A HYDRAULIC STUDY OF COOLING WATER INTAKE STRUCTURE

Aljaž ŠKERLAVAJ*

Turboinštitut, Slovenia

Franci VEHAR

Turboinštitut, Slovenia

Rok PAVLIN

Turboinštitut, Slovenia

Andrej LIPEJ

Turboinštitut, Slovenia

ABSTRACT

Computational fluid dynamics (CFD) calculations of pump sumps are troublesome due to the nature of the flow. Pump sump flow is turbulent and unsteady, and pump sump dimensions are large compared to diameter of vortices occurring near the sump walls or in the pump column. Therefore, to capture the general and important phenomena of the flow, the computational grid should be fine enough at certain areas of the sump. Combined with unsteady calculations, this usually results in computationally expensive cases. The decision for a suitable turbulent model plays an important role in adding or reducing the computational costs. The present study first compares some different turbulent models on a fine computational grid to the published experimental model. The intention of the present work is to get an answer whether the Unsteady Reynolds Averaged Navier-Stokes (URANS) model really fails in predictions of vortex modeling, since the usage of Large Eddy Simulation (LES) model for industrial cases would represent huge computational power demands. In the second part of the paper a real case pump sump is analyzed.

KEYWORDS

CFD, Turbulent Models, Pump Sump, Unsteady Vortices, Vortex Rope

1. INTRODUCTION

Each year's progress in computational capabilities enables a wider usage of computationally more demanding turbulent models in CFD calculations. At the moment the general usage of turbulent models spans from two-equation linear eddy viscosity RANS models, such as k-ε or k-ω models, to more demanding RANS models, such as Reynolds Stress Models (RSM), or even Large Eddy Simulation (LES) turbulent models.

**Corresponding author:* TRI department, Turboinštitut d.d., Rovšnikova 7, 1210 Ljubljana, Slovenia
phone: +386 1 5820152, fax: +386 1 5820112, email: aljaz.skerlavaj@turboinstitut.si

For a CFD calculation, computational cost is a result of a turbulent model, a calculation mesh and a calculation time step used. The turbulent model, mesh and time step are interconnected. The usage of more demanding turbulent model generally requires finer meshes and lower time steps. If suitable for a specific case, two-equation turbulence models are preferred due to the computational cost.

One of the most often used two-equation models in Ansys CFX is a Shear Stress Transport (SST) turbulent model [1]. It is a combination of k-ε and k-ω models, with k-ε model being used in the free-stream zone and k-ω model being used near the wall, thus bringing the best of both models. SST also limits the eddy-viscosity. Although the SST model can predict the separation point quite well, the extent of the separated region can be overpredicted ([1], [2], [3] and [4]).

If two-equation models do not give good results, one is confronted with the choice of more expensive models. RSM models solve seven additional equations instead of two, as do the two-equation models, which makes the model much slower to compute. Even the usage of "pure" RSM models might not prevent a CFD user from incorrectly estimating turbulent mixing in the separated shear layer [5] and hence getting an overpredicted extent of a separated region. On the other hand, a difference in computation time for one time step on the same computational mesh between SST and LES models is small. However, LES needs small time steps with a Courant number of CFL<1, which, along with the denser meshes, makes the overall computation much slower than the one with the SST model.

Besides RSM and LES models, hybrid model alternatives exist. One of them is Scale-Adaptive-Simulation (SAS) SST model [6], which is an improved unsteady RANS method that develops LES-like solutions in unstable flow regimes. It is a SST model with an additional production term in the ω equation which increases when the flow equations start to go unsteady. The increase of the SAS term results in slower decay rate of the Reynolds stresses due to the smaller turbulent viscosity [7].

The second alternative is Detached Eddy Simulation (DES) model [8], which is a hybrid between RANS model, used at the boundary layer, and LES model in detached regions. The detachment point for DES model might be sensible to the grid size [1]. Like LES models, it also requires small time steps with a Courant number of CFL<1.

The decision for the turbulent model chosen in a CFD calculation is not easy. Test cases are important, either numerical [5] (LES or even DNS) or experimental ones. In order to decide for an appropriate turbulence model for a pump sump case, LES and SST numerical results ([9] and [10]) from experimental test case [11] have been recalculated and reevaluated. Based on the decision, an industrial case pump sump has been calculated.

2. PUMP SUMP NUMERICAL TEST CASE MODEL

Geometry of a pump sump numerical model (Fig.1, Tab.1) is based on the experimental model [11], as described in [10]. The inlet section is divided into two channels with unequal discharges, 0.905 and 0.385 m^3/min. Water flows through the channels, mixes, and enters the pump column at pump bell and exits at the top of the pump column. Due to non-equal inlet flow rate a floor vortex is formed in a pump bell area.

In the study [9], [10] a comparison between LES and SST turbulent models has been made. The results have been compared to PIV data of a pressurized pump sump [11]. The steady SST calculation has been done on a mesh with approximately 1.5 million cells, and the LES calculation on a mesh with approximately 5 million cells. The LES model results showed good qualitative and quantitative agreement with PIV data, while the SST model seemed to completely fail to predict the turbulent kinetic energy of the bottom part of the main floor attached vortex.

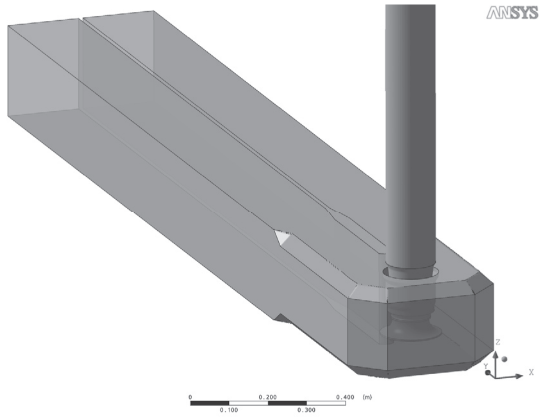

Fig.1 Geometry of a pump sump test case model.

Outlet pipe diameter D	129.8 mm		Bell mouth distance from floor	0.62 D
Bell mouth maximum diameter	1.23 D		Peer to pipe center distance	1.35 D
Pump sump height	1.91 D		Simulated pump column height	12.8 D
Inlet channels width	1.49 D		Simulated pump sump model length	20.5 D

Tab.1 Characteristic geometry dimensions of the pump sump.

In the present paper, in order to get good inlet and outlet velocity profiles approximation, the inlets have been moved upstream and the outlet further downstream. This results in the pump sump model length of 20.5D instead of 7.7D and the pump column length of 12.8D instead of approximately 2D, as in [9], [10].
Since it has been assumed that the SST model failure in [9] was due to the coarse meshes, it has been decided to test all turbulent models on the same computational mesh. For the test case a structured mesh with approximately 35 million nodes has been created (Fig. 2). The first mesh element at the floor below the pump bell is 0.001D high, which is twice as high as in [9]. The first element from the floor elsewhere has been set to 0.0015D. The first element at the inner side at the bottom of the pump bell is 0.00028D high and the ones at the narrowest part of the pump bell are $6.6 \cdot 10^{-5}$D high. In [9] the height of the elements on the interior pump column walls was 0.0002D. The highest y+, in a range between 10 and 100, was observed at the floor in the vortex core. At the lower inner pump bell walls y+ values were in the range from 3 to 10, while at the rest of the inner column walls they were mostly from 1 to 3.

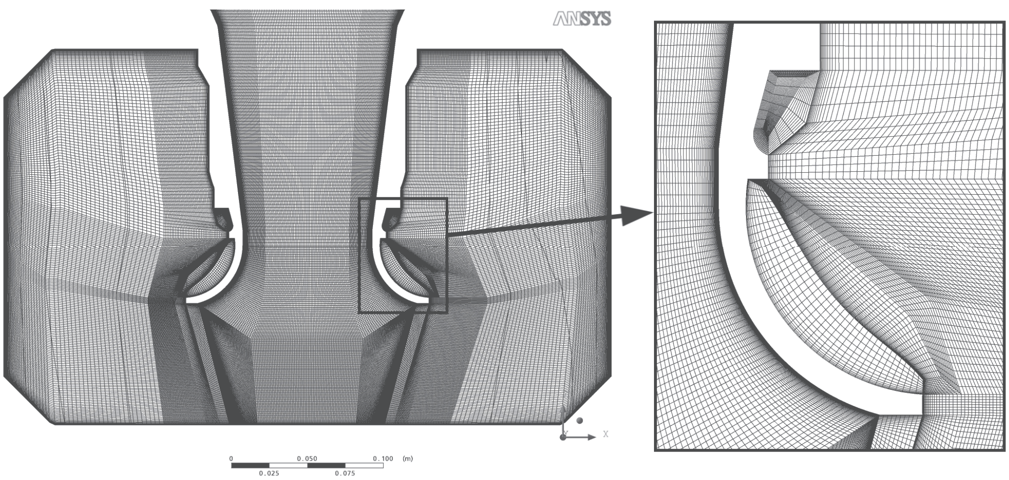

Fig.2 Cross-section through the calculation mesh at the center of the pipe.

Minimum orthogonality angle of the mesh was equal to 23.5 degrees, maximum aspect ratio was 242.8, and maximum expansion factor was 39. As in [9], time step for transient calculations was set to $0.002 \cdot D/U_o$, where U_0 is the mean channel flow velocity. Combined with the mesh, this resulted in a maximum Courant number of approximately 10, while the RMS Courant number was around 0.3. Time step for a steady SST calculation was set to $0.02 \cdot D/U_o$.

A commercial code Ansys CFX 11.0 has been used for the calculations. First a steady SST calculation has been done for approximately 8600 iterations, so that the flow would travel for $172 \cdot D$ with a mean velocity U_o. RMS velocity residuals were below $1.4 \cdot 10^{-4}$ and the RMS pressure residuals were smaller than 10^{-6}. Maximal velocity residuals were below $1.3 \cdot 10^{-2}$ and the RMS pressure residuals were smaller than $2.7 \cdot 10^{-4}$. The result was used as an initial for unsteady SST (URANS), SAS-SST and LES Smagorinsky model calculations. At each physical time step we iterated until RMS velocity residuals were smaller than 10^{-5}. The maximal velocity residuals were below 10^{-2} for all three calculations. SST and SAS-SST (further denoted as SAS) calculations usually needed two iterations per time step, while LES calculation needed three iterations per time step. Total simulated time for the unsteady SST calculation was $33.2 \cdot D/U_0$ (16600 time steps), for the SAS calculation $29 \cdot D/U_0$ (14500 time steps) and for LES calculation $41.2 \cdot D/U_0$ (20600 time steps).

We have focused on the simulation of a main submerged vortex. The most interesting quantities are velocity magnitude and turbulent kinetic energy (TKE). Although the Yulin experimental PIV data included only the two time-averaged in-plane velocity components, a 3-component velocity magnitude is presented in Fig. 3 since the computed 2- and 3-component results were qualitatively similar. In [9], the TKE (denoted as *k*) for the experimental results was estimated from in-plane velocity fluctuations u'_{P1} and u'_{P2} (Eq. 1). In Fig. 4, the TKE for steady SST is based on a solution of transport equation for TKE (modeled TKE). TKE for transient models is a sum of 3-component velocity fluctuation flow statistics using Eq. 2 (resolved TKE) and modeled TKE, denoted as total TKE.

$$k = \frac{3}{4} \cdot \left(\overline{(u'_{P1})^2} + \overline{(u'_{P2})^2} \right) \quad (1)$$

$$k = \frac{1}{2} \cdot \left(\overline{(u'_1)^2} + \overline{(u'_2)^2} + \overline{(u'_3)^2} \right) \quad (2)$$

For comparison reasons it was decided to use the same scales in Fig. 3 and Fig. 4 as used in [9]. Velocity magnitude in Fig. 3 was normalized by U_o, whereas total TKE in Fig. 4 was normalized by U_o^2.

In contrast to [9], the steady SST calculation predicted similar velocity magnitude values below the pump bell level as LES and experiment (Fig. 3). Compared to LES, the V-shape of the vortex velocity magnitude seems to be better predicted with SAS than with URANS.

In [9], the TKE of the steady SST calculation produced an annulus-like area of higher TKE values in the pump column at an approximate diameter $0.5 \cdot D$. Our calculations show better qualitative agreement of the SST model (Fig. 4, top frame). High TKE values are located on the pipe centerline.

In figures of the experiment [9], the highest TKE values (approximately $0.4 \cdot U_0^2$) seem to be evenly located along the core axis below the pump bell. The values inside the pump bell are not available due to PIV limitations. From location of high TKE values in different planes in Fig. 4 it seems that the vortex has a 3-dimentional shape and that the two planes do not give the clear picture. Drawing an iso-surface of TKE values, this assumption shows as a correct one (Fig. 5). For this reason a comparison between the experiment and Fig. 4 would be questionable, therefore URANS and SAS results are rather compared to a LES model result.

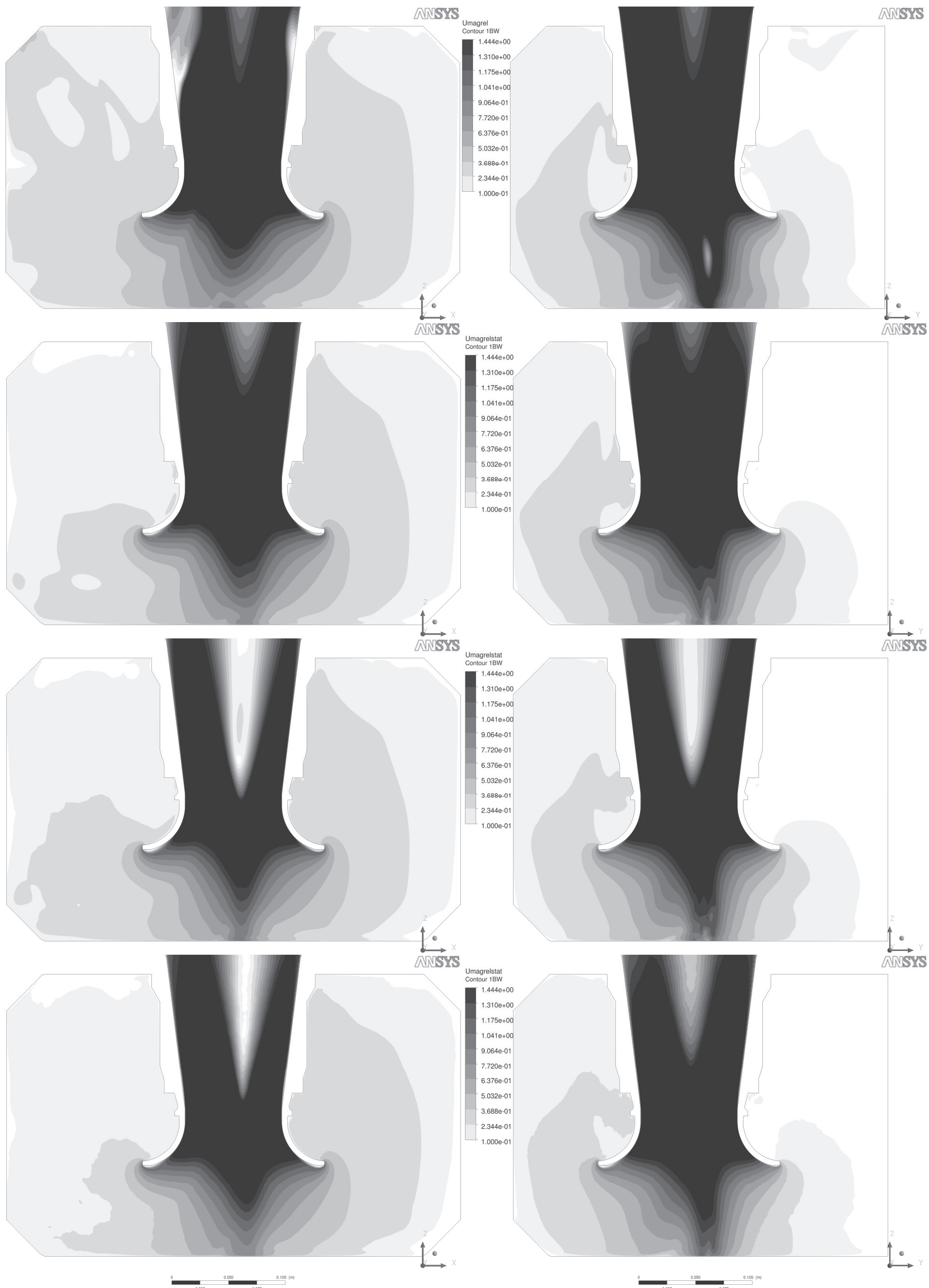

Fig.3 Normalized time-averaged 3-component velocity magnitude at the rectangular cross-sections through the pipe center. From top to bottom: Steady SST model (final result), transient SST model, SAS model and LES model. From left to right: cross-sections through the pipe centerline normal to Y-direction towards the inlet channels and normal to X-direction.

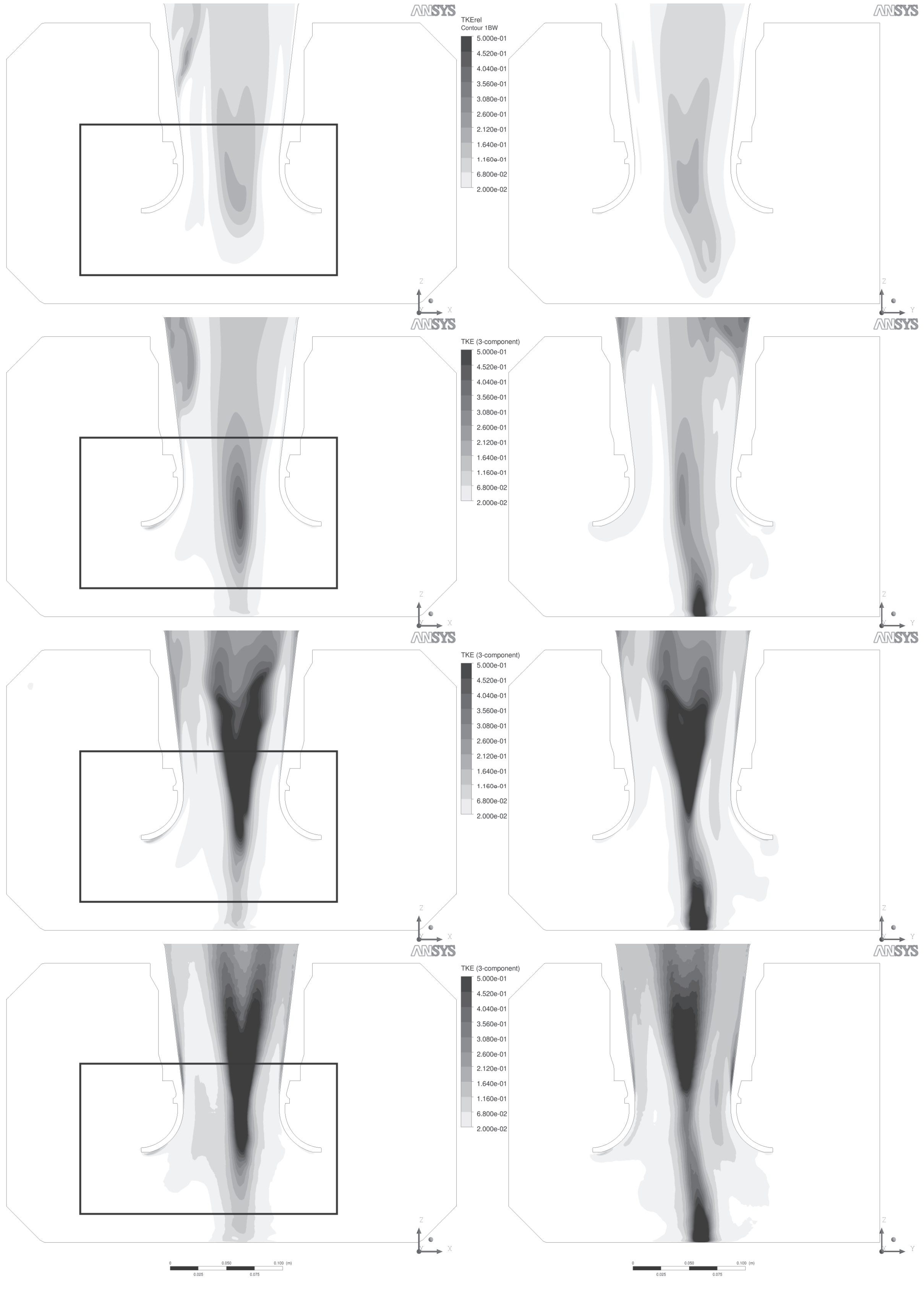

Fig.4 Normalized time-averaged total turbulent kinetic energy at the rectangular cross-sections through the pipe center. From top to bottom: Steady SST model, transient SST model, SAS model and LES model. A square represents an approximate location of the measuring window. From left to right: cross-sections through the pipe centerline normal to Y-direction and normal to X-direction.

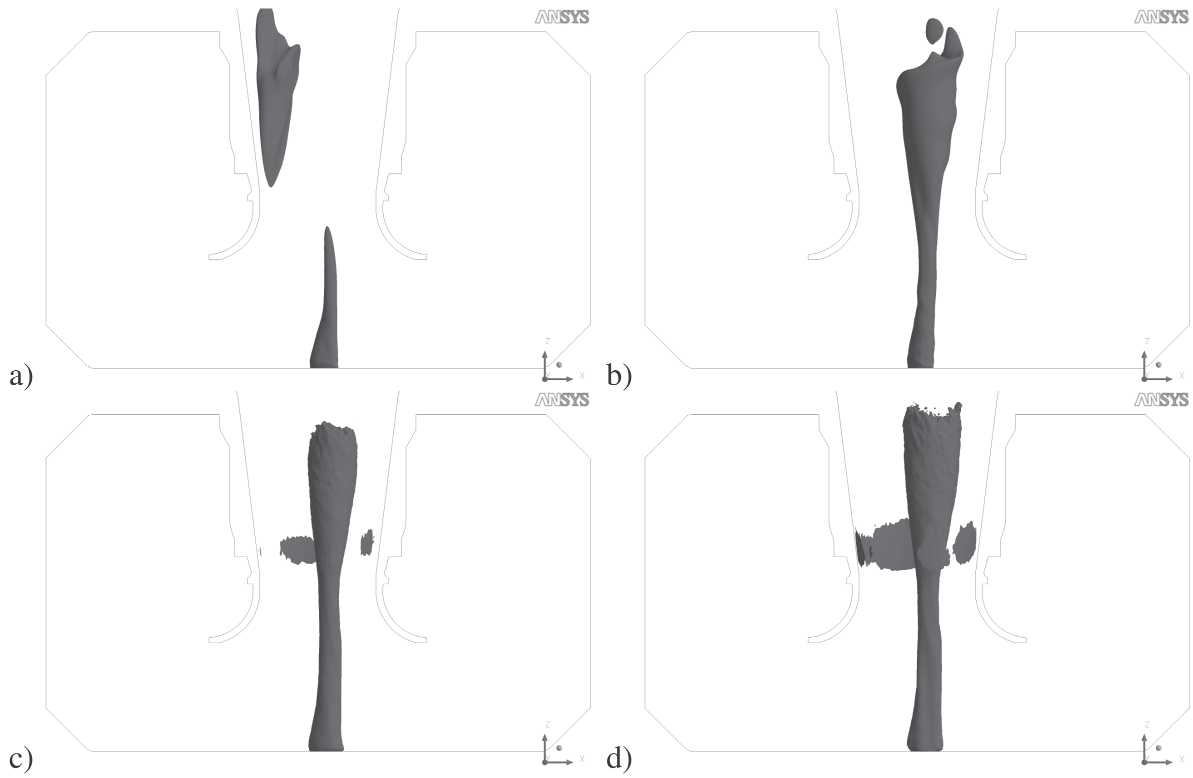

Fig.5 Iso-surface (value: 0.4) of normalized time averaged total turbulent kinetic energy for URANS (frame a), SAS (frame b) and LES (frame c) calculations. View towards the inlet channels. Frame d: the same iso-surface of 2-component total TKE for LES model calculation.

The shape of a total TKE isosurface with SAS model is approximately the same as the one with LES model. The URANS calculation produces values of TKE which are lower than the ones by LES. Using different values of iso-surface, it seems that the vortex only has one local maximum, which is located at the floor, and TKE values gradually decrease away from the floor. LES, on the other hand, produces two local maximal values, one at the floor and one in the pump bell. SAS does produce two local maximal values, although the local maximal value in the bell is not so distinctive. A shape of the total TKE isosurface for LES calculation is approximately the same if using Eq. 1 (Fig. 5, frame d) or Eq. 2 (Fig. 5, frame c). A local maximal time-averaged 3-component total TKE value at the floor is $2.3 \cdot U_0^2$ and the same local maximal value in the pipe is $1.8 \cdot U_0^2$. The local maximal time-averaged 2-component total TKE value normal to the inlet channel at the floor is $2.0 \cdot U_0^2$ and the same local maximal value in the pipe is $2.2 \cdot U_0^2$.
The patches of high TKE values at bell walls occur due to the flow detachment. Local maximal values of total normalized TKE of the detachment patches are 0.64 for transient SST, 0.38 for SAS and 0.5 for LES model. The location of such values is approximately the same for LES and URANS, while for SAS model the patches are located just above the surface level. It seems that URANS produces too large detached region, which in turn probably inhibited the vortex growth in the pump bell area.
A vortex shape in Fig. 6 is depicted by using different iso-surfaces of Q-criterion for the last time-step of each turbulent model. The figure reveals the shape of the vortex, called vortex rope, which may occur in diffuser part of a water turbine at part loads. It seems that the URANS calculation failed in vortex rope prediction, probably due to the large detachment region. The SAS model predicted a stable single rope, while an unstable double vortex rope was predicted by LES calculation.

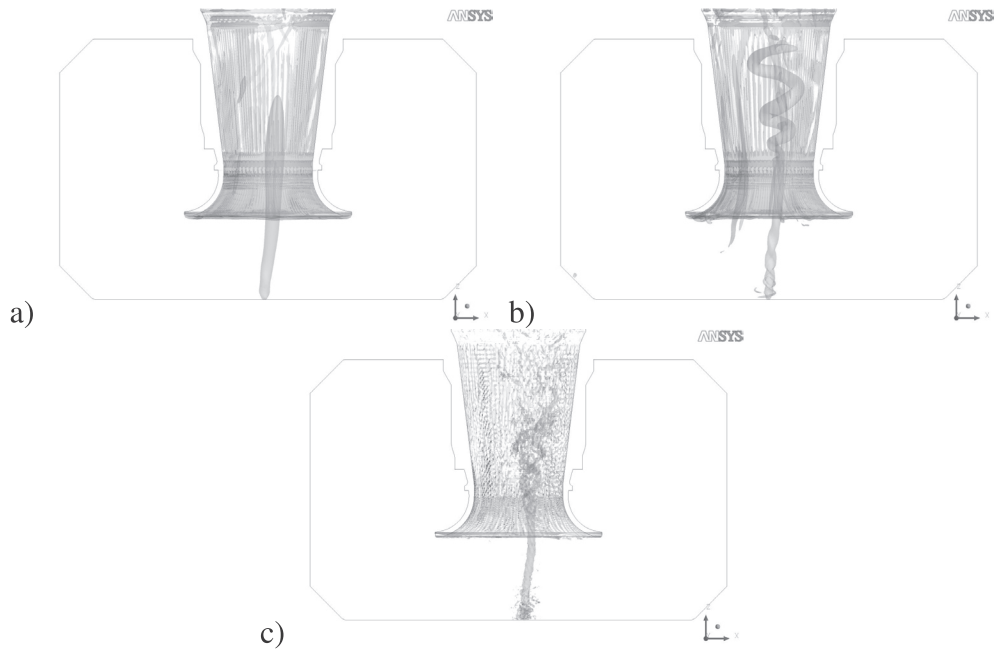

Fig.6 Iso-surface of Q-criterion of vortex identification for URANS (frame a; value: 50000 s^{-2}), SAS (frame b; value: 50000 s^{-2}) and LES (frame c; value: 500000 s^{-2}) calculation. View towards the inlet channels.

3. PUMP SUMP INDUSTRIAL CASE MODEL

Industrial pump sump CFD calculations are very demanding. Dimensions of sumps used in industry are large compared to vortices that may form in the vicinity of the pump columns. The characteristics of such flows are usually highly unsteady and intermittent vortices. Vortices may start at the floor, side walls or even at the water surface. In the last case, an air entrainment is possible. Therefore, at least in the area where the vortices of the interest occur, which is generally in the vicinity or inside the pump column, the computational mesh should be fine.

As defined by Hydraulic Institute standard [12] pump sump experimental models should not produce strong vertices, bell mouth to walls connected vertices, non-uniform velocity profiles at a pump impeller inlet or swirling flows at the same position. There are two testing time intervals defined, in which the flow should match certain criteria, the longest one being a 10-minute interval. At the moment CFD calculations could not replace an experimental model. Among other problems it would be impractical to test the pump sump model for such a long interval since such calculation would demand a lot of calculation time. For instance, although a time step could be a bit higher for the former numerical test case model, only 3 seconds of the total time by a LES simulation have been simulated, for which approximately 40 days of calculation time on 128 processor cores were needed.

On the basis of the above results from the test case model a real case pump sump has been modeled [14]. The industrial case is a CW intake structure (Fig. 7) with three pumps. The reason for the decision of the pump sump modeling was a possible air entrainment vortices occurrence. A 1:16 scale has been used for an experimental and a numerical model.

For CFD calculations three interconnected domains have been created: a domain of the approaching flow, with coarse unstructured mesh of 1.6 million elements (580000 nodes), a porous stationary domain of six travelling screens each with a structured mesh of 110000 elements, and a main pump sump domain with an unstructured mesh of 25.5 million elements (7 million nodes).

Again, Ansys CFX 11.0 code has been used for the calculations. Since the URANS calculation was able to predict the existence of the vortex of the test to some extent, a decision has been made to use the SST turbulence model for a transient calculation. It should be pointed out that to save some computational time a coarse mesh in the approaching flow has been used which means that the LES turbulent model would be inappropriate for this area. We did not want to calculate the domains separately due to their flow interconnections.

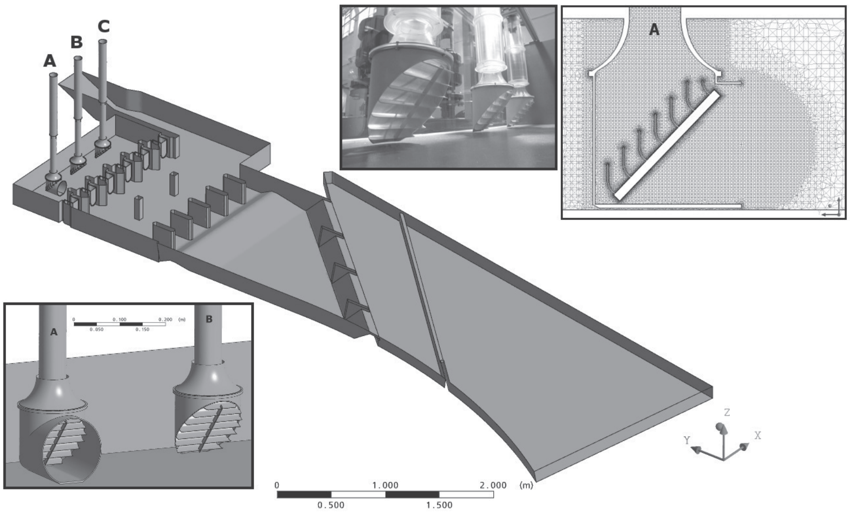

Fig.7 Geometry of the industrial case model (scale 1:16). Top frame: close-up of the original geometry layout. Main window: modified geometry layout (columns B and C represent initial geometry; column A represents modified inlet geometry). Lower left frame: close-up of modified geometry. Upper right window: cross-section through computational mesh of modified pump bell A.

A one-phase numerical model has been used, with water surface being modeled as a flat surface with a free-slip boundary condition assumption, as in [13]. A LO-LO water sump level was used as a surface level. Flow discharge was scaled on the basis of the Froude number similarity between the prototype and the model. The model outlet diameter is 0.075 m and the scaled outlet flow per pump is 8.276 kg/s.

Two pump sump models have been calculated: the initial one where all three pump columns were designed as B and C columns in Fig. 7, and the modified one where only the column A was modified (Fig. 7). Values of y^+ were below 35 in the sump area; at the bottom, in the vicinity of the pumps, the values were mostly between 3 and 6, with values up to 10; at the pump bell blades, the values were between 0.5 and 15; at the pump column, the values were between 0.5 and 30. A computational mesh size is represented in Fig. 7 by a cross-section through the pump bell A. Time step for transient calculations was 0.005 s. The time step is a compromise between coarse and very fine mesh parts. RMS velocity residuals were below $2.2 \cdot 10^{-4}$. Approximately ten seconds of total time have been simulated by transient calculations.

The results of the initial configuration revealed that almost steady vortices occurred behind the pump bell blades. With the dye injection method of flow visualization on the experimental model it was not possible to either confirm or deny the existence of such vortices. Otherwise, both CFD and experimental case have shown some similarities, such as surface vortices position. The original experimental case met the usual demands of the standard and no air entrainment was observed. To see if it was possible to damp the vortices a pump bell extension has been designed (Fig. 7, pump A). Although the extension did improve the vortex occurrence by lowering the strength of the vortices (Fig. 8a, pump A), at the experimental model the air entrainment vortices occurred.

In order to find out why the dye injection visualization method seemed to be unsuccessful, a SAS-SST calculation has been made on the same mesh and with the same settings as URANS calculation. In Fig. 8b it can be seen that in contrast to URANS calculation, numerous small vortices were produced in the bell area, which would result in dye diffusion. Again, the strength of the vortices of pump A in the column area is lower than the strength of such vortices in unmodified pumps.

Although at the moment it is unlikely that CFD calculations would replace the experimental models, CFD simulation can be a helpful tool in the process of pump sump improvement. URANS calculations might produce relatively good results, but some caution at their interpretation is needed.

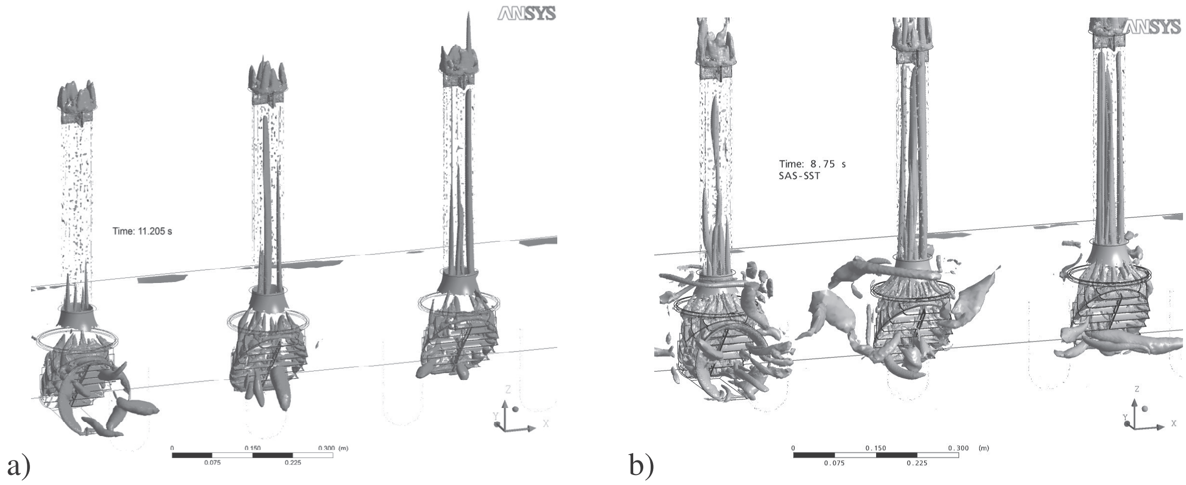

Fig.8 Representation of vortices in pump sump based on Q-criterion of vortex identification. a) SST calculation. b) SAS-SST calculation.

4. CONCLUSION

In the first part of the article it has been shown that in contrast to [9] even the steady SST calculation predicted basic quantities reasonably well. This is either due to quality calculation mesh used or due to usage of a modified SST model [1] instead of a standard one. Steady SST calculation could not predict TKE well due to transient nature of the flow. For URANS calculation the TKE values were under predicted, while the SAS model produced approximately the same shape of TKE values as LES. The URANS predicted large separation region which seemed to inhibit the vortex creation. Being aware of the model limitations it seems that unsteady SST is still suitable for industrial cases since it might be used on meshes which are fine only in the area of interest. In the second part an industrial case has been presented where CFD was a part of the pump sump testing process.

5. ACKNOWLEDGEMENTS

The research was partially funded by the Slovenian Research Agency ARRS - Contract No. 1000-09-160263.

6. REFERENCES

[1] Menter, F.R., Kuntz, M., Langtry, R.: Ten Years of Industrial Experience with the SST Turbulence Model. *Turbulence, Heat and Mass Transfer 4*, Begell House, Inc., New York. 2003

[2] Hadžić, I., Hadžić, H., Muzaferija, S.: *CASE 9.2: Periodic flow over a 2-D hill - Description of numerical aspects and turbulence models*. 9th ERCOFTAC/IAHR Workshop on Refined Turbulence Modelling, Darmstadt University of Technology, Germany. 2001

[3] Skoda, R., Schilling, R.: *Computation of the Test Case 9.2: Periodic flow over a 2D hill*. 9th ERCOFTAC/IAHR Workshop on Refined Turbulence Modelling, Darmstadt University of Technology, Germany. 2001

[4] Hellsten, A., *Description of the Computations for the Test Case 9.2*. 9th ERCOFTAC/IAHR Workshop on Refined Turbulence Modelling, Darmstadt University of Technology, Germany. 2001

[5] Jang, Y. J., Temmerman, L., Leschziner, M. A.: Investigation of Anisotropy-Resolving Turbulence Models by Reference to Highly-Resolved LES Data For Separated Flow. *ECCOMAS CFD Conference 2001*. Swansea, Wales, UK. 2001.

[6] Menter, F.R., Egorov, Y.: A Scale-Adaptive Simulation Model using Two-Equation Models. *AIAA paper 2005-1095*. Reno, Nevada. 2003

[7] Davidson, L.: Evaluation Of The SST-SAS Model: Channel Flow, Asymmetric Diffuser And Axi-Symmetric Hill. *ECCOMAS CFD 2006*. TU Delft, The Netherlands. 2006

[8] Spalart, P. R.: Strategies for turbulence modelling and simulations. *Int. J. Heat Fluid Flow*. **21**. 2000. pp. 252-263.

[9] Tokyay, T. E., Constantinescu, S. G.: Large Eddy Simulation and Reynolds Averaged Navier-Stokes Simulations of Flow in a Realistic Pump Intake: A Validation Study. *World Water and Environmental Resources Congress*, EWRI, Alaska. 2005

[10] Tokyay, T. E., Constantinescu, S. G.: *Investigation of Flow Physics of Pump Intake Flows Using Large Eddy Simulation*. IIHR Technical Report No. 445. College of Engineering, The University of Iowa, Iowa, USA. 2005

[11] Yulin, W., Yong, L., Xiaoming, L.: *PIV Experiments on Flow in a Model Pump Suction Sump*. Research Report, Tsinghua University, China.

[12] American National Standard: *Pump Intake Design*. Hydraulic Institute, USA. ANSI/Hl 9.8-1998.

[13] Constantinescu, G. S., Patel, V.C.: Numerical Model for Simulation of Pump-Intake Flow and Vortices, *J. Hydr. Engrg.*, ASCE, **124(2)**, 1998. pp. 123-134.

[14] Vehar, F., Pavlin, R., Škerlavaj, A., Ermenc, D.: *Circulating Water Pumps - Model Intake Flow Analysis*. Report No. 2958, Turboinštitut, Slovenia. 2009. (in Slovene)