Mechanical Engineering – Scientific Journal, Vol. **39,** No. 1–2, pp. 25–32 (2021)
Number of article: 639
Received: Novemberr 8, 2021
Accepted: December 10, 2020
In print: ISSN 1857–5293
On line: ISSN 1857–9191
UDC: 621.224.7.018:004.942

Original scientific paper

ANALYSIS OF FREE SURFACE FLOW IN PELTON TURBINE USING LATTICE BOLTZMANN METHOD

Andrej Lipej

University of Novo mesto, Faculty of Mechanical Engineering,
Na Loko 2, 8000 Novo mesto
E-mail: andrej.lipej@fs-unm.si

Abstract: Computational fluid dynamics (CFD) has become an indispensable tool in the development and optimization of various machines and devices in the last decade. In most cases, researchers deal with single-phase flows and internal flows where all externally wetted walls are defined. In some cases, it is also necessary to analyse multiphase flow and free-surface flow. Using classical numerical methods such as the finite element method (FEM) or the finite volume method (FVM), the results are highly dependent on the size and quality of the computational grids. Particular attention should be paid to the conditions at the boundary between the individual phases of the fluid. In most cases, it is necessary to consider non-stationary conditions and the necessity of using non-stationary numerical methods. All the above properties of multiphase flows and their consequences in performing numerical analyses lead us to very long computational times and consequently the use of very powerful computers. Industry often needs results in a relatively fast time therefore many simplifications need to be considered. Recently a new approach for numerical simulations of multiphase and free-surface flows based on the Lattice Boltzmann method (LBM) started to be used. The method is very useful especially for development of small-scale hydraulic turbines, where the expense for development process is usually very limited. In this paper, some examples of the use of LBM in the calculation of free-surface flows and compare the results using classical CFD methods have been presented.

Key words: free surface flow, computational fluid dynamics, Pelton turbine, Lattice Boltzmann Method

АНАЛИЗА НА СТРУЕЊЕ НА СЛОБОДЕН МЛАЗ КАЈ ПЕЛТОН ТУРБИНА СО ПРИМЕНА НА МЕТОДОТ ЛАТИС БОЛЦМАН

Апстракт: Во последната деценија, пресметковната динамика на флуиди (CFD) прерасна во незаменлива алатка за развој и оптимизација на разни машини и уреди. Најчесто, истражувачите се занимаваат со проучување на еднофазни струења и струења во затворени канали чии надворешни ѕидови (граници) во допир со флуидот се дефинирани. Во некои случаи, потребно е да се анализира повеќефазно струење и струење во отворен канал. Со користење на класичните нумерички методи како што е методот на конечни елементи (МКЕ) или методот на конечни волумени (МКВ), резултатите во голема мера зависат од големината и квалитетот на нумеричката мрежа. Посебно внимание треба да се обрне на границата меѓу одделните фази на флуидот. Во повеќето случаи, неопходно е да се разгледуваат нестационарни услови и да се применат нестационарни нумерички методи. Карактеристиките на повеќефазните струења и последиците од нив при спроведувањето на нумерички анализи води кон подолг процес на пресметка и потреба од примена на помоќни компјутери. Со оглед на тоа што за индустријата се потребни резултати добиени релативно брзо, се наметнува потребата од поедноставувања. Во последно време почнува да се применува нов пристап за нумеричко симулирање на повеќефазни струења и струења во отворени канали, базиран на методот на Lattice Boltzmann (LBM). Методот е особено корисен за развој на хидраулични турбини во мали хидроцентрали, каде обично трошоците за развојниот процес се многу ограничени. Во овој труд се презентирани некои примери со примена на LBM за пресметка на струења во отворени канали и споредба на резултатите користејќи класични CFD методи.

Клучни зборови: струење во отворени канали; пресметковна динамика на флуиди; Пелтон турбина; метод Lattice Boltzmann

1. INTRODUCTION

In recent decades, CFD has been widely used in the development phase of various rotating machines. In the case of compressor and pump, internal flow analysis is used in most applications where the geometry is well defined and the boundary conditions at the inlet and outlet and on the walls are known.

In some cases, it is also necessary to analyse multiphase flows and free surface flows. Free-surface flows are a particular problem, as the geometry of the space occupied by the liquid is not initially defined and changes during the analysis itself. In methods where a quality computational grids are needed to solve the system of differential equations, must be paid special attention to the conditions at the liquid surface or at the boundary between different liquids (e.g. air, water).

Using numerical schemes such as finite difference method (FDM), finite volume method (FVM), finite element method (FEM), PDE is converted into a system of algebraic equations. These equations are usually solved iteratively until satisfactory results are obtained. In such cases, different methods of computational grids adaption are used, where prescribed changes in flow conditions occur.

Some software packages for CFD have a built-in so-called method for automatic computational grid adaption, where special parameters can be defined that allow optimal implementation of finer grids in the right locations. The consequences of using such methods are a large increase in the number of elements and, consequently, an extension of the computational times. Such types of flows are in most cases also non-stationary and it is necessary to analyse a large number of time steps which further result in longer computational times.

As an example of this type of numerical simulations, numerical analysis in Pelton turbines will be presented. Recently, many scientific papers have been published on the development of Pelton turbines using the solution of a system of classical Navier-Stokes nonlinear partial differential equations. A paper [1] presents the analysis of the flow conditions in a bucket of a Pelton turbine. Comparisons between numerical analyses on a simplified model and the results of model measurements still show a large difference between the absolute results, but if a comparison of relative results is made, the trends are quite good.

The paper [2] focuses more on the casing design of Pelton turbines. As well as the differences between numerical and experimental results. It is again emphasized that the qualitative results of the flow form are quite good, while the quantitative ones still differ. A very detailed analysis of the flow conditions in the Pelton turbine is presented in the article [3], where the numerical analyses also considered the cavitation that can be detected during the operation of the Pelton turbines. Compared to other articles, the differences between the numerical and experimental results were smaller, especially when cavitation was considered.

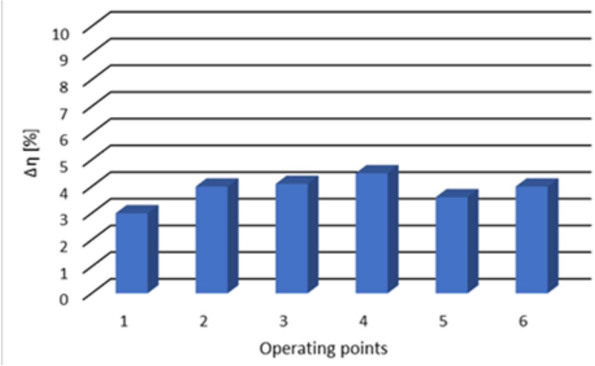

Fig. 1. Comparison of Pelton turbine efficiency between model test results and CFD (N-S) analysis

In numerical analyses of the flow conditions in the Pelton turbine using N-S equations, several comparisons of numerical and experimental results were performed [4]. Figure 1 presents a comparison of numerical simulations and model measurements where comparable computational grids were used, as in the presented article, and all the input parameters required for numerical analysis are the same. The results show that under different operating conditions where the flow rates were changed, the differences between the measured and numerically calculated efficiencies vary between 3% and 4.5%.

Certainly, these results can be further improved, but they are presented here for later credible comparison of the results obtained using the N-S equations and the LBM method

As an alternative, a new efficient method for the numerical analysis of free-surface flows has recently emerged. The Lattice Boltzmann Method is a relatively unknown and new method in computational fluid dynamics applications. It was derived from lattice gas vending machines and is still under development. The basic steps of LBM (collision, current, boundary conditions, macroscopic quantities) will be presented.

The Lattice Boltzmann (LBM) method with its simplified kinetic descriptions, emerged as an important tool for hydrodynamic simulation. In a heterogeneous computing environment, it is often an advantage because of its flexibility and better parallel scaling.

Another possible method is based on the simulation of small particles on a microscopic scale. It is molecular dynamics. The main equation is Hamilton's equation, where the location and velocity of each particle by molecular dynamics need to be identified. But the amount of data would simply be too large to process.

Such simulation of real problems is not feasible. It is known that in one cubic decimetre of air is approximately 10^{22} molecules. Such method can be avoided because it is not necessary to know the position of each particle. Only total effect of the resultant [5], such as wind, is significant.

LBM bridges the gap between the macro and micro scale. The method considers the behaviour of a group of particles as a single group particle [5]. The basis of LBM is a microscopic model complemented by mesoscopic kinetic equations. The basic idea of LBM is to build simplified kinetic models that incorporate the basic physics of microscopic processes so that the macroscopic average properties take into account the necessary macroscopic equations [6].

The reason why simplified kinetic models can be used is that the macroscopic dynamics of a fluid is the result of the common behaviour of many microscopic particles in the system. The property of particle aggregation is represented by the distribution function. LBM is becoming increasingly popular in the field of CFD because LBM is addressed locally. It has a high degree of parallelization, making it ideal for parallel computing on multiple processor supercomputers.

LBM has its roots in lattice gas automata (LGA), a kinetic model with a discrete lattice and a discrete time. Starting with LGA on a hexagonal lattice, Frish, Hasslacher, and Pomeau first obtained the correct Navier-Stokes equations [7]. This model is known as the HLC model. As already stated, lattice gas automata are constructed as simple particle dynamics in the discretization of space and time. As a result, all particle velocities are also separated where particles can move around, but only inside network nodes.

The basic difference between the system of Navier-Stokes equations and LBM is shown in the following two equations:

Navier Stokes equation

$$\rho(\partial u/\partial t + (u.\nabla)u) = -\nabla p + \mu\nabla^2 u \qquad (1)$$

Lattice Boltzmann equation

$$\partial f/\partial t + e.\nabla f = -\ 1/\tau\ (\ f - f^{EQ}\ ) \qquad (2)$$

Comparison between Navier-Stokes (N-S) equation and Lattice-Boltzmann (L-B) equation shows that N-S equation is second order and L-B equation is a first-order partial differential equation, where f is distribution function, f^{EQ} is equilibrium distribution function, τ is the rate of relaxation towards local equilibrium and e is velocity [8].

In our case the calculation can be done with two different lattice models D3Q19 and D3Q27, which is presented in the Figure 2.

Given the fact that not much research on this topic is presented in the literature, it is certainly useful to determine the effects of certain parameters on the quality of the results.

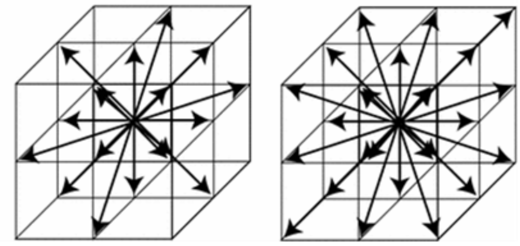

Fig. 2. Lattice models

LBM accuracy testing was performed by calculating the jet force hitting the vertical wall and compared with the theoretical calculation of the jet force. Figure 3 shows a good match of the results.

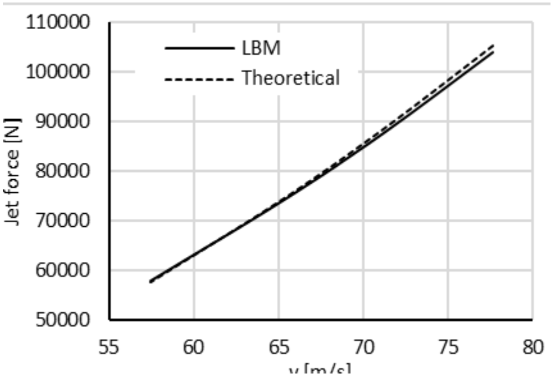

Fig. 3 Comparison of jet force calculation

2. COMPARISON OF N-S EQUATAION AND LATTICE BOLTZMANN METHOD

When Navier-Stokes equations are used for flow analysis, special attention should be paid to the quality of computational grids. Due to the use of wall functions or using the Low Reynolds method, appropriately sized elements near the walls and gradual enlargement of elements that are further away from the wall are required. This requirement is controlled using a non-dimensional parameter y^+, which is known to be between about 1 and 50, depending on the used numerical method.

Since at least two different phases need to be analyzed for free-surface flows, another important consideration in computational grids is the boundaries between the different phases. Water and air are mainly used. A dense computational grid is also required at the boundary between the phases to accurately determine the shape of the free surface. Since the location of the free surface of the liquid is generally unknown before the calculation, an automatic method of adjusting the computational grids may be used in some software packages.

Furthermore, it is necessary to have a quality mesh in places where the liquid detaches from the wall if the jet comes out of the nozzle, because otherwise it can be obtained a very distorted shape of the jet, which affects the quality of further results. Sometimes there are also problems with defining determination of the elements along the walls in connection with the automatic adaptation of the grids.

In most cases, the size of the computational grid which is done in a few steps, is larger by a factor two or more at the end, after all the automatic adaptations have been done. For oversized finite computational grids, certain restrictions on increasing the number of elements in individual densification steps can be applied.

For free-surface flow analysis using LBM, it is not necessary to divide the computational domain into elements in order to calculate the flow conditions, but it is only necessary to determine the size of the individual particle under consideration. A very small size means that the number of particles that interact with each other is higher in a given computational domain and accordingly, the number of computational operations increases and finally the computational times may be longer.

Figure 4 shows the distribution of the water velocity along the centre of the jet with respect to different elementary particle sizes in the calculation using LBM. Due to the interaction with the wall the size of the particles affects the thickness of the jet and consequently the flow conditions coming out of the nozzle. The larger are particles, the thinner is jet coming out of the nozzle. As a result, the average speed shown in the figure increases, representing approximately 4% of the speed difference in this case. Since the particle size also affects the length of the computational time, this must be considered when choosing the particle size. In the case shown in Figure 4, the computational times for the smallest particles are more than three times longer than for the largest ones.

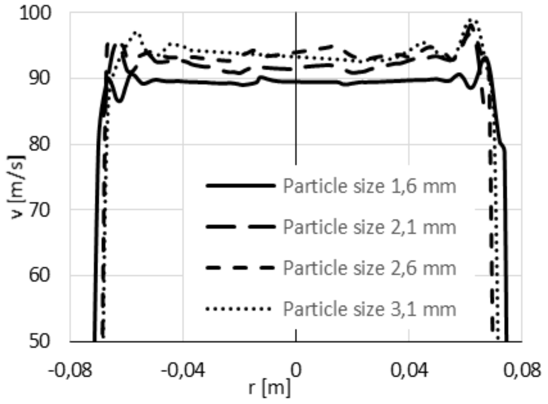

Fig. 4 Velocity distribution as a function of particle size

It is necessary to determine the appropriate size so that it does not affect the shape of the water jet too much and the calculation times are still acceptable.

2. NUMERICAL OPTIMIZATION OF JET DEFLECTOR

Pelton turbines are designed for operation where a large height difference in water drop and relatively small flows are exploited. For various reasons, it is often necessary to shut down the turbine quickly.

In such cases, a so-called water hammer can occur in the pipeline, the consequences of which can be catastrophic. To prevent these unpleasant phenomena, Pelton turbines use a jet deflector (Figure 5), which can redirect the jet of water past the runner in the case of a rapid shutdown of the turbine.

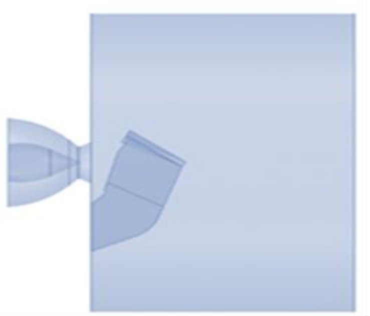

Fig. 5. Basic geometry of nozzle, jet deflector and computational domain

The conclusions of the research present a comparison of the results obtained by solving N-S equations (Equation 1) using element-based finite volume method, which involves discretising the spatial domain using a mesh [9] and Lattice Boltzmann method [10] in the development process of the optimal shape of the jet deflector, so that the forces and torques when stopping the turbine, despite the high-water velocities, are minimal.

The development of the jet deflector [11] was started with the initial geometry shown in Figure 6. This figure presents the shape of the jet for both calculations. The figure on the top shows the result of using N-S equations and the LBM on the bottom. In the continuation of the research, different forms of deflectors were analysed. Only some examples of numerical analyses will be presented in the article.

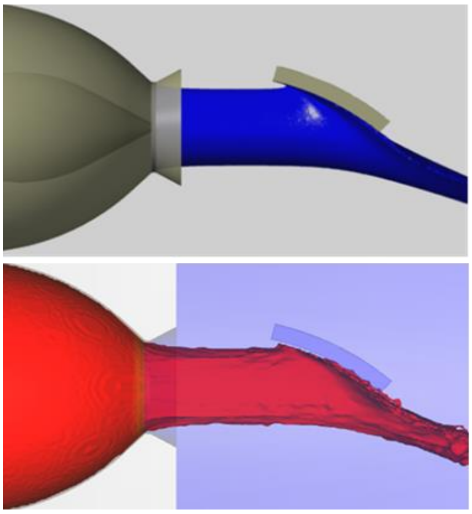

Fig. 6. Water jet shape for both methods – Navier Stokes (top), Lattice Boltzmann (bottom)

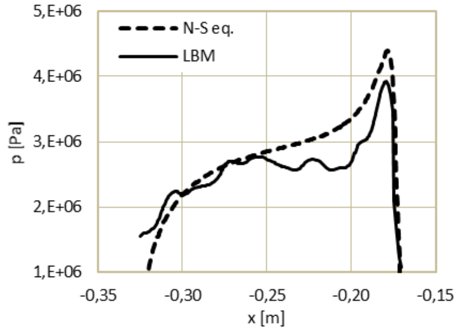

Fig. 7. Pressure distribution for initial geometry of deflector

It was necessary to pay attention to the quality of the jet calculation during development, because the pressure distribution over the surface of the deflector was the main result used to determine the forces and torques.

Figure 8 shows the pressure distribution over the deflector surface for both methods of numerical analysis. As can already be seen in Figure 6, using the N-S equations, the water jet is fairly smooth compared to the result obtained with LBM.

The same can be observed in Figure 8, but the comparison generally gives a good match of the results, presented in the form of a graph in Figure 7.

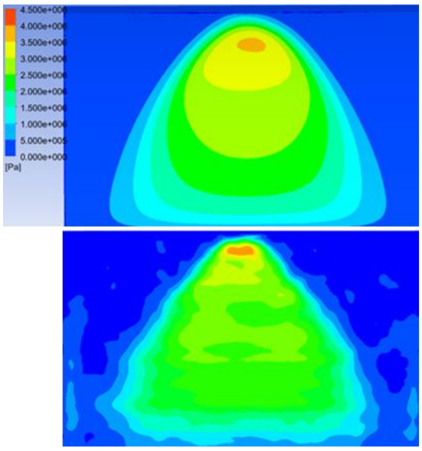

Fig.. 8. Pressure distribution for initial geometry of deflector - Navier Stokes (top), Lattice Boltzmann (bottom)

Figure 9 shows a comparison of the pressure distribution also for the case of a flat deflector where the results are similar to those of the basic geometry. The first conclusion of the comparison between two different methods for free surface flow analysis is the fact that with a simpler method, where pre-processing is simpler and complete numerical analysis is much faster, comparable and useful results can be obtained.

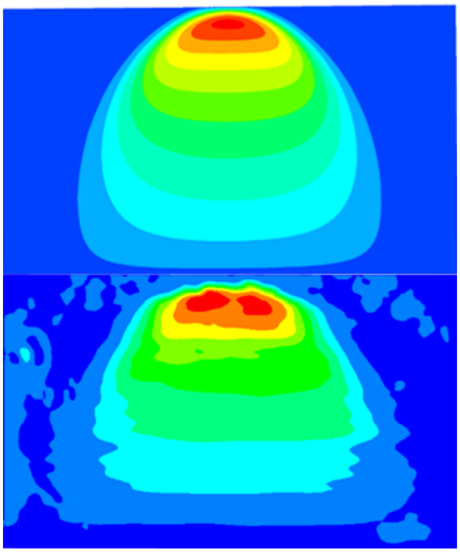

Fig. 9. Pressure distribution for flat deflector - Navier Stokes (top), Lattice Boltzmann (bottom)

The final optimal shape of the deflector is a combination of straight and curved, so that part of the jet deflects not only downwards but also on both sides left and right. This makes it possible to reduce a certain component of the force responsible for the magnitude of the torque.

A comparison of pressure distribution in the middle cross section of the deflector is presented in Figure 10. In all cases the calculated pressure with LBM is slightly lower than pressure obtained using N-S equations. The main reason is the inaccurate calculation of the flow conditions near the walls which is at the moment one of the disadvantages of LBM.

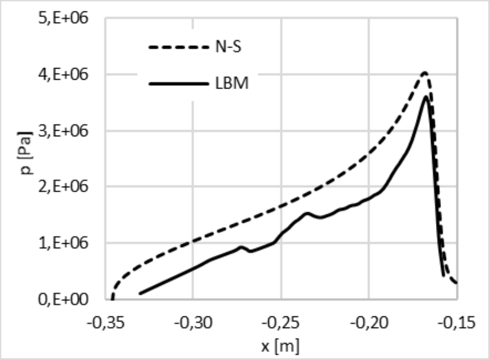

Fig. 10. Pressure distribution for final optimized geometry

These results are partially improved by choosing different particle size. As presented in the introductory part of the article, no significant improvements are obtained.

The actual pressure distribution for the final geometry over the deflector surface is shown in Figure 11.

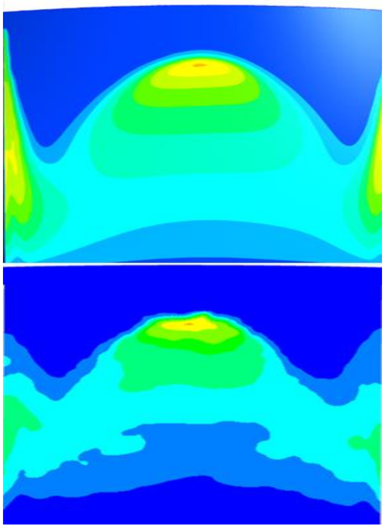

Fig. 11. Pressure distribution for final optimized geometry - Navier Stokes (top), Lattice Boltzmann (bottom)

Table 1

Comparison of improvements for force and torque between Navier – Stokes and LBM (%)

Numerical model	ΔF_R	ΔM_h
Navier - Stokes	10.9	28.2
LBM	11.6	30.4

The deflector optimization performed with both presented methods shows significant improvement in terms of force and torque reduction, as shown in Table 1, where the final torque is reduced around 30% compared to the initial one.

In the case of using the LBM method, the improvements are slightly higher in percentage, which is due to the difference in the results of the pressure distribution over the deflector surface. However, the differences between the two methods are not large, because in LBM calculations, smaller pressure values are obtained in all cases.

3. NUMERICAL ANALYSIS OF THE FLOW IN PELTON RUNNER

The flow conditions in the Pelton turbine are quite complex. The flow is turbulent and non-stationary, a large number of time steps need to be considered in the analysis. Since the flow is also two-phase with a free surface, there is also an issue with the preparation of computational grids. Especially on a non-predefined boundary between the phases - water and air. Due to all the above phenomena, numerical analysis of the flow in the Pelton turbine runner is a very time-consuming simulation.

Sometimes very accurate results in the initial stage of development are not so significant, so a fast calculation method that allows qualitatively good results is very welcome. The LBM method, presented in the first part of the article in the development of a jet deflector in a Pelton turbine, proved to be a good substitute for solving the system of N-S equations for free-surface flows.

The paper also presents the results of numerical analysis of the flow in a Pelton turbine runner.

As a test case, a given geometry of the PT runner and the results of calculations using N-S equations has been used, which were also previously verified by model measurements. The calculations were performed on the case of one blade which is rotated by a few degrees according to the jet, and for the part of the runner with four blades. All calculations were performed at the same boundary conditions in terms of inlet speed.

A comparison of the reflected water jet shows that the two methods yield qualitatively very similar results (Figure 12). The computational times in the case of N-S method are approximately 5 times longer.

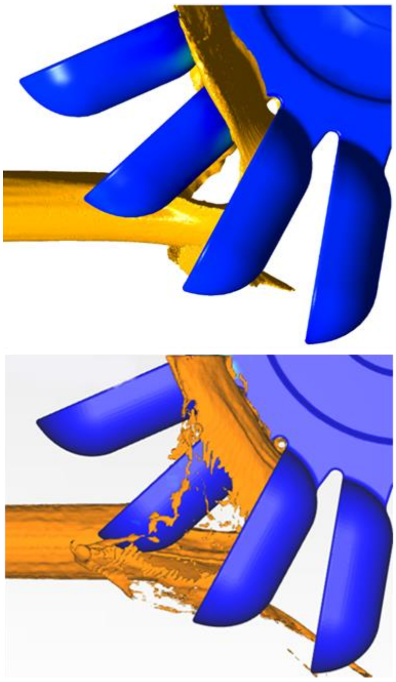

Fig. 12. Water jet shape for the flow in Pelton turbine runner - Navier Stokes (top), Lattice Boltzmann (bottom)

There is also a difference in the complexity of the preparation of the calculation (pre-processing), since with LBM it is not necessary to generate complex computational grids in the space around the runner blades. With the LBM method, it is only necessary to define the geometry of the runner and the area where the water (fluid) is located.

The biggest difference is due to the calculation of the flow in the nozzle, because the LBM method does not allow such an accurate calculation of the flow conditions along the wall.

A comparison was also made to distribute the pressure over the surface of the bucket and Figure 13 shows that the results match quite well. If the calculation considers the flow in the whole nozzle, then in the free jet the LBM method gives a slightly lower jet velocity than in solving the N-S equations. However, it is possible to consider the calculation domain without the whole nozzle but for a quality result the right particle size must also be taken into account.

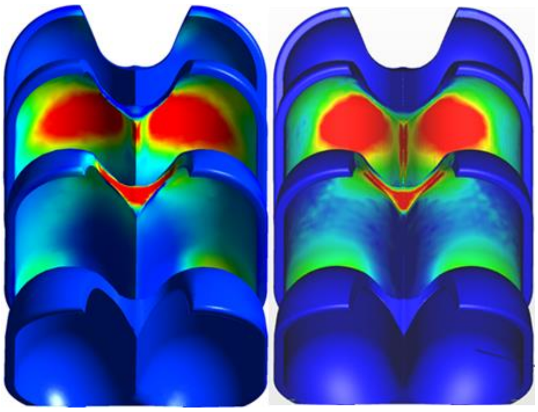

Fig. 13 Pressure distribution inside the bucket of Pelton runner - Navier Stokes (left), Lattice Boltzmann (right)

In some cases, a slight touch of the water jet may also be obtained on the outside of the driver blades and Figure 14 shows the pressure distribution for both methods also on the outside of the runner.

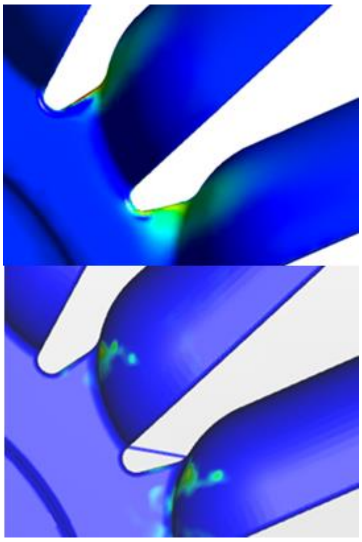

Fig. 14. Pressure distribution outside the bucket - Navier Stokes (top), Lattice Boltzmann (bottom)

4. CONCLUSIONS

In numerical simulations of flow conditions in hydraulic machines, problems arise when it is necessary to analyze multiple phase flows and free surface flows. In most cases, such numerical simulations require quality preparation of geometries, computational grids and all the necessary parameters for calculation. It takes quite extensive duration for the entire pre-processing and at the same time it is known that due to the flow conditions, the computational times are also very long.

In industry, it is usually necessary to obtain the results of various analyzes relatively quickly, despite the fact that the quality of the results may also be partially degraded due to the speed of numerical simulations.

The paper presents the Lattice Boltzmann Method, where the preparation time is shorter as it is not necessary to generate computational grids, and at the same time the computational times are shorter compared to the methods where N-S equations are solved.

A comparative analysis of both methods (N-S and LBM) on the examples of calculating the Pelton turbine deflector and calculating the flow conditions in the Pelton turbine runner showed that if the qualitative results are considered, it can be argued that they are quite good. In some cases, the quantitative results show a large difference, which can be greater than 15 % compared to the N-S equations. In most cases, the differences range from 5 % to 10 % which could be considered as still acceptable results for the rapid preliminary analyses that are often needed in industrial research.

In particular, it can be emphasized that model measurements cannot be afforded in development process for small hydro power plants, nor is it always possible to perform expensive and time-consuming numerical analyses.

Therefore, the presented LBM is even more suitable for analyses of Pelton turbines and all other applications where free surface flows must be analyzed.

It cannot be argued that LBM is equivalent to methods using N-S equations, but with further development, comparatively better results can be expected in the future.

REFERENCES

[1] Kumashiro, T., Fukuhara, H., Tani, K.: Unsteady CFD simulation for bucket design optimization of Pelton turbine runner. IOP Conference Series: *Earth and Environmental Science*. **49** (2) (2016). ISSN 17551315. Doi:10.1088/1755-1315/49/2/022003.

[2] Petley, S., S., Aggidis, G. A.: *Transient CFD and experimental analysis for improved Pelton turbine casing designs.* IOP Conference Series: Earth and Environmental Science. **240** (2) (2019). ISSN 17551315. Doi:10.1088/1755-1315/240/2/022005.

[3] Jošt, D., Škerlavaj, A., Pirnat, V., Morgut M., Nobile, E. (2019): *Numerical prediction of efficiency and cavitation for a Pelton turbine.* IOP Conf. Series: Earth and Environmental Science 240 062033, doi:10.1088/1755-1315/240/6/062033.

[4] Jošt, D., Mežnar, P., Lipej, A. (2010): *Numerical prediction of Pelton turbine efficiency.* IOP Conf. Series: Earth and Environmental Science 12 012080, doi:10.1088/1755-1315/12/1/012080.

[5] Mohamad, S. A., (2011): *Lattice Boltzmann method,* Springer-Verlag London.

[6] Chen, S., Doolen, G. (1998): Lattice Boltzmann method for fluid flows, *Annu. Rev. Fluid Mech.* 30, 329-364.

[7] Frish, U., Hasslacher, B., Pomeau, Y. (1986): Lattice gas automata for the Navier-Stokes equation, *Phys. Rev. Lett.* 56, 1505-1508.

[8] Sokup, M. C., Thorne, D. T. (2006): *Lattice Boltzmann Modeling*, Springer-Verlag Berlin Heidelberg.

[9] ANSYS CFX – *Solver Theory Guide.*

[10] N. OMNIS™ tools, *LB solver*, OMNIS 4.2 © NUMECA Int., - version: EN202010131605

[11] Popovski, B., Lipej, A., Markov, Z., Popovski, P. (2019): *Optimisation of Pelton turbine jet deflector using CFD analysis*. IOP Conf. Ser.: Earth Environ. Sci. 240 022031, doi: 10.1088/1755-1315/240/2/022031.