Approach to 3D Unsteady CFD Analysis of a Single-Blade Pump

Kurilla Matej[1,*], *Knížat* Branislav[1] , and *Olšiak* Róbert[1]

[1]Slovak University of Technology in Bratislava, Nám. slobody 17, 812 31 Bratislava, Slovakia

Abstract. Single-blade centrifugal pumps are hydraulic machines used in many industrial areas. A unique screw shaped blade enables liquids containing solids and fibrous matters to be pumped. Owing to good pump hydraulic properties on the one hand and unfavourable impeller mechanical properties on the other have single-blade pumps become recently more interesting for researchers regarding the CFD simulations. In this case a conventional CFD approach for multi-blade pumps cannot be applied due to the lack of symmetry of the single-blade impeller. Possible approaches to the CFD simulation of a single-blade impeller in the Ansys Fluent and the Ansys CFX are compared in this paper. A comparison of two CFD meshing tools showed substantial element number decrease of the whole CFD model. This paper presents also the laboratory experiment results of the investigated single-blade pump. The paper describes a new approach to the single-blade CFD simulation through Ansys Fluent which is faster and more user-friendly then the conventional approach.

1 Introduction

Many academic papers deal with the single-blade pump CFD simulation topic and offer new techniques how to carry out the numerical simulation in order to achieve qualitative and quantitative agreement with laboratory measurements but there are still many questions to be answered in this research field.

In 1994 Menter [1] presented a new SST turbulence model which combines advantages of the k-ω turbulence model in the near-wall region and advantages of the k-ε turbulence model in the free flow region. This model has become one of the most preferable turbulence model used in hydrodynamic pump CFD simulations.

Keays and Meskell [2] provided a numerical investigation of a single-blade impeller hydraulic behaviour. They found out a strong unsteady character of investigated parameters. The parameters convergence was achieved after 8 full revolutions of the impeller.

De Souza et al. [3] reported a qualitative and quantitative agreement of the CFD and experimental results of the single-blade pump. The full transient CFD calculation, with the time step chosen to reach an angular resolution of 6° per iteration, was carried out.

* Corresponding author: matej.kurilla@stuba.sk

Benra et al. [4] validated results of the 3D unsteady CFD simulation with the PIV optical method. A velocity field in the single-blade impeller was studied and compared to the PIV results. The k-ω SST turbulence model and time step according to 3° of the angular resolution was used.

A blade asymmetry results in a hydraulic parameter fluctuation, described by Schiffer et al. [5] as a clocking effect. The clocking effect may have its roots in the insufficient quantitative accuracy of the steady state calculation. So the transient mode has to be applied in order to get a satisfactory qualitative and quantitative agreement [5].

Kurilla et al. [6] showed different approaches of the single-blade pump steady state numerical simulation and the best agreement was attained with the BSL EARSM turbulence model and the Stage-Interface.

This paper addresses the CFD simulation of the single-blade pump where two different CFD solvers are compared. The Ansys Fluent with the Fluent meshing tool and the Ansys CFX with the Ansys meshing tool are objects of this research. The CFD results were plotted against the laboratory measurement of the single-blade pump prototype. Clean water was used during the simulations and laboratory measurements.

2 Data of the investigated pump

The investigated single-blade pump is a prototype designed to fulfil given hydraulic parameters. The efficiency peak should be reached at a flow rate of Q = 10 l/s and a head of H = 10 m. A requested ball passage diameter was $ØD$ = 50 mm and the rotational speed of n = 2900 min^{-1}. A suction diameter of D_{in} = 50 mm was the same as an outlet diameter of D_{out} = 50 mm. The single-blade impeller, with the diameter of D_2 = 138 mm, was manufactured on a 3D printer from the ABS plastic.

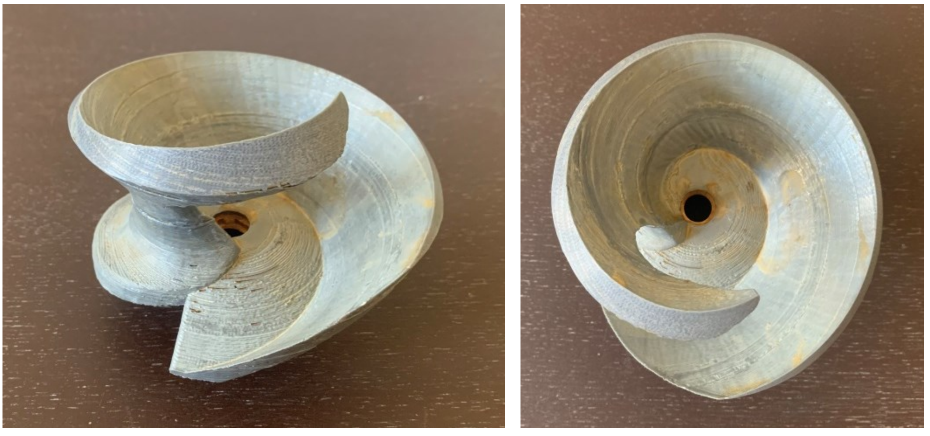

Fig. 1. Investigated single-blade impeller.

3 CFD model

The numerical simulation model consists of three domains. The model is divided into the stationary suction pipe, the volute casing and the rotational impeller. External 3D CAD software was used to prepare the tested model geometry. The Ansys 19R2 software, which includes the CFX and the Fluent module, was used to carry out the CFD simulation. The aim of this research is to gain the comparison between the conventional unsteady CFD approach, presented by Schiffer et al. [5], and a new Fluent-based approach. The Ansys Fluent should replace the CFX from the leading position in the Ansys family, what does the research field of the turbomachine simulations concern. The main advantage, presented by manufacturer, is the new Fluent meshing tool available since the Ansys version 19R2. This meshing too supports the generation of the Poly-Hexcore mesh with the substantial decrease of mesh elements compare to the Ansys meshing.

The Fluent-based approach as well as the CFX-based approach was applied on the investigated single-blade pump model.

3.1 The Fluent-based approach

Reducing the element number is an effective way to improve the calculation time. In order to achieve fast and accurate results the element number optimization of the simulation model has to maintain satisfactory mesh quality.

Each domain of the simulation model was meshed separately hence the conformal interface wasn´t ensured. All domains were meshed with the same mesh setup described below. The “Curvature and Proximity” function was applied to generate the surface mesh. Due to allow the solver the transient simulation to be execute, the Share Topology option wasn´t allowed. The first boundary layer height of 0.03 mm was set in order to capture the velocity profile in the near-wall region. Total number of boundary layers was 12 and the uniform volume mesh consisted of hexahedral, polyhedral and prismatic elements. The “Improve volume mesh” function was applied to remove mesh errors. A usage of the sizing function in the gap between the blade and the front tip of the pump might improve the mesh quality in this area so the loss flow rate was included in the numerical model.

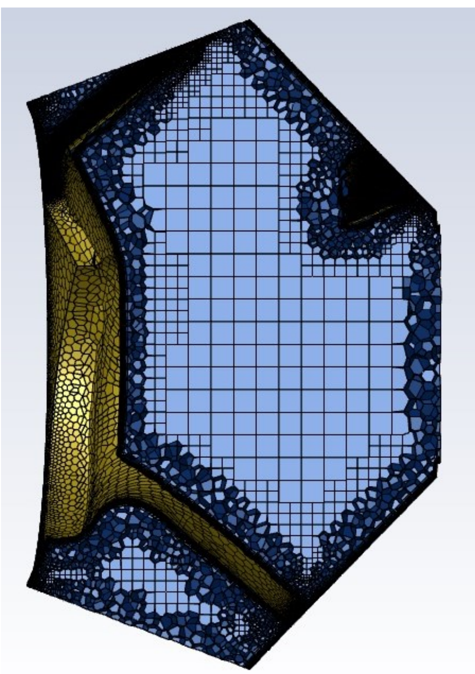

Fig. 2. The impeller meshed in the Fluent meshing.

The transition SAS SST turbulence model was used and as a boundary condition a constant static pressure of 1 atm was defined at the outlet. An adjustable flow rate arranged the operating point change of the pump. The connection between stationary and rotating domain was set as the Transient Rotor-Stator interface. The transient time step of 0.000172 s responded the impeller angular resolution of 3° and maximum 6 iterations per time step was chosen. A “Fixed” time stepping method was used. The example of the poly-hexcore mesh is in Figure 2.

3.2 The CFX-based approach

The Ansys meshing, included in the CFX module, is a meshing tool supporting the generation of tetrahedral and hexahedral meshes. The produced mesh can be either hexahedral or tetrahedral because there is no poly-hexcore function, which provides smooth transition in affected areas. As a result the CFD model mesh has to consist of only tetrahedral elements and the total element number is considerably higher than by the Fluent one. Despite higher element number the Ansys meshing allows all simulation domains to be meshed together. Researcher´s afford was to attain as compact prismatic layers in the near-wall region as was achieved with the Fluent meshing. The overall element size was set in the range from 6 mm to 12 mm and the “Proximity” function was applied. Required element density in the gap between the blade and the front tip was achieved with the sizing function of 0.25 mm. Prismatic layer settings were the same as in the Fluent-based approach. Comparison of element numbers of the CFD model domains can be seen in Table 1 and the impeller mesh built up with tetrahedral elements is depicted in Figure 3.

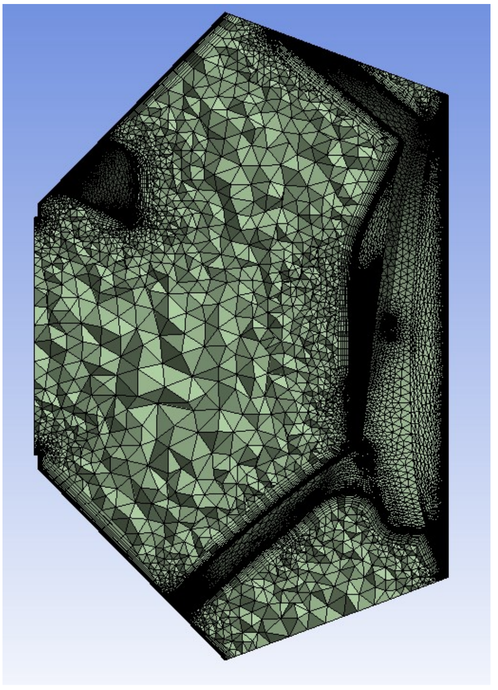

Fig. 3. Impeller meshed in the Ansys meshing.

To carry out the transient simulation the solver setup described by Schiffer et al. [5] was used. Turbulence model, interface, boundary conditions, convergence control and transient time step were chosen same as in the Fluent-based approach. In addition, the CFX module offers the choice of on advective scheme and in this caste the Central difference with marked “Bounded CDS” option was chosen.

Table 1. Overview of element numbers.

Domain	Fluent meshing	Ansys Meshing
Suction pipe	41 804	138 783
Impeller	893 709	3 015 473
Spiral casing	217 167	3 877 860
Σ	1 206 680	7 032 116

4 CFD results end experimental data

To achieve satisfactory convergence of simulation data the iteration number was set to 1200 what responded to 10 full revolutions of investigated impeller. According to Keays and Meskell [2] should be the minimal number of impeller revolutions 8 but the research results, shown in Figure 4., revealed substantial faster convergence after 5-6 full impeller revolutions. Figure 4. shows the comparison of evaluated specific energy simulated in Fluent and CFX. Different mesh density led to faster and better convergence of the CFX result, on the one hand, but considerably lengthened the computational time, on the other. The CFX predicted specific energy was also lower than the Fluent one. The explanation can be found in the insufficient mesh density of the Fluent mesh which doesn´t consider all flow features required way. Simulated parameters, in Figure 4., were computed in the designed BEP at the flow rate of Q = 10 l/s. Slower and worse convergence can be expected at more distant flow rates. The clean water was used as a pumped medium.

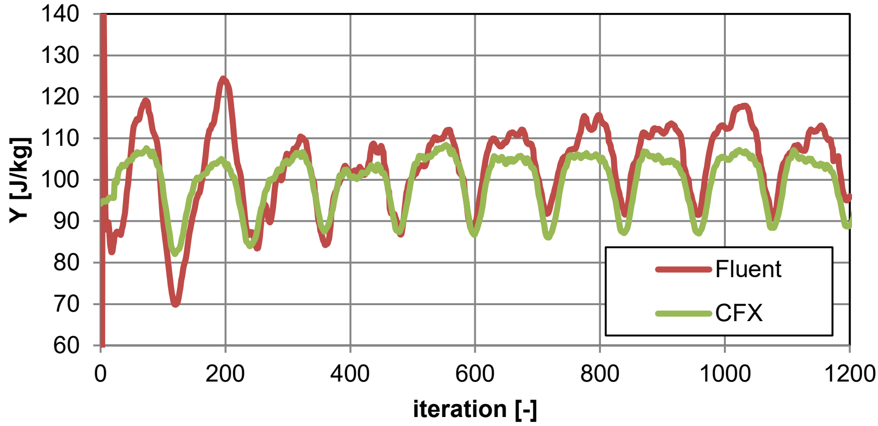

Fig. 4. Convergence of evaluated specific energy.

In the figures below are compared measured with simulated data. Calculated and measured specific energy is viewed in Figure 5. Both approaches achieved sufficient qualitative agreement with the laboratory measurement result but the specific energy overestimation in the BEP was 18.1 %, in the CFX case, and 22.9 %, in the Fluent case. The differences were caused by imperfect calculation mesh and incomplete CFD model. The mesh of both solvers did not meet quality requirements and did not capture all important flow features such as backflows and boundary layer breakage. The CFD model did not include a mechanical losses in bearings and a mechanical seal and also the flow in a non-hydraulic area (behind the hub) wasn´t considered.

Simulated hydraulic efficiency was, analogously to the specific energy, overestimated at the whole range of measured flow rates. The CFX-based approach predicted the hydraulic efficiency about 4.5 % higher than the measured one and the BEP location was moved to the right. The Fluent-based approach estimated the hydraulic efficiency about 9 % higher than the measured one and BEP position was identical to the CFX result.

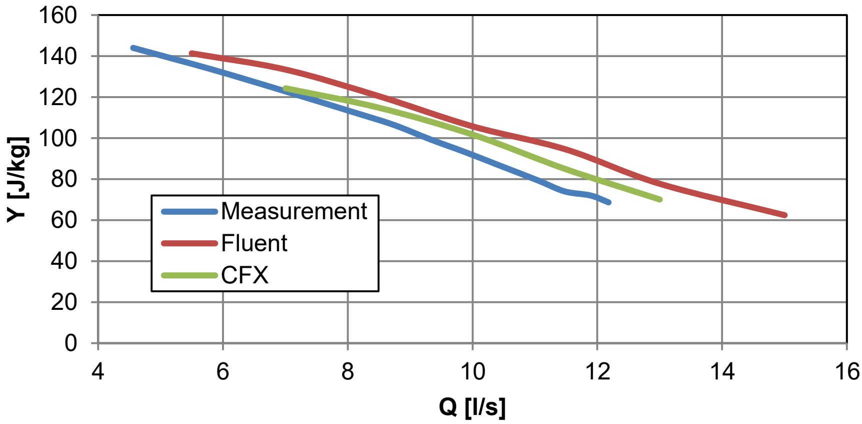

Fig. 5. Comparison of simulated and measured specific energy.

Although the qualitative agreement was achieved, the BEP location and accurate efficiency prediction wasn´t attained. Possible reasons, for such overestimation, are described in the paragraph above.

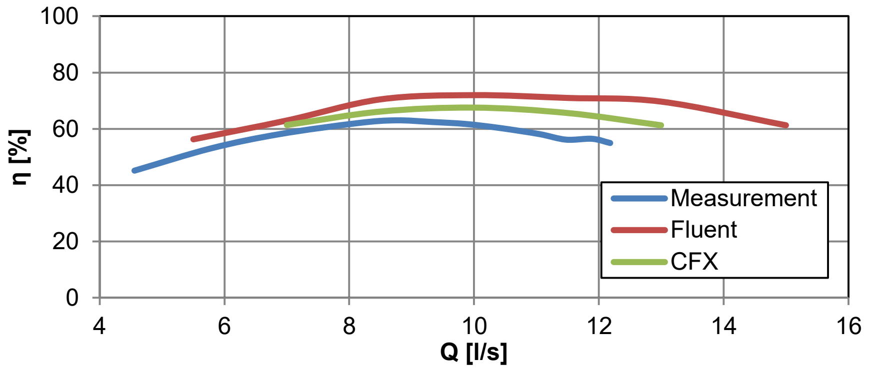

Fig. 6. Comparison of simulated and measured efficiency.

5 Conclusions

This paper addresses the CFD simulation of a single-blade pump. A new Fluent-based approach is compared to the conventional CFX-based approach and results are compared to the laboratory measurements. A CFD analysis was carried out with two different meshes but similar solver setups in the CFX and the Fluent module. After the evaluation of the CFD analysis the following conclusions are presented.

- The Fluent meshing enabled to mesh the whole CFD model with considerably lower element number than the CFX meshing.
- Both approaches achieved sufficient qualitative agreement with the laboratory measurements but the quantitative agreement wasn´t successfully attained. The solvers overestimated specific energy and hydraulic efficiency.
- It turned out that the mesh density did not influenced the qualitative accuracy of CFD results and the inconsistency of CFD and experimental results could be a consequence of imperfect CFD model, especially poor consideration of mechanical losses in bearings and mechanical seal.
- New Fluent-based approach enables the 3D unsteady CFD simulation to be carried out with shorter computational time demand, compare to CFX one.

The described Fluent-based approach is a powerful tool for carrying out the unsteady simulation. It turned out that the tested CFD model is appropriate for a fast indicative simulation of a single-blade pump. To get more accurate results the mesh quality should be higher and the CFD model has to include all flow features and mechanical losses.

Acknowledgement

This contribution was created on the basis of the project "Research centre ALLEGRO" (ITMS project code: 26220220198), supported by Operational Programme Research and Development funded by the European Regional Development Fund.

The authors gratefully acknowledge the contribution of the Scientific Grant Agency of the Slovak Republic under the grant VEGA 1/0743/18.

References

1. F. R. Menter, *Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications*, In AIAA Journal Vol. **32**, No. 8, (1994)
2. J. Keays, C. Meskell, *A study of behaviour of a single-bladed waste-water pump.* In Proc. ImechE 2006, Vol. **220**, Part E : J. Process Mechanical Engineering, (2005). DOI : 10.1243/09544089JPME60
3. B. De Souza, J. Daly, A. Niven, P. Frawley, *Simulation of Transient Flow through Single Blade Centrifugal Pump Impeller with Tipgap Leakage,* In Proc. of the 4th WSEAS International Conference of Fluid Mechanics and Aerodynamics, pp 349 – 354, (2006)
4. F.-K. Benra, H. J. Dohmen, M. Sommer. *Periodycaly Unsteady Flow in a Single-Blade Centrifugal Pump – Numerical and Experimental Results,* In Proc. of FEDSM, ASME Fluid Engineering Division Summer Meeting and Exhibition, Houston, USA, (2005)
5. J. Schiffer, Ch. Bodner, H. Jaberg, S. Korupp, L. Runte. *Performance analysis of a single-blade impeller pump based on 3D numerical simulation,* International Rotating Equipment Conference, Düsseldorf, (2016)
6. M. Kurilla, B. Knížat, R. Olšiak, P. Slovák, CFD Analysis of flowin a single-blade impeller. AIP Conference Proceedings **2118**, 030023 (2019). DOI : 10.1063/1.5114751