Comparison of CFD models for multiphase flow evolution in bridge scour processes

A. Bayón-Barrachina, D. Valero,
F.J. Vallès Morán, P. A. López-Jiménez

Dept. of Hydraulic and Environmental Engineering

arbabar@iiama.upv.es

August 28^{th} 2014

UNIVERSITAT POLITÈCNICA DE VALÈNCIA

Table of contents

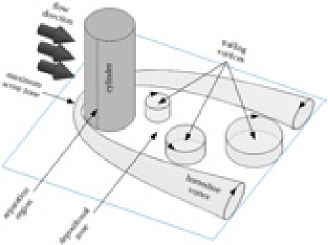
Introduction

- Erosion is one of the most potentially hazardous environmental phenomena in semi-arid regions (UNEP, 2000).
- Scour has been the cause of more than 60% of bridge collapses occurred since the 1950s (Landers, 1992).

Credit of the pictures: Left, S.D. Craig (USGS); Right, P. Johnson (Penn State Univ.)

Introduction

- The horseshoe vortex is the flow structure that contributes most to the occurrence of scour on bridge piers (Baker, 1979).



- Scour prevention methods (Melville & Coleman, 2000):
  - Flow alterations
  - Armoring protections

Introduction

- A good comprehension of the flow behavior when passing around cylinders is crucial to protect bridges successfully.
- An extensive literature has addressed this classical problem of fluid mechanics both numerically and experimentally.
- In CFD cases of application, LES is the most used approach.
- LES models are more accurate than RANS, although their computation times are 10 times longer (Rodi, 1997).

Goal

The aim of this work is to compare two of the most used numerical models in hydraulic engineering applications, namely:

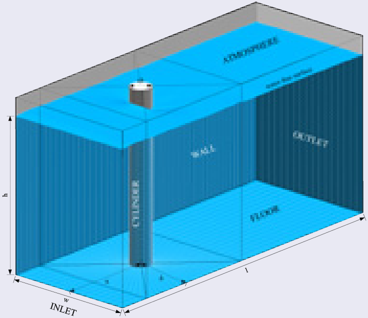
- The open source platform **OpenFOAM**
- The commercial software **FLOW-3D**

Materials and methods

A case study is run to compare the performance of both models.

Geometry

- The study case consists of a single cylindrical bridge pier.
- The pier is attached to the bottom of a sandy streambed channel of rectangular walls.
- The results are compared to experimental data obtained in an open sediment transport channel.



Materials and methods

Mesh

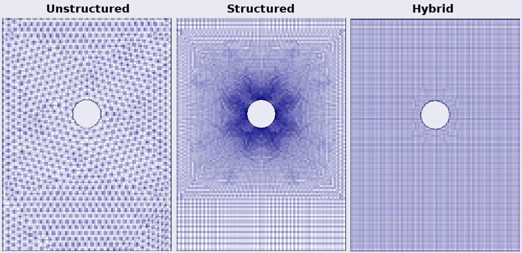
- Three meshing approaches are assessed using OpenFOAM:

Mesh type	**Pros**	**Contras**
Unstructured	- Refinable - Adaptive	- Problems to reproduce water free surfaces
Structured	- Refinable - Fast simulations	- Poor capability to adapt to complex geometries
Hybrid	- Adaptive - Fast simulations	- Bad refinement in solid contour vicinity

Materials and methods

Mesh

- The third option (hybrid meshing approach) is considered the most suitable in the OpenFOAM case.
- The same approach using the FAVOR method is the only option in FLOW-3D.



Materials and methods

Numerical model

- The Saint-Venant and the Boussinesq Equations can model the flow around cylinders with some simplifications.
- A full three-dimensional approximation of this kind of flow can only be done approximating the Navier-Stokes Equations.
- The Navier-Stokes Equations are discretized using the Finite Volume Method (FVM), see Versteeg & Malalasekra (2007).
- The treatment of two coexisting flows is done using the Volume of Fluid (VOF) approach (Hirt & Nichols, 1981).
- The VOF implementation and how water-air interfaces are tracked differs slightly from one model to the other.

Materials and methods

Numerical model

- The flow turbulent fluctuations are fully modeled using the RNG RANS k-ε model (Yakhot *et al.*, 1992).
- Simulations are run using a Courant-dependent variable time step algorithm to ensure convergence.
- Simulations are considered to have converged when the domain turbulent kinetic energy and volume of water stabilize.
- Variables are averaged using a 4-second window (one order of magnitude larger than quasi-periodic phenomena periods).
- Variables are obtained only from 3.75ø upstream of the pier to 28.00ø downstream of it to minimize the BC effects.

Materials and methods

Boundary conditions

Inlet: constant water depth and inlet velocity and null k & ε (an initial reach is simulated for the flow to develop).

Outlet: constant water depth (hydrostatic profile imposed) and null Neumann condition for the rest of variables.

Sky: atmospheric boundary condition (the flow can freely leave the domain with no gradients whatsoever).

Walls: no-slip condition and high-Reynolds-number rough wall function to reduce computation times.

Materials and methods

Case study

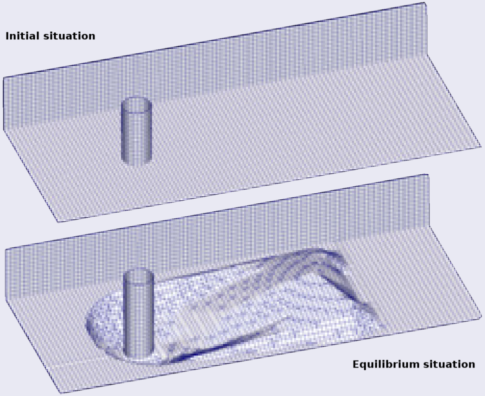
The case study is aimed at comparing both numerical models, OpenFOAM and FLOW-3D, using experimental data:

- Channel dimensions: $0.800m \times 0.064m$
- Pier diameter: $\o = 0.0107m$ (at $L_1 = 0.300m$ from inlet)
- Sand grain diameter: $d_s \in [0.40, 0.63]mm$
- Flow rate: $q = 0.75 \cdot 10^{-3} \frac{m^3}{s}$
- Water level: $h_{inlet} = 0.0792m$, $h_{outlet} = 0.0786m$
- Diameter-based Reynolds number: $Re = 1,570$
- Approaching Froude number: $Fr_1 = 0.167$

Materials and methods

Case study

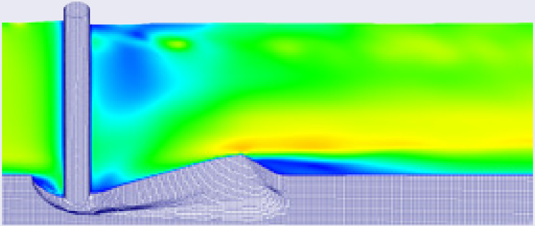
- Two scenarios are considered:



Materials and methods

Case study

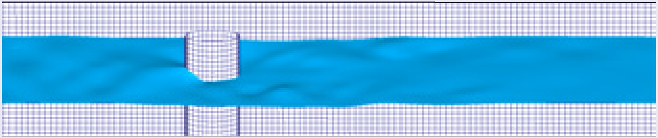
- A mesh sensitivity analysis for four different element sizes ($\Delta x = [5, 4, 3, 2] mm$) is conducted using the variables:
  - Recirculation zone length (L_D)
  - Vortex shedding period (T_{VS})
  - Max. vertical dimensionless coordinate (y^+_{max})



Analysis of results

Reproduction of the physics

- The results reproduce the physics of the problem observed at the experimental facility in both cases.
- Stagnation produces a water level rise upstream of the pier.
- A water level descent occurs downstream of the pier due to flow acceleration.



Analysis of results

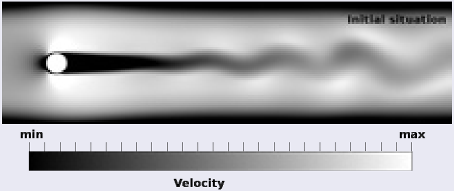
Mesh sensitivity analysis

- The three variables analyzed showed no significant improvement beyond a mesh size of $\Delta x = 3mm$.
- Computational times soar when using finer meshes.
- The maximum y^+ coordinate of the elements in touch with solid boundaries converged in both models to 60.
- The recirculation length (L_D) converged to $0.406m$ and $0.418m$ in OpenFOAM and FLOW-3D, respectively.

Analysis of results

Mesh sensitivity analysis

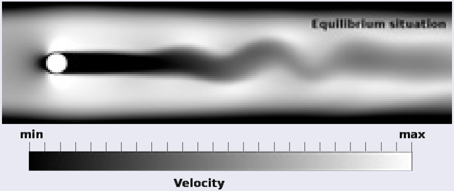
- The vortex shedding period (T_{VS}) converged to 0.334s and 0.414s in OpenFOAM and FLOW-3D, respectively.
- Blevins' (1990) method to estimate the Strouhal number as a function of the diameter-based Reynolds number is used.
- $T'_{VS} = 0.372s$ is obtained, which implies a relative error of -10.2% and 11.3% in OpenFOAM and FLOW-3D, respectively.



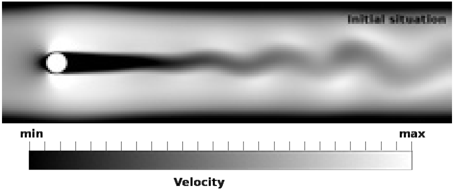
Analysis of results

Mesh sensitivity analysis

- The vortex shedding period (T_{VS}) converged to 0.334s and 0.414s in OpenFOAM and FLOW-3D, respectively.
- Blevins' (1990) method to estimate the Strouhal number as a function of the diameter-based Reynolds number is used.
- $T'_{VS} = 0.372s$ is obtained, which implies a relative error of -10.2% and 11.3% in OpenFOAM and FLOW-3D, respectively.



Analysis of results

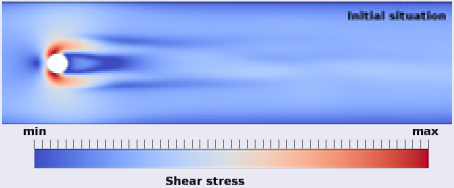


Play video

Analysis of results

Shear stress

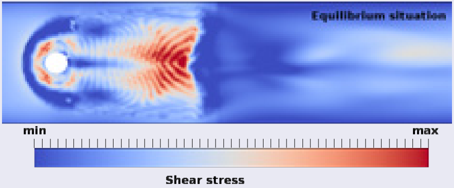
- The largest shear stress in the initial situation occurs where the horseshoe vortex forms (future scour hole ubication).
- The smallest value corresponds to the region where the sediment mound will eventually appear.



Analysis of results

Shear stress

- In the equilibrium situation, the shear stress becomes lower and more uniform in the pier vicinity.
- Unexpectedly large stresses occur on the sediment pound that can overcome the critical shear stress (τ_c).
- The Shield's diagram cannot foresee the triggering of erosion processes on uneven streambeds. Is τ_c useful? Does it exist?



Conclusions

- A comparison of performance of OpenFOAM and FLOW-3D applied to hydraulic structure modeling is presented.
- A study case is conducted using both codes and experimental data obtained in an open sediment transport channel.
- Both models manage to reproduce the physics of the problem under study and all variables converged to stable solutions.
- OpenFOAM slightly overestimates the recirculation length in detached flows with respect to FLOW-3D results.
- Compared to Blevins' (1990), OpenFOAM underestimates the vortex shedding period, whilst FLOW-3D overestimates it.
- Large shear stresses are estimated in zones where scour will occur, while lower values are found in deposition regions.
- It is observed that the presence of a scour hole weakens the wake vortex formation to a certain extent.

Thank you very much for your attention!

Comparison of CFD models for multiphase flow evolution in bridge scour processes

A. Bayón-Barrachina, D. Valero,
F.J. Vallès Morán, P. A. López-Jiménez

Dept. of Hydraulic and Environmental Engineering

arbabar@iiama.upv.es

August 28^{th} 2014