4th International Congress
of Croatian Society of Mechanics
September, 18-20, 2003
Bizovac, Croatia

4thICCSM

Complete Francis turbine flow simulation for the whole range of discharges

Z. Čarija and Z. Mrša

Keywords: *CFD, Numerical simulation, Francis turbine, Fluent, Parallel computing, Linux cluster*

1. Introduction

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 can be solved numerically. Computational result strongly depends on methods used for converting governing to algebraic equations and on the choice of the fluid flow domain. Chosen technique depends on available computer resources and on possibility to simplify domain of interest. Mean techniques used for simulating fluid flow in hydraulic machines will be shortly described.

Simulating fluid flow through different turbine components separately is today standard procedure. The main idea is to assume steady state uniform conditions in circumferential direction at the spiral-casing outflow. Accordingly only one geometrically periodic part of tandem cascade and runner blades has to be considered. This simulation results in severe saving of computer resources but with less accurate results. The obtained results are still sufficiently accurate for the routine everyday design, but are inaccurate for optimisation.

Fluid flow simulation of the entire Francis turbine is desirable when spiral casing has non-uniform distribution of the incoming water or when strong interaction between turbine components exists [3]. Obtained results are more accurate because calculation domain contains all cascade channels and no averaging is applied in circumferential direction.

Above methods assumes that the flow field is steady and such methods are sometimes called "frozen rotor" approach.

For more accurate results, unsteady fluid flow simulation of entire turbine is required where interaction of turbine components is taken into account [6]. Resulting computational grids are huge and computationally expensive. Such grids can be handled by parallel computing using domain decomposition.

In this paper nonuniform circumferential distribution of the flow through the output of the spiral casing was treated. Fluid flow simulation was done for the complete Francis turbine and not only one geometrically periodical part [1]. Another reason for simulating entire flow is the strong interactions between the components (especially between guide vanes, runner and draft tube). For more accurate results it is inevitable to take care of these interactions into the fluid flow numerical simulation, which can only be done by unsteady flow simulation [6].

CFD is also involved in turbine design and shape optimisation of turbine components either in design of completely new turbine or in revitalization of old ones where only selected parts of the turbine have to be optimised. It is challenging work because each turbine has to be designed according to the local situations like given head, discharge etc. Many geometry variations are necessary to obtain optimal shapes with superior hydraulic efficiencies. Today the shape optimisation process is completely based on CFD methods. Ruprecht presented two different approaches for the shape optimisation 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, analysing flow interactively [7]. Another approach is based on mathematical optimisation tools where suitable 'quality function' has to be defined which will be criteria in optimisation process. The quality function depends on specific situation and requires a lot of experience to define it very well. Gradient [2] and genetic [4], [5] methods are widely used today in the shape optimisation of turbine parts. Gradient methods can detect local minimum instead of the global whereas genetic methods are more robust, non-sensitive on local minimum but requires more optimisation cycles.

Methods for runner blade shape construction and shape optimisation with genetic algorithm as optimisation tool is under development.

2. Investigated geometry and computational grid

Francis turbine fluid flow numerical simulation will be presented. The investigated turbine consists of spiral casing, tandem cascade with 10 stay vanes and 20 guide vanes channels, runner with 15 blades and draft tube.

Geometry definition and grid generation was carried out with the FLUENT preprocessor GAMBIT, where the turbine geometry and mesh definition was parameterised using GAMBIT journal files. Additional program was developed to interactively change these parameters and rebuild computational mesh. Because of the unsymmetrical distribution of incoming water from the spiral casing, the complete turbine including all flow channels in the tandem cascade and in the runner was considered. The whole calculation domain is divided into five components (spiral casing, stay vane, guide vane, runner and draft tube). For each component the computational grid domain is generated individually. These component domains are finally merged using FLUENT merge utilities into one computational domain with the non-conformal meshes on grid interfaces.

Final grid is huge and it is decomposed for the parallel processing. The computational mesh of the complete turbine is shown in Fig.1 where different colours represent different processors for the calculation. Calculation was performed using parallel network of workstation (cluster of five AMD Athlon1.6 workstations) running Linux OS.

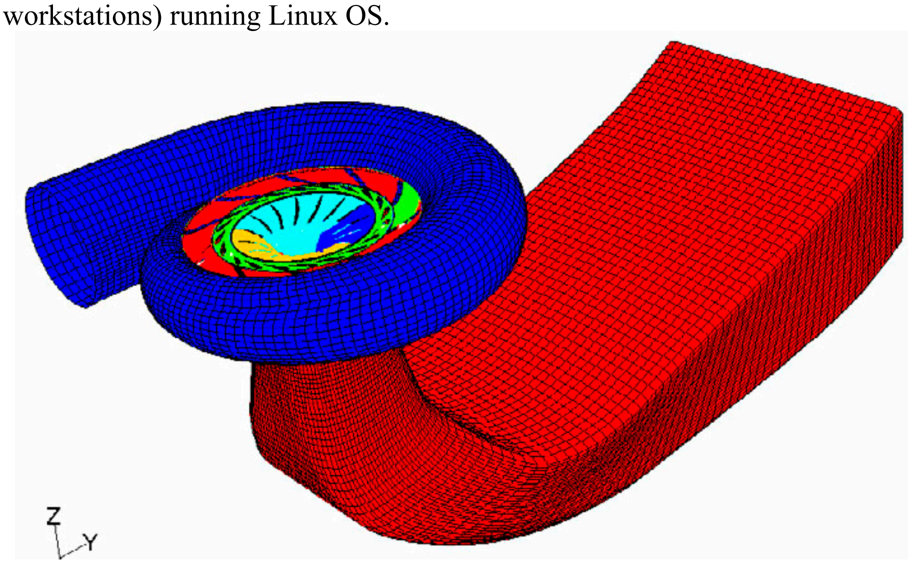

Figure 1. Computational grid

The whole computational grid consists of more then 2.3 million volume cells. Computational grid of that size can be generated in approximately 20 minutes with parameterised mesh definition using GAMBIT journal files.

3. Numerical modelling of turbulent fluid flow

For the fluid flow analysis of the entire Francis turbine commercial computer code FLUENT was used. Reynolds averaged Navier-Stokes equations are applied on finite volumes where Reynolds stress is calculated from the standard k-ε model. Governing equations for predicting turbulent fluid flow are mass and momentum conservation, Eqs (1), (2):

$$\frac{\partial \rho}{\partial t} + \left(\rho v_i\right)_{,i} = 0 \tag{1}$$

$$\frac{\partial \left(\rho v_i\right)}{\partial t} + \left(\rho v_i v_j\right)_{,j} = -p_{,i} + \left(\mu\left[v_{i,j} + v_{j,i} - \rho \overline{v_i' v_j'}\right]\right)_{,j} \tag{2}$$

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

FLUENT uses a control-volume-based technique to convert the governing equations to algebraic equations that can be solved numerically. Steady state incompressible fluid flow was assumed. Governing equations are solved sequentially (segregated solver). Multiple reference frame option was used to simulate fluid flow in moving zones (runner). This is a steady-state approximation where individual cell zones move at different rotational speeds. This approach is appropriate when the flow at the boundary between these zones is nearly uniform. This model can be used for a turbomachinery application in which rotor-stator interaction is relatively weak or as initial condition for the unsteady fluid flow simulation. The turbulence is taken into account by the standard k-ε model, which is mostly used in industrial applications today. For the unsteady problems this model is not suitable because it tends to over-predict eddy viscosity. Non-conformal boundaries are used at the interfaces between five grid sub-domains. This boundary condition permits not coincident grid node locations at the boundaries where two sub domains meet, which is very helpful in the process of the grid generation. Fully developed pipe flow at the spiral casing inlet and free outflow at the draft tube outlet were assumed for boundary conditions.

4. Results

Numerical simulation and measurements were done for the same specific head ψ (prototype at the head H=125m) and different specific flow rates φ.

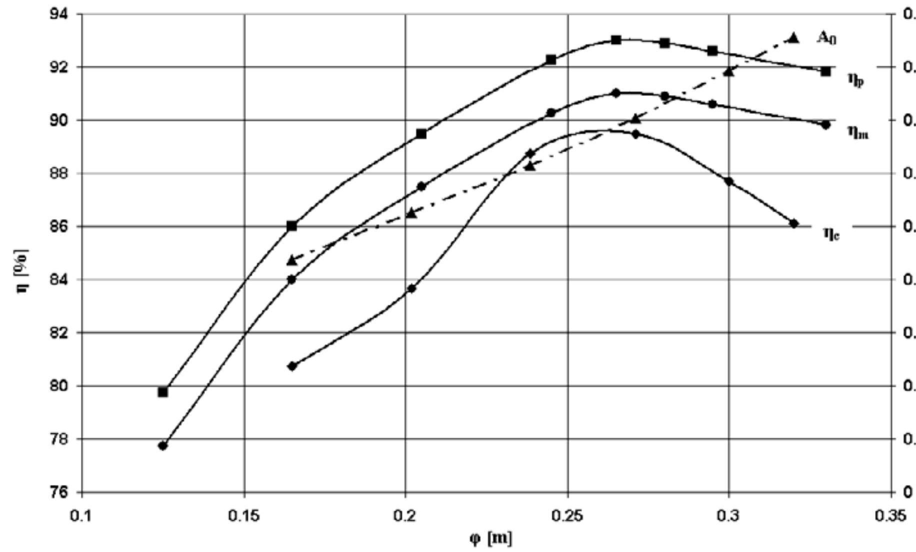

Figure 2. Measured and calculated efficiencies

Results of the numerical simulation were compared with the measurements. Model measurements were carried out on the Turboinstitut test ring and these data are then converted to prototype data.

According to the IEC 60193 standard, prototype efficiency is 2% higher then model efficiency. Calculated, model and prototype efficiency are shown on Fig.2 (***p*** subscript stands for prototype, ***m*** for model and ***c*** for calculated efficiency) with the wicket gate-opening curve. Calculated results show in average 2% lower efficiency then measured ones. This is due to the still coarse grid and limitation of the applied turbulence model. These differences are in good agreement with the measurements and observations at the Turboinstitut. Numerical simulation and measurements predict the highest efficiencies (optimal operation point) at the same specific flow rate with minimum difference. Much higher drop of calculated efficiency was found left and right from the optimum point of operation. 2nd order upwind discretization gave superior results compared to first order for which this difference was 4-5% bigger.
Path lines (colored by velocity magnitude) in draft tube for two different points of operation are shown on Fig. 3-6. Each point is represented with the path lines released from the draft tube inlet section and with path lines released only from the line on that section. From figures it is clearly visible that the draft tube flow strongly depends on the point of operation. At the low load operation point (A_0=0.39), path lines go downstream in clockwise direction entering the draft tube.

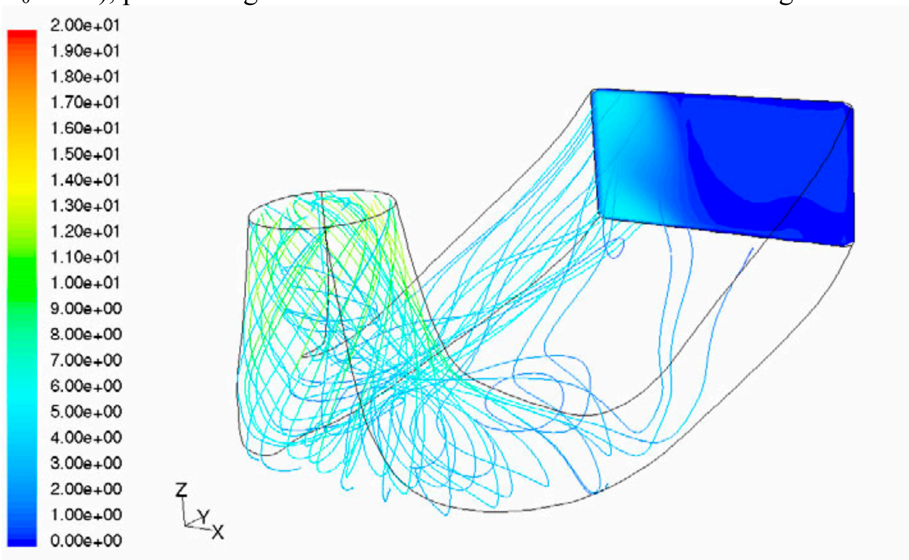

Figure 3. Path lines in draft tube (A_0=0.39) released from inlet section

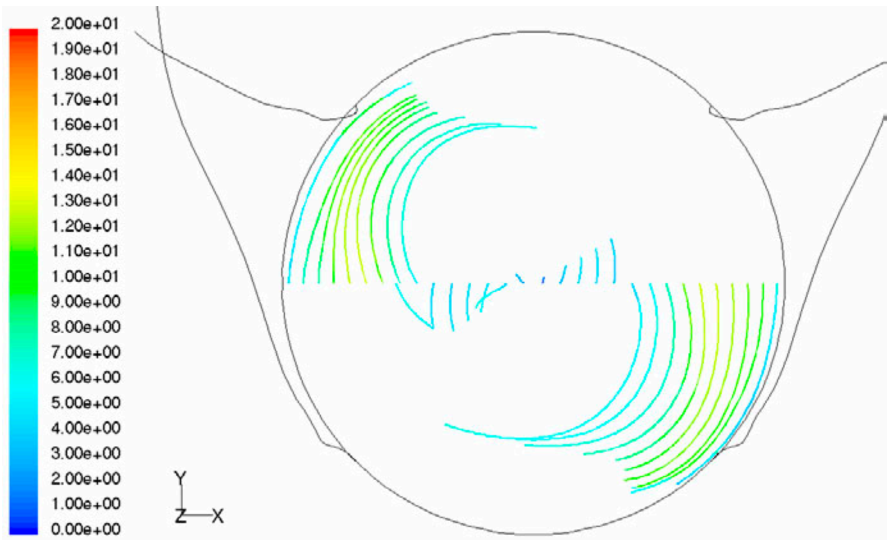

Figure 4. Path lines in draft tube (A_0=0.39) released from line on inlet section

Vortex change its direction at the optimal point and at the A_0=0.86 position of wicket gate (high load) it has counter clockwise direction. At the optimal point of operation incoming water from the runner has almost axial direction without vortex, but elbow draft tube generate secondary flow in horizontal part of draft tube. Such secondary flow in draft tube for non-optimal point of operation (A_0=0.86) is shown on Fig.5 where two vortexes are clearly visible in horizontal part of draft tube. At the low load operation point (A_0=0.39) path lines concentrate at one side of the draft tube, while at higher load (A_0=0.86) discharge is on both sides of draft tube according to the secondary elbow generated two vortexes.

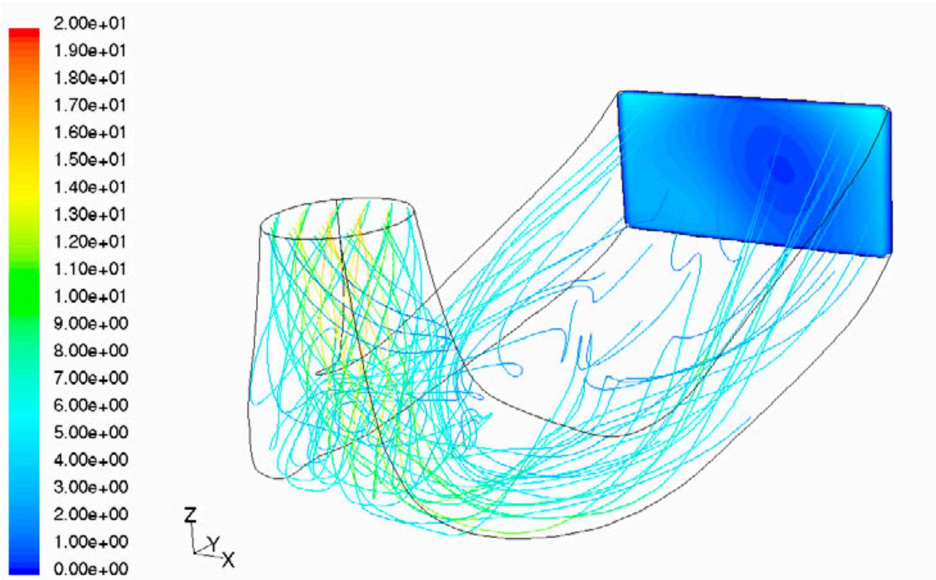

Figure 5. Path lines in draft tube (A_0=0.86) released from inlet section

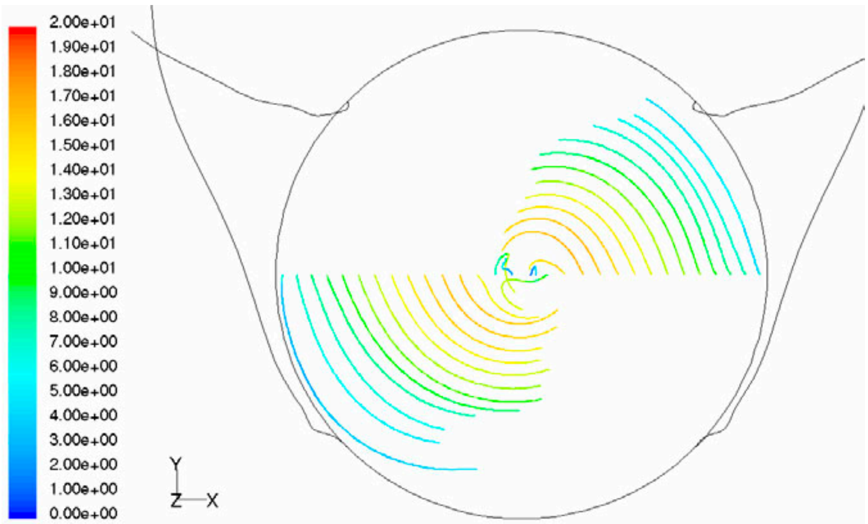

Figure 6. Path lines in draft tube (A_0=0.86) released from line on inlet section

Pressure distributions at the runner blades for two points of operations are shown on Fig.7 and Fig.8. Pressure and suction sides of runner blades are clearly visible and stagnation places on runner blades. Pressure differences from the pressure and suction sides are higher at the higher load (A_0=0.86) and these differences result in torque on the shaft. The lowest pressures are on the suction side near the trailing edge. This is the position where cavitations occurs at the high loads but presented numerical model does not take cavitations into account so there are unrealistic low pressures at the suction side of runner blades.

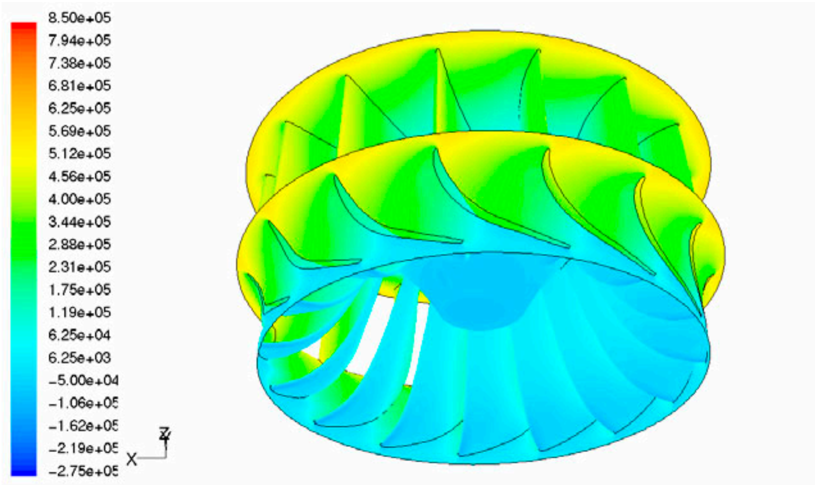

Figure 7. Runner static pressure distribution (A_0=0.39)

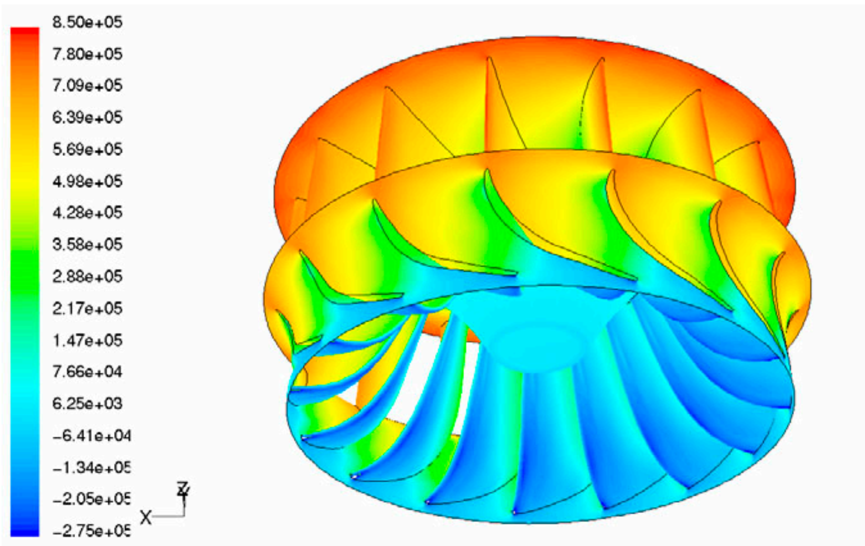

Figure 8. Runner static pressure distribution (A_0=0.86)

5. Conclusion

Simulation of the steady fluid flow with moving reference frame in the entire Francis turbine consisting of the spiral casing, draft tube, 10 stay vanes, 20 guide vanes and 15 runner blades is presented. Resulting computational grid is huge and flow must be computed in parallel by domain decomposition. Calculation was performed using commercial fluid flow solver FLUENT on a Linux cluster. The obtained efficiencies in different points of operation are in average 2% lower then measured ones due the still coarse grid and limitations of applied turbulence model. Predicted and measured optimal operation points are found at the same specific flow rate. This is also the po-

sition of the lowest differences between calculated and measured efficiencies.
CFD can be used in design process of different turbine parts because obtained results are sufficiently accurate compared to measurements.
Numerical simulation of entire Francis turbine flow shows clearly the influence of the wicket-gate opening on the draft tube vortex existence and direction. Vortex changes its direction from clockwise to counter clockwise when going from small to height discharges. It disappears at optimal point of operation lowering draft tube and overall turbine losses. The main part of the whole turbine losses are draft tube losses.

Using CFD it is possible to get significant insight into issues such as energy transfer and fluid flow details in hydraulic turbines. Only the last fine-tuning has to be done experimentally.

References

[1] Čarija Z, Mrša Z. Complete Francis Turbine Fluid Flow Simulation. Proceedings of the 13th Interntional 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.

Zoran Čarija, assist.
Faculty of Engineering, University of Rijeka, Department of Fluid Mechanics and Computational Engineering, Vukovarska 58, Rijeka, Croatia, Phone: 0038551 651554, Fax.: 0038551 651-490, 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, Fax.: 0038551 651-490, e-mail: mrsa@rijeka.riteh.hr