Determine of velocity field with PIV and CFD during the flow around of bridge piers

I. Mistrová^{1,a}, D. Picka^{1,b}

¹Institute of Water Structure, Faculty of Civil Engineering, Brno University of Technology, 602 00 Brno, Veveří 331/95

Abstract. The article describes the processing of specific junior research FAST-J-11-51/1456 which dealt with physical and CFD of the velocity field during the flow around of bridge piers. Physical modelling has been carried out in Laboratory of water management research in Institute of Water Structures in Brno University of Technology - Faculty of Civil Engineering. To measure of the velocity field in profile of bridge piers were used laser measuring method PIV (Particle Image Velocimetry). The results of PIV served as a basis for comparing experimental data with CFD results of this type of flow in the commercial software ANSYS CFX.

1 Introduction to the issue

^aIf we place in the path of the flowing fluid barrier, the fluid begins to wrap her. The most famous examples of real objects include wrapping, such as water in a river flows around the piers or flowing air flows around objects and bodies on the earth's surface (air wrapping around static objects and cars or aircraft). The body wrap also occurs when the fluid at rest and the body moves in the space - as a ship floating in the sea. When you wrap any objects created water flow tangential stress strain their surface, which gradually degrades the material from which they are formed. Wrap may be either pressure or vacuum. During this hydraulic phenomenon leads to boundary layer separation, which is a thin layer along flowarounding object, and the wakes of different shape in dependence on the flow velocity (a type of flow - laminar and turbulent) and the shape of the body itself. In the boundary layer creates a strong deceleration current, which in some cases leads to a discontinuity of the current and the formation of vortices. Flow separation and wake are always associated with a large loss of energy and intense pulsation speeds and pressures, and therefore we try to remove it and most appropriate treatment shaped structures [1].

2 Objectives of the project

The aim of the project was measured by PIV velocity field for three selected shapes gridiron pillars and three angles of deflection of the pillars of the longitudinal axis of flow. The pillars are arranged in the profile trough so that they are always at least three pillars located in the central part of the profile above the glass tray at the bottom of the hole (Fig. 1).

Fig. 1. 3D model placement pillars in luminous efficacy trough

The project was divided into two parts, and physical and numerical modeling. The findings and outputs obtained from physical modeling served to show basic characteristics of the flow along the pillars, the pillars of the influence of the shape and influence the direction of deflection of the pillars on the flow in the space behind the pillars. The main objective of this research project was to obtain enough quality data for the initial calibration and subsequent verification of CFD model.

^a mistrova.i@fce.vutbr.cz

^b daniel.picka@seznam.cz

3 Hydraulic trough, models of piers and measuring technology

3.1 Hydraulic trough - measuring track

Model bridge piers was installed in a hydraulic flume width of 1 m, length 12 m and maximum achievable flow $160 \text{ l}\cdot\text{s}^{-1}$. Construction is reinforced concrete trough with vertical glass walls. Inflow of water into the trough was channeled through perforated plate with circular holes that measured in specific area to ensure, where possible, the most consistent flow and created a stable flow profile. Drain water from the trough was controlled using segmented plate cap fitted at the end of the trough.

3.2 Models of piers

Overall, it was necessary to measure the velocity fields for three shapes gridiron pillars (Fig. 2), the three angles of deflection of the pillars of the longitudinal axis and the filling stream: 0° , 30° and 45° , and for three flow rates: minimum (small) - from 1.3 to $1.6 \text{ l}\cdot\text{s}^{-1}$, medium (middle) - from 24.0 to $24.5 \text{ l}\cdot\text{s}^{-1}$ and the largest (large) - from 40 to $43 \text{ l}\cdot\text{s}^{-1}$. Flow rate was dependent on the water level in the settling trough levels in the range of 245 to 255 mm from the bottom of the trough - see table 1 and table 2.

Fig. 2. Scheme of the three pillars of shapes gridiron

For fixing pillars into the flowing water in the trough was made structure consisting of stainless steel with welded hollow metal around the edges, increasing its rigidity. The shorter side of plate were also welded to the vertical hollow with holes for mounting screws because of the possibility strutting beam between gutter walls (Fig. 3).

Fig. 3. View of the spacer structure and method of fixing the glass wall trough

Table 1. Overview of the measured states - flow rates, type of gridiron and depth of water in the trough - recorded during the measurement

Shape gridiron of pier	Angle of rotation [°]	Flow [$\text{l}\cdot\text{s}^{-1}$]			Water level [mm]
		Label	Value set	Actual value	
Rectangle	45	large	42,5	42,5	245
		middle	24	24	255
		small	1,3	1,3	250
Rectangle	30	large	43	42,9	253
		middle	24	24	257
		small	1,2	1,3	245
Rectangle	0	large	44	44	245
		middle	25	25	245
		small	1,5	1,5	248
Semicircle	45	large	43	42,7	250
		middle	24,5	24,5	258
		small	1,6	1,6	253
Semicircle	30	large	43	43,1	250
		middle	25	24,9	243
		small	1,5	1,5	250
Semicircle	0	large	44	44,2	253
		middle	24	24	245
		small	1,6	1,6	255
Spike	45	large	40	40	250
		middle	24,5	24,4	250
		small	1,6	1,6	253
Spike	30	large	40	39,6	250
		middle	24,5	24,7	252
		small	1,6	1,5	249
Spike	0	large	41	41,2	253
		middle	24,5	24,4	251
		small	1,6	1,6	250

Table 2. Load cases - the values of Re, the corresponding velocity and flow rate - calculated before physical modeling

Re [-]	v [$\text{m}\cdot\text{s}^{-1}$]	Q [$\text{m}^3\cdot\text{s}^{-1}$]	h [m]
2000	0,00261	0,00065	0,25
75000	0,09803	0,02451	0,25
122500	0,16011	0,04003	0,25

3.3 Measuring technique used to measure the velocity field - measuring kit for PIV

The main part of this kit includes a double pulse laser type Nd: YAG, manufacturer New Wave Research Gemini PIV type with adjustable repetition rate of 0-15 Hz and a maximum energy of 120 mJ in a flash and camera FlowSense 2M, the Danish manufacturer Dantec resolution 1600×1200 pixels. To synchronize all elements of the set in the measurement was used Central System Controller Hub also made by Dantec. In the selected horizontal cut channel set was recorded 100 instantaneous velocity field states with a sampling frequency of 1 Hz in the first set and a frequency of 5 Hz in the second set of measurements. Estimating the value of the expanded uncertainty water velocity in the measured field, with a confidence level of 95% is less than 10%. PIV method is not intended to measure the point velocity, but to detect the spatio-temporal context. In Figure 4 it is possible to see the overall view of the measurement technique for measurement by PIV spaced around the track density in the laboratory.

Fig. 4. Overview of PIV measurement technology deployed

4 Velocity field measurements in space wrapping around the pillars of flowing water

Before measurement of velocity field by PIV were carefully considered ways to illuminate space when wrapping the pillars of the flowing water and shooting options illuminated field camera to the measurement results as much as possible to account for the fact that the flow in the selected area. It was decided that the area of interest will be recorded in a single horizontal cut (incision parallel with the bottom trough) and at a distance of about 125 mm from the bottom of the trough. Illuminate the area of interest was carried out through a mirror mounted in a special sealed aquarium which could be placed directly into the gutter flooded area (downstream for a number of pillars). From the top of the aquarium open to him to be able to insert the head Laser source (Fig. 5). Scanner for laser illuminated area of interest (horizontal cut) was deposited on the floor of the laboratory under the bottom of the trough. Image capture area of interest, therefore, takes place through a mirror set sloping (ie the inclination angle of 45° from the bottom of the trough) under the glass hole in the bottom of the trough.

Fig. 5. View of the aquarium with a mirror and the general disposition of the model in the trough

5 Results of physical measurements using PIV

After getting the huge amount of data during the measurement in the laboratory, it was decided to evaluate the collected data. Two-images were further processed using the software FlowManager v4.71. Of vector fields, which are the output PIV measurement methods laid down in a horizontal plane (see Chapter 3) can best track the location of flow separation of water from the wall surface of the reference object of flow-arounding pillar. Fig. 6a shows the time-mean values of image projections of the velocity vector in the horizontal plane with a rectangular gridiron pillars, pillars of diversion from the longitudinal flow direction by 30° and the flow is $1.3 \text{ l}\cdot\text{s}^{-1}$. We can see the range wake in the space around the pillars and the stream behind the pillar. At such low flow rates a reverse flow is very small.

Fig. 6a. Output from PIV - rectangular gridiron, the angle 30° and flow rate $Q = 1.3 \text{ l}\cdot\text{s}^{-1}$

Fig. 6b. Output of the CFD - rectangular gridiron, the angle of 30° and $Q = 1.5 \text{ l}\cdot\text{s}^{-1}$ – detail

In Figure 7 we can see the size and scope of the rectangular gridiron wake pillars, pillars of diversion from the longitudinal flow direction by 30° and the flow is 42.9 l.s^{-1} . Here we see, compared to the vector field in Figure 6a, much clearer in the wake flow back along the side wall pillars and the area downstream of it. The maximum speed measured in the direction of flow (flow at 42.9 l.s^{-1}) was 0.45 m.s^{-1} and the velocity of reverse flow in the wake 0.19 m.s^{-1} . For comparison, at the flow rate 1.3 l.s^{-1} , the value of the speed of 0.018 m.s^{-1} and 0.008 m.s^{-1} .

Fig. 7a. Output from PIV - rectangular gridiron, the angle 30° and flow rate $Q = 42.9 \text{ l.s}^{-1}$

Fig. 7b. Output of the CFD - rectangular gridiron, the angle of 30° and $Q = 40.0 \text{ l.s}^{-1}$ – detail

Other images (Fig. 8a and 9a) show time-mean values of image projections of the velocity vector in the horizontal plane at a semicircular pillars gridiron, deflection of the pillars of the longitudinal flow direction by 45° and flow values 1.6 l.s^{-1} and 24.5 l.s^{-1} . We see that when the flow is small (1.6 l.s^{-1}) is as follows swim for deviating pillar wider than in the case of large flow (24.5 l.s^{-1}). The values of maximum velocity measured in the direction of flow were to: 1 small flow rate 0.018 m.s^{-1} , the second for larger respectively mean flow of 0.29 m.s^{-1} . Measured values of the maximum velocity in the wake backflow were to: 1 small flow rate 0.006 m.s^{-1} , the second for larger respectively mean flow of 0.08 m.s^{-1} .

Fig. 8a. Output from PIV - semicircular gridiron, the angle 45° and flow rate $Q = 1.6 \text{ l.s}^{-1}$

Fig. 8b. Output of the CFD - semicircular gridiron, the angle of 45° and $Q = 1.5 \text{ l.s}^{-1}$ - detail

Fig. 9a. Output from PIV - semicircular gridiron, the angle 45° and flow rate $Q = 24.5 \text{ l.s}^{-1}$

Fig. 9b. Output of the CFD - semicircular gridiron, the angle of 45° and $Q = 25.0 \text{ l.s}^{-1}$ - detail

The last figures in Chapter 4 (Fig. 10a and 11a), it is possible to see the image projections of the time-mean values of the velocity vector in the horizontal plane at the pointed pillars gridiron, deflection of the pillars of the longitudinal flow direction by 30° and flow values $1.5 \text{ l}\cdot\text{s}^{-1}$ and $39.6 \text{ l}\cdot\text{s}^{-1}$. The values of maximum velocity measured in the direction of flow were to: 1 small flow rate $0.014 \text{ m}\cdot\text{s}^{-1}$, the second for the largest flow rate $0.33 \text{ m}\cdot\text{s}^{-1}$. Measured values of the maximum velocity in the wake backflow were to: 1 small flow rate $0.006 \text{ m}\cdot\text{s}^{-1}$, the second for the largest flow rate $0.16 \text{ m}\cdot\text{s}^{-1}$.

Fig. 10a. Output from PIV - spike gridiron, the angle of 30° and flow rate $Q = 1.5 \text{ l}\cdot\text{s}^{-1}$

Fig. 10b. Output of the CFD - spike gridiron, the angle of 30° and $Q = 1.5 \text{ l}\cdot\text{s}^{-1}$ – detail

Fig. 11a. Output from PIV - spike gridiron, the angle of 30° and flow rate $Q = 39.6 \text{ l}\cdot\text{s}^{-1}$

Fig. 11b. Output of the CFD - spike gridiron, the angle of 30° and $Q = 44.0 \text{ l}\cdot\text{s}^{-1}$ - detail

6 CFD modeling

6.1 Numerical model pillars

Data obtained from measurements on a physical model served as the entrance and boundary conditions for calibration and simulation of flow in the numerical model. Numerical calculations were performed in the solver Ansys CFX, which is part of the software package ANSYS Workbench.

2D geometry model created in Autodesk AutoCAD software environment served as a basis for a discrete area. Computing and discrete area were created in programs that are part of the ANSYS Workbench.

The task was schematized as a 2D problem in a vertical plane through the longitudinal axis in the direction of flow. CFX solver calculates 3D tasks in general, so it had to be set to the minimum width of 0.003 m computational domain, while the ceiling was set in the area of discrete symmetry condition. The described type of problem falls within the turbulent flow regime. For his good agreement with the results of numerical modeling of measurement was used for solving turbulence generally known Shear Stress Transport (SST) model. It was derived Mentera in 1994. In the modified model (compared to the standard $k-\omega$) the definition of the turbulent viscosity, which takes into account the transport of turbulent shear stress. SST is one of the so-called two-layer models that are used in the inner shear area (ie sublayer and logarithmic region) $k-\omega$ model and in the outer region of the shear layer standard $k-\omega$ model. For the thin shear layers makes this change only small differences compared to the standard approximations.

Turbulent $k-\omega$ model is based on the averaged Navier-Stokes equations, known as RANS, the closure equations is performed on the basis of the principle of Boussinesq turbulent eddy viscosity.

Computational domain of size $1.0 \times 1.91 \text{ m}$ was divided into three sub required by grid computing density (Fig. 12). The area around the pillars were covered densely. Discretization of the computational domain was using tetrahedra, whose size ranged from 2.0 mm to 7.0 mm . Total computational grid contains 383,000 nodes and

1,141 million cells. The field was applied boundary conditions: wall (wall - no slip) at the bottom of the pillars and the longitudinal walls of the trough. At the inlet was administered condition known velocity obtained from physical measurements using PIV. The outlet was applied hydrostatic pressure distribution corresponding to the discharge space. Reference pressure of 0.1 MPa was assumed [4].

Fig. 12. Computing area for CFD simulation

6.2 The results of numerical modeling

Outcomes of CFD (streamlines from numerical module) and physical measurements of velocity field in the wrap pillars are similar (see Figure 6b to 11b). Other outputs from numerical modeling can be seen in Figure 13 to 15. CFD method used proved to be satisfactory. In the future, it would be a good idea to wrap the pillars numerically modeled in 3D space. This method is particularly challenging for several computing power and time [4].

Fig. 13. Output from CFD - velocity field - rectangle gridiron, the angle of 30° and flow rate $Q = 1.5 \text{ l}\cdot\text{s}^{-1}$ – detail

Fig. 14. Output from CFD - velocity field - semicircle gridiron, the angle of 30° and flow rate $Q = 1.5 \text{ l}\cdot\text{s}^{-1}$ - detail

Fig. 15. Output from CFD - velocity field - spike gridiron, the angle of 30° and flow rate $Q = 44.0 \text{ l}\cdot\text{s}^{-1}$ - detail

7 Conclusion

Contactless measurement of secondary ingredients and pulse velocity flow by PIV in the specified horizontal section are expected to allow a detailed study of current fields, mainly in the area of flow separation from the surface of the pillars and reverse flow and vortex fields in the area downstream of the pillars of various shapes and angles of deflection from the longitudinal direction of flow.

Measurement of the velocity field was to wrap the pillars of the model used has already been completed and the model of trough space completely dismantled and stored. Complete final report of specific solutions junior grant No. FAST-J-11-51/1456 titled "Mathematical and physical modeling of the velocity field in the wrap bridge piers" is prepared and stored in the archives of WRL-IWS-FCE-BUT. In case of further possible research problem of this model will be re-fitted to the hydraulic chute and perform other measurements related to measurements carried out in the framework of this research project. In March 2012 an application for a specific junior grant, which would build on the research already completed.

The flow, ie. Wrap flowing water around bridge piers, was in terms of time and required simplification solved as a 2D problem. However, it is certain that this is actually a three-dimensional character of the flow, which is a very difficult test case for numerical simulation.

After comparing the results of the CFD results with measurements by PIV PIV method is proved to be a suitable solution for obtaining the necessary calibration and verification of numerical model. The data obtained can be used to specify the initial conditions in the numerical model in solving complex problems of hydrodynamic type. Especially when dealing with hydraulic phenomena with free surface. The disadvantage of this solution is a large demand on qualified personnel. Best method requires two experienced workers. One should have a long experience with the measurement and evaluation of data from the PIV. The second should understand the problems of mathematical and numerical modeling, and should have knowledge of numerical computing software.

References

1. M. Vlachová, Techmania. Background in Physics, Mechanics, Czech Republic, (2008).
2. I. Mistrová, P. Zubík, Measurement of velocity field at flow around of bridge piers by method PIV, Proceedings of the Symposium Hydrotechnik 2011, Slovak University of Technology in Bratislava, Bratislava, (2011).
3. I. Mistrová, Measurement of velocity field at flow around of bridge piers by non-contact optical method PIV, Proceedings of conference Juniorstav 2012, Brno University of Technology, Brno, (2012).
4. I. Mistrová, P. Zubík, Mathematical and physical modeling of velocity field during the flow around of bridge piers, Final report of a specific solution Junior Grant No. FAST-J-11-51/1456, Brno, (2012).