USAGE OF SUPERCOMPUTERS IN DEVELOPMENT PROCESS OF HYDRAULIC MACHINES

Andrej Lipej
Assoc. prof., Ph. D.
University of Novo mesto,
Faculty of mechanical engineering
Na Loko 2, Novo mesto, Slovenia

andrej.lipej@fs-unm.si

Numerical analysis of flow in water turbines and pumps is important because it allows a reduction in expensive and time-consuming measurements and gives insight into the flow in all machine parts. On the basis of numerical results it is easier to find the reasons for low efficiency or for cavitation, and to improve the hydraulic shapes of all turbine and pump parts. For small projects, model tests are too expensive and Computational Fluid Dynamics - CFD analysis is the only way to foresee whether the required efficiency and cavitation characteristics of the prototype will be obtained. To obtain accurate results, very fine computational grids, appropriate turbulence models and the unsteady models are required. These conditions are responsible for a very long computational time with the calculations often taking several weeks. The computational time is a million times longer than the real time of the analysed process for modelling an unsteady phenomenon. Our goal is to achieve numerical results in real time. This can be attained using very fast processors or massively parallel computer clusters. Very long computational times are present using optimization algorithms in the development process, when automatically perform the design based on parametrized basic geometry, boundary conditions and prescribed objectives.

Keyword: *water turbine, pump, unsteady flow, multi-phase flow, optimization methods*

1 Introduction

CFD simulations in hydraulic machines are nowadays mainly based on Reynolds-Averaged Navier-Stokes (RANS) turbulence models. It is clear that some applications can be better covered by turbulence models when a part of the turbulence spectrum is resolved in at least a part of the computational domain. Such methods are known as Scale-Resolving Simulation (SRS) models [1]. Nowadays in different commercial CFD software, many SRS models are possible to use.

There are many reasons for using SRS models instead of RANS formulations. The first important reason for using SRS models is the need for more information of the flow that cannot be obtained using the RANS models. Examples can be found in multi-physics effects like vortex cavitation where the unsteady pressure field is the cause of cavitation. In such situations, the need for SRS can exist even in cases where the RANS model would in principle be capable of computing the correct time-averaged flow field.

The second reason for using SRS models is the need for high accuracy, which is very important in hydraulic machinery analysis. We know from the experience that RANS models have some limitations in accuracy in certain applications. The flow in hydraulic machines is mainly wall-bounded and RANS models have shown their strength essentially for such flows. For some special cases where we have free shear flows, the performance of RANS models is less efficient. There are wide variety of such examples, start from simple flows such as jets, mixing layers and wakes to flows, flows with strong swirl, massively separated flows and many more.

Paper deals with two types of hydraulic machines, water turbines and pumps. Water turbines are essential for the realisation of the Sustainable Development Goals (SDGs) and the Paris Agreement. They are still one of the important pillars of the renewable energy sources (RES). Investment in renewable electricity in the near future will be huge and important portion will be reserved for hydropower. The water turbines are important also because a single turbine can have a huge capacity (around GW) in comparison with other type of thermal, nuclear, wind or solar, complete power stations.

Other important machines are pumps, which are one of the biggest consumer of the electric energy. Their energy consumption in the use phase is the most significant environmental aspect of all life-cycle phases with their annual electricity consumption amounting in Europe of 109 TWh in 2005, corresponding to 50 Mt in CO_2

emissions. If the use of energy in the future will not be reduced, it is predicted that energy consumption will increase to 136 TWh in 2020. It has been concluded that use-phase electricity consumption can be reduced significantly.

All above-mentioned facts are sufficient reasons to continue with the research and development in the area of hydraulic machinery. In all industrial problems, the flow is turbulent. The analysis of turbulent flow can cause many problems.

Basic equations: continuity and momentum

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho U) = 0$$

$$\frac{\partial (\rho U)}{\partial t} + \nabla (\rho U \otimes U) = -\nabla p + \nabla \bullet \tau + S_M$$

Rotational forces

For flows in a rotating frame of reference, rotating at a constant angular velocity, additional sources of momentum are required to account for the effects of the Coriolis force and the centrifugal force:

$$S_{M,rot} = S_{Cor} + S_{cfg}$$

where Coriolis force is presented by

$$S_{Cor} = -2\rho\omega \times U$$

and centrifugal force is

$$S_{cfg} = -\rho\omega \times (\omega \times r) \ .$$

In the article, we will discuss phenomena that require very powerful computers to achieve high quality and useful results. First, the pressure fluctuation in draft tubes is presented. Hydraulic instability associated with pressure fluctuations is a serious problem in hydraulic machinery. Pressure fluctuations are usually a result of a strong vortex created in the centre of a flow at the outlet of a runner. Second, the rotor stator interaction in reversible pump turbine (RPT) is analysed. The main reason was investigation of unsteady calculation of rotor-stator interaction in pump mode to find out the time dependent distribution of the torque on guide vanes. The third part presents problems in low-head turbines. Fourth case deals with the analysis of free surface flow in Pelton turbine. Fifth part of the paper deals with the dynamic problems in pumps. In the last part of paper, some new approaches in CFD analysis for the drastic reduction of CPU time and optimization methods for efficient design of different hydraulic machines based on multi-objective genetic algorithms are presented.

2 Supercomputers

The organization Top 500 presents the list of the most powerful computers in the world twice a year. In June 2018, the leadership took over the computer named Summit. It is an IBM-built supercomputer now running at the Department of Energy's (DOE) Oak Ridge National Laboratory (ORNL), captured the number one spot with a performance of 122.3 petaflops on High Performance Linpack (HPL). Installed power of the system is 8.8 MW. It consists of 2,282,544 processor cores. This is not the highest number of processors but because of powerful 22-core Power9 CPUs, and NVIDIA Tesla V100 GPUs, the system is the fastest in the world. It took the first place from the Sunway Taihu Light supercomputer from China, which consists of 10,649,600 processor cores (15.4 MW).

3 Pressure fluctuations in draft tubes

The draft tubes are a necessary part of hydraulic turbines because they are responsible for conversion of kinetic energy at the runner outlet into pressure energy. At off-design conditions, the flow in the draft tube has a transport and a circumferential component. One difficulty in hydraulic turbines is the appearance of the vortex rope (Fig. 1) which is related to the strength of the swirl at the runner outlet [2]. The vortex rope in draft tube causes pressure oscillations. Final consequences are dynamic shaft forces and vibrations, which usually lead also to bearing problems. Hydraulic instability associated with pressure fluctuations is a serious problem in hydraulic machinery.

Fig. 1 Vortex rope in draft tube of FT

It is possible to use Unsteady Reynolds Averaged Navier-Stokes (URANS) models for the transient calculation but the URANS simulation produces only the large-scale unsteadiness. The Scalable Adaptive Simulation (SAS) Shear Stress Transport (SST) method is an improved URANS formulation with the ability to adapt the length scale to resolved turbulent structures.

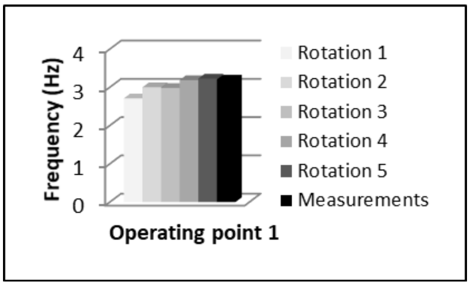

Fig 2. Frequency after several complete vortex rotation and comparison with measurements

The SAS concept is based on the introduction of the von Karman length-scale into the turbulence scale equation. The information provided by the von Karman length-scale allows SAS models to dynamically adjust and to resolved structures in a URANS simulation, which results in a Large Eddy Simulation (LES)-like behaviour in unsteady regions of the flow field. At the same time, the model provides standard RANS capabilities in stable flow regions.

Using SAS model, the prediction of pressure fluctuation frequencies and amplitudes is quite accurate. If we compare the accuracy of frequencies and amplitudes, we can claim that we mainly obtain more precise results in the analysis of frequencies than in the amplitude analysis.

In order to achieve a convergent solution, we need to have sufficiently long computational times. This is seen in Fig. 2, where the results are displayed after each turn of the vortex. The exact value approaches the experimental solution only after a few full revolutions of the vortex.

4 Rotor stator interaction in RPT

Conditions in a typical RPT mostly point out the importance of the unsteadiness of the flow, above all because of its instability caused by the viscous effects – the boundary layer separation, vortex spreading and particular the influence between rotating and non-rotating part [3]. These effects cause pressure pulsations with different orders of magnitude of the frequencies and amplitudes. The numerical simulation of the two mentioned and completely different causes for the unsteadiness must enable effective solution – using normal computational time – of very large computational grids and enable simultaneous calculation of large number of time steps.

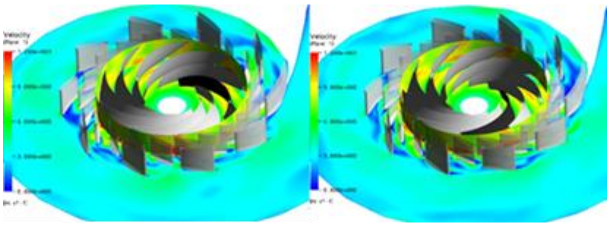

Fig. 3 Unsteady flow in reversible pump-turbine – two time steps

The numerical analysis of the torque on guide vane pivots (Fig. 3) in reversible pump-turbine show some difficulties in obtaining the reliable results.

For pump mode, the unsteady analysis is necessary. Stable results are obtained only after a long computation time and using sufficiently small time steps. Figure 4 shows the time distribution of the torque to the guide vane pivots for all twenty guide vanes.

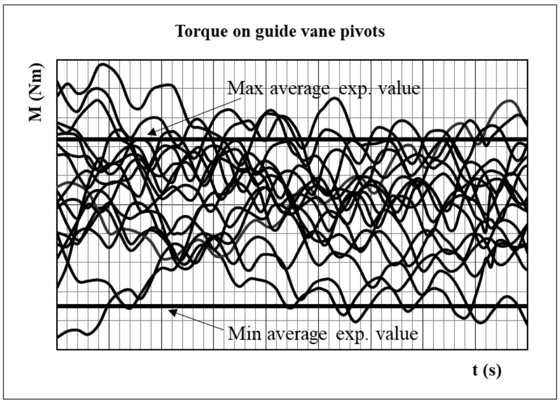

Fig. 4 Time dependent distribution of torque – pump mode

The minimum time of calculation has to be adequate to several runner revolutions and the length of a single time step must not be larger than two degrees of the runner revolution.

5 Analysis of low head turbines

Numerical analysis of low head turbines is not always successful because at full rate the losses in draft tube can be over predicted. The results are better, if the unsteady analysis instead of steady state calculation is used. Usually SRS models can resolve turbulence structures in the draft tube more detailed (Fig. 5).

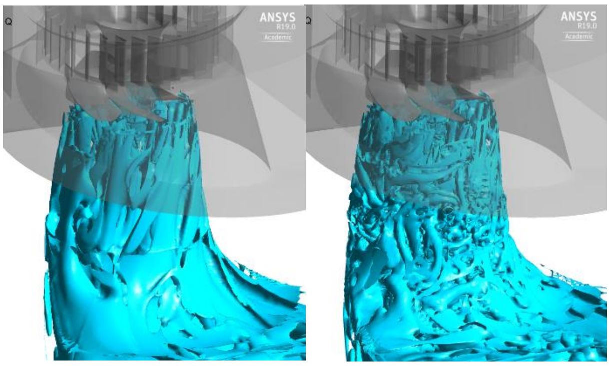

Fig. 5 Turbulence structure – velocity invariant Q (URANS SST – left, SAS-SST – right)

If we expect any problems with numerical accuracy, it is useful to check the turbulence structures visually by using, an iso-surface of the Q-criterion,

$$Q=1/2(\Omega^2-S^2),$$

where S is Strain rate and Ω - vorticity. Q-criterion is the second invariant of the velocity gradient tensor.

Energy losses at full load are very high, if the ratio of eddy viscosity and dynamic viscosity is large. It can be seen that the above-mentioned ratio can be around ten times smaller when the appropriate SRS turbulence model is used instead of RANS turbulence models.

The comparison between model test and numerical results shows that at full load the difference of efficiency can be improved from Δη=8 % to Δη=0,5 %.

For such results improvements we usually need to use a Zonal LES (ZLES) model [4], with much longer CPU time and usage of very powerful computers. Inside the prescribed zone, the LES model is used to resolve the flow. In CFX, the model source term in the k-equation forces the eddy viscosity to be equal to the LES subgrid-scale viscosity inside the user-specified zone. The synthetic turbulence at the RANS-LES boundary is based on a Harmonic Flow Generator acting through a special source term in the momentum equation.

6 Two phase flow

Two-phase flow (water, air) in Pelton turbines is turbulent and unsteady (Fig. 6). While useful results for Francis and Kaplan turbines can be obtained by steady state analysis this is usually not possible for Pelton turbines. Free surface flow has to be modelled by a multiphase model [5]. Numerical analysis of flow in a Pelton turbine is therefore much more complex and time consuming. Free surface flows refer to a multiphase situation where the fluids are separated by a distinct resolvable interface. They can be modelled with using a homogeneous or inhomogeneous model. The inhomogeneous model should be used when two fluids are being mixed and later separated.

The homogeneous model assumes that the transported quantities (with the exception of volume fraction) for the process are the same over all phases.

$$U_\alpha = U, \qquad p_\alpha = p, \qquad 1 \le \alpha \le \mathrm{Np}$$

It is therefore sufficient to solve bulk transport equations for shared fields instead of solving individual transport equations.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = \sum_{\beta=1}^{N_p} \Gamma_{\alpha\beta}$$

$$\frac{\partial}{\partial t}(\rho U) + \nabla \cdot \left(\rho U \otimes U - \mu(\nabla \mathrm{U} + (\nabla U)^T)\right) = S_M - \nabla p$$

$\Gamma_{\alpha\beta}$ in equation is the mass flow rate per unit of volume from phase β to phase α.

Density and viscosity are calculated from density and viscosity of all phases in the fluid:

$$\rho = \sum_{\alpha=1}^{N_p} r_\alpha \rho_\alpha \quad \mu = \sum_{\alpha=1}^{N_p} r_\alpha \mu_\alpha$$

A detailed description of multiphase models and modelling of free surface flows can be found in many papers. The near-wall modelling is based on the wall-function approach as an extension of the method of Launder and Spalding. In the log-law region, the near wall tangential velocity is related to the wall-stress by means of a logarithmic function. The logarithmic relation for the near wall velocity is given by:

$$u^+ = \frac{U_t}{u_\tau} = \frac{1}{k}\, ln(y^+) + C$$

The definition of the y^+ variable is given by the standard definition of y^+ generally used in CFD modelling,

$$y^+ = \frac{\rho\, \Delta y\, u_\tau}{\mu}$$

Where Δy is the distance between the first and second grid points off the wall.

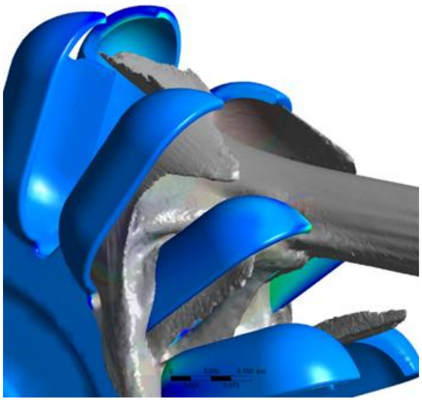

Fig. 6 Free surface flow in Pelton runner

For Pelton turbines, the homogeneous model is usually used [6]. Slightly better results can be obtained with the inhomogeneous model but with a higher cost. The homogeneous model assumes that the transported quantities (with the exception of volume fraction) for the process are the same over all phases.

Pressure distribution on the Pelton runner and velocity vectors are presented in Fig. 7.

Numerical analysis can help us to optimize the Pelton Turbine push-out jet deflector shape (Fig. 8). With such optimization it is possible to minimize the forces, stresses, deformations and torque of jet deflector servomotor.

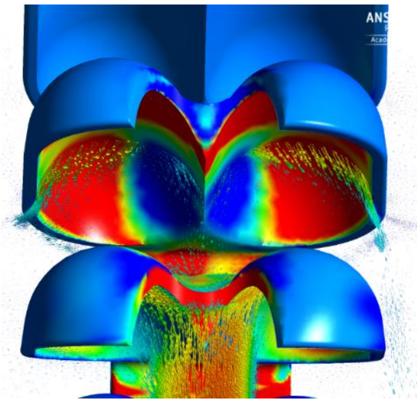

Fig. 7 Pressure distribution and velocity vectors

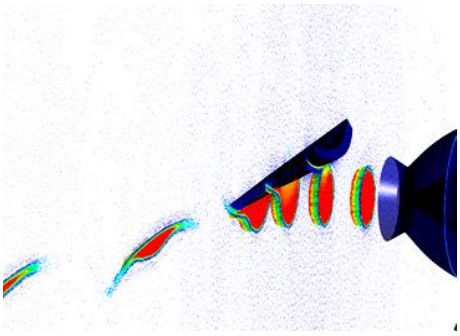

Fig. 8 Velocity distributions in the jet of Pelton turbine deflector

7 Pump intakes and inlet recirculation

A pump intake has usually very important impact on a pump operation (Fig. 9) due to the production of strong unsteady vortices [7] which may cause air intake problems. Several physical methods and turbulence models, as well as time step magnitude and wall boundary layer influences can be used in such applications: Euler, laminar, Shear Stress Transport turbulence model (SST), SST curvature correction (SST-CC) turbulence model, Scale Adaptive Simulation with curvature correction (SAS-CC) turbulence model, and Large Eddy Simulation (LES) turbulence model.

They all have particular demands of computational time. The most appropriate turbulence model for such case, in terms of accuracy and CPU time, would be the SAS-CC model.

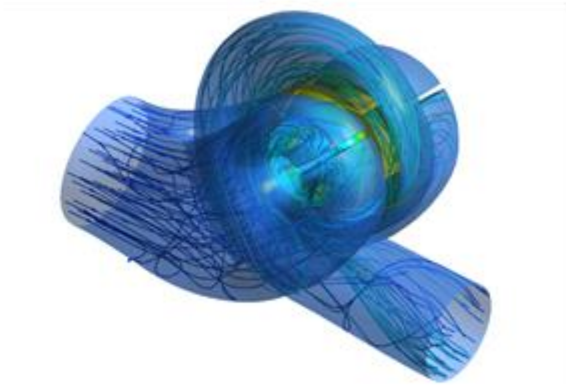

Fig. 9 Flow distribution in inlet chamber and spiral casing

The development of heavy-duty process pumps, usually based on various design criteria, depends on the pump's application. The most important criteria are Q-H, efficiency and NPSH characteristics. Cavitation due to inlet recirculation is not often one of the design criteria [8], although many problems in pump operation appear because of inlet recirculation when the operation range is from 0.5-0.8 Q_{opt}.

Flow recirculation at the inlet of centrifugal pumps (Fig. 10) may cause different (harmless) effects, such as noise, vibration, erosion damage and large forces on the impeller.

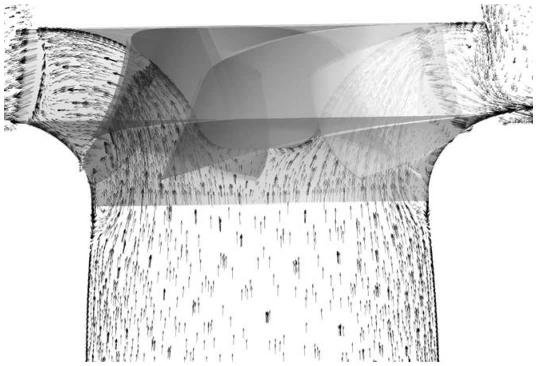

Fig. 10 Inlet recirculation

It can also cause cavitation due to recirculation (Fig. 11). The chance of damage is heavily dependent on the suction energy level, specific speed of the pump, the NPSH margin in the pump and the nature of the flow provided to the suction piping. According to experience, low suction energy pumps are not susceptible to damage from suction recirculation. Cavitation due to recirculation reduces the level of pump reliability and the danger of impeller damage is substantial, although $NPSH_a$ is much higher than $NPSH_{req}$ of the pump.

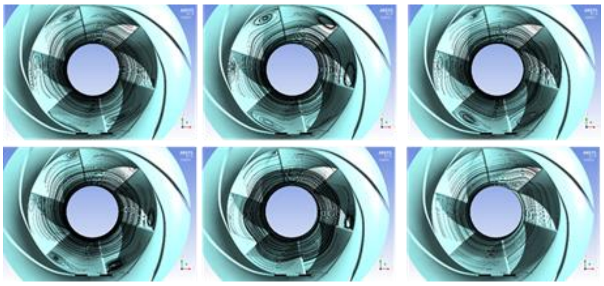

Fig. 11 Streamlines for different flow rates

There are also others numerical analysis problems in different types of pumps. One of the important issue is cavitation [9]. Another serious problem is an influence of wall roughness on the energetic and cavitation characteristics [10]. The analysis of multi-stage pumps is also numerically very demanding especially for nuclear power plants applications (Fig.12).

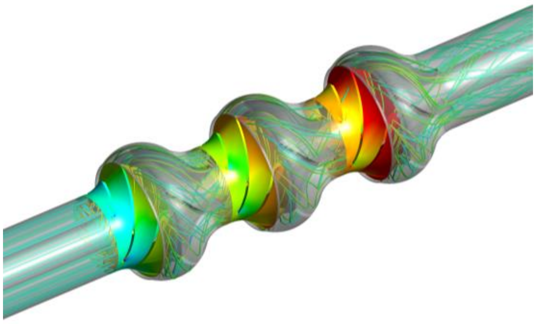

Fig. 12 Multi-stage pump

Highly unsteady flow with huge velocity gradients is in special ejector pumps (Fig. 13), which are used in Nuclear Power Plants.

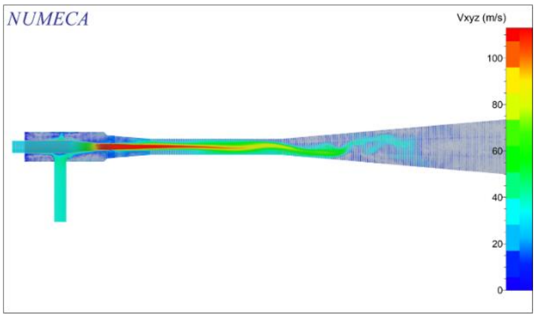

Fig. 13 Unsteady flow in ejector pump

8 CPU Reduction

In turbomachinery, unsteady phenomena is generated by rotor-stator interaction. Usually such unsteady analysis takes a lot of time. To reduce CPU time, so-called Non-Linear Harmonic (NLH) method has been presented by Numeca International. This approach, introduced by He and Ning (1998), can be considered as a hybrid method that conjugates the advantages of classical steady state and full unsteady calculations: it provides an approximate unsteady solution at affordable calculation costs [11].

The main idea is that the flow perturbations that make the flow unsteady are written about a time-averaged value of the flow and are Fourier decomposed in time. Casting the unsteady Navier- Stokes system into the frequency domain, transport equations are obtained for each time frequency. Both steady and harmonic equations are coupled via deterministic stresses that are derived from the time averaging of products of periodic fluctuations. For turbo machinery applications, the frequencies that are solved are the blade passing frequencies of adjacent rows and their sub-harmonics. More details on the method and the way all the equations are solved can be found in Vilmin et al. [12]. Because of the transposition to the frequency domain, only one blade channel is required like for a steady flow simulation. The presented method also features an improved treatment that enhances the flow continuity across the R/S interface by a reconstruction of the harmonics and the time-averaged flow on both sides of the interface. Since its introduction into NUMECA Fine/Turbo^{TM}, the NLH method has been applied and validated on a wide range of academic and industrial application.

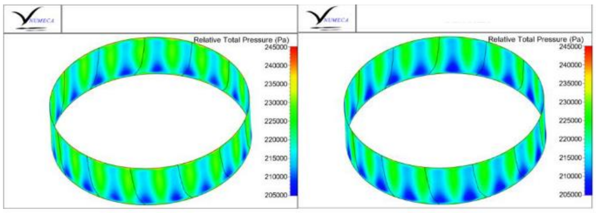

Fig. 14 Comparison of pressure distribution – unsteady analysis (left); NLH method (right)

Different rotor-stator treatments were analysed for the case of Francis turbine (Fig. 14) in order to find out the accuracy of above-mentioned method for engineering simulations. A mesh consisted of a Guide Vane, Runner and a Draft tube. In the case of using NLH method, the acceleration of computation is around order of magnitude. Using NLH method, the results are very similar in comparison to those obtained with standard method [13].

9 Optimization of hydraulic machines

If the shape of individual parts of hydraulic machines can be presented by a certain number of an arbitrary array of parameters and provide all details for the generation of automatic computational grid, automatic optimization algorithm can be used for new hydraulic design.

Genetic algorithms are search algorithms based on the mechanics of natural selection and natural genetics. For a natural process the survival of the fittest individuals is the main criteria, for engineering systems the survival criteria can be defined in a very arbitrary manner. For the case of hydraulic machines, the objectives are usually obtained by CFD or stress analysis. The criteria have to determine the success of the individuals in each generation. The main goal in the development of the optimization procedure is the demand for a robust method, especially in searching for the global extreme.

Multi-objective genetic algorithm proved to be very suitable for the optimization of water turbines [14] and pumps [15]. The only major issue is very long computation time, because time-consuming CFD calculations are used to obtain the optimization objectives.

9 Conclusion

In the development of hydraulic machines, we often have a situation where we need to use very powerful computers. This is a result of using various mathematical models or different physical theories.

There are many problems in the industry where the turbulent flows have to be solved, but no single turbulence model can solve all of them with high accuracy. Sometimes it is possible to categorize flows into different types. The application engineers usually have a difficult task to select appropriate model for a given problem. Some flows are very difficult to categorize.

Unfortunately, there is no unique CFD model to solve all industrial problems. Each individual model has its own advantages or disadvantages. It is important that the users of a CFD codes understand the theoretical background of the turbulence models in order to be able to select the appropriate one and to use it efficiently.

In many cases the comparison of different turbulence models shows that the most accurate and economic is Scalable Adaptive Simulation (SAS) Shear Stress Transport (SST) method - SAS-SST.

If the geometry and the application allow the definition of well-defined interfaces for flows with strong local instabilities, ZLES model is a better solution. Synthetic turbulence should be introduced at these interfaces in order to preserve the balance between the RANS and LES turbulence content.

For individual parts of the turbines and pumps, the LES turbulence model can be used, but in most cases, the companies have a need for a complete simulation, which is not entirely feasible now. However, LES can play important role in particular detailed analysis of individual elements of hydraulic machines.

Table 1 Computing power for a single turbomachinery blade

Method	Cells	Time	Inner loops per time step	Ratio to RANS
RANS	$\sim10^6$	$\sim10^2$	1	1
LES	$\sim10^8$-10^9	$\sim10^4$-10^5	10	10^5-10^7

Nevertheless, LES turbulence model needs very long computational time or a huge computation capacity (Table 1).

If CFD analyses will replace model tests in the near future, accuracy and computational time of CFD analyses should fulfil demands of development engineers. To obtain accurate results for steady state and unsteady flows, appropriate turbulence models have to be used. It is important to note that in most cases, with the prolongation of computational times, we do not gain on the quantity of results, but on the quality.

Twenty years ago, High-Performance Computing (HPC) was primarily used by the larger enterprise organizations for engineering simulations. HPC now becoming more common in smaller and mid-sized companies because powerful computers are more affordable and because of the possibility of different on-demand CFD cloud services for a temporary period of time.

10 References

[1] F. R. Menter, "Best Practice : Scale - Resolving Simulations in ANSYS CFD," ANSYS Germany GmbH, November, 2015.

[2] A. Lipej, D. Jošt, P. Mežnar et al., "Numerical prediction of pressure pulsation amplitude for different operating regimes of Francis turbine draft tubes," International Journal of Fluid Machinery and Systems, vol. 2, no. 4, pp. 375–382, 2009.

[3] A. Lipej, D. Jošt, P. Mežnar et al., "Numerical analysis of rotor-stator interaction in a reversible pump-turbine – pump mode," 23rd IAHR Symposium, Yokohama, October, pp. 8, 2006.

[4] D. Jošt, A. Škerlavaj, and A. Lipej, "Numerical flow simulation and efficiency prediction for axial turbines by advanced turbulence models," IOP Conf. Series: Earth and Environmental Science 15 (2012) 062016.

[5] D. Jošt, A. Lipej, and P. Mežnar, "Numerical analysis of dynamic behaviour of flow in a Pelton turbine," in *IAHR Int. Meeting of WG on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems Barcelona, 28-30 June 2006*, 2006, no. June, pp. 28–30.

[6] D. Jošt, P. Mežnar, and A. Lipej, “Numerical prediction of Pelton turbine efficiency,” *IOP Conf. Ser. Earth Environ. Sci. 15 022009 26th IAHR Symp. Hydraul. Mach. Syst.*, 2010.

[7] D. Florjančič, *Trouble-shooting Handbook for Centrifugal Pumps*. Turboinštitut d.d., 2008.

[8] D. Mitruševski, “Cavitation due to the fecirculation - important criteria for design of heavy duty process pumps,” in *6th IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, September 9-11, 2015, Ljubljana, Slovenia*.

[9] A. Škerlavaj, M. Titzschkau, R. Pavlin et al., “Cavitation improvement of double suction centrifugal pump HPP Fuhren,” *IOP Conf. Ser. Earth Environ. Sci. 15 022009 IOP*, 2012.

[10] A. Lipej, S. Muhič, and D. Mitruševski, “Wall roughness influence on the efficiency characteristics of centrifugal pump,” *Stroj. Vestnik/Journal Mech. Eng.*, vol. 63, no. 9, pp. 529–536, 2017.

[11] “Numeca International FINE/Turbo Theory Guide v12.1,” 2017.

[12] Vilmin S., Lorrain E., Hirsch C. et al., “Unsteady flow modeling across the rotor/stator interface using the nonlinear harmonic method,” *ASME Turbo Expo*, 2006, Barcelona, Spain, GT2006-90210.

[13] A. Lipej, D. Čelič, B. Tartinville et al., “Reduction of CPU time for CFD analysis of hydraulic machinery development process,” *IOP Conf. Ser. Earth Environ. Sci.*, vol. 15, p. 8, 2012.

[14] A. Lipej, “The Optimization of Axial Turbines,” Journal of Mechanical Engineering, vol. 49, pp. 185–195, 2003.

[15] A. Škerlavaj, M. Morgut, D. Jošt et al., “Optimization of a single-stage double-suction centrifugal pump,” *IOP Conf. Ser. J. Phys. Conf. Ser. 796 012007*.

