ENGINEERING MECHANICS 2004
NATIONAL CONFERENCE
with international participation
Svratka, Czech Republic, 10 - 13 May 2004

# EXPERIMENTAL AND NUMERICAL MODELLING OF FLOW AROUND THE OBSTACLES PLACED ON A CHANNEL BOTTOM

**B. Hoření*, Z. Chára***

**Summary:** *The paper deals with an application of integral equations on a wake flow. Two square cylinders oriented perpendicularly to flow direction are placed on the bottom of an open channel. The results of integral method are compared with LDA measurements and with Fluent k-ε simulation. Besides the averaged results that offers Fluent simulation, the integral method can also simulate dynamical properties of the flow.*

## 1. Introduction

River engineering nowadays utilises numerous numerical models to estimate flow discharges and water surfaces profiles in river systems like HEC-RAS. The computational procedures are usually based on the solution of one-dimensional energy equation where the energy losses are evaluated through friction coefficient. When the detail descriptions of flow around hydraulic objects are needed, then it is necessary to utilise more sophisticated approach like CFD. In this paper we have focused on a numerical simulation based on an application of integral equations. The simulations are tested on a wake flow behind two square cylinders placed on the bottom of the channel.

## 2. Wake-flow modelling

Wake flow is characterised by a very complicated time-dependent structure. The application of complete differential equations of non-stationary flow field solved with help of an appropriate mathematical model (finite volume or finite element methods, etc.) represents one of the possible approaches how to model wake flow structure. However, this approach requires high-efficient computer technique and the results obtained have hitherto strongly differed from the experimental ones for high Reynolds numbers.

The second approach is based on the solution of the simplified equations describing a flow field. It assumes that significant viscosity effects are restricted to a relatively narrow region of a boundary layer in the close proximity of the walls. Boundary layer separation profiles surfaces exhibiting discontinuous velocities in flow field. These surfaces are unstable, deformable and are located in wake-flow region. A model using flow field discontinuities usually corresponds to the reality the better the higher a Reynolds number is,

* Ing. Bohumír Hoření CSc., Ing. Zdeněk Chára, CSc. Institute of Hydrodynamics ASCR, Pod Patankou 30/5, 166 12 Prague 6, Czech Republic;
Phone: +420-233109011, Fax: +420-233324361, E-mail: horeni@ih.cas.cz

(Batchelor, (1970), Loicyanskii, (1970)).

Models of the potential flow with free surfaces of discontinuity are based on the second Green Identity written in a special form

$$4\pi G = \int_S \left(\frac{1}{r}\frac{\partial G}{\partial n} + G\frac{\mathbf{r}\circ\mathbf{n}}{r^3}\right)\,\mathrm{d}S - \int_V \frac{1}{r}\Delta G\,\mathrm{d}V,$$

where $G$ is an unambiguous function continuous up to second derivative inside a domain $V$ bounded by the surface $S$ with external normal $\mathbf{n}$. A vector $r$ connects a point inside the domain with a point on the surface $S$. Function $G$ represents a velocity potential $\varphi$ in a flow field.

The boundary can also be placed along both sides of free surfaces of discontinuity in the flow field. An integral equation for a potential on the boundary $S$ can be obtained by a suitable arrangement of the second Green Identity under preservation of boundary conditions. In the incompressible domain the solution can be simplified by neglecting the volume integral containing divergence.

The integral equation for determination of the potential has been widely used solution of numerous problems, (Hess & Smith, (1964), Argyris & Scharpf, (1970)), lately especially in a variant of boundary elements. However in geometrically complicated 2D problems its applicability is questionable with respect to the ambiguity of the potential in multiple connected regions. In 3D problems the discontinuity of potential represents another source of complications of the application of the numerical models. For determination of a motion of free planes of discontinuity it is necessary to obtain velocity as a derivation of the potential. To attenuate above mentioned problems it is suitable to transform the original equation to the equation expressing velocity $\mathbf{grad}\varphi$. In the incompressible domain it results in the following equation

$$4\pi\,\mathbf{grad}\,\varphi = \int_S \mathbf{grad}\,\left(\frac{1}{r}\right)\frac{\partial\varphi}{\partial n}\,\mathrm{d}S + \int_S \varphi\,\mathbf{grad}\,\left(\frac{\mathbf{r}\circ\mathbf{n}}{r^3}\right)\,\mathrm{d}S.$$

This relation forms a basis for the method presented in this paper. Using this relation it makes possible to define more accurately also methods derived by other procedures like superposition of flow, singularity methods, methods of discrete or continuously scattered vortexes (Prandtl (1918), Lavrentev (1932), Belocerkovkij & Nist (1978)), which are, in combination of boundary layer solutions, successfully applied to complicated technical problems, Drela (1989), Bal (1999).

In this paper the integral equation in the vector form is used for modelling the wake behind the obstacles under a water free surface. The obstacles have sharp edges this enables sufficiently precise prediction of separation points without otherwise necessary detailed solution of the boundary layer.

## 3. Mathematical 2D model of flow around obstacles

The flow patterns in a horizontal hydraulic flume of a rectangular cross section with prismatic obstacles across the whole width of a channel can be simplified and modelled as 2D flow in a vertical cross section. The initial equation has a similar form to a 3D flow but it is not completely the same. The following relation fulfilled in an arbitrary point inside the region

$$2\pi\,\mathbf{grad}\,\varphi = \int_S \mathbf{grad}\,\left(\ln\frac{1}{r}\right)\frac{\partial\varphi}{\partial n}\,\mathrm{d}S + \int_S \varphi\,\mathbf{grad}\,\left(\frac{\mathbf{r}\circ\mathbf{n}}{r^2}\right)\,\mathrm{d}S.$$

The solution subjects on the boundary to the condition $\mathbf{grad}\,\varphi \circ \mathbf{n} = v_n,$ where $v_n$ denotes a velocity component on the boundary in the normal direction (including possible mass transport across the boundary).

Distribution of the potential $\varphi$, or the value $\partial\varphi/\partial n,$ on the body surface can be approximated by suitably chosen functions. The Hermite polynoms were used ensuring continuity of the potential and its first derivation at the end points of the individual boundary segments. If the boundary segment contains an internal point where a jump change of potential occurs, the term $x$ was replaced by $\operatorname{sign}(x)$ in the approximating polynom.

Boundary geometry was approximated in a similar manner. Hermite polynoms in complex variable ensure continuity of a tangent of smooth boundary at the end points of the segments. If the boundary contained angle points with jump change of normal direction the segment in the vicinity of this point was obtained by the conformal mapping of the smooth segment. This approach considerably eliminates a necessity of an abnormal shortening of the elements near the edge that is otherwise necessary for precise description of sharp changes of the potential near the edge.

If a sufficient number of control points, where the integral equation for potential gradient and boundary conditions are satisfied, is chosen the method results in a solution of a system of linear equations for unknown coefficients of Hermite polynomials approximating the solution.

In general there is a possibility to model the planes of discontinuity (vortex planes) in the flow field by continuos functions, Hoření (1976). This procedure is applicable only for simple geometries of vortex planes arising for example behind aircraft wings or for initial stage of an development of breaking away vortex planes. For a general problem it is practically impossible to create an algorithm solving in detail continuous vortex planes and their mutual interaction as well as interaction with overflowed bodies in an acceptable computational time. Therefore a function constant on the individual segments was applied to approximate the potential on vortex planes in the wake. The velocity induced in a domain point by a closed vortex line of intensity Γ satisfies the following relation (Loicyanskii (1970))

$$\mathbf{v} = -\frac{\Gamma}{4\pi}\,\mathbf{grad}\int_\sigma \frac{\partial}{\partial n}\left(\frac{1}{r}\right)\mathrm{d}\,\sigma,$$

where $\sigma$ is an arbitrary open plane bounded by the vortex thread. This equation applied to the approximation of the potential by the partially constant function leads to the mathematical model where the vortex plane is substituted by a grid of discrete vortices. Such models (derived from the idea of superposition of flow patterns without any relation to the distribution of the potential on the vortex plane) are used for example in the aerodynamics of aircraft wing for many years, Belocerkovkij (1965). A massive development of the computational techniques have enabled in recent years a return to the methods of vortex grids at quantitatively higher level. They are frequently presented as Discrete Vortex Method and they use the Lagrangian framework to solve Euler and Navier-Stokes equations, Clarke & Tutty (1994), Takeda et al.(1999).

The principal difference of modern methods of discrete vortices in comparison with classical approach is an attempt to include into the solution not only convective vortex movement but also a diffusive part of the Navier-Stokes equations which are usually used in the form containing vorticity $\mathbf{\Omega} = \mathbf{rot}\mathbf{v} = \nabla \times \mathbf{v}$. It can be derived the relation, Loicyanskii (1970)

$$\frac{\partial\mathbf{\Omega}}{\partial t} + (\mathbf{v}\circ\nabla)\,\mathbf{\Omega} = (\mathbf{\Omega}\circ\nabla)\mathbf{v} + \nu\nabla^2\mathbf{\Omega},$$

or in the Lagrangian form

$$\frac{\mathrm{d}\mathbf{\Omega}}{\mathrm{d}t} = (\mathbf{\Omega} \circ \nabla)\mathbf{v} + \nu\nabla^2\mathbf{\Omega},$$

In a plane flow the Lagrangian form is simplified due to the reciprocal perpendicularity of the vectors $\mathbf{\Omega}$ and $\mathbf{v}$ to the form

$$\frac{\mathrm{d}\mathbf{\Omega}}{\mathrm{d}t} = \nu\nabla^2\mathbf{\Omega}.$$

The well-known solution of this equation is a diffusion of the plane vortex in the course of time. The result is a distribution of the tangential velocity component induced by a vortex of circulation $\Gamma$ in a dependence on radius $r$ and time $t$

$$v = \frac{\Gamma}{2\pi r}\left(1 - e^{-r^2/4\nu t}\right).$$

Classical vortex methods neglected the diffusion term, this corresponds to the limiting case $\nu \to 0$, or $\mathrm{Re} \to \infty$.

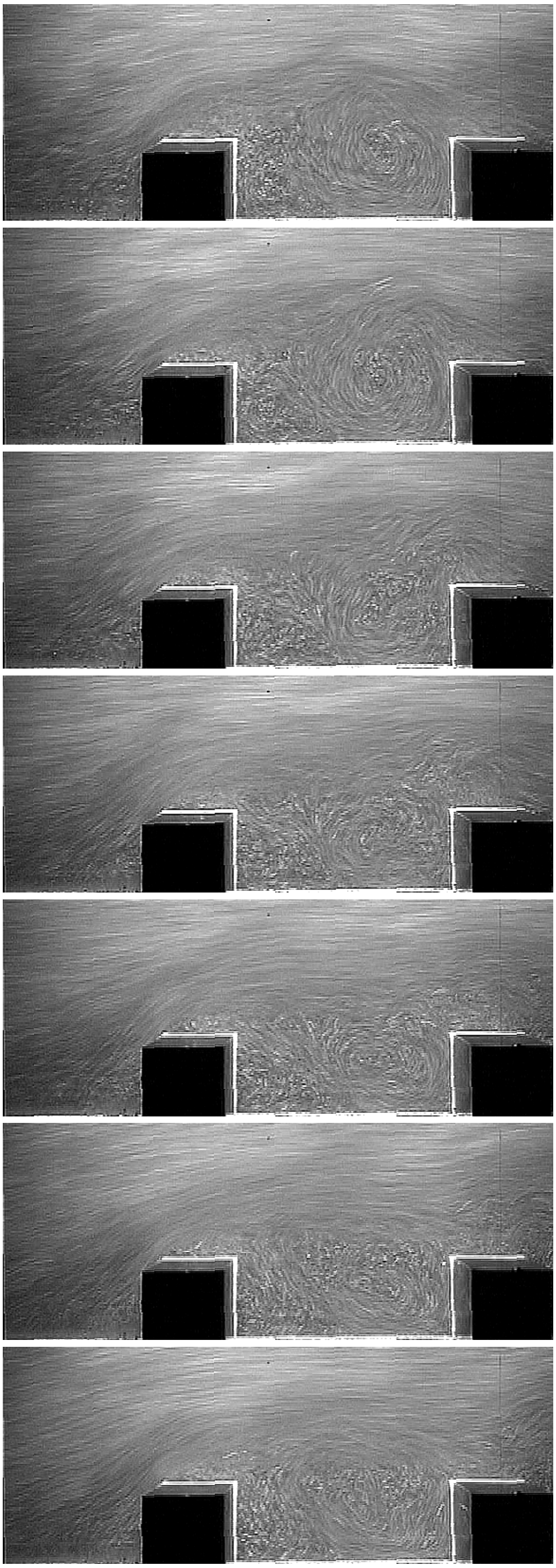
In this the paper described method uses the simplest model of the vortex planes. The planes are substituted by a system of discrete vortices, which flow field changes from the origin of the vortex as that isolated vortex. To verify the applicability of the presented simple method (considering simplified interaction of vortices with opposite signs and the vortex with boundary layer) an experiment was carried out and the results were compared also with the differential method.

## 4. Flow in a vicinity of two square cylinders

### 4.1. Experimental arrangement

Experiments were performed in an open hydraulic flume of cross section 0.25 x 0.25 and length 6 m. Bottom of the channel is covered by a brush paper, sidewalls are made from glass. Two square cylinders of diameter 3 x 3 cm were placed on the bottom of the channel across whole cross section of the channel. The space between the cylinders was three times the cylinder height (9 cm), see. Fig. 2. The channel outlet was equipped with a weir to regulate a water level. In these experiments the water depth was kept on a constant level 12 cm and the flow discharge was 8.8 l/s. Cylinder Reynolds number based on the average velocity was Re=8500.

The velocity filed was measured by means of LDA technique consisting of an Ar-ion laser, two-component fiber optical system Dantec, photomultipliers operating in backscatter mode and two BSA processors. Focal length of the transmitting lens was 310 mm. Since the square cylinders were made from a non-transparent material the measurements near the solid surfaces were limited. To partially improve the near wall measurements the measuring point was shifted to the distance 8 cm from the sidewall. Besides the LDA measurements we also performed a visualization with help of a digital video camera Panasonic DX1. Part of a video sequence is shown in Fig. 1 with time step 80 ms between two successive pictures. The flow direction is from the left to the right. In this sequence we can see that the flow is highly unstable – a large vortex is formed in front of the second cylinder then it totally collapses and a formation of a new vortex is starting.



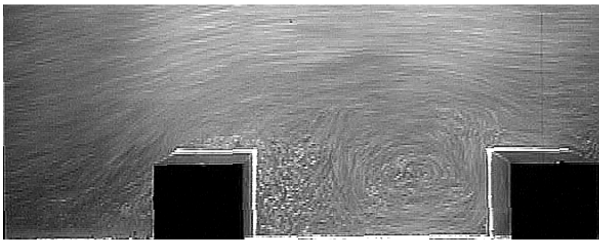


Fig. 1 Visualisation of flow patterns. Time step is 80 ms.

### 4.2. Method of the integral equations

The flow is modelled in the domain schematically shown in Fig. 2. The free surface that changed during the experiment negligibly was replaced by a wall with no friction. The origin of the co-ordinates is located on the channel bottom at the upstream face of the first square cylinder.

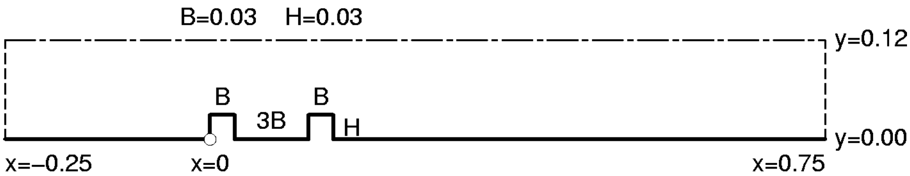



Fig. 2 Flow domain used for theoretical solution (dimensions are in meters)

The points of separation were located on the upper edges of both square cylinders. The motion of vortex plane originated on the cylinder surfaces was integrated by the simple method predictor-corrector of the second degree with a time step 2 ms. The flow patterns for selected time periods are shown in Figs. 3-8. From Figs. 7 and 8 it is evident that the flow field does not change in a monotonic way even after a long time.

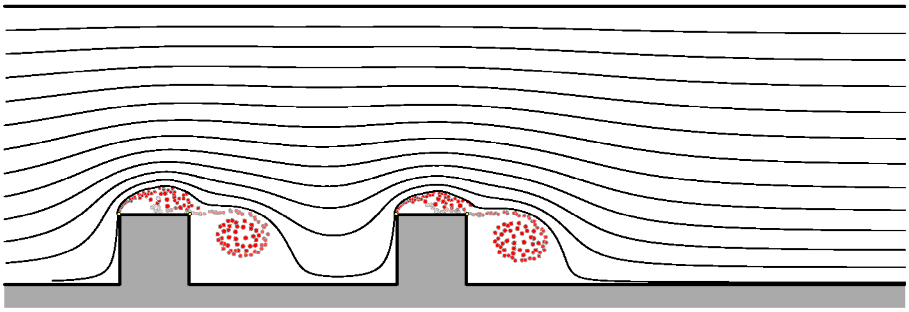


Fig. 3 Flow field at the time 0.2 s (integral method)

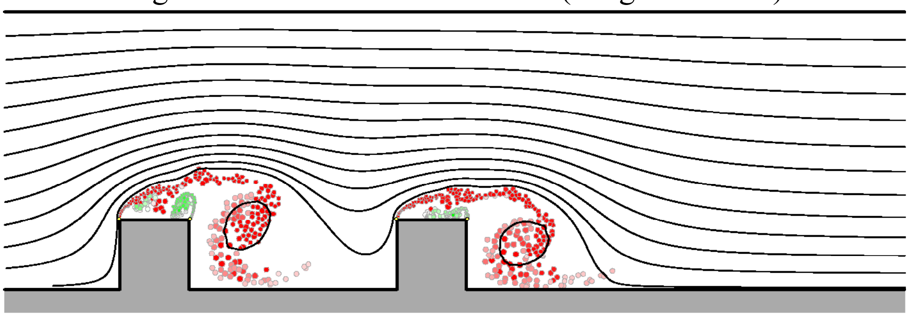


Fig. 4 Flow field at the time 0.4 s (integral method)

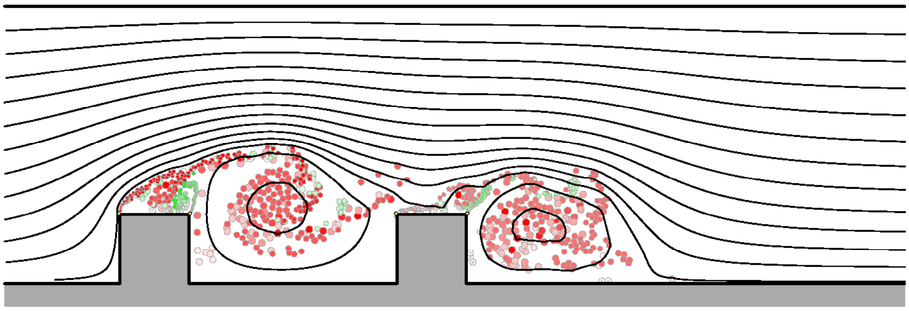


Fig. 5 Flow field at the time 0.6 s (integral method)

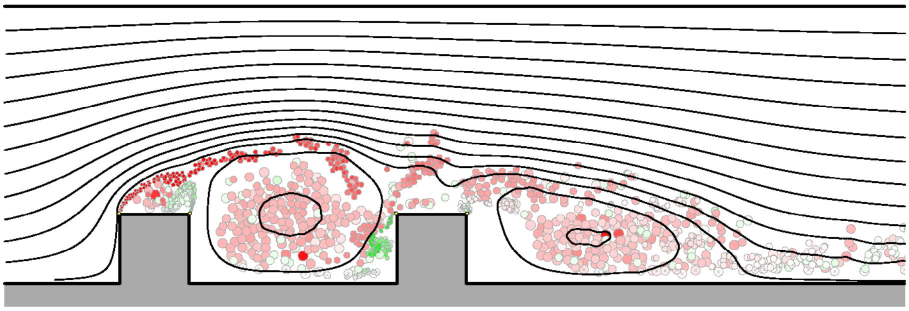


Fig. 6 Flow field at the time 1.2 s (integral method)

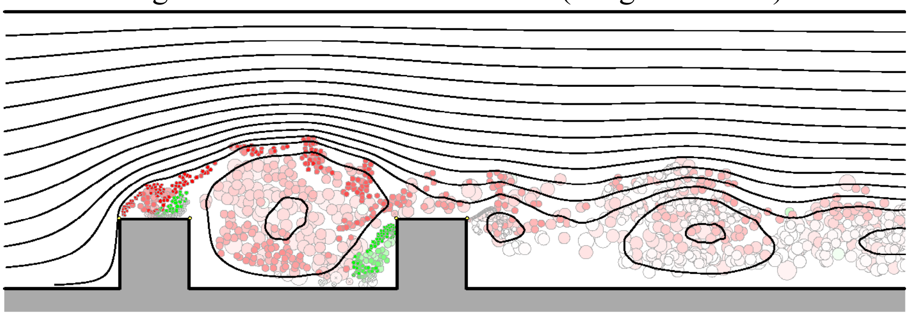


Fig. 7 Flow field at the time 11 s (integral method)

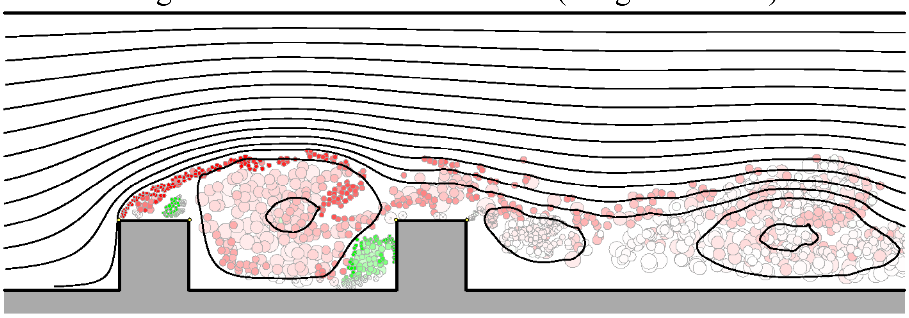


Fig. 8 Flow field at the time 12 s (integral method)

**4.3. Differential method**

To compare the experimental and integral equations results it was performed also a solution of the same geometrical arrangement with help of Fluent 6.0. In the flow domain (see Fig.2) there was built a mesh grid of square elements (approximately 200000 elements). For numerical simulation there was applied the k-ε turbulent model with enhance wall treatment. On the upper wall no friction was assumed. The problem was solved as unsteady with time step 0.1 ms in the initial stage, after stabilisation the step was prolonged up to 20 ms. The results evidently converged to the case of stationary wake (i.e. to the time averaged values), Fig.9. The solution stabilised after 5 seconds, the computation was stopped after 100 sec of model time. The velocity changes in any point of the domain are described only through the local turbulence characteristics. In Fig. 10 there are shown izolines of turbulent kinetic energy.

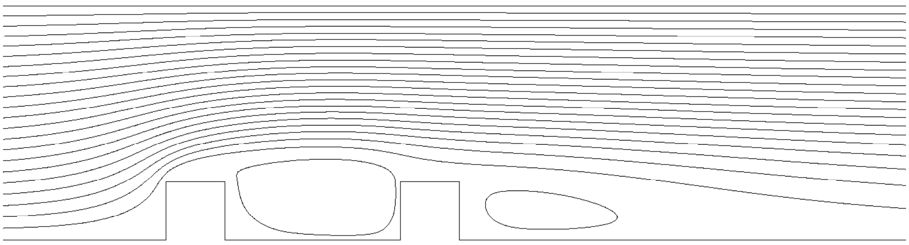


Fig. 9 Steady flow filed (Fluent solution)

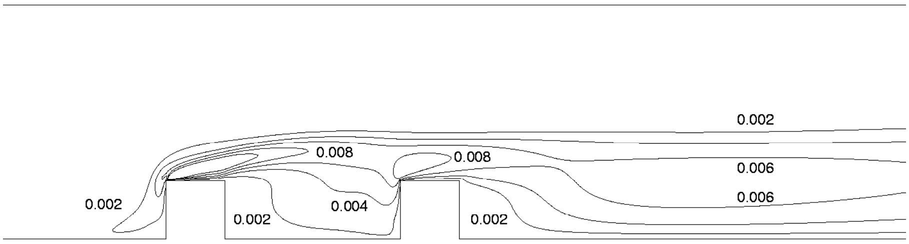



Fig. 10 Kinetic turbulence energy $[m^2/s^2]$ (Fluent solution).

## 5. Results

In Fig. 11 there are shown the time series of horizontal ($\boldsymbol{v_x}$) and vertical ($\boldsymbol{v_y}$) velocity components in the location between the cylinders and in the distance 42 mm above the channel bottom. The figures on the left side show results of the integral method, the figures on the right side show the experimental data. The computed time series should in detail depend on a degree of vortex fractionation in the wake. But as can be seen in Fig. 11 the computed time series is not going to steady state but similarly as on the experiment non-regular disturbances take place with repetition frequency of several hertz (groups of vortices are breaking away to main stream). While the Fluent solution (standard k-e model) does not allow a dynamical picture of the flow field, the integral method is partially able to give a better view, which is comparable with experimental observation.

Comparison of averaged values from experiment, Fluent and integral equations method is presented in Figs. 13 -15. In Figs. 13 and 14 there are plotted profiles of horizontal and vertical velocity in the selected positions, respectively, in Fig. 15 there are shown profiles of turbulent kinetic energy calculated according to the relation

$$k = \frac{(v'_x)^2 + (v'_y)^2}{2}$$

where $v'_x$ and $v'_y$ are root mean squares of velocity fluctuations in horizontal and vertical directions, respectively.

As can be seen in Figs. 13-15 Fluent results of velocity components and turbulent kinetic energy follow experimental data very well. The integral method gives somewhat worse results but still on an appropriate level. To improve the results of integral method mainly in the area between the cylinders (x=75 mm) it will be necessary to apply more complex model of vortex dissipation.

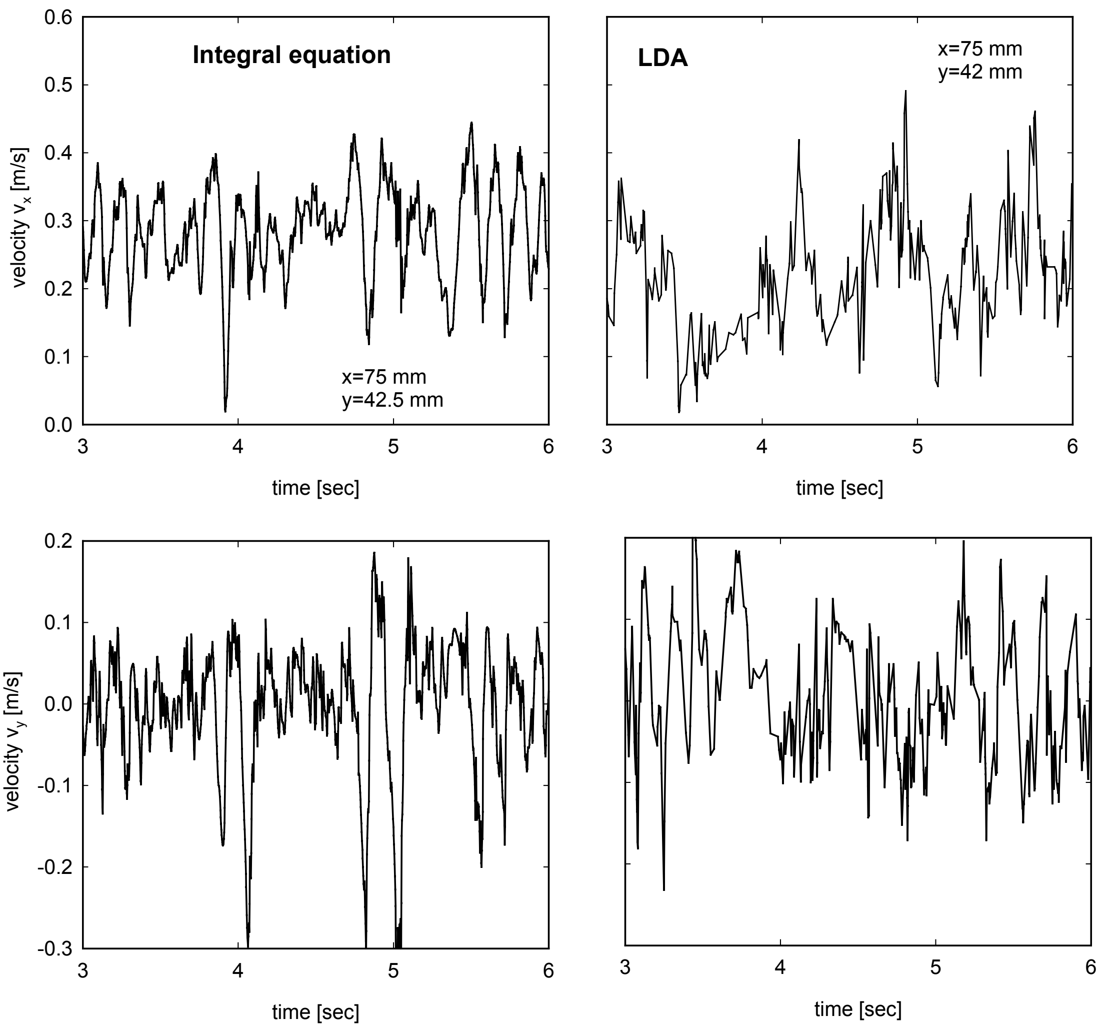



Fig. 11 Time series of horizontal and vertical velocity components

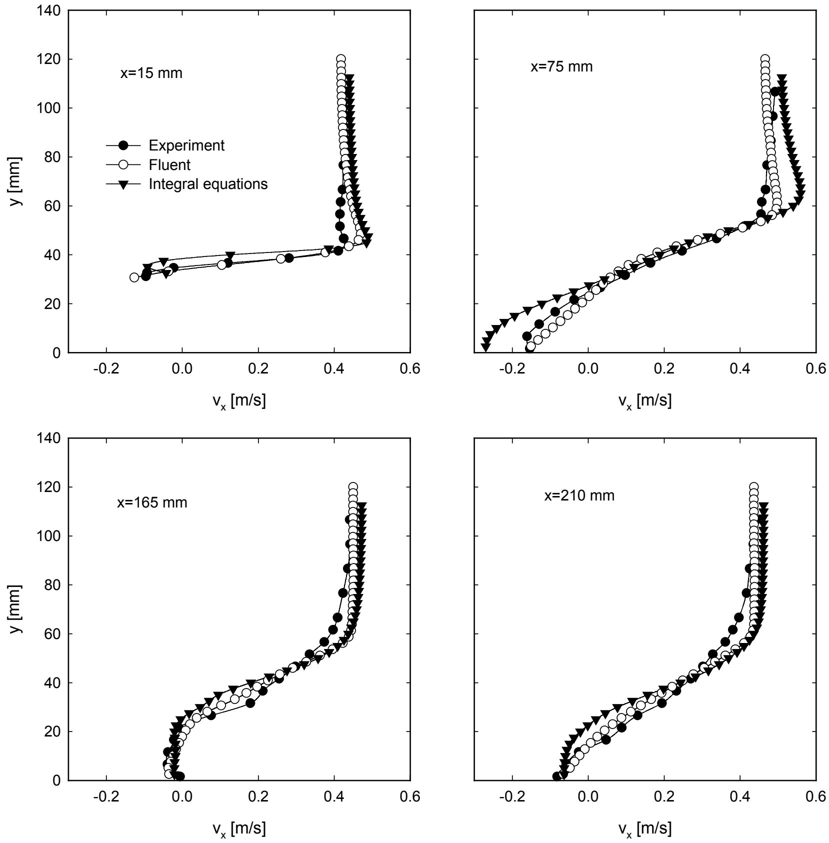



Fig.13 Profiles of horizontal velocity component

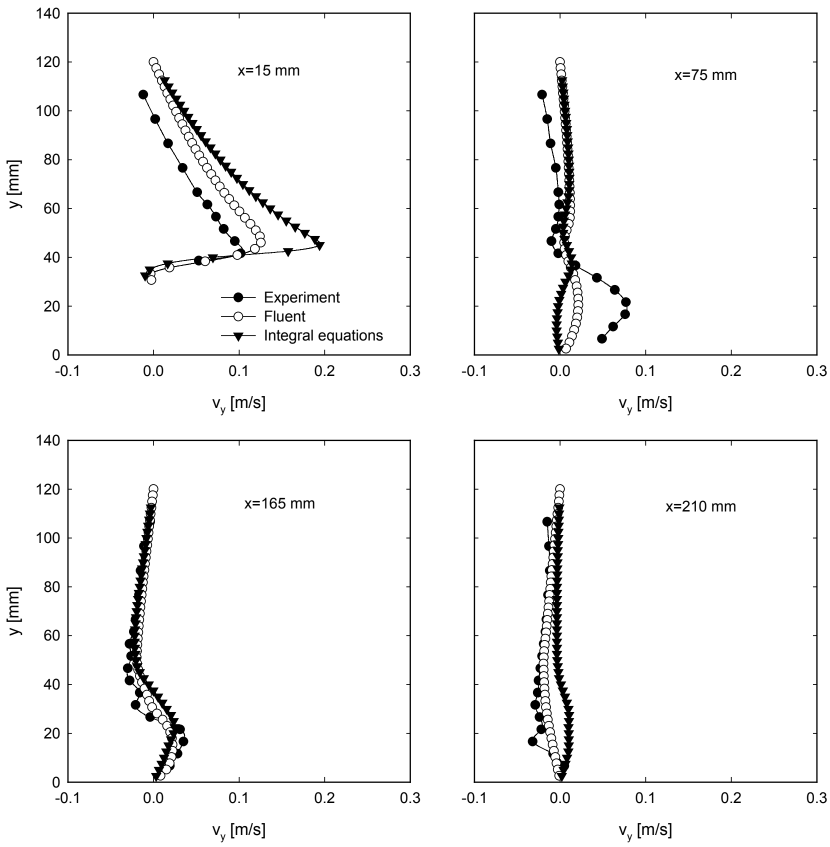



Fig.14 Profiles of vertical velocity component

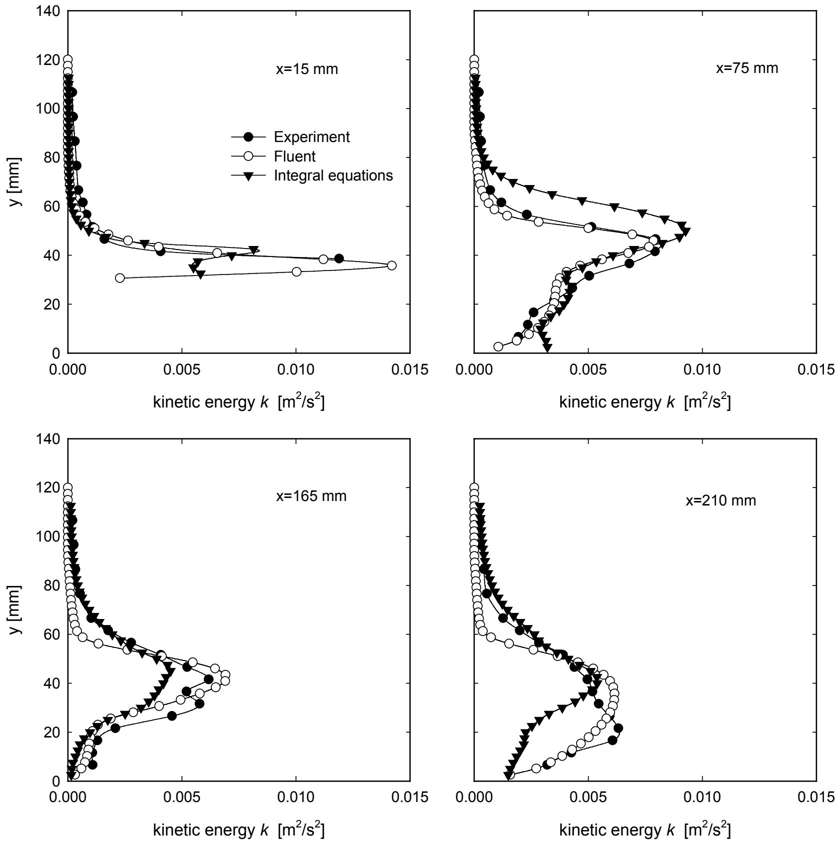



Fig.15 Profiles of turbulent kinetic energy

## 6. Acknowledgement:

The research was partly supported by the grant of the Grand Agency of the Czech Republic No. 103/03/0724.

## 7. References

Argyris, J.H, & Scharpf,D.W.(1970) Two and three-dimensional potential flow by the method of singularities,*The Aeronautical Journal RAS*, 1.

Bal, S.(1999) A potential based methjod for two-dimensional hydrofoil, *Ocean Eng*. 26, pp. 343-361.

Batchelor,G.K. (1970) *An Introduction to Fluid Dynamics*, Cambridge.

Belocerkovkij, S.M. & Nist, M. I. (1978) Otryvnoe i bezotryvnoe obtekanie tonkych kryljev idealjnoj zidkostju, *Nauka*, Moskva.

Belocerkovkij, S. M. (1965) Tonkaja nesuscaja poverchnost v dozvukovom potoke gaza, Nauka, Moskva.

Clarke,N.R. & Tutty, O.R.(1994) Construction and validation of discrete vortex method for two-dimensional incompressible Navier-Stokes equation, *Comp Fluids*, 23, No.6, pp.751-783.

Drela, M. (1989) XFOIL: An analysis and design system for low Reynolds number airfoils, in Low Reynolds number aerodynamics, Springer, June.

Hess, J.L. & Smith, A.M.O. (1964) Calculation of non-lifting potential flow about arbitrary three-dimensional bodies, *J.Ship Research*, 8,2, pp.22-24.

Hoření, B.(1976) Neustálený pohyb profilu v ideální nestlačitelné tekutině, sborník symposia "Nestacionární aerodynamika, aeroelasticita a dynamika lehkých konstrukcí z hlediska snížení jejich hmotnosti", VAAZ Brno.

Lavrentev, M.A. (1932) O postroenii potoka obtekajuscego dugu zadannoj formy, *Trudy CAGI* 118.

Loicyanskii, L.G. (1970) Mechanika židkosti i gaza, Moskva.

Prandtl, L.(1918) Tragflügeltheorie I,II, Göttingen Nachrichten.

Schlichting, H. (1974) Grenzschicht-Theorie, Nauka, Moskva.

Takeda,K., Tutty, O.R. & Nicole, D.A.(1999) Parallel discrete vortex methods on commodity supercomputers; an investigation into bluff body far wake behaviour, ESAIM: Procedings, Vol.7, pp.418-428.