

Comparison of CFD Models for Multiphase Flow Evolution in Bridge Scour Processes

A. Bayón-Barrachina, D. Valero,

F. Vallès-Morán, P.A. López-Jiménez

¹Dept. Of Hydraulic and Environmental Engineering

Universitat Politècnica de València

Camí de Vera, s/n, València – 46022

SPAIN

E-mail: arbabar@iia.upv.es

Abstract: *The present work presents a performance comparison between two widely-used CFD codes, namely: the open source platform OpenFOAM and the commercial software FLOW-3D, applied to hydraulic structure modeling. To do so, a case study, consisting of a rectangular channel with a cylindrical bridge pier attached to its rough bottom is modeled using both codes. The flow is assumed to be turbulent transient incompressible multiphase and viscous and is simulated using the finite volume method (FVM) and the volume of fluid approach (VOF). Turbulence is modeled by means of the RANS model RNG $k-\epsilon$. Two scenarios are considered: the initial situation, where the channel bottom is even, and the equilibrium situation, where it is eroded. The eroded streambed geometry is extracted from laboratory experiments using an open sediment transport channel. Several variables of interest, such as shear stresses and vortex shedding period, are estimated and compared among both numerical models and results available in the literature. The main purpose of this study is to assess the accuracy of these solvers when modeling the hydrodynamics of common sediment transport problems.*

Keywords: *scour, erosion, CFD, RANS, OpenFOAM, FLOW-3D.*

1. Introduction

As the United Nations Environment Program states, erosion is one of the most potentially hazardous physical phenomena occurred in semi-arid regions (UNEP, 2000). Its consequences can compromise human and animal welfare as erosion jeopardizes agricultural yields, biodiversity of ecosystems and water quality, as well as it promotes climate change (Blanco & Lal, 2008). In urban areas, rainfall-induced erosion can also wash off asphalt, vehicle tire chips and all sort of urban debris, so polluting the hydrosphere (Bouteglier *et al.*, 2002). From the technical point of view, turbulent water flows can have calamitous consequences on dam spillways, river banks, bridge piers and other hydraulic structures. Indeed, as regards bridges, Landers (1992) found that foundation scour was behind 60% of all reported bridge failure cases occurred since the 1950s, so this is a very active research topic.

Erosion occurs when flows exert shear stresses on solid boundaries over a certain threshold, which depends on soil type, granulometry, temperature, etc. (Moody *et al.*, 2005). The highly three-dimensional and chaotic nature of environmental flows makes erosion processes difficult to foresee. Furthermore, sudden increases of shear stress can occur due to changes of flow regime (Wu & Rajratnam, 1996) or geometry (Pitt *et al.*, 2007).

The present work focuses on a specific case of erosion process caused by a sudden geometry variation, which is the flow encountering a cylindrical obstacle. This classical problem of fluid mechanics can be often observed in environmental hydraulics, e.g. in flows over vegetated floodplains. However, in this case more emphasis is done on river flows passing around bridge piers. As Melville & Coleman (2000) state, the intensity of scour processes in these cases is extremely variable. It depends on a wealth of factors, such as the flow alteration caused by piers and their geometry, their relative position with respect to the river banks and the eventual presence of scour protection systems.

Most protection works on bridge piers focus on preventing the erosive effects of the horseshoe vortex (see Fig. 1): the most potentially hazardous flow structure formed in the upstream region of cylinders

surrounded by fluids in motion (Baker, 1979). Many authors devoted their careers to the study of these protection methods. According to Melville and Coleman (2000), two main approaches exist in this regard, namely: flow alteration actions and armoring protections.

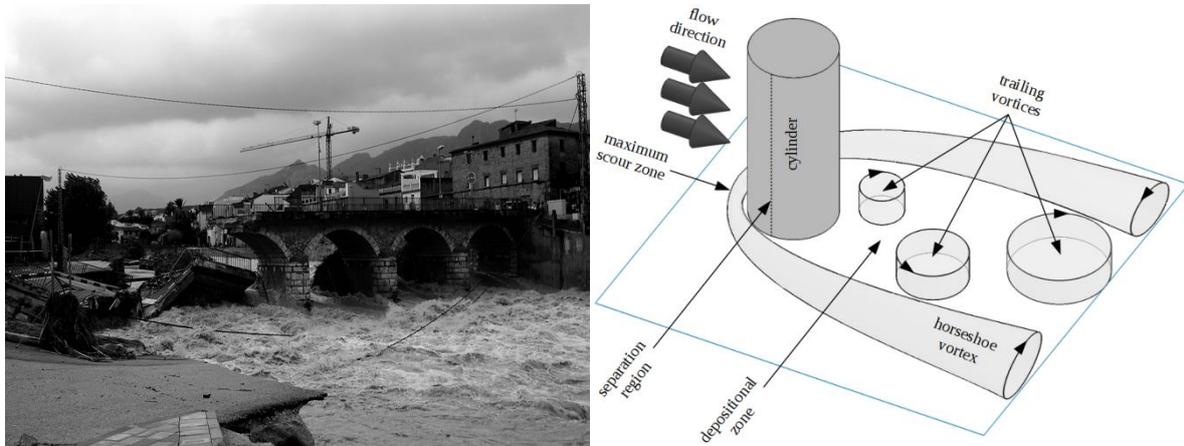


Figure 1 – Left: collapse of Beniarbeig Bridge (València, Spain) due to flood-induced pier scour (Server, 2007). Right: main flow structures formed around a circular cylinder.

Along the lines of the first type of systems, a wealth of studies on the use of inclined bridge piers (Bozkus & Cesme, 2010), non-circular sections (Melville & Sutherland, 1988) and sacrificial piles (Melville & Hadfield, 1999) has been reported. As regards armoring protections, some authors analyzed the use of collars around the pier itself (Moncada-M *et al.*, 2010; Zarrati *et al.*, 2004), whereas others focused on the implementation of protection systems on the foundations, such as riprap (Parola, 1993), wire gabions (Yoon, 2005) and tetrahedral frames (Tang *et al.*, 2009). More in-depth information on this topic can be found in the specific literature (Tafarojnoruz *et al.*, 2010; Chiew, 1992; Parker *et al.*, 1998; Deng & Cai, 2009).

Nevertheless, all actions and efforts to design pier protection systems against scour are sterile unless a complete understanding of the flow around these structures is achieved. For this reason, a wealth of studies can be found in the literature. Most of them have focused on the analysis of a single cylinder, both experimentally (Sumner *et al.*, 2004) or numerically (Frohlich & Rodi, 2004). In the latter case, most studies are conducted using techniques of computational fluid dynamics (CFD), although some cases of use of statistical models have been reported (Hong *et al.*, 2012). Many authors analyzed variations of the cylinder geometry, ranging from square (Liu, 2010) or oscillating cylinders (Mittal & Kumar, 2001) to dispositions of several cylinders into an array (Doolan, 2009; Stanescu *et al.*, 1996). Erosion-sedimentation processes using numerical approaches have not been so extensively studied, although some cases can be found (Souza *et al.*, 2010; Liu & Garcia, 2008).

The literature on hydraulic structure modeling, be numerically or experimentally, is extensive. For this reason, choosing the best model to reproduce this kind of phenomena is not a trivial decision. Thus, the goal of this work is to compare the performance of two widely-used CFD codes, the open source platform OpenFOAM and the commercial software FLOW-3D. To do so, a case study is analyzed simulating two different scenarios: initial and equilibrium state. The geometry of the eroded streambed is obtained using an open sediment transport channel. In order to assess the accuracy of both codes, their results are compared to previous works.

2. MATERIALS AND METHODS

2.1. Geometry and mesh

The analyzed geometry consists of a single emerging cylinder attached to the bottom of a rectangular channel with a rough streambed. As shown in Fig. 2, the three-dimensional spatial discretization of this geometrical domain is conducted using three different meshing approaches. First, an unstructured

adaptive mesh of tetrahedral elements is used for its good ability to adapt to complex geometries (Biswas & Strawn, 1998) and to refine only those regions where it is necessary (Kim & Boysan, 1999). However, this kind of meshes present divergence problems when defining water free surfaces even using very fine meshes, so its use is ruled out after some preliminary test simulations.

A six-block structured adaptive mesh of hexahedral elements is also tested to overcome the aforementioned convergence problem. According to several authors, such as Keyes *et al.* (2000), this kind of meshes ensure a more systematic storage of information and so a faster access to memory during simulations. Nevertheless, preliminary simulations demonstrate that this approach is not always able to refine selectively. Therefore, excessively dense meshes are necessary to achieve good results, so the use of this kind of mesh is also ruled out.

At last, a quasi-structured non-adaptive mesh of hexahedral elements is used. This mesh is only refined longitudinally from the inlet and the outlet to the cylinder with a scale factor of 2. The elements in the other two directions have the size of the longitudinally finest region. Despite the simplicity of this meshing approach, it yields the best results at the lowest computational cost and does not show any problem in reproducing properly the modeled geometry. Despite both numerical models meshes are exactly alike, how mesh elements are intersected by solid surfaces is treated in a different manner according to the numerical code. FLOW-3D uses a porosity approach called FAVOR (Hirt & Sicilian, 1985) whereas OpenFOAM readapts mesh element configuration so that element boundaries become tangential to solid surfaces.

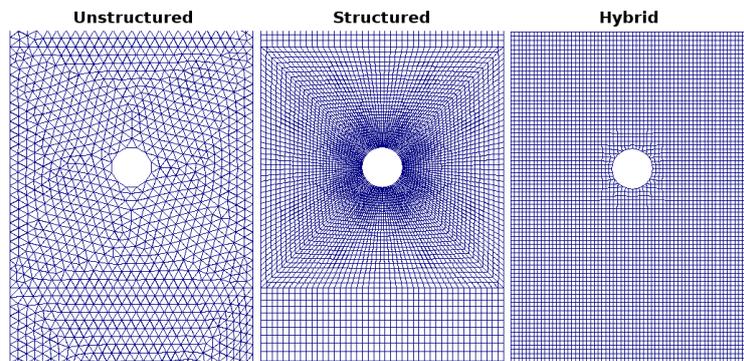


Figure 2 – Top view of the three meshing approaches used using the same number of elements.

2.2. Numerical model

Despite the Saint-Venant and Boussinesq equations can be applied to the flow around cylinders, in cases like this, where a full three-dimensional description of the flow is required, the approximation of the Navier-Stokes equations becomes compulsory. The numerical models used in this investigation discretize these equations assuming transient multiphase incompressible isotropic turbulent viscous flow using the finite volume method (FVM), extensively described by Versteeg & Malalasekera (2007). The treatment of two coexisting fluids (i.e. water and air) and the definition of a neat interface between them are achieved thanks to the implementation of the volume of fluid method (VOF), described by Hirt & Nichols (1981).

As regards the treatment of turbulence, most of the cases found in the literature are based on a LES approach (the use of DNS is out of the question due to its unaffordable computational costs). Nevertheless, according to Rodi (1997), despite LES models yield significantly better results than their RANS counterparts, their computation times are more than ten times larger. For this reason, RANS models are often preferred as a good compromise between accuracy and computational cost. In this case, RANS equations as described by Pope (2000) are numerically solved. In particular, the RNG $k-\epsilon$ model (Yakhot *et al.*, 1992) is preferred for its good performance in this kind of applications (Bradshaw, 1996).

The definition of the model initial and boundary conditions is crucial due to the large effect they exert on the final results. In order to force a subcritical developed flow with a hydrostatic profile throughout

the modeled domain, a constant water level boundary condition is imposed at both the inlet and the outlet. A Dirichlet boundary condition is imposed to the velocity and the turbulence model variables (i.e. k and ϵ) at the inlet. An initial reach is simulated in order for the flow to develop completely. At the outlet, a null von Neumann condition is imposed to k and ϵ . The open air patch is treated by means of an atmospheric boundary condition in OpenFOAM, which allows the flow to leave the domain if necessary. In FLOW-3D, pseudo-single-phase approach is employed, not being necessary to solve the flow equations in the void domain (Prosperetti & Tryggvason, 2007).

As regards solid boundaries, a no-slip condition is imposed. In order to save computational resources, boundary layers are not resolved down to their lowest scale, but they are modeled instead. To do so, a high-Reynolds-number rough wall function is used. This allows the treatment of solid walls roughness by adding an extra term to the smooth wall function implementation (Cebeci & Bradshaw, 1977).

Simulations are run using a Courant-dependent variable time step algorithm to ensure convergence; which forces the maximum Courant number to remain below 0.75. All variables are averaged using 4-seconds time windows measured once simulations have converged to a quasi-periodical state. Simulations are considered converged when fluid fraction and turbulent kinetic energy (k) in the domain become stable in time.

2.3. Case study

As stated above, a case study is simulated numerically and compared to previous studies. Two scenarios are reproduced using both numerical codes, OpenFOAM and FLOW-3D. First, the initial situation, where the channel streambed is still even, is simulated. A second simulation is run with the eroded streambed in the equilibrium state (see Fig. 3). To do so, the channel bottom surface is experimentally tracked using a point gauge when erosion-sedimentation processes stabilize after several hours of experiment. The physical model used consists of a sediment transport open channel of methacrylate walls, PVC piers and sandy streambed of grain diameter between 0.63 and 0.40mm. The channel is 0.8000m long and 0.0640m wide and a single cylindrical pier of diameter 0.0107m is inserted on the channel center at 0.3000m of the inlet. The flow rate imposed is $0.75 \cdot 10^{-3} \text{ m}^3/\text{s}$ and the water surface level is 0.0792m at the inlet and 0.0786m at the outlet, respectively.

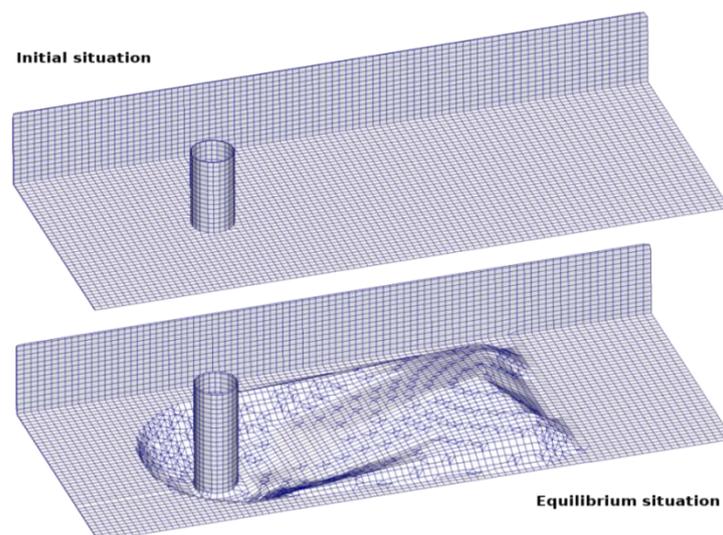


Figure 3 – Two scenarios simulated: initial and erosion-sedimentation equilibrium state.

All the conditions exposed above, assuming a water density of $1000 \text{ kg}/\text{m}^3$ and a kinematic viscosity of $10^{-6} \text{ m}^2/\text{s}$, yield a diameter-based Reynolds number of 1,570 and an approaching Froude number of 0.167. Given the experimental model dimensions, scale effects could be significant when comparing it to a real-life scale prototype. However, as the numerical model is set up according to the experimental model size, scale is not expected to bias results. Fig. 4 sketches the experimental facility configuration.

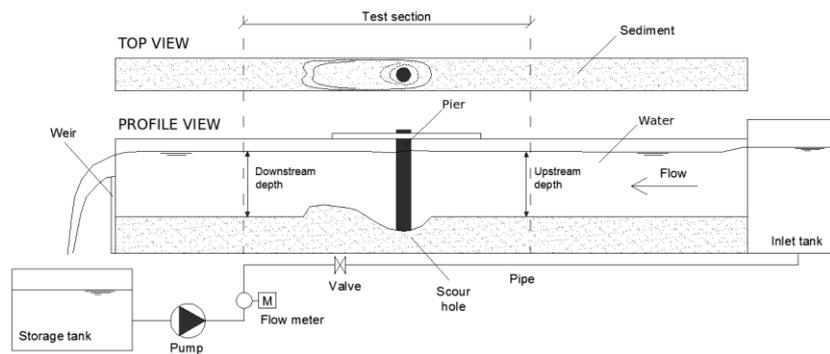


Figure 4 –Experimental facility.

A mesh sensitivity analysis is conducted in order to determine at which refinement level the accuracy of results is mesh independent. Four different mesh element sizes are tested, namely 5, 4, 3 and 2mm, which imply a number of elements ranging from 0.8 to $3.0 \cdot 10^6$. Also preliminary simulations are run to ensure that no significant differences are observed when varying the length of the simulated channel stretch. Only certain undesirable effects may appear when bringing inlet and outlet boundary conditions extremely close to the pier. Nevertheless, the channel length analyzed in this case is far away from such situation. The insensitivity to this parameter is corroborated by observing the experimental facility behavior.

Among the variables analyzed, three of them are used to determine the mesh independence of results, namely: the recirculation zone length after the deposition region, the vortex shedding period and the maximum y^+ coordinate on the streambed. Also the shear stress exerted by the flow on the channel bottom is analyzed due to the important role that it plays in erosion-sedimentation process triggering. In order to neglect the effects exerted by the domain boundary conditions, all variables are taken into account only 3.75 and 28.00 diameters upstream and downstream of the bridge pier, respectively.

3. ANALYSIS OF RESULTS

The results of the simulated case study seem to reproduce all the expected features of the system analyzed. The water surface is nearly even in the initial situation simulations, except for the region upstream of the bridge pier, where stagnation produces a slight water level rise, observed in both the numerical and experimental models. The constriction produced by the pier accelerates the flow, with the subsequent water level decrease downstream of the pier. In the equilibrium situation, small undulations occur at the free surface downstream of the pier. This is due to the action of vertical flows caused by the presence of the scour hole.

As regards the mesh sensitivity analysis, the three variables used as criterion of convergence demonstrated, both in OpenFOAM and FLOW-3D, that very little improvement is obtained beyond mesh element sizes of 3mm, whereas computational costs increase significantly from this point on. The flow recirculation length (downstream of the deposition zone) is a very sensitive variable in RANS models, which is why it is considered a good criterion variable. In this case, OpenFOAM predicted an average recirculation length of 0.406m, whereas in FLOW-3D this variable reached 0.418m. The second variable used to test the results sensitivity to the mesh size, the maximum y^+ coordinate value on the streambed surface, remains in both cases around 60.

The vortex shedding period also converged but, in this case, to different values according to the model used: OpenFOAM estimates a vortex shedding period of 0.334s, whereas FLOW-3D yields a value of 0.414s. Both results are compared to the value obtained following the method proposed by Blevins (1990) to estimate the Strouhal number as a function of the diameter-based Reynolds number. The value calculated using this method yields a period of 0.372s, which implies a relative error of -10.2% and 11.3% committed by OpenFOAM and FLOW-3D, respectively. It is important to remark that these values correspond exclusively to the initial situation simulations. This is due to the fact that, as it can be observed in Fig. 5, the formation of wake vortices in the equilibrium situation occurs in a more

irregular and less perceptible way. Apparently, the presence of the scour hole makes the recirculation zone downstream of the pier longer and tends to mitigate the vortex formation to a certain extent.

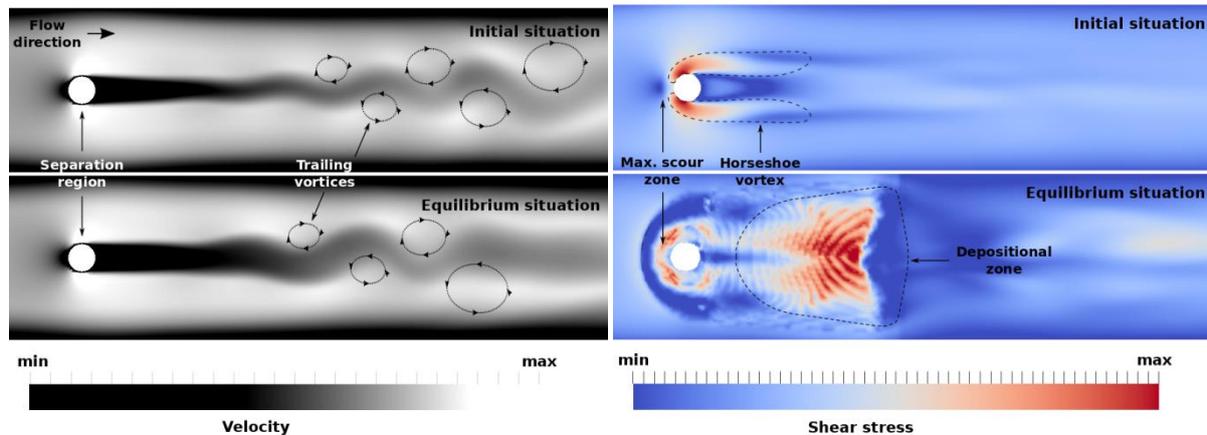


Figure 5 – Left: vortex shedding (velocity at $z = 0.40\text{m}$). Right: shear stress distribution on streambed.

As regards the shear stresses exerted by the flow on the channel streambed, several interesting facts arise from the simulation results. First, it can be observed that the larger stresses in the initial situation occur in the zone of formation of the horseshoe vortex, in agreement with that stated by Baker (1979). It can also be seen that the lower stresses correspond to the future sediment deposition zone.

When analyzing the equilibrium situation, it is found that shear stresses tend to distribute in a more uniform way in the immediate vicinity of the bridge pier. This makes sense bearing in mind that it is an erosion-sedimentation process equilibrium state. Nevertheless, large shear stresses occur on the deposition mound, which can probably be explained by the acceleration caused by the flow constriction that it produces.

Authors have not found an explanation to the existence of this unexpectedly large shear stress in the equilibrium state; however the slope of the bed in combination with the shear stress could yield a stable state. It can be observed how this stresses are even larger than those caused by the horseshoe vortex formation and so are likely above the critical shear stress. However, the latter value, which can be easily obtained using the Shield's Diagram, is not always able to determine the triggering of erosion processes. On the one hand, it attempts to reproduce in a simplistic way an extremely chaotic phenomenon, where lift, drag, buoyancy, soil cohesion and other forces interact in a nonlinear way. On the other hand, the critical shear stress can only be estimated as long as the flat streambed hypothesis is fulfilled. E.g. the presence of obstacles, such as bridge piers, cause non-tangential flows that can trigger sediment transport at unexpectedly low velocities. In this case, on the ascending slope of the deposition mound, the flow has to exert a higher shear stress than the critical value to maintain the equilibrium, as it has to overcome the effect of gravity. All these reasons have led some authors, such as Lavelle & Mofjeld(1987), to wonder if it is useful (or even possible) the use a threshold to determine the incipient particle movement.

As it is discussed above, OpenFOAM and FLOW-3D results are slightly different. Despite the model setup has been set intentionally as similar as possible in both cases, this only affects general aspects of its implementation: certain details may differ from one model to another. E.g. the free surface definition method used in FLOW-3D is the so-called one-fluid VOF, whereas OpenFOAM uses another variation of the VOF involving the resolution of both phases and the use of a surface compression algorithm to obtain neat free surfaces. Differences in evaluation of advection and viscous terms can also yield the observed differences in the models. Besides, the meshing approach is not exactly similar, as discussed above, which can also exert significant effects on this kind of simulations. Therefore, it is important to remark that a sediment transport model implemented in both numerical models could require a different calibration since hydrodynamic results are different.

4. CONCLUSIONS

A comparison of two widely-used CFD models in hydraulic engineering, OpenFOAM and FLOW-3D, is performed. To do so, a case study consisting of a circular cylinder attached to the rough bottom of an open air channel, is conducted. The results are analyzed in order to determine the result independence to mesh cell size and time-averaging window size. Among the studied variables, the most important ones from the hydraulic point of view are the water free surface level, the formation of wake vortices, the recirculation region length and the shear stress distribution throughout the channel streambed.

The physics of the phenomenon analyzed is properly reproduced by both numerical models and mesh independent results are obtained at mesh sizes below 3mm. The length of the recirculation region downstream of the deposition mound is estimated higher by OpenFOAM than by FLOW-3D, whereas the latter tends to overestimate shear stresses with respect to the first.

Flow pattern variations are observed when passing from the initial to the equilibrium situation, as well as changes in the shear stress distribution are detected. While in the initial situation the largest stresses occurred in the horseshoe vortex formation region, in the equilibrium situation these values move to the deposition mound ascending slope. This raises some important questions about the use of a critical shear stress value that are addressed in the previous section.

As regards vortex shedding, when comparing to the results found in the bibliography, OpenFOAM underestimates its period by 10.2%, whilst FLOW-3D overestimates it by 11.3%. It is also found that, in the equilibrium situation, the presence of a scour hole tends to weaken the wake vortex formation downstream of the pier. This could be interesting from the point of view of the design of structures to mitigate vortex shedding in cylinders in order to prevent undesirable vibrations.

5. ACKNOWLEDGEMENTS

This research was conducted thanks to the funding provided by the VALi+D R&D Program of the Generalitat Valenciana (Spain). It would not have been possible without the contribution of Beatriz Nácher, (Laboratory of Hydraulics, School of Civil Engineering – Universitat Politècnica de València).

6. REFERENCES

- Baker, C. (1979), *The laminar horseshoe vortex*, Journal of fluid mechanics, Cambridge University Press, 95 (02), 347-367.
- Biswas, R. and Strawn, R. C. (1998), *Tetrahedral and hexahedral mesh adaptation for CFD problems*, Applied Numerical Mathematics, Elsevier, 26 (1), 135-151.
- Blanco, H. and Lal, R. (2008), *Principles of Soil Conservation and Management*, Springer.
- Blevins, R. D. (1990), *Flow-induced vibration*, Van Nostrand Reinhold, New York.
- Bouteligier, R., Vaes, G. and Berlamont, J. (2002), *Sensitivity of urban drainage wash-off models: compatibility analysis of HydroWorks QM and MouseTrap using CDF relationships*, Journal of Hydroinformatics, 4, 235-243.
- Bozkus, Z. and Cesme, M. (2010), *Reduction of scouring depth by using inclined piers*, Canadian Journal of Civil Engineering, NRC Research Press, 37 (12), 1621-1630.
- Bradshaw, P. (1996), *Understanding and prediction of turbulent flow*, International journal of heat and fluid flow, Elsevier, 18 (1), 45-54.
- Cebeci, T. and Bradshaw, P. (1977), *Momentum transfer in boundary layers*, Washington, DC, Hemisphere Publishing Corp., New York, McGraw-Hill Book Co., 1.
- Chiew, Y.-M.(1992), *Scour protection at bridge piers*, Journal of Hydraulic Engineering, American Society of Civil Engineers, 118 (9), 1260-1269.
- Deng, L. and Cai, C. (2009), *Bridge scour: Prediction, modeling, monitoring, and countermeasures—Review*, Practice periodical on structural design and construction, American Society of Civil Engineers, 15 (2), 125-134.

- Doolan, C. J. (2009), *Flow and noise simulation of the NASA tandem cylinder experiment using OpenFOAM*, AIAA paper, 3157.
- Frohlich, J. and Rodi, W. (2004), *LES of the flow around a circular cylinder of finite height*, International Journal of Heat and Fluid Flow, Elsevier, 25 (3), 537-548.
- Hirt, C. and Nichols, B. (1981), *Volume of fluid (VOF) method for the dynamics of free boundaries*, Journal of Computational Physics, 39 (1), 201-225.
- Hirt, C.W. and Sicilian, J.M. (1985), *A porosity technique for the definition of obstacles in rectangular cell meshes*. In Proc. Fourth International Conf. Ship Hydro.
- Hong, J.-H., Goyal, M. K., Chiew, Y.-M. and Chua, L. H. (2012), *Predicting time-dependent pier scour depth with support vector regression*, Journal of Hydrology, 468-469, 241-248.
- Keyes, D., Ecer, A., Satofuka, N., Fox, P. and Periaux, J. (2000), *Parallel Computational Fluid Dynamics' 99: Towards Teraflops, Optimization and Novel Formulations*, Elsevier.
- Kim, S.-E. and Boysan, F. (1999), *Application of CFD to environmental flows*, Journal of Wind Engineering and Industrial Aerodynamics, Elsevier, 81 (1), 145-158.
- Landers, M. N. (1992), *Bridge Scour Sata Management, Hydraulic Engineering: Saving a Threatened Resource-In Search of Solutions: Proceedings of the Hydraulic Engineering sessions at Water Forum '92, Baltimore, Maryland (USA)*, American Society of Civil Engineers.
- Lavelle, J. W. and Mofjeld, H. O. (1987), *Do critical stresses for incipient motion and erosion really exist?*, Journal of Hydraulic Engineering, American Society of Civil Engineers, 113 (3), 370-385.
- Liu, Z. (2010), *Square Cylinder Large Eddy Simulation Based on Random Inlet Boundary Condition*, Journal of Applied Fluid Mechanics, 3 (1).
- Liu, X. and Garcia, M. H. (2008), *Three-dimensional numerical model with free water surface and mesh deformation for local sediment scour*, Journal of waterway, port, coastal, and ocean engineering, American Society of Civil Engineers, 134 (4), 203-217.
- Melville, B. W. and Coleman, S. E. (2000), *Bridge scour*, Water Resources Publication.
- Melville, B. W. and Hadfield, A. C. (1999), *Use of sacrificial piles as pier scour countermeasures*, Journal of Hydraulic Engineering, American Society of Civil Engineers, 125 (11), 1221-1224.
- Melville, B. and Sutherland, A. (1988), *Design method for local scour at bridge piers*, Journal of Hydraulic Engineering, American Society of Civil Engineers, 114 (10), 1210-1226.
- Mittal, S. and Kumar, V. (2001), *Flow-induced oscillations of two cylinders in tandem and staggered arrangements*, Journal of Fluids and Structures, Elsevier, 15 (5), 717-736.
- Moncada-M, A., Aguirre-Pe, J., Bolívar, J. and Flores, E. (2009), *Scour protection of circular bridge piers with collars and slots*, Journal of Hydraulic Research, 47 (1), 119-126.
- Moody, J. A., Smith, J. D. and Ragan, B. W. (2005), *Critical shear stress for erosion of cohesive soils subjected to temperatures typical of wildfires*, Journal of Geophysical Research: Earth Surface, 110 (F1).
- Pope, SB. (2000), *Turbulent flows*. Cambridge university press.
- Prosperetti, A. and Tryggvason, G. (2007), *Computational methods for multiphase flow*. Cambridge University Press.
- Parker, G., Toro-Escobar, C. and Voigt Jr, R. L. (1998), *Countermeasures to protect bridge piers from scour*, St. Anthony Falls Laboratory.
- Parola, A. (1993), *Stability of riprap at bridge piers*, Journal of Hydraulic Engineering, American Society of Civil Engineers, 119 (10), 1080-1093.
- Pitt, R., Clark, S. E. and Lake, D. (2007), *Construction site erosion and sediment controls - Planning, design and performance*, DEStech Publications, Inc.
- Rodi, W. (1997), *Comparison of LES and RANS calculations of the flow around bluff bodies*, Journal of Wind Engineering and Industrial Aerodynamics, Elsevier, 69, 55-75.
- Server, B. (2007). *El pont sobre el Riu Girona a Beniarbeig (Marina Alta, Pais Valencia) unes poqueshoresdespres de caure*, Flickr.
- Souza, L., Schulz, H., Vilella, S. and Gulliver, J. (2010), *Experimental Study and Numerical Simulation of Sediment Transport in a Shallow Reservoir.*, Journal of Applied Fluid Mechanics, 3 (2).
- Stanescu, G., Fowler, A. and Bejan, A. (1996), *The optimal spacing of cylinders in free-stream cross-flow forced convection*, International Journal of Heat and Mass Transfer, Elsevier, 39 (2), 311-317.

- Sumner, D., Heseltine, J. and Dansereau, O. (2004), *Wake structure of a finite circular cylinder of small aspect ratio*, Experiments in Fluids, Springer, 37 (5), 720-730.
- Tafarjnoruz, A., Gaudio, R. and Dey, S. (2010), *Flow-altering countermeasures against scour at bridge piers: a review*, Journal of Hydraulic Research, 48 (4), 441-452.
- Tang, H.-W., Ding, B., Chiew, Y.-M. and Fang, S.-L. (2009), *Protection of bridge piers against scouring with tetrahedral frames*, International Journal of Sediment Research, Elsevier, 24 (4), 385-399.
- UNEP (2000), *Guidelines for erosion and desertification control* management, United Nations Environment Program (UNEP) and Food and Agriculture Organization (FAO).
- Versteeg, H. and Malalasekera, W. (2007), *An introduction to computational fluid dynamics*, Pearson Prentice Hall.
- Wu, S. and Rajaratnam, N. (1996), *Transition from Hydraulic Jump to Open Channel Flow*, Journal of Hydraulic Engineering, 122 (9), 526-528.
- Yakhot, V., Orszag, S., Thangam, S., Gatski, T. and Speziale, C. (1992), *Development of turbulence models for shear flows by a double expansion technique*, Physics of Fluids A: Fluid Dynamics (1989-1993), AIP Publishing, 4 (7), 1510-1520.
- Yoon, T. H. (2005), *Wire gabion for protecting bridge piers*, Journal of hydraulic engineering, American Society of Civil Engineers, 131 (11), 942-949.
- Zarrati, A. R., Gholami, H. and Mashahir, M. (2004), *Application of collar to control scouring around rectangular bridge piers*, Journal of Hydraulic Research, 42 (1), 97-103.