. It leaves them in unsafe conditions requiring maintenance and occasionally results in loss of life. Damage of hydraulic structure because of local scouring is a global concern, and it has been studied by many researchers experimentally and numerically for several decades such as [13, 14].

Mao [22] studied the interaction between a pipeline and erodible bed. Author observed the scour around horizontal cylinders in steady current and wave conditions, as well as with different Reynolds numbers \((Re)\), Shields parameters, and pipeline gaps. These experiments examined scour features such as shape and size of the scour hole, and the time scale of the scour formation. Later this work was further investigated by Jensen [23]. They investigated experimentally the flow around a pipeline placed initially on a flat, erodible bed at five characteristics stages of a progressive process in currents. The results showed that as the scour develops with time and space, the mean flow field and turbulence around and the forces on a pipeline undergo considerable changes. This study aims to investigate the effect of turbulence models on the flow field behavior at different five scouring phases and study the effect of scouring on flow parameters such as drag coefficient \(C_d\), lift coefficient \(C_l\).
2 Methodology

2.1 Geometrical structure and Boundary conditions

Fig. 1 shows the schematic of the two dimensional (2D) geometrical domain used in the present study along with the corresponding boundary conditions. A logarithmic velocity profile as presented in Fig. 3 is created using user-defined functions (UDF) in Fluent based on the following formulation [24].

![Figure 1: Geometrical model of computational domain and boundary conditions](image1)

![Figure 2: The grid for model calculation](image2)
\[ U_{\infty} = \frac{u}{K} \ln \frac{y}{y_0} \]  

(1)

Where \( U_{\infty} \) is an approach velocity (m/s), \( u_* \) is friction (or shear) velocity, \( u_* = \left( \frac{r}{\rho} \right)^{1/2} \) (m/s), \( y \) is water (or flow) depth (m) and \( y_0 \) is roughness height (m).

The velocity profile is applied at the inlet. The profile from the presented CFD model is compared with that from the experimental study of Dudley [25] for consistency, see Fig. 3. Zero pressure outlet boundary condition is applied at the flow exist. The water surface (top wall) is set as a symmetry boundary condition. No-slip boundary condition is applied on the pipe surface and the scour bed. Gravity acts in the negative \( y \)-direction.

![Figure 3](image-url)

**Figure 3**: Comparison of the horizontal logarithmic velocity-inlet (\( U_\infty \)) in the total water depth between present numerical investigation and experimental work of Dudley RD [25].

### 2.2 Governing equations

The continuity and the momentum equations for the present case are as given below:

**Continuity equation**:

\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \]  

(2)

**X-component of the momentum equation**:

\[ \rho \left( u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = - \frac{\partial P}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \]  

(3)
Y-component of the momentum equation:

\[
\rho \left( u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right) = - \frac{\partial p}{\partial y} + \rho g + \nu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)
\]  

(4)

2.3 Turbulence modeling

Turbulence models of two-equation \( k - \varepsilon \) and Large Eddy Simulation (LES) models are used in the present research and their results are compared with experimental data from the literature.

2.3.1 \( k - \varepsilon \) models

Two-equation \( k - \varepsilon \) models, turbulent kinetic energy \( k \) and turbulent dissipation \( \varepsilon \), are the simplest and the most widely used models among all turbulence models that aim to study the effect of turbulence in the flow. Two-equation model signifies that it includes two extra transport equations to represent turbulence properties of the flow. There are three different models that are derived from \( k - \varepsilon \) model standard \( k - \varepsilon \) model, Realizable \( k - \varepsilon \) model and Renomalization Group model (RNG). Despite of having the two general equations, these turbulence models use the different ways to calculate the principle form of the eddy viscosity equation.

2.3.2 Large Eddy Simulation (LES)

A large-eddy simulation (LES) model explicitly calculates the large-eddy field and parameterizes the small eddies. The large eddies in the atmospheric boundary layer are believed to be much more important and insensitive to the parameterization scheme for the small eddies. In large-eddy simulation (LES), the large three-dimensional unsteady turbulent motions are directly resolved while the smaller scale motions are modeled. In terms of
computational effort LES lies between RANS and DNS, and it can be expected more accurate and reliable than Reynolds-stress models for flows where large-scale unsteadiness is significant [26-30].

2.3.2.1 Smagorinsky-Lilly model

The first SGS model developed was the Smagorinsky-Lilly model, which was developed by Smagorinsky. It models the eddy viscosity as:

\[ v_r = \left( C_r A_x \right)^2 \sqrt{2S_y S_y} - \left( C_r A_x \right)^2 |S| \]  

(5)

Where \( \Delta_x \) the grid is size and \( C_r \) is a constant. This method assumes that the energy production and dissipation of the small scales are equilibrium.

2.4 Numerical methods

The commercial CFD software FLUENT 14.0 [31] which is based on Finite Volume Method (FVM) is used to solve the Large Eddy Simulations (LES) equations for an incompressible flow. The transport governing equations are discretized using the second order upwind spatial discretization method. The Pressure-Implicit with Splitting of Operators (PISO) scheme was used for the coupling of the pressure and the velocity fields. The under-relaxation factor of all the components, such as velocity components and pressure correction is kept at 0.3. The scaled residuals of 1x10^{-6} are set as the convergence criteria for the continuity and momentum equations. Transient model based on implicit scheme with a time step were used in the current numerical study. The typical wall treatment function \( y^+ (= yU_l/\nu) \) value of the first node in all turbulence model near the bed profile is less than 1.
2.5 Simulation cases

A total of 7 cases were simulated in the present study, see Table 1. In the first part of this study, the effect of different turbulence models such as standard $k$-$\varepsilon$ model and Large Eddy Simulation (LES) model on horizontal velocity and kinetic energy square root has been investigated. For studying this effect, we have adopted the domain proposed by Jensen et al.[23] to validate the results of Mao et al. [22]. This domain is of 0.5 m length and 0.1 m height with pipe diameter of 0.03 m. Turbulence models were tested for scour gap at time 0 min, 1 min, 6 min, 30 min, and 300 min and four positions (X/D = -3.0, 1.0, 4.0, and 8.0) infront and behind the pipe. Qualitatively the results of a particular turbulence model were same for all scour gaps, so, the results of Cases (1-2) for scour gap at time 0 min are presented. Consequently, the turbulence model that reproduces a similar result as of experimental investigation of Jensen et al. is chosen for all further simulation cases in the current study.

In the second part of this paper, a parametric study has been carried out by using five bed profiles suggested by Mao [22] at 0 min, 1 min, 6 min, 30 min, and 300 min. Five simulation cases (7-11) were run to obtain the drag coefficient $C_d$ and lift coefficient $C_l$ around the pipe with different gaps at position X/D=0 using LES turbulence model.

**Table 1: Simulation cases**

<table>
<thead>
<tr>
<th>Sim.*</th>
<th>Domain</th>
<th>Cases</th>
<th>Turbulence Models</th>
<th>Positions (X/D)</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(1,2,3,4,5)</td>
<td>1-2</td>
<td>Standard $k-\varepsilon$</td>
<td>-3.0,1.0,4.0,8.0</td>
<td>To identify the preferred turbulence model</td>
</tr>
<tr>
<td></td>
<td>(0.1,6,30,300 min)</td>
<td></td>
<td>LES</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>(1,2,3,4,5)</td>
<td>3-7</td>
<td>LES</td>
<td>0</td>
<td>To evaluate $C_d$ and $C_l$</td>
</tr>
<tr>
<td></td>
<td>(0.1,6,30,300 min)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

* Simulation

2.6 Mesh independence and time step test

The computational mesh constructed by FLEUNT is based on the square grid. The grid used in this simulation is a variable 0.001m grid along a scoured bed wall and around the cylinder
region by which the finer mesh is required near solid boundaries including the bed's surface and cylinder in order to solve the flow details near the solid regions. Accurate representation of flow near the wall region leads to a successful prediction of the turbulent core of the flow away from the wall and an accurate calculation of the bottom shear stress. Then, the finer square grids near the scour bed and around cylinder are expended linearly to a coarse grid of 0.009 m towards the upper boundary (Fig. 2). In addition, the wall treatment functions is applied by which in the current study Enhanced wall treatment with $y^+$ are applied near solid region such as bed scoured. Different wall treatment functions were tested and results showed no differences can be produced especially with using $y^+ < 1$. Mesh independence is carried out using five different types of grids having cell number of 40,000, 82,500, 130,225, 223404, and 315,623. A fine grid is used near the bed. The domain with 223,404 cells is selected throughout this study, because it shows reasonable accuracy and the lowest deviations of the velocity profile and turbulent kinetic energy (TKE) for turbulence models compared to Jensen laboratory data. Four different time step sizes ($\Delta t = 0.0002, 0.002, 0.02$, and 0.2) were tested and 0.002 is used throughout this current numerical study as there were no deviations observed below this value of time step. The number of iterations for this study are kept between 3000-3500.

![Figure 4: Mesh independence test for Mean $U_x$ (m s) at location X/D= -3.0 for different types of grids cell numbers, ---- Jensen, ---- 40000, O 82500, ---- 130225, ---- 223404, x 315623.](image)

3 Results and discussion

Current numerical study has been carried out in two major sections. First section deals with the effect of turbulence models such as standard $k-\varepsilon$ model and Large Eddy Simulation (LES) model on horizontal velocity profile and kinetic energy square root while the second one deals with the effect of scour gap on drag $C_d$ and lift $C_l$ coefficients. A large number of simulations have been run to study the effect of turbulence models on unsteady horizontal velocity profile and turbulent kinetic energy square root, and the results are discussed in section 3.1. The LES turbulence model identified using above study is used for further investigation of effect of scour gap on $C_d$, $C_l$ which is discussed in section 3.
3.1 Effect of different turbulence models

Cases 1-7 are used to predict the horizontal velocity profile (Ux) and the turbulent kinetic energy (TKE) at some axial length (i.e., X/D = -3.0, 1.0, 4.0, and 8.0) using different type turbulence models and the results are presented in Fig.5 and Fig.6 respectively. For comparisons, the experimental results reported in Jensen et al. [2] are also presented in the figure. Note that, the location the axial position covers the front and rear part of the pipe (or obstruction). Referring to Fig.5a tp 5b, which is presented at X/D = -3.0, 1.0, 4.0 and 8.0 respectively, the standard k-ε turbulence model prediction is much closer to experimental data at various water depth; some of them overlap each other. Nearly the same can be said for the LES model but some deviation seen at some of water depths; for example, at water depths below 0.03m and those between 0.05 to 0.06 m at X/D = 4.0 (Fig. 5d). TKE is well predicted by standard k-ε and horizontal velocity profile (Ux) and the turbulent kinetic energy (TKE).

Figure 5: Unsteady horizontal velocity (Ux) at different positions. (a) X/D = -3.0, (b) X/D = 1.0, (c) X/D = 4.0, and (d) X/D = 8.0 with different turbulent models — standard k-ε, o ELS and Red Jensen.
Figure 6: Turbulent Kinetic Energy (TKE) at different positions, (a) $X/D = -3.0$, (b) $X/D = 1.0$, (c) $X/D = 4.0$, and (d) $X/D = 8.0$ with different turbulent models — standard $k$-$\varepsilon$, o ELs and Red Jensen.
3.2 Effect of scouring depth

Fig. 10 shows the drag and lift coefficients respectively for five different gap phases. When the flow approaches to cylinder, the cylinder is subjected to different forces such as lift and drag that cause the formation of gap scouring. As it is seen in the figure, the lift and drag coefficients decrease as the gap becomes deeper. In addition, it is noticed that the pipe subjects to negative lift coefficients as long as the bed erosion comes into action and it remains throughout the scour process. The negative lift can be elaborated by the strong suction (or gap) below and behind the cylinder and the angle attack of the approaching flow (the position of the stagnation point). As a result of lift coefficient elimination, the drag coefficient is reduced with time as well. A reduction of 26.3% in $C_d$ and 48.3 % in $C_l$ was observed between the time phases of 10 min and 300 min.

![Figure 7: Bed profiles during the development of scouring](image-url)
4. Conclusions and future work

Two dimensional (2D) CFD analyses were carried to investigate fluid flow over an obstruction under different bed profiles using a number of turbulence models. Unsteady horizontal velocity profile and the kinetic energy square root at few axial directions are examined. The effect of scour depth on the drag coefficient, the lift coefficient of the obstruction body were numerically investigated. The conclusions of the current study are as follows:

The standard k-ε model and LES both can predict accurate results in comparison to other turbulence models when compared to experimental data for unsteady horizontal velocity and turbulent kinetic energy square root but it consumes time and needs a high performance computer to perform the scouring simulation.

The drag and lift coefficients decrease as the gap under the pipe increases. A reduction of 26.3% in Cd and 48.3% in Cl was observed between the time phases of 10 min and 300 min.

Acknowledgement

The financial support by the high impact research Grants of the University of Malaya (UM.C/625/1/HIR/61, account no. H-16001-00-D000061) and Exploratory Research Grant Scheme (ERGS: ER013-2013A) are gratefully acknowledged.

References


