CFD Simulation of Pressure Fluctuations in Plunge Pools: In Search of a New Method

[1]**Shayan Maleki,** *Ph.D, Senior Dams Engineer, GHD Pty Ltd*

[2]**James Apostolidis,** *CFD Modelling Specialist, GHD Pty Ltd*

[3]**Tom Ewing,** *CFD Modelling Specialist, GHD Pty Ltd*

[4]**Virgilio Fiorotto,** *Full Professor, University of Trieste*

The stability analysis of dam spillways and stilling basin chutes requires the knowledge of the spatially fluctuating pressure at the bottom of the structure with reference to the large vortex system with dimensions comparable with the structure characteristic length of the order O (0.1 – 1 m). In this context only the small frequency pressure fluctuations (smaller than 1 – 10 hz in prototype) must be analyzed in Large Eddy Simulation (LES) context; while the higher frequency pressure fluctuations could be filtered given their negligible importance in relation to stability computations with reference to the spatial Taylor macroscale and fluctuating pressure variance evaluation. These two quantities allow us to define the variance of the force acting on the structure, and as a consequence via statistical analysis, the design force on the structure. This procedure is historically performed via physical hydraulic modelling (PHM) where these quantities are measured in a laboratory setup. Considering the limits of current industry approach to Computational Fluid Dynamics (CFD), the use of Detached Eddy Simulation (DES) could become a valid low cost solution and could potentially be a valid method to perform preliminary studies in order to refine the design while avoiding expensive physical model modifications.

In this paper, the pressure field at the base of a rectangular impinging jet is measured in laboratory flume setup and is compared with the numerical results obtained via equivalent DES simulations conducted in CFD.

Maximum values and the structure of spatial correlation of the anisotropic field of fluctuating pressures are described in view of their relevance to the structural design of the lining of spillway stilling basins and other dissipations structures, as well as in view of their relevance to rock stability analysis. The comparison of the laboratory study with DES simulations presented in this paper shows a good agreement indicating that this approach may eventually provide a lower cost substitute for physical model studies in the design of stilling basins and plunge pools. However, it is acknowledged that virtually all stilling basins and plunge pools present a three-dimensional hydraulics complexity, and numerous further studies need to be done.

Keywords: *Pressure Fluctuation; CFD; Turbulence; Energy Dissipation, Plunge Pool*

Introduction

The stability of concrete slabs or rock blocks at plunge pool bottom depends on the instantaneous pressure field (e.g. Bollaert and Schleiss, 2003; Melo et al., 2006; Asadollahi et al., 2011; Barjastehmaleki et al. 2016 a, b). This pressure propagates under lining elements through rock fissures, open or failed joints, can generate an uplift force that can lead to dislodging of the lining. This instability can be investigated separately with reference to the mean dynamic pressure and the fluctuating components in order to reach a comprehensive analysis (Fiorotto et al, 2016).
The mean dynamic pressure component is studied by Fiorotto et al. (2016) and the ability of CFD in evaluation of the mean dynamic pressure is here investigated and preliminary results are presented in this paper. The paper is a part of a full paper including more complex conditions in different physical hydraulic models.
The pressure field is not statistically homogeneous along the flow direction with a decrease in pressure variance increasing the distance from the impingement section. For this reason, an accurate experimental description of the jet behaviour must be sought, with reference to the bottom pressure fluctuations in the impingement. Therefore, extensive experiments are carried out and pressure fluctuations are measured at different points under jet impact and the results are compared with the pressure fluctuations results evaluated using DES CFD approach.
The aim of the paper is to investigate the ability of a new CFD approach in evaluation of pressure fluctuations at bottom of a plunge pool with reference to the problem of rock block or concrete slabs stability.

Experimental Setup

The experiments were conducted in a horizontal rectangular flume, 0.5 m wide, 0.5 m height and 10 m long, in the Hydraulics Laboratory of the University of Trieste, Italy. The flume had a Plexiglas bed and sidewalls and a vertical tailgate for controlling the water level. The water was supplied by a head-tank to the jet device and the discharge was measured with an inline magnetic flowmeter (Toshiba LF620) with an accuracy of ±2%. The maximum discharge used in the experiments was 40 l/s and it was controlled by a valve. The jet device, place below the water surface, had a thickness of 0.01 m and width of 0.50 m and it was adjustable in height in order to change the jet distance from the channel bottom (Fig. 1). Its distance from the upstream channel wall was equal to 2.0 m. This distance can be changed using a movable wall that can be inserted in the channel.

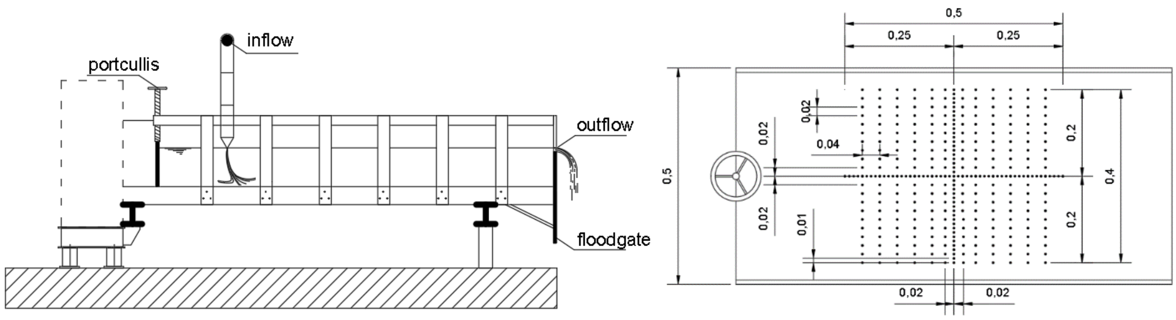


Fig. 1 Experimental setup and test area with indication of the tap positioning

The test area was 0.4 m wide and 0.5 m long and with the centre at 2.15 m downstream of the upstream channel wall. Pressure taps with diameter of 2 mm were inserted along the centre line of the flume and 12 different cross sections. The tap diameter was established based on the investigations of Fiorotto and Rinaldo (1992b), which show that increasing of the tap diameter reduces the magnitude of the measured fluctuating pressures. It is due to the effect of spatial averaging of a very small scale of turbulence that reduces the variance of the signal and the value of the maximum and minimum fluctuating pressures. The distance between two successive taps was 1 cm in stream-wise direction while in cross-stream direction, where the pressure is statistically homogeneous in the space, the distance between the taps is equal to 2 cm arranged, in their position, in order to obtain a gap in the spatial correlation function of 1 cm (Fig.1). This allows to get precise assessments of the spatial correlation functions. The water level in stilling well was measured with accuracy equal to 0.1 mm in a mm Vernier scale. The water surface was measured with a conventional point gauge with accuracy to the nearest 0.1 mm.

The fluctuating pressure was measured by means of AEP TP14 type pressure transducers calibrated in the range of 0-250 mbar. In the linear working range (25-225 mbar) the transient time to the Heaviside function with amplitude of 50 mbar was lower than the microscale time of the pulsating pressure measured in the previous experimental investigations (Fiorotto and Rinaldo 1992a). The pressure transducers were connected to the taps by a rigid tube with the inner diameter of 4 mm and length of 0.2 m. A computer was linked to the transducers via a 16-channel capture card (United Electronic Instruments PD2MFH). Sampling was done by dasyLab 6.0 software. Since previous spectra analysis had shown that the dominant frequencies of pressure fluctuations were less than 30 Hz (Bowers and Tsai, 1969), a sampling rate of 100 Hz was adopted. The acquisition time was set equal to 30 min to obtain a large enough number of data in order to get accurate evaluations of the spatial correlation functions. The uniform distribution of the flow from the jet device was checked via standard deviation analysis of the fluctuating pressure in cross stream direction. Due to the statistical spatial homogeneity of the pressure field in this direction the same value of standard deviation is expected; an acceptable approximation smaller than ±3% was measured for all the runs.

Special care was taken to avoid from the possible effects of inaccuracy in the transducers signal due to the effect of entrapped air inside the measurement cells and tubes, vibration of the channel floor etc (Barjastehmaleki et al. 2016 a).

CFD

In order to investigate whether that numerical techniques can similarly yield a valid dataset for analysis of pressure fluctuations related to concrete slab and plunge pool stability, CFD models of the experimental flume setup were developed for comparison to the equivalent physical scenario.

All modelling was undertaken using Siemens Star CCM+, which is a CFD modelling package that is used worldwide by commercial and research organisations.

The CFD models featured an equivalent geometry setup to the physical flume experiment, with the exception of the use of a ‘free-slip’ wall boundary on the lid of the model domain to approximate the free surface within the flume experiment. This was done in order to limit the CFD simulations to a single phase, and is considered appropriate where the undulation of the free surface is not critical to the phenomena being analysed.

The time of the simulations was equal to 30 min and a sampling rate of 100 Hz was adopted to provide results comparable to the experiment.

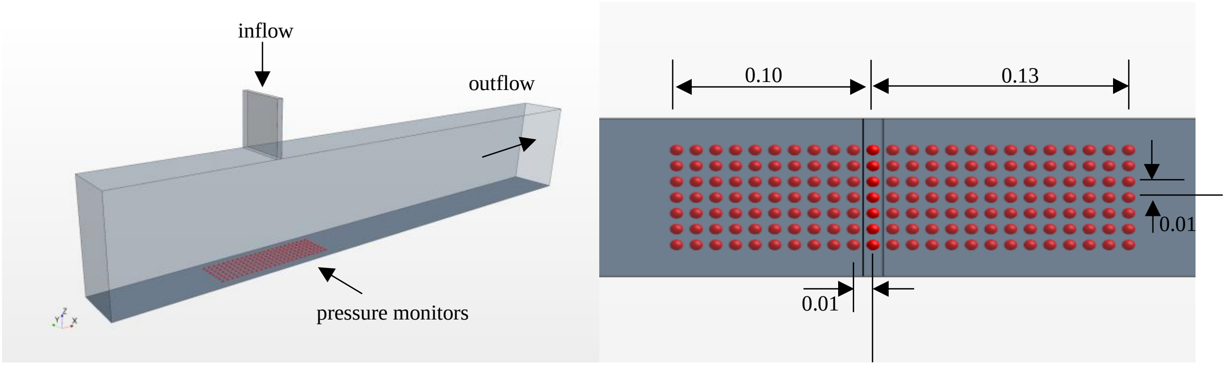


Fig 2 CFD model setup and test area with indication of the tap positioning

The CFD models employed the following boundary conditions:

- The inlet to the model was configured as a flow boundary condition in which a fixed discharge value is distributed over a cross sectional flow area;
- The outlet of the model was configured as a zero pressure boundary to allow the free exit of flow from the model domain;
- A no-slip shear wall condition was applied on the model base and back wall to simulate the effects of wall roughness;
- A free-slip shear wall condition was applied on the model sidewalls and lid.

The CFD simulations featured the following key physics models:

- Single phase flow (water only)
- Isothermal flow conditions
- Constant density and dynamic viscosity
- Volume of Fluid (VOF) approach;
- Gravity (9.81 m/s^2 in the negative Z axis);
- Implicit unsteady solver with an applied timestep of 0.001s; and
- Two alternate turbulence models:
  - K-Epsilon (k-ε) turbulence model; and
  - Detached Eddy Simulation (DES) turbulence model.

The volume mesh that was generated for the modelling task featured a variable cell size ranging from 2 mm to 4 mm and comprised approximately 1.6 million cells. Prism meshing (inflation of the mesh along wall boundaries) and targeted cell size refinement was utilised throughout the CFD model domain to ensure sufficiently fine mesh at all solid boundaries and key regions of interest. As a part of the CFD modelling, different mesh resolution were tested to cope for the best performance in precision/computational time. The error in outputs using smaller mesh size resulted less than 2%. Furthermore, a control mesh volume was used for the jet part, so that the jet was represented by 20 cells of 0.5 mm

Two simulations were run with two different turbulence models in the course of the study; one using a k-ε turbulence model and one using a DES turbulence model. It is well understood in hydrodynamics that k-ε turbulence models are typically insensitive to adverse pressure gradients (Ansys Fluent 12.0). As a result, in the context of concrete slab and plunge pool stability analysis such models are not able to correctly reproduce the detailed turbulent eddies and associated local pressure fluctuations at the water-rock interface, and can result in overly optimistic design evaluations for flows that separate from smooth surfaces.

Acknowledging these limitations, the decision was made to develop and run the first CFD model featuring k-ε turbulence as a ‘control’ case for comparison purposes, as the k-ε turbulence model is historically the most widely used turbulence model in industrial CFD applications. Robustness, economy and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow simulations.

A second version of the CFD model was then run using K-omega based DES turbulence model. The aim of the DES version of the CFD model was to produce a dataset of pressure fluctuations using numerical methods and validate these readings using the pressure fluctuation results from the equivalent physical flume experiment. A successful validation of the pressure results would thus demonstrate the potential for CFD (using the appropriate solver parameters) to provide a low-cost adjunct to physical models. It is realised, however, the quantification of pressure fluctuations related to plunge dissipation structures will require extensive scaled models (both physical and numerical) before considering this CFD approach as a general solution for evaluation of the pressure fluctuations in dissipation structures.

DES is a modification of a RANS model in which the model switches to a sub-grid scale formulation in regions fine enough for LES calculations. Regions near solid boundaries and regions where the turbulent length scale is less than the maximum grid dimension are assigned the RANS mode of solution. As the turbulent length scale exceeds the grid dimension, the regions are solved using the LES mode – in this way the grid size can be chosen with reference to the scale of turbulence relevant to the problem. This method of simulating turbulence has the potential to correctly reproduce the detailed turbulent eddies and associated local pressure fluctuations in the regions of interest, while remaining less computationally expensive than undertaking a full LES model.

Results and Discussion

The CFD results are discussed in terms of the following formats:

- Velocity vectors plotted through a plane along the centreline of the flume;
- Wall pressure plotted on the base of the flume; and
- Transient pressure results from the pressure monitors.

Velocity Results

The following observations are noted from a comparison of the velocity vector results (Fig. 3):

- The trajectory of the inlet jet is similar in both DES and k-ε models;
- The k-ε model only shows two large eddies; one between the rear wall and the jet, and one between the jet and the outlet. Conversely, the DES model shows finer details of the turbulence and small eddies throughout the model domain;
- Higher velocities can be observed in the wall jet region in the DES model compared to the k-ε model.

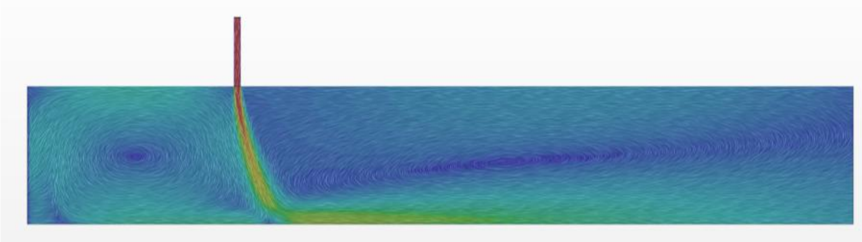
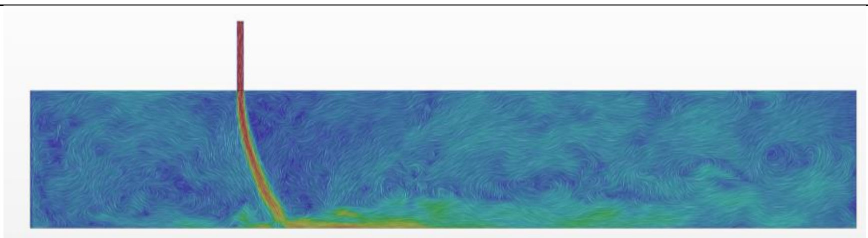
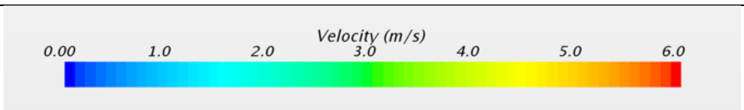
k-ε CFD model	
DES CFD model	
Legend	

Fig 3 Comparison of the velocity vectors

Wall Pressure Results

- The k-ε model shows a symmetrical & graduated pressure distribution along the base of the flume (Fig. 4). This is not considered realistic given the 3D violent nature of the turbulence;
- The DES model gives a better estimation of the pressure distribution on the base of the flume. This is discussed further in transient pressure results section;
- It is noted that the peak wall pressure recorded in the DES model is approximately 3 times greater than the peak wall pressure recorded in the k-ε model.

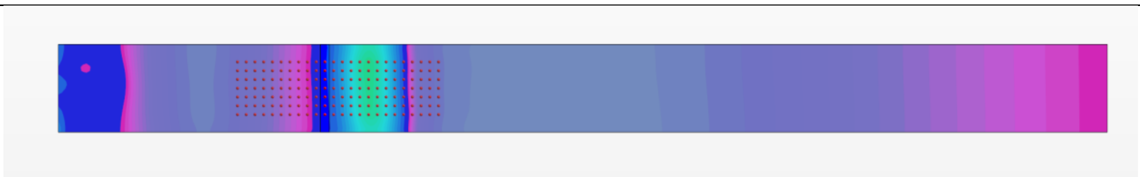
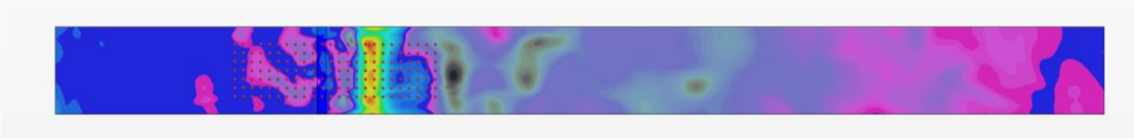
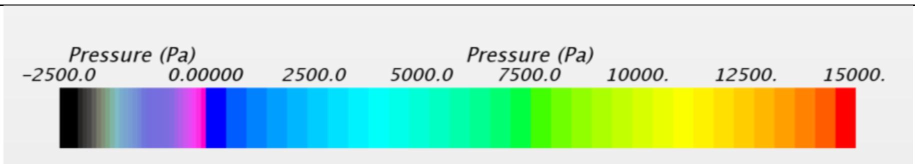
K-Epsilon CFD model	
DES CFD model	
Legend	

Fig 4 Comparison of the wall pressure results

Transient Pressure Results

Using experimental setup, for the ratios H/y_e equal to 20 and discharge equal to 20 l/s, the fluctuating pressure at the bottom was measured; with the assumption of stationary in time, the following statistical properties were evaluated:

The mean value of the pressure (Barjastehmaleki et al. 2016a):

$$\bar{p}\,(x,y) = \lim_{T\to\infty}\frac{1}{T}\int_0^T p(x,y,t)dt \qquad (1)$$

where *p(x,y,t)* = pressure measurement at time *T* in (*x*,*y*) and *T* = acquisition time. The space-time covariance function *R*:

$$R(x,\xi,y,\eta,\tau) = \lim_{T\to\infty}\frac{1}{T}\int_0^T p"(x,y,t)p"(x+\xi',y+\eta,t+\tau)dt \qquad (2)$$

where ξ = longitudinal distance from a pivot point in (*x*,*y*), η = transversal distance from the pivot point and *p"*(*x*,*y*,*t*) = *p* (*x*,*y*,*t*) - $\bar{p}$(*x*,y). Noting that the pressure field is statistically stationary in space in the transversal direction the coefficient *R* is independent of the *y* direction, so that the correlation surface ρ at any pivot point (*x*,*y*) as a function of ξ and η, is defined by:

$$\rho(x,\xi,0,\eta,0) = \frac{R(x,\xi,0,\eta,0)}{\sigma_p(x,0)\sigma_p(x+\xi,\eta)} \qquad (3)$$

where σ_p = standard deviation of the pressure fluctuations.

In Fig. 5, the pressure maximum and minimum pressure fluctuations deviation from the mean dimensionless with the jet kinetic head are reported. These quantities are:

$$c_p^{\pm} = \frac{\Delta p}{\gamma v_e^2/2g} \quad (4)$$

where Δp is the value of the maximum pressure deviation from the mean measured in 24 hours of acquisition time. The c_p^{+} coefficient is related to the maximum pressure and it is positive, while the c_p^{-} coefficients, relate to the minimum pressure, is negative (Fig. 5). Values of $c_p^{\pm}$ related to the results of the flume studies of a jet impinging in a flume, K epsilon CFD model and DES CFD model are presented in Fig 5.

The Values of $c_p^{\pm}$ evaluated by DES CFD are in good agreement with PHM results, while K epsilon model underestimate the peak values that can cause underestimation of uplift force in dissipation structures and failure of structures.

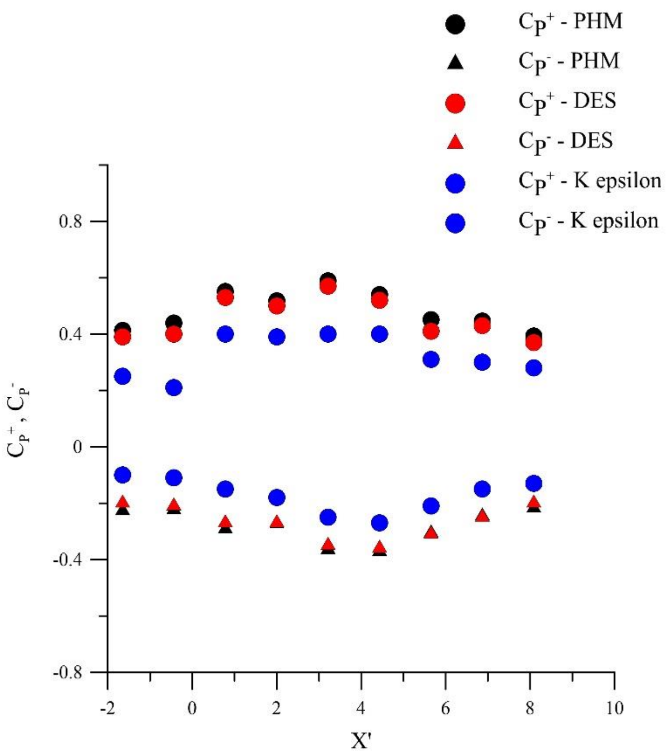
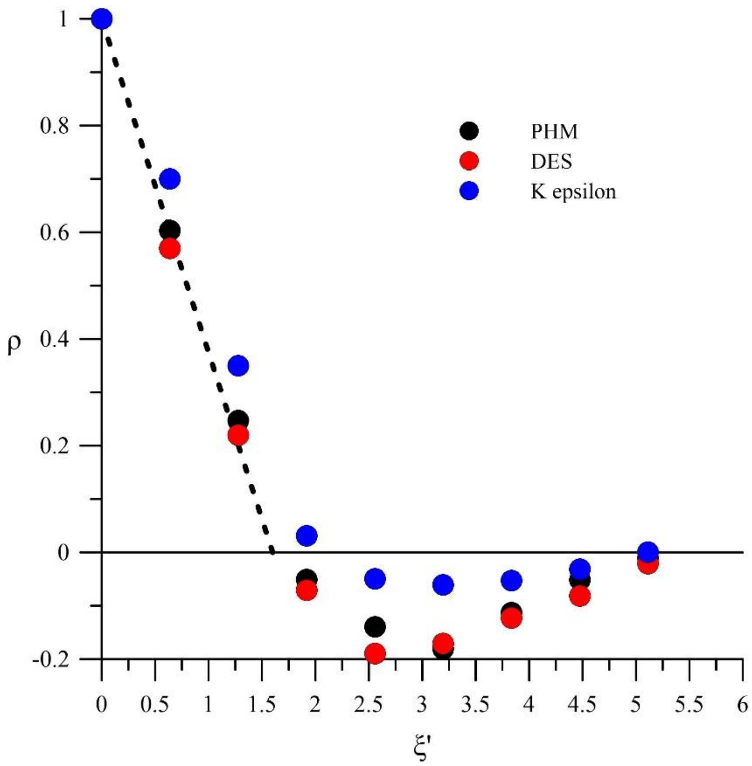
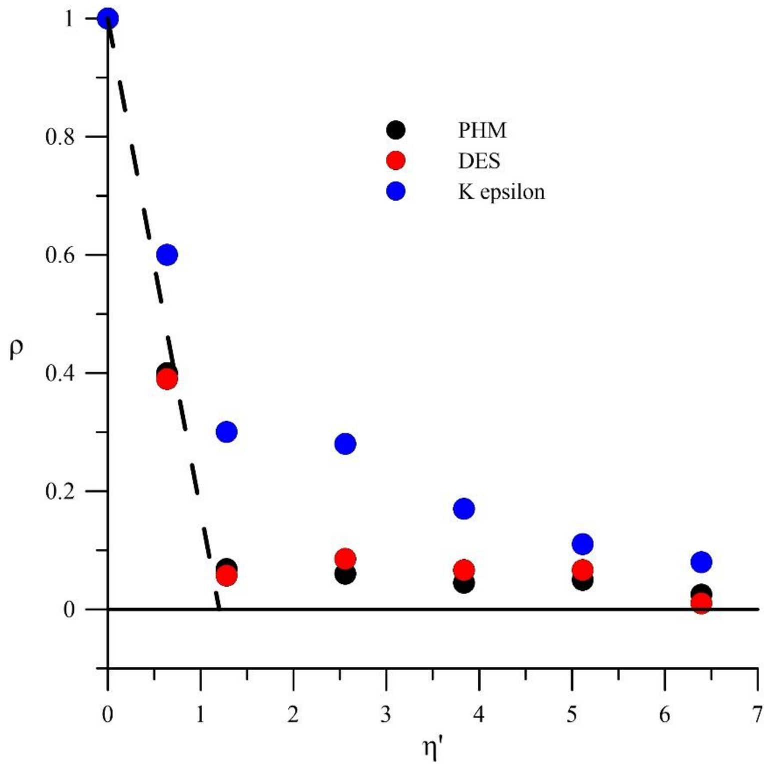


Fig. 5 Experimental bottom fluctuating pressure parameters $c_p^{\pm}$

The evaluation of stability of Rock plunge pool and concrete blocks in dissipation structures requires the calculation of the correlation function in the plane under the jet impact/hydraulic jump (as well as geology in the case of plunge pools). Fig. 6a and 6b show the longitudinal and transversal pressure correlation function under the jet impact region, respectively. The comparison of the results of PHM with DES CFD model and K epsilon CFD model shows that there is a good agreement between the results of PHM and DES CFD and highlights the capability of DES CFD approach in evaluation of the correlation function of the pressure fluctuations (similar values of zero crossing point for DES model and PHM –Fiorotto and Rinaldo 1992a and 1992b). Fig 6a and 6b show a significant difference between results of K epsilon CFD model and PHM.



a)



b)

Fig. 6 Experimental bottom fluctuating pressure correlation function: a) stream wise direction b) cross wise direction

Conclusion

The following conclusions can be drawn from the present study:

- The experimental results are in agreement with the DES CFD results and this highlights the capability of the new CFD approach in evaluation of the pressure fluctuations in this lab study case.
- The comparison of the results of k-ε CFD model (typically used in industry) with experimental results shows the inaccuracy and limitation of RANS models in evaluation of the pressure fluctuations downstream of large dams.

The paper presents the preliminary results on evaluation of pressure fluctuations using new DES CFD approach. A full paper describing this new CFD method including results, applications of the approach in real life case, errors etc is under review and for publication in Journal of Hydraulic Engineeting-ASCE.

References

Ansys Fluent 12.0 (2009) "Theory guide"

Asadollahi, P., Tonon, F., Federspiel, M., and Schleiss, A., (2011). Prediction of rock block stability and scour depth in plunge pools." *J. Hydraul. Res*., 49 (6) 750 – 756 .

Barjastehmaleki, S., Fiorotto, V., and Caroni, E. (2016a). "Spillway Stilling Basins Lining Design via Taylor Hypothesis." *J. Hydraul. Eng.*, 10.1061/(ASCE)HY.1943-7900.0001133, 04016010.

Barjastehmaleki, S., Fiorotto, V., and Caroni, E. (2016b). "Design of Stilling Basin Linings with Sealed and Unsealed Joints." *J. Hydraul. Eng.*, 10.1061/(ASCE)HY.1943-7900.0001218, 04016064.

Bollaert, E., and Schleiss, A. (2003). "Scour of rock due to the impact of plunging high velocity jets. Part I: A state-of-the-art review ; Part II: Experimental results of dynamic pressures at pool bottoms and in one - and two-dimensional closed-end rock joints." *J. Hydraul. Res.*, 41(5), 451–480.

Bowers, C. E., and Tsai, F. H. (1969). "Fluctuating pressure in spillway stilling basins." J. Hydraul. Div., Vol.95, HY6, 2071-2080.

Fiorotto, V. and Rinaldo, A. (1992a). "Fluctuating uplift and lining design in spillway stilling basins." *J. Hydraul. Eng.*, 118(4), 578-596.

Fiorotto, V. and Rinaldo, A. (1992b). "Turbulent pressure fluctuations under hydraulic jump.", *J. Hydraul. Res.*, 30(4), 499-520.

Fiorotto, V., Barjastehmaleki, S., and Caroni, E. (2016). "Stability Analysis of Plunge Pool Linings." *J. Hydraul. Eng.*, 10.1061/(ASCE)HY.1943-7900.0001175, 04016044.

Melo, J. F., Pinheiro, A. N. and Ramos, C. M. (2006). "Forces on plunge pool slabs: influence of joints location and width." *J. Hydraul. Eng.*, 132(1), 49-60.