

TURBULENT FLOW SIMULATION IN KAPLAN DRAFT TUBE

Z. Čarija, Z. Mrša and L. Dragović

Keywords: CFD, Numerical simulation, Kaplan turbine, Turbulence models, Fluent

1. Introduction

This work was motivated by a need for validation of commercial CFD fluid flow solver for hydraulic turbine flow simulations. For the same reason, the ERCOFTAC draft tube has been used for many years as a base for evaluating the state-of-the-art CFD (Computational Fluid Dynamic) codes in the field of hydropower generation.

CFD (Computational Fluid Dynamic) is today standard procedure for simulating and analysing fluid flow through hydraulic machines. This is a process where fluid flow domain is divided into many small volumes where governing equations are converted into algebraic equations, which are then solved numerically. Computational result strongly depends on applied mathematical model and numerical methods used for converting governing to algebraic equations.

Fluid flow simulation results through complete hydraulic turbines, using steady state approach, correspond very well to measured engineering quantities (power, torque, flow rate, efficiency etc.). Almost negligible difference between calculated and measured results enables the use of CFD fluid flow simulations for designing completely new turbines or revitalization of old ones, where geometrical shapes of selected turbine parts have to be optimised [1], [3], [6].

Ruprecht presented two different approaches for the shape optimization in the field of turbomachinery. Geometry manipulation as well as the flow evaluation can completely be done in 'Virtual Reality' where designer (expert in turbines) can work in intuitive way, analyzing flow interactively [7]. Another approach is based on mathematical optimization tools where suitable 'quality function' has to be defined which will be criteria in optimization process. The quality function depends on specific situation and requires a lot of experience to define it very well. Gradient [2] and genetic algorithm methods [4], [5] are widely used today in the shape optimization of turbine parts. Gradient methods can detect local minimum instead of the global, whereas genetic algorithm methods are more robust, non-sensitive on local minima, but requires more optimization cycles.

Probably one of the most interesting parts, from the aspect of efficiency, is a hydraulic turbine draft tube. Its role is to increase the pressure head by converting outlet kinetic energy into static pressure and therefore impacts directly on the total efficiency of the hydropower plant.

Draft tube efficiency can be additionally improved with appropriate draft tube design, or by correcting existing one, especially in the elbow draft tube type, when hydraulic losses are reduced with direct impact on total turbine efficiency.

Draft tube fluid flow is difficult to predict numerically due the highly swirled, fluctuating and non-uniform flow that leaves the rotating runner and enters the draft tube. Furthermore, draft tube fluid flow is characterized by adverse pressure driven fluid flow, with several zones of fluid flow separation. Fluid flow complexity in draft tube arises when working out from optimal point of operation, where higher quantity of kinetic fluid flow energy enters the draft tube.

Without visualization of the fluid flow simulation results, only pure intuition existed of what the fluid flow in hydraulic turbines looks like. From the flow simulations results the fluid flow behav-

ior inside draft tube can be understood in sufficient detail and this knowledge can be used to design new generation of the draft tubes or revitalize old ones.

2. Problem description

In this paper the turbulent fluid flow in the Hölleforsen hydropower draft tube model is simulated with the commercial fluid flow code Fluent 6.2. The results were compared with the available experimental data. Experimental laboratory provided measured data for two different points of operation. Hydraulic turbine comprises spiral casing with stay ring, 24 adjustable guide vanes, draft tube and runner with four adjustable runner blades. Turbine model was mounted in VAUB's turbine rig at the Alvkärleby laboratory in Sweden. First experimental measured database, conducted at 60% load of Kaplan turbine, correspond to near optimal point of operation (case T) and the second one is positioned at the right leg of propeller efficiency curve (case R). Comparison between experimental and calculated results in this paper was performed only for near optimal point of operation which is close to the best efficiency for the system according to the fixed single runner blade angle. Experimental settings were $H=4.5$ m (head), $n=595$ rpm (runner speed) and $Q=0.522$ m³/s (flow rate). Draft tube geometry is shown on Fig 1. with the characteristic cross sections where experimental measurements were done.

The real hydro power plant, which is situated in Indalsälven in Sweden, works with the 27 m head and consists of three Kaplan turbines with a 5.5 m runner diameter, maximum power of 50 MW and flow rate of 230 m³/s per turbine.

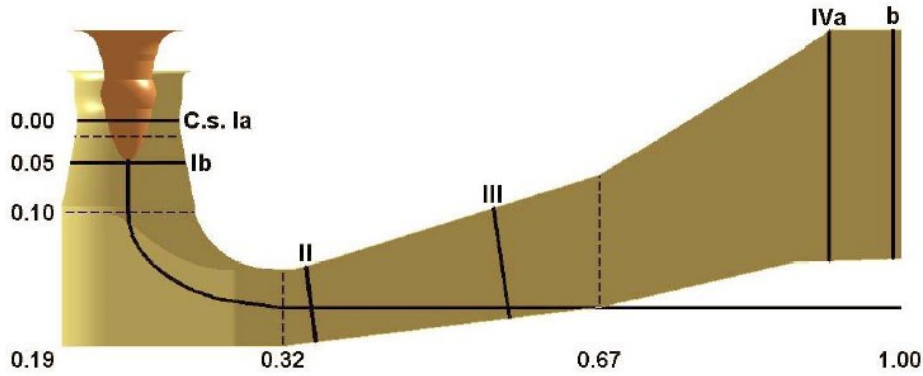


Figure 1. Draft tube geometry

2.1 Experimental data

Velocity LDV measurements were made across the inlet section as well as pressure measurements at the draft tube outlet sections and along upper and lower draft tube centerlines. The velocity components were measured with a two-component fiber optic LDA probe while three-hole Pitote tube probe measurements have been carried out to evaluate static and dynamic pressure. The location of a single point measurement is specified by the radius. Measured mean velocity components are the axial velocity (U ; in the negative z -direction) and the tangential velocity (W ; positive tangential velocity has the same direction as rotating runner.) Corresponding to these mean values, root mean square (RMS) and the cross correlation between the two components has been also determined (u - the RMS value of U , w - the RMS value of W and uw - the cross correlation of U and W). Inlet section LDV measurements are performed 127 mm below the runner hub centre at angle $\alpha=-80^\circ$. It was assumed that the velocity components and turbulent quantities are axis symmetric. Additional assumptions must be made for the radial velocity (V) and the turbulent quantities (v^2 , uv , vw) in order to complete experimental database which will be suitable for fluid flow simulation. The radial velocity (V) at the inlet section is defined from the measured axial (U) and tangential velocity (W) component assuming the flow is attached to the runner and the draft tube wall. According to that assumption radial velocity component equals to:

$$V(r) = U(r) \cdot \tan \Theta \quad (1)$$

where angle Θ is obtained by following expression:

$$\Theta = \Theta_{cone} + \frac{\Theta_{wall} - \Theta_{cone}}{R_{wall} - R_{cone}} \cdot (r - R_{cone}) \quad (2)$$

The unmeasured turbulent quantities (v^2 , uv , vw) are related to the measured turbulent quantities as follow:

$$v^2 = w^2 ; uv = vw = uw \quad (3)$$

The wall pressure distribution at the draft tube outlet is supplied for setting a suitable outlet condition in the fluid flow simulations. Axial and tangential velocity components are influenced by the upstream runner. Influence of runner rotation to tangential velocity can be seen on runner cone ($R=0.10\text{m}$) where, according to no-slip conditions, high tangential velocity components were observed.

Inlet boundary conditions for turbulence models based on k and ε turbulence quantities were calculated from measured normal Reynolds stresses components.

$$k = \frac{1}{2} \cdot (u^2 + v^2 + w^2) \quad (4)$$

$$\varepsilon = \frac{C_\mu^{\frac{3}{4}} \cdot k^{\frac{3}{2}}}{l_\varepsilon} \quad (5)$$

$$l_\varepsilon = 0.1 \cdot (R_{wall} - R_{cone}) \quad (6)$$

Resulting boundary conditions profiles applied at the inlet section are shown on Fig 2.

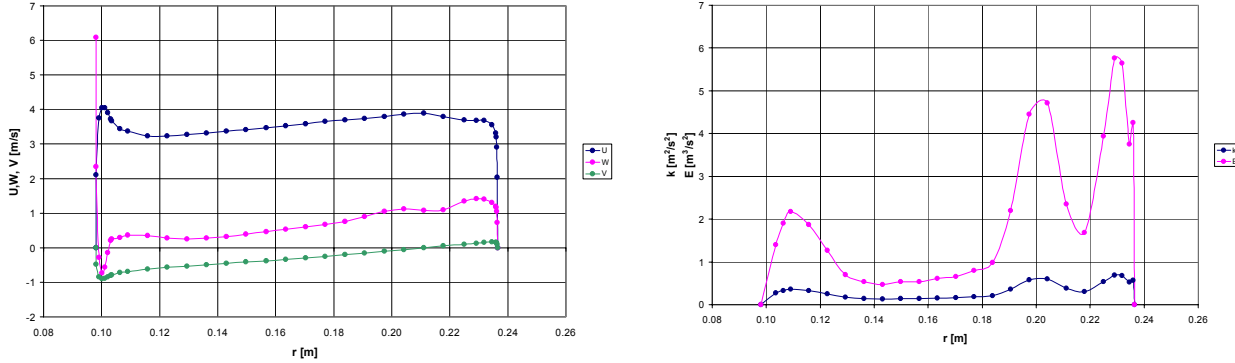


Figure 2. Inlet boundary conditions

2.2 Mesh

Fluent CFD solver was applied for the fluid flow simulation on the same problem using meshes with different number of finite volume cells. Multi block structured grid topology was used for hexahedral mesh generation enabling unstructured mesh generation with different number of finite volume cells with minimal user interaction. All meshes used in this work were created according to turbulence model requirements where each wall adjacent volume cell's centroid is located within the log-law layer ($30 < Y^+ < 300$). In order to satisfy this condition generated mesh was continuously corrected in near wall region (according to the fluid flow solution) until above conditions is not satisfied in the whole computational domain. To investigate influence of the mesh size on results four different meshes (100k, 240k, 500k and 1000k volume cells), each one approximately double sized, were created with the approximately same mesh quality.

3. Numerical modeling

For the fluid flow analysis in the Kaplan draft tube a commercial computer code Fluent (version 6.2) was utilized. Fluent solves Reynolds averaged Navier-Stokes equations applied on finite volumes, where Reynold's stress is calculated from the applied turbulence models. Governing equations for predicting turbulent fluid flow are mass and momentum conservation, Eqs (7), (8):

$$\frac{\partial \rho}{\partial t} + (\rho v_i)_{,i} = 0 \quad (7)$$

$$\frac{\partial (\rho v_i)}{\partial t} + (\rho v_i v_j)_{,j} = -p_{,i} + \left(\mu [v_{i,j} + v_{j,i} - \overline{\rho v_i' v_j'}] \right)_{,j} \quad (8)$$

where v , p , μ and ρ are velocity, pressure, dynamic viscosity and density.

Steady state incompressible fluid flow was assumed for fluid flow simulation in draft tube. Fluent uses a control-volume-based technique to convert the governing equations to algebraic equations that can be solved numerically. Governing equations are solved sequentially (segregated solver). A variety of pressure-based algorithms are available in Fluent. For the present steady-state computations, the SIMPLE [9] algorithm is adopted and second order upwind scheme is used for convection terms discretization in the momentum and turbulent quantities transport equations. The resulting system of equations is solved using an algebraic multigrid method for faster convergence.

Influence of applied turbulence model on calculated results is investigated using standard $k-\varepsilon$ model (SKE), realizable $k-\varepsilon$ model (RKE) and RNG turbulence model [8] with standard wall functions. The major difference between those models is the method for calculation of turbulent viscosity and the way of defining generation and destruction terms in the turbulence dissipation rate equation. Standard $k-\varepsilon$ model [10] is mostly used in industrial applications today due to its robustness and satisfactory accuracy. Enhanced $k-\varepsilon$ turbulence models are better defined and are more consistent with the physics of turbulent flow.

Measured velocity profiles and estimated turbulence quantities are applied at the velocity inlet boundary conditions and free outflow at the draft tube outlet. Measured pressure at the outlet section was used for setting boundary conditions regarding pressure. Draft tube walls are assumed to be solid with no-slip conditions.

4. Results

It is possible to separate the calculated results for C_{pr} distribution along upper centerline in several different parts depending on the manner that the experimental data are followed. In the first part (from the inlet to the beginning of the elbow) calculated results are very similar to the measured data especially at the inlet part where the calculated C_{pr} is almost equal to the measured one. Reaching the elbow the deviation from the experimental data is getting greater, and the greatest deviation is in the elbow part. In the horizontal part of the draft tube, the calculated results approach the experimental data more quickly. Towards the end of domain of the draft tube these results are getting almost equal to the experimental ones.

Although the great deviation in this part, the results clearly shows the tendencies of the experimental measurements because there is sudden pressure fall at the beginning of the elbow and pressure recovery at the end of the elbow (Fig. 3.).

In the elbow part of the lower centerline calculated results are firstly bigger, but afterwards, reaching the end of the elbow, they are getting smaller then the experimental results. This kind of behavior of the results is identical for all sizes of computational meshes. Reason for this might be in limits of the chosen turbulent model for predicting such complex fluid flow in the elbow part of draft tube. Close to the end of the draft tube, in the horizontal part, the calculated results approach the experimental data more quickly and finally at the end of the draft tube (outlet) results are almost equal to the experimental data (Fig. 4.).

RKE turbulent model is getting almost accurate results and the SKE results are somewhat lower

than the experimental ones. From the shown figures it is possible to see that the sizes of the computational meshes have great impact on the calculated results. With the computational mesh refinement results are getting more accurate. Results of the RNG turbulent model, applied on all mesh sizes, are significantly different from the experimental data with lower values of the C_{pr} throughout the whole draft tube domain. Therefore RNG turbulence model should be avoided for fluid flow simulation in this and similar cases.

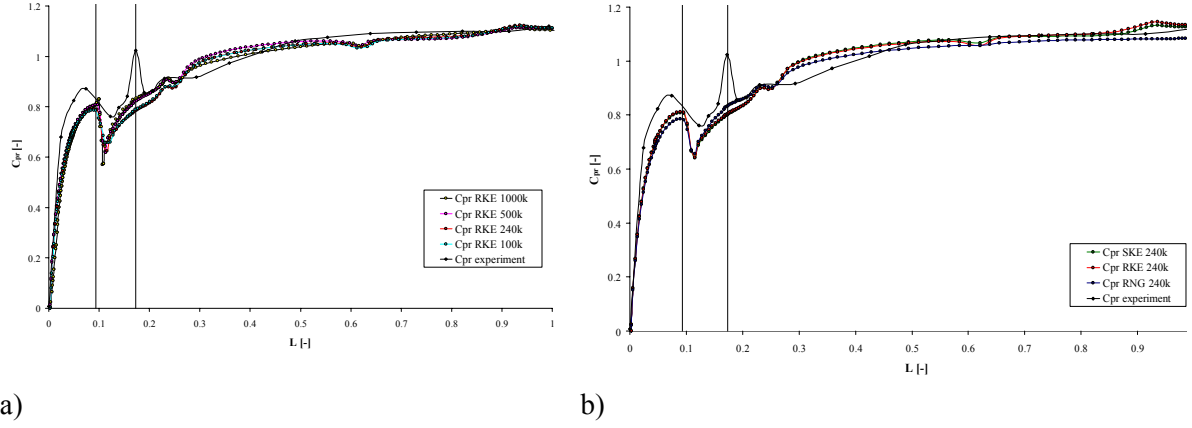


Figure 3. C_{pr} distribution along upper wall centreline ; a) RKE turbulence model, b)240k mesh size

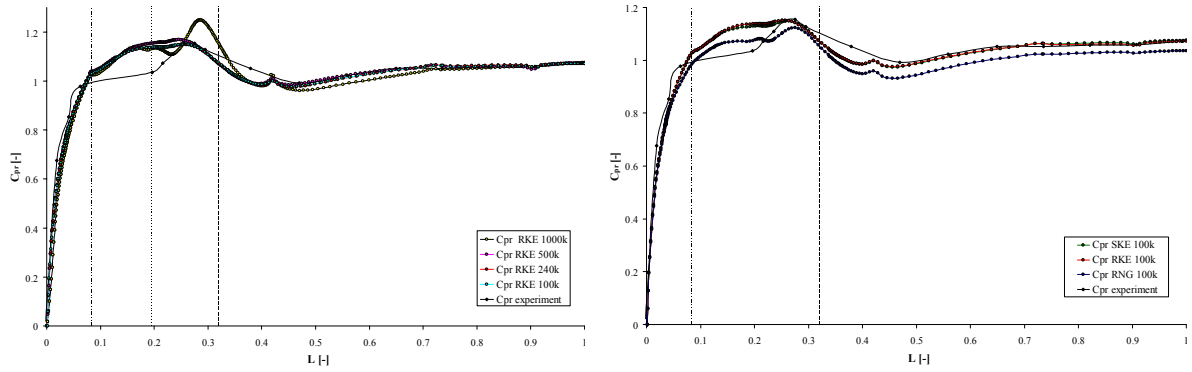


Figure 4. C_{pr} distribution along lower wall centreline ; a) RKE turbulence model, b)100k mesh size

Previous analysis shows that the best results are obtained using RKE turbulence model applied on 1000k mesh size. Therefore following fluid flow visualization will be performed for that case.

Velocity magnitude contours plots at cross section II, III and IVa are shown on Fig. 5. Maximum values of calculated velocities are found on all horizontal draft tube sections at the left draft tube side (seen from the flow direction). Velocity magnitude values decrease continuously approaching outlet draft tube section where height recirculation zone was found. This recirculation zone is clearly visible on Fig. 6. where iso-surface of constant X velocity components (iso-surface of $V_x=0$), colored with static pressure contours, is shown. This recirculation zone negatively affects fluid flow decreasing effective fluid flow area; locally increasing velocity magnitude (as well as fluid flow losses) and finally decreasing overall draft tube pressure recovery. By appropriate draft tube design such recirculation regions should be avoided or minimized as much as possible.

The estimated radial velocity profile applied at the inlet section keeps the flow attached to the rotating runner cone and outer wall (Fig. 7.). Only at the end of runner cone a small recirculation zone, under the influence of re-circulation region below the runner cone, was observed. The flow downstream of the runner cone is developed in the wake, with negative axial velocities region just below runner cone.

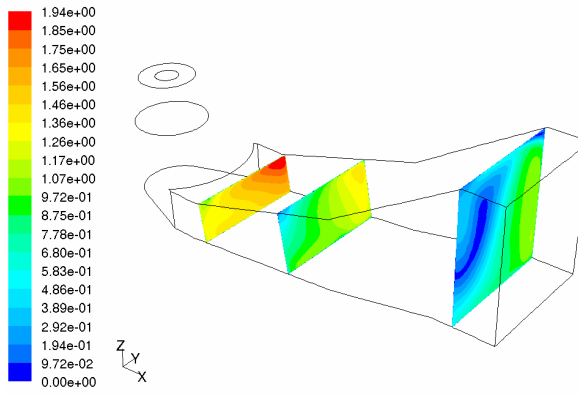


Figure 5. Velocity magnitude plots

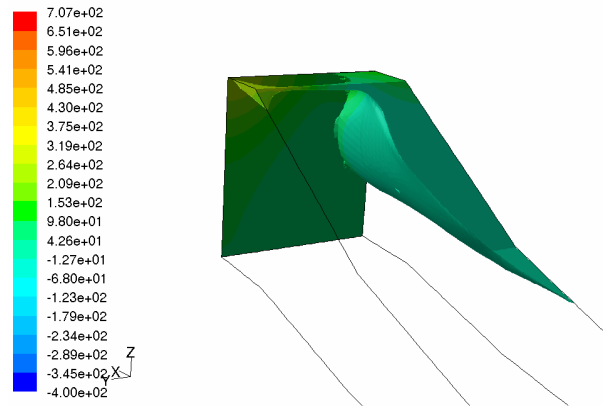


Figure 6. Recirculation zone

Secondary flow patterns can be investigated by plotting velocity vectors on cross sections II and III (Fig.7. b). From Fig.7. b) it is obvious that one dominant vortex structure exist in domain with its center slightly aligned on draft tube right side. For the investigated optimal point of operation, incoming water from the runner has almost axial direction with slightly vortex character according to the tangential velocity profile applied at domain inlet section. Incoming vortex, from turbine runner, has the same direction as runner rotation. Elbow type draft tube additionally generates secondary flow in the horizontal part of draft tube. Dominant vortex structure is additionally amplified by the central vortex generated by runner cone rotation (Fig. 9.).

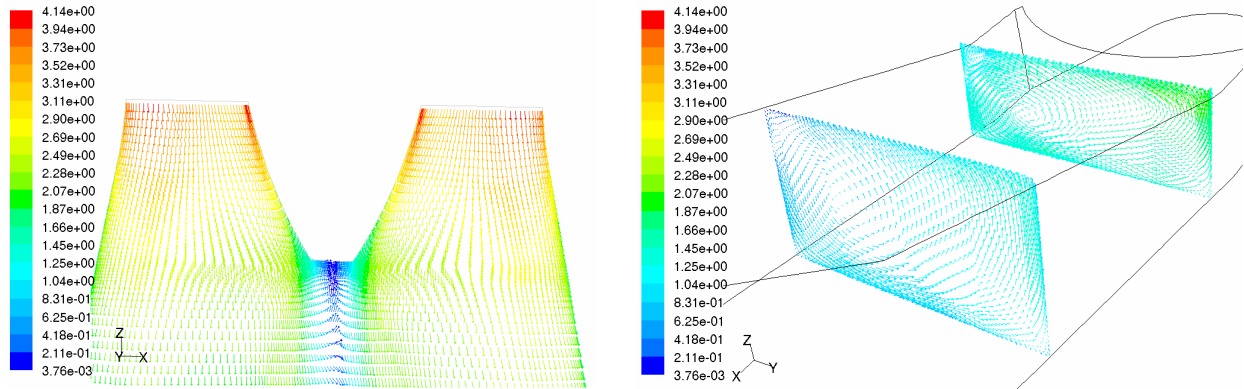


Figure 7. Velocity vectors plots; a) central surface, b) cross sections II and III

In Fig. 8. pressure contours are shown plotted on central surface section along draft tube. Strong pressure recoveries with large pressure gradients are visible in the starting, vertical, part of draft tube. At the draft tube bend there is a large increase off static pressure due to the fluid flow deceleration in that region. Main draft tube pressure recovery part is archived in vertical part of draft tube while after draft tube elbow pressure slowly grows to its final value at the outlet section. Flow behavior in draft tube can be visualized with path lines analyses performed by releasing fluid particles from the curves of constant radial-coordinate values at the inlet section. Path lines released from inner inlet sections (lower radial coordinate values) are surrounded with path lines released from outer sections forming large dominant vortex in the whole flow domain. Path lines released from inner section (closer to runner cone) contains higher swirl intensity, generated by rotating runner cone, which is propagated towards draft tube outlet section.

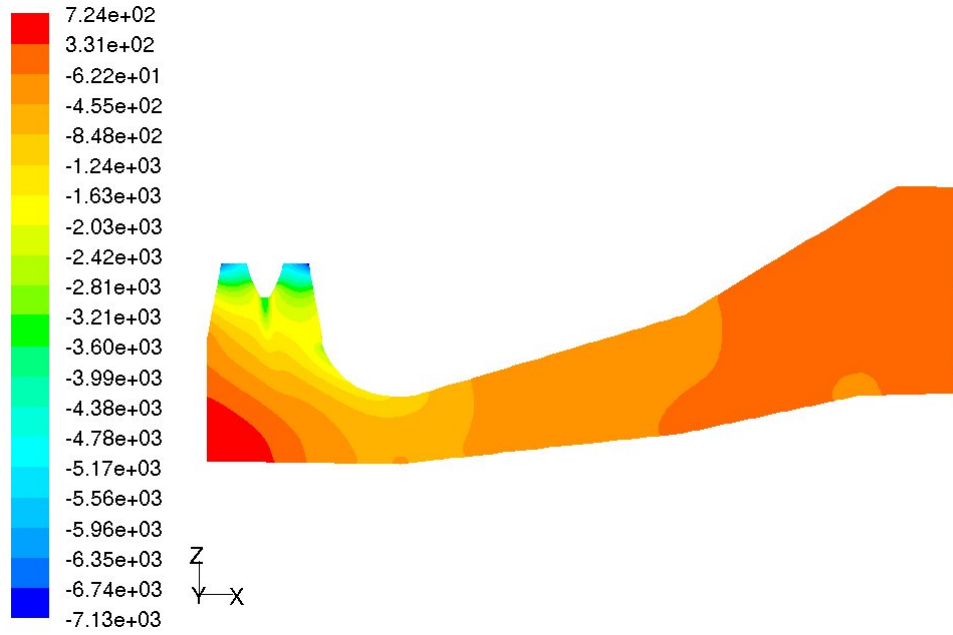


Figure 8. Pressure contours

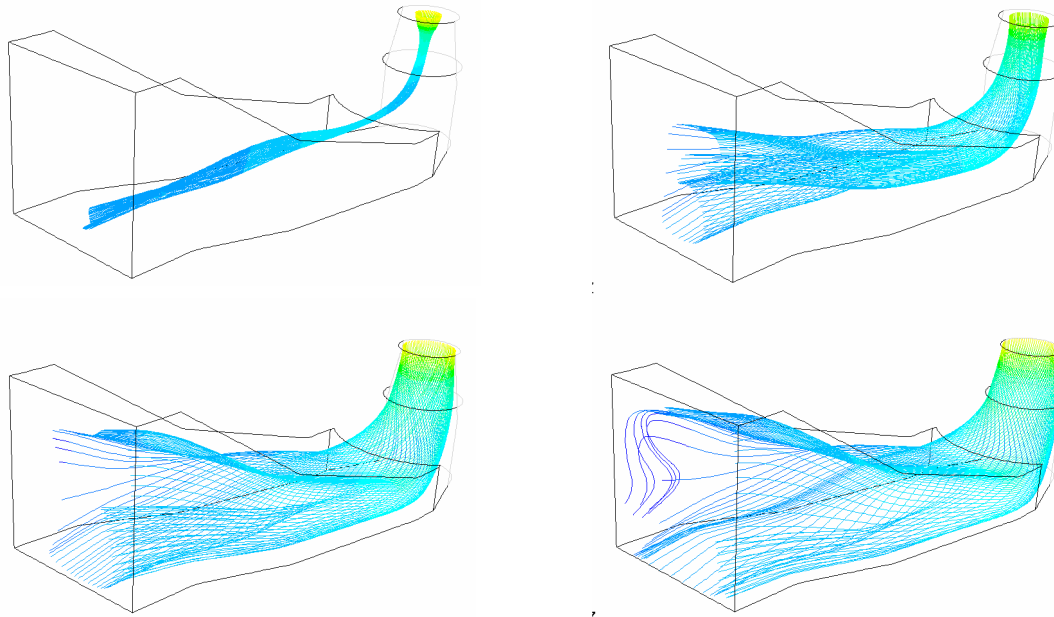


Figure 9. Path lines in draft tube released from different inlet radial sections

Recirculation zone, at the draft tube outlet region, influences dominant vortex by compressing (in horizontal direction) and stretching it (in vertical direction). Elbow type draft tube also induces vortex deflection from right wall (just after draft tube elbow) when fluid core is directed to the left (in flow direction) of the draft tube centerline. This deflection could be a reason why recirculation zone is forming at the domain outlet.

5. Conclusion

Fluid flow simulation in Hölleforsen hydropower draft tube model, using commercial fluid flow solver Fluent, was presented. Fluid flow analyses was investigated using three $k-\epsilon$ turbulence models (SKE, RKE and RNG) applied on several hexahedral meshes each one approximately double sized (100k, 240k, 500k and 1000k volume cells).

Fluid flow simulations are performed for the optimal efficiency point of operation on propeller curve. Inlet boundary conditions are given by measured data at draft tube inlet section. Data that could not be measured (radial velocity, some components of the Reynolds stresses and dissipation length scale at the inlet section of the draft tube) but are essential for fluid flow simulations were assumed in order to make more accurate numerical simulations.

Calculated pressure recovery distributions along upper and lower wall centre line compared with the experimental data are very good. Larger deviations from experimental results are found in elbow part of draft tube. This is reasonable because this is a region with highly complex fluid flow structure where RANS turbulence models are probably not enough appropriate for fluid flow prediction. After the elbow in horizontal draft tube part calculated pressure recovery almost equals to measurements.

RKE and SKE turbulence models have shown the robustness with suppressing level of accuracy predicting draft tube engineering quantities very close to measured one. RNG turbulence model appears bad, predicting lower value in whole domain and its application should be avoided for this and similar problems. Strong grid size influence was observed on calculated results especially on pressure distribution near inlet section where non realistic high pressure region are found when using coarser grids. This influence disappears with grid refinement in this area. Anyway special attention should be dedicated to mesh generation near draft tube inlet section, in order to correctly propagate applied boundary conditions to the fluid flow domain.

References

- [1] Čarija Z, Mrša Z. Complete Francis Turbine Fluid Flow Simulation. Proceedings of the 13th International DAAAM Symposium 2002; ISBN 3-901509-13-5
- [2] Eisinger R, Ruprecht A. Automatic Shape Optimisation of Hydro Turbine Components based on CFD. Seminar "CFD for turbomachinery applications" 2001.
- [3] Heitele, M., Helmrich, T., Maihofer, M., Ruprecht, A. "Hew Insight ito an Old Product by High Performance Computing". 5th European SGI/CRAY MPP Workshop, Bologna, 1999..
- [4] Lipej A, Poloni C. "Design of Kaplan Runner Using Multiobjective genetic algorithm optimisation", Journal of Hydraulic Research 2000; Vol. 38.
- [5] Mrša Z, Sopta L, Vuković S. "Shape optimisation method for Francis turbine spiral casing design", ECCOMAS 2000.
- [6] Ruprecht A, Eisinger R. "Numerical Simulation of a Complete Francis Turbine including unsteady rotor/stator interactions", 20th IAHR Symposium on Hydraulic Machinery and Systems 2001.
- [7] Ruprecht A, Eisinger R, Göde E, Rainer D. "Virtual Numerical Test Bed for Intuitive Design of Hydro Turbine Components", IHA 1999.
- [8] Fluent 6.2 User manual, Fluent.Inc, 2006.
- [9] Patankar S. V., Numerical Heat Transfer and Fluid Flow, Hemisphere, Washington, D.C., 1980.
- [10] Launder B. E., Spalding D. B. The Numerical Computation of Turbulent Flows. Computer Methods in Applied Mechanics and Engineering, 1974.

Zoran Čarija, mr.sc.

Faculty of Engineering, University of Rijeka, Department of Fluid Mechanics and Computational Engineering, Vukovarska 58, Rijeka, Croatia, Phone: 0038551 651554, e-mail: zcarija@rijeka.riteh.hr

Zoran Mrša, prof.dr.sc.

Faculty of Engineering, University of Rijeka, Department of Fluid Mechanics and Computational Engineering, Vukovarska 58, Rijeka, Croatia, Phone: 0038551 651554, e-mail: mrsa@rijeka.riteh.hr

Luka Dragović, dipl.ing., e-mail: lukad@rijeka.riteh.hr