

[https://doi.org/10.52326/jes.utm.2023.30\(2\).01](https://doi.org/10.52326/jes.utm.2023.30(2).01)
UDC 621.671



DETERMINING OPTIMAL SIMULATION SETTINGS FOR THE CENTRIFUGAL PUMP PARTS OPTIMIZATION PROCESS

Viorel Bostan¹, ORCID: 0000-0002-2422-3538,
Andrei Petco^{1,2*}, ORCID: 0009-0004-0577-3296

¹ Technical University of Moldova, 168 Stefan cel Mare Blvd., Chisinau, Republic of Moldova

² S.R.L. "CRIS" 68/2-69 Albisoara str., Chisinau, Republic of Moldova,

*Corresponding author: Andrei Petco, andrei.petco@tcm.utm.md

Received: 05. 17. 2023

Accepted: 06. 20. 2023

Abstract. This paper explores the optimal simulation settings required by the optimization process of centrifugal canned motor pump parts using the CH 6,3/32-2,2-2 pump as an example. The optimization of centrifugal pump parts is particularly important in order to increase pump characteristics: pump efficiency, net positive suction head, as well as other optimization criteria. The optimal conditions to obtain a geometric model and optimal mesh properties, initial and boundary conditions, solver settings were investigated. The mathematical models used in the simulation were also presented. In conclusion, there have been revealed optimal simulation settings.

Keywords: *Centrifugal pumps, Numerical analysis and optimization methods, Computational Fluid Dynamics, Ansys CFX.*

Rezumat. Articolul explorează setările optime de simulare necesare procesului de optimizare a pieselor pompei centrifuge cu motor capsulat, folosind pompa CH 6,3/32-2,2-2 ca exemplu. Optimizarea pieselor pompei centrifuge este deosebit de importantă pentru a crește caracteristicile pompei: randamentul, rezerva de cavitație, precum și alte criterii de optimizare. Au fost investigate condițiile optime pentru obținerea unui model geometric și proprietățile optime ale rețelei de discretizare, condițiile inițiale și la limită, setările solutorului. Au fost prezentate și modelele matematice utilizate în simulare. În concluzie, au fost relevate setările optime de simulare.

Cuvinte cheie: *Pompe centrifuge, Analiza numerică și metode de optimizare, Dinamica fluidelor computaționale, Ansys CFX.*

1. Introduction

The use of Computational Fluid Dynamics (CFD) methods in the pump manufacturing industry is becoming an industry standard. CFD tools are used to obtain geometry of the pump pieces, in the optimization process or in the case of applying reverse engineering to investigate the parameters of the already produced pump.

This study presents the investigations on the optimal simulation settings in order to increase the efficiency of the optimization process of the centrifugal pump main parts, whose generalized optimization algorithm is presented in Figure 1.

We can notice that optimization is an iterative process, so choosing the optimal simulation settings is crucial to obtain an optimized geometric model with minimum wasted resources.

The methodology is presented based on the geometry of the canned motor hermetic centrifugal pump CH 6,3/32-2,2-2 (Figure 2).

2. Creating a geometric model

Geometric model is performed by "extracting" the geometry of the flow zone from the geometric model of the pump. Figure 3 shows that the CH-type hermetic pump is a monobloc construction composed of two parts [1]:

- single-stage pumping part;
- three-phase asynchronous canned electric motor.

The liquid is pumped into the pumping part, the fluid passes inside the pump casing through the suction connection, enters the impeller, where by transforming the kinetic energy of rotation into hydrodynamic energy, the fluid pressure increases, the liquid being transported to the discharge connection of pump casing. On the front and back side of the impeller, there are two cavities necessary to balance the axial forces.

A part of the pumped liquid from the discharge zone passes through the sealing slot behind the impeller, thus the lubrication, the cooling of the bearings and the inner cavity of the electric motor are carried out by means of the pumped liquid. A cooling jacket can also be used to cool the stator.

As we can see from Figure 2, a centrifugal pump is a complex hydraulic system. Due to the large difference in the ratio between the thicknesses of the flow channels in the pump it can be approx. 10^2 - 10^3 (between the sealing gap and the diameter of the pump casing), it is difficult to achieve the simulation of the flow in the entire pump.

To make the optimization process more efficient, it is reasonable to exclude from the calculation secondary circuit (the flow of the internal cavity of the electric motor), „balancing” cavities, the cooling circuit of the motor, etc. The exclusion will not significantly influence the calculation results [1].

The simulation is carried out in the following areas: the suction connection, the internal volume of the pump impeller, the area of the volute and the area of the discharge connection (Figure 3).

In order to represent correctly the velocity and pressure field, the area of discharge and suction connection is elongated by approx. 3-5 $Dy(dy)$, as it is necessary for the true modeling of pump work within a hydraulic system [1].

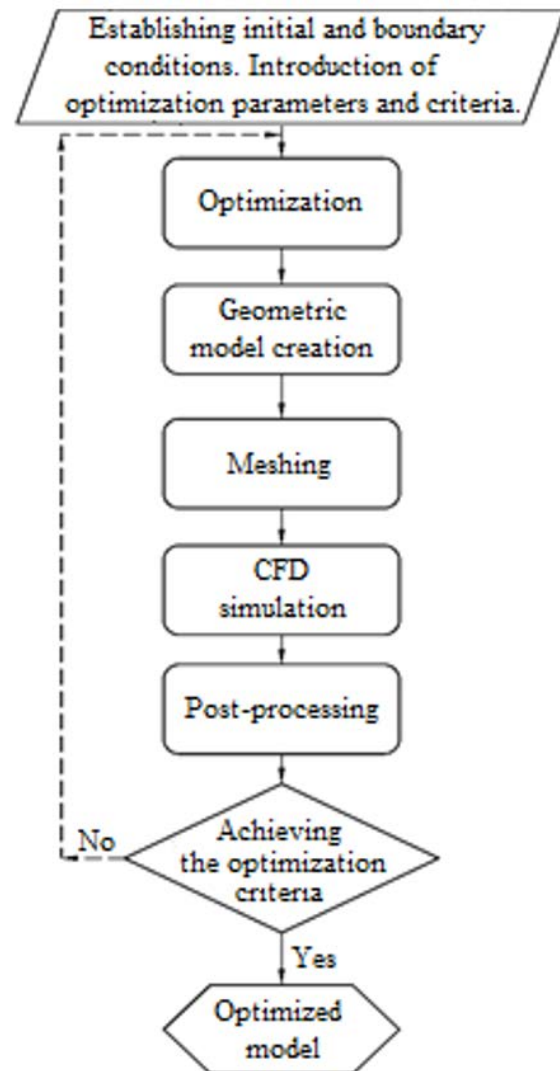


Figure 1. Optimization scheme.

When researching the flow of fluid through the pump, the following approaches can be applied: the pump impeller can be divided axially symmetrically, in relation to the axis of rotation. The contact between the dividing surfaces being joined by the Periodic Interface or undivided impeller. In the first case, the volume of the researched domain is reduced, which leads to an increase in the calculation speed, but may reduce its accuracy. The first approach is used in the preliminary phase of the process optimization, when the rotor can be investigated separately from the other working organs of the pump, and the second in the final phase, when maximum calculation accuracy is required. In this study, optimal parameters of the undivided impeller simulation were investigated.

It should be mentioned that the exposed algorithm is valid only if we already have the geometric model of the pump, otherwise we need to obtain some initial geometry intended for optimization. In this case we can apply simplified calculation methods, namely:

The two-dimensional calculation (ANSYS Vista CCD) simplified approach considering the impeller geometry in two dimensions, frequently used on first calculation iteration, to obtain, primarily, the meridian section of the impeller. This approach, unfortunately, may not capture all the three-dimensional flow phenomena accurately, but requires minimal computation.

The quasi-three-dimensional calculation (ANSYS BladeModeler, Cfturbo), represents a two-dimensional method, with the application of some assumptions intended for the description of the flow in the channels between the blades, taking into account the effects formed in the boundary layer of the flow. This method is more accurate than the two-dimensional method, which allows to obtain quickly the initial geometry of the pump parts, but in turn is inferior to *the three-dimensional simulation method* (ANSYS CFX, Fluent, Poliflow, etc.) used for calculations in the final phase of the optimization process to receive the precise image of the flow in boundary layers, secondary torrents, etc., an example of the use of which is given below.

3. Discretization methods selection The fluid flow inside the pump represents a complex hydrodynamic phenomenon, described by Navier-Stokes equations, which cannot be

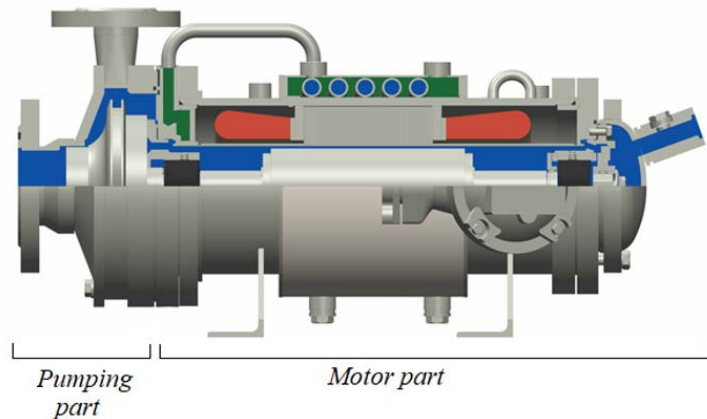


Figure 2. Canned motor pump of CH series (axial section).

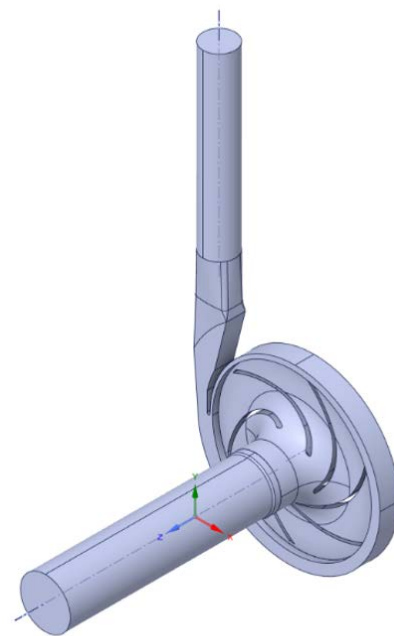


Figure 3. CH 6,3/32-2,2-2 pump geometry, represented in the Ansys SpaceClaim module.

solved analytically in general. An alternative is to numerically solve these equations by dividing the domain into finite elements or volumes. In the numerical simulation of fluid flow processes in pumps, through ANSYS CFX, used as a solver in the case of the given study, the Finite Volume Method (FVM) is used as a discretization method.

The ANSYS Workbench environment has several modules for obtaining a finite volumes mesh: ANSYS Meshing, ANSYS TurboGrid, ANSYS ICEM CFD, ANSYS Fluent Meshing, ANSYS HFSS Meshing. In this study, the optimal parameters of unstructured meshes obtained by ANSYS Meshing were considered and compared with the structured mesh obtained in conjunction with ANSYS TurboGrid and ANSYS ICEM CFD.

ANSYS Meshing is a universal mesh generator (Figure 4) used to create structured and unstructured meshes for various simulation applications (computational fluid dynamics, solid mechanics, electromagnetism and heat transfer, etc.). ANSYS Meshing is the most frequently applied mesh generator due to ease of use, as it allows for convenient customization and control over mesh parameters such as mesh density, element sizing, and boundary layer resolution. However, when studying the flow in a pump, ANSYS Meshing is used to form an unstructured grid consisting of tetrahedral elements occupying the internal volume of the flow path and a boundary layer consisting of prisms which allow to make a more correct description of the flow in the immediate vicinity of the wall, namely, a strong change in velocity caused by the phenomenon boundary layer [2]. Also, this combination allows to achieve a high degree of automation to the meshing procedure.

Figure 5 highlights that the size of the finite volumes of the mesh is uniform in all three domains of the geometric model. To establish the optimal parameters of the mesh, a series of simulations was performed on different sizes of the finite volume, at different number of layers at the boundary surfaces. 16 unstructured discretization grids were created with finite volume size $1 \div 2.5$ mm and $5 \div 20$ inflation layers.

For advanced control of the distribution of finite volumes, the option Advanced Size Function: On Proximity and Curvature was applied, an option that is applied to increase the resolution of the network in the area with large curvature and in narrow areas.

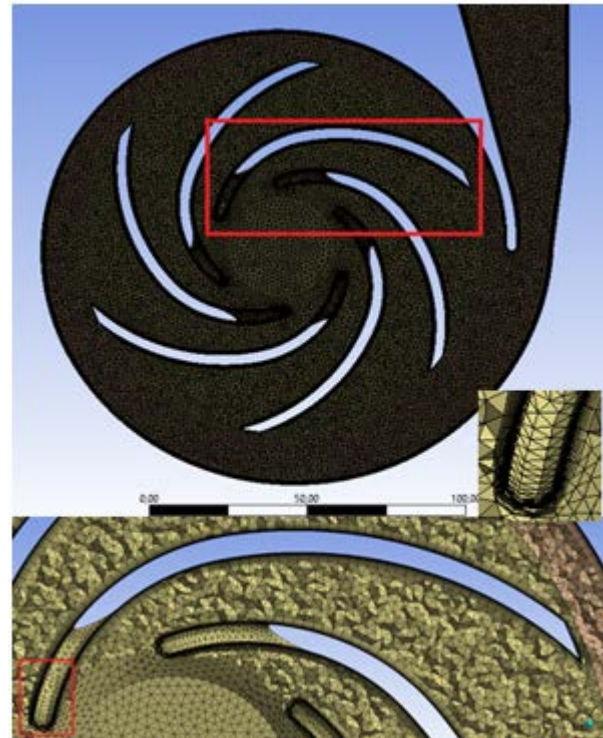


Figure 4. Unstructured mesh, obtained in ANSYS Meshing [1].

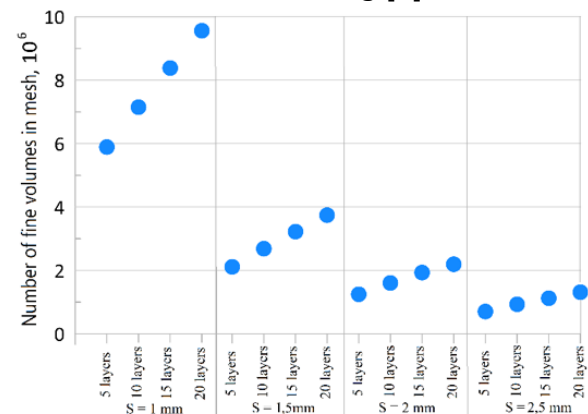


Figure 5. Correlation between fine volume size, number of inflation layers, and number of final volumes in the mesh.

The Inflation process was applied to the surface of the walls with the setting of the Total Thickness option, the Growth Rate growth coefficient of 1.2 and the thickness of the inflation layers proportional cell finite volume size.

ANSYS ICEM and ANSYS TurboGrid were used to construct a structured grid.

ANSYS TurboGrid is a powerful meshing tool specifically designed for turbomachinery applications (pumps, axial compressors and turbines). It offers an automated mesh generation of algorithms that can generate complex blade geometries mesh of inducers, impellers, vanes etc. Figure 6 shows the mesh generated in ANSYS TurboGrid. The desired achievable parameter y^+ equal to 1 was used as the meshing parameters, with the assumed Reynolds number of fluid flow in the impeller equal to 5×10^5 .

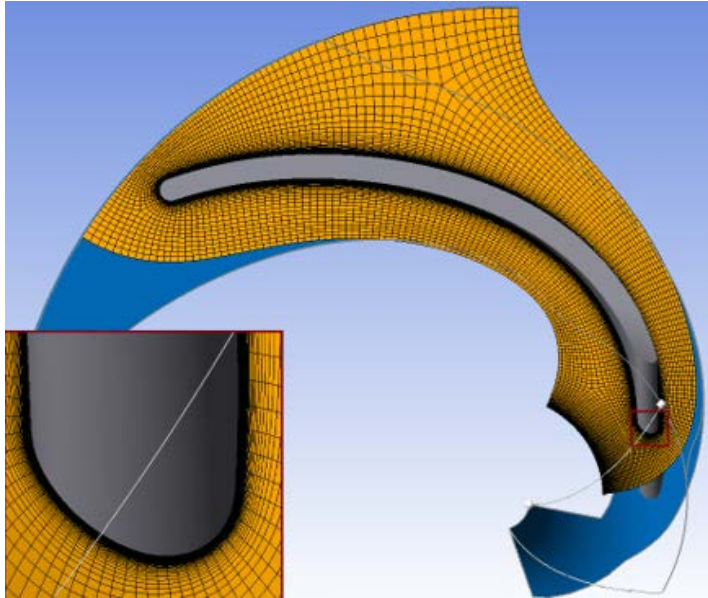


Figure 6. Structured Mesh, obtained in ANSYS TurboGrid.

The *ANSYS ICEM* tool is the most finely tuned mesh generator, primarily designed to generate high quality structured hexahedral meshes, includes a wide range of methods for creating various types of mesh models. The module is difficult to master, while having the most extensive toolkit of all modules provided by ANSYS. It should be noted that if TurboGrid has a ready-made topological set that determines the distribution of volumes, in turn, to build a mesh in ICEM, the block structure must be built manually. This greatly increases the time to get the finished mesh and increases the requirements for the skills of the engineer.

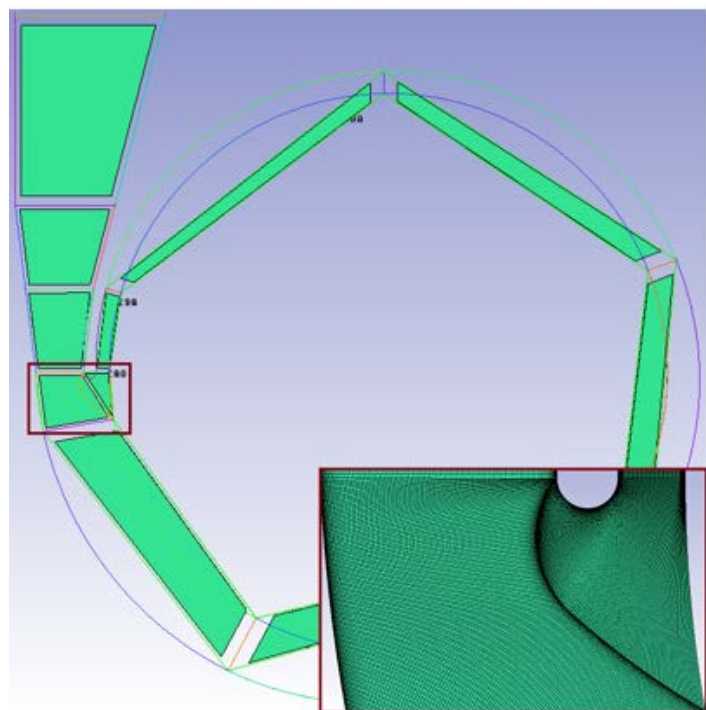


Figure 7. A block system that defines the distribution of elements and a fragment of a structured grid obtained in ANSYS ICEM.

Figure 7 shows the block system on which the finite volumes mesh is built. A fragment of the grid

built in ICEM is also shown, the boundaries of the transition between the blocks associated with the geometry are visible. Since the spiral of the pump housing has a complex geometry, the engagement becomes a difficult task, so the use of a block structured mesh cannot be used only in the final optimization phase, when validating the results.

4. Establishing the mathematical model

Centrifugal pumps work by converting the mechanical energy of a rotating impeller into the energy of a fluid flow, such a complex flow is described by the Navier-Stokes equations, used to model the fluid motion inside the pump, considering complex physical phenomena such as vortex formation, reverse flows, turbulence and predict flow characteristics such as pressure and velocity field distribution, hydrodynamic losses etc. The Navier-Stokes equations represent a set of partial differential equations consisting of mass conservation and conservation of momentum equation.

The differential form of the continuity equation is:

$$\frac{\partial u}{\partial x} + \nabla(\rho u) = 0 \quad (1)$$

where: ρ is the density of the fluid, u - the velocity of the fluid and t - denotes the time.

Conservation of Momentum equation, in three dimensions are form:

$$\frac{\partial u}{\partial x} + u \cdot \nabla u = f - \frac{1}{\rho} \nabla p + \nu \cdot \Delta u \quad (2)$$

where: p - the pressure, f - the mass unit force, related to the mass, τ - the viscous stress tensor, defined by the constructive equation of the Newtonian fluid:

$$\tau = 2\nu(S - \frac{1}{3}(\nabla \cdot u)I)$$

ν - the kinematic viscosity, I - the unit tensor of the second order, and S is the strain rate tensor:

$$S = 2\nu(\nabla u + \nabla u^T)$$

The left-hand side of Equation (2) represents the unitary inertial forces, and the terms on the right-hand side represent the mass forces, pressure forces, and viscous friction forces, respectively. [2].

Nonlinear character of equations system with partial derivatives (1) and (2), is due to the inertia term $u \cdot \nabla u$. This nonlinear term introduces complex interactions between structures of different scales in the fluid motion, being considered the primary source of turbulence. The nonlinearity can become relatively weak, if the inertial forces, which have a destabilizing role, are small in relation to the frictional forces. In this case, the Navier-Stokes equations can be solved exactly or integrated numerically without additional simplifying assumptions. Generally, solving the Navier-Stokes equations is extremely difficult. At present, only the existence of weak solutions has been demonstrated, and the presence of strong solutions being valid only for small time intervals. Therefore, various simplifications are considered to facilitate finding the solutions, if not exact, at least approximate [2].

Since an analytical solution of the Navier-Stokes equations, representing an exact mathematical description of the flow, cannot be achieved, these equations can be solved in an approximate manner, dividing a complex flow domain into a multitude of small cells, by numerical methods [3]. The numerical solution, although it is only a numerical approximation of real physical processes in a fluid flow, it is still the most rational approach both for solving problems of calculating the theoretical characteristics of a pump and for solving the problem of optimizing pump parts.

It should also be noted the main problem in the calculation of the fluid flow, namely, modeling the turbulent process, which consists in pronounced fluctuations of the flow field in time and space, which can have a significant impact on the flow characteristics. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterized by a high Reynolds Number. It is a complex process, mainly because it is three dimensional, unsteady and consists of many scales [3, 4].

If the velocities in equation (1) are considered as unsteady, the Navier-Stokes equations would be sufficient to calculate turbulent flows. This *Direct Numerical Simulation (DNS)* of turbulent flows in pump components is, however, beyond present computing capabilities. In order to exactly describe all turbulent fluctuations, extremely fine calculation grids and excessive computing times are required. Number N of elements needed in a grid for DNS can be estimated from $N \approx Re^{9/4}$. For the liquid flowing through the pump impeller, the $Re = 10^6$, the recommended number of finite volumes is $N = 10^{13}$ [3].

Turbulence models are used to correctly describe the turbulent flow. These models are needed to simplify mathematical representation of the turbulent flows through numerical methods, which simulate the behavior of turbulent fluid. This allows to describe the interaction between turbulent eddies and the averaged parameters of the flow. The choice of the turbulence model and the pertinent turbulence parameters thus harbors one of the main uncertainties of Navier-Stokes calculations of turbomachines [3].

It should be noted that in the case of simulating the flow of fluids through the pump flow section, the Navier-Stokes equations are simplified by adopting a series of simplifying assumptions: treating the fluid as a Newtonian fluid, in an incompressible, isothermal flow with a constant fluid dynamic viscosity [1].

The most commonly used modeling approach is *RANS (Reynolds-Averaged Navier-Stokes)* allowing to modeling turbulent flows based on averaging the Navier-Stokes equations over time. The main difference of the approach is that the averaged velocity and pressure are divided into averaged components and turbulent components. The application of the averaging of the Navier-Stokes equations requires the introduction of additional terms that are interpreted as apparent stresses and apparent thermal fluxes associated with the turbulent motion, being formulated depending on the average parameters through the turbulence models that are necessary to close the system of averaged Reynolds equations [2].

Turbulence models introduce additional assumptions, which, as a rule, are no longer a strict reflection of general conservation principles. One of these assumptions being: the linear Boussinesq approximation $\tau_{ij}^R = \nu_T \cdot S_i$, which describes the linear dependence of the Reynolds tensor τ_{ij}^R and the strain rate tensor S_i , then equations (3) and (4) can be reformulated as follows:

$$\frac{\partial \bar{u}_i}{\partial x_j} = 0 \quad (3)$$

$$\bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} - \frac{\partial}{\partial x_j} \left[(\nu - \nu_T) \left(\frac{\partial \bar{u}_i}{\partial x_j} - \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] + \frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_j} = 0 \quad (4)$$

Turbulence models based on the RANS equations (3) and (4) are known as Statistical Turbulence Models due to the statistical averaging procedure employed to obtain the equations. Due to the use of averaging procedures, we are forced to introduce additional unknown terms containing products of fluctuating quantities that act as additional stresses in the fluid. These terms are called Reynolds stresses.

Reynolds stresses need to be modeled by additional equations of known quantities in order to achieve "closure." Closure implies that there is a sufficient number of equations for all the unknowns, including the Reynolds-Stress tensor resulting from the averaging procedure. The equations used to close the system define the type of turbulence model [4].

It should be noted that in addition to the RANS approach (Reynolds Navier-Stokes averaging) used in steady-state calculation, there are modified URANS models (Unsteady Reynolds-Averaged Navier-Stokes) used for transient calculation. Unlike the RANS model, which averages flow variables over time, URANS models take into account unsteady flow phenomena by including time-dependent terms in the governing equations, which allows the simulation of a time-varying flow pattern.

The turbulence models which can be used to simulate the fluids flow of the pump's parts are:

- *Standard k - ϵ model*: one of the most prominent turbulence models, is considered the industry standard model [5]. This is the baseline turbulence model in CFX, which solves transport equations for turbulent kinetic energy (k) and its dissipation rate (ϵ). This model is widely used because convergence is better than with other turbulence models. Standard k - ϵ model is a universal model suitable for a wide range of turbulent flows, but its use due to the overestimation of the turbulent kinetic energy in places with a strong velocity gradient, the flow pattern is distorted [6]. The use of this model may lead to loss of accuracy on flows on curved paths, in geometries such as bends or diffusers. Also, the k - ϵ model does not describe well the swirling flows and the strong secondary flows. It may be difficult to simulate the flow through rotating pumps components, since the body forces influence the boundary layers. These phenomena can be found in such parts of the pump as impellers, diffusers, volutes, etc. [3].
- *Standard k - ω Model*: a common two-equation turbulence model, used as an approximation for the RANS equations, involves solving the transport equations of the turbulent kinetic energy k and the specific dissipation rate ω . Advantages of this model is the near wall flow treatment [2]. The model does not involve the complex nonlinear damping functions required for the k - ϵ model and is therefore more accurate and more robust [4]. This model was developed specifically for flows against strong pressure gradients (a similar flow pattern appears diffuser) [3]. The model is used mainly for historical reasons, since it has been supplanted by modified k - ω models such as Menter's SST and BSL models due to the strong dependence of the solution on free flow ω values outside the shear layers [6].
- *SST (Menter's Shear Stress Transport) k - ω model*: This hybrid model combines the k - ϵ and k - ω models to capture the advantages of both models [2]. The SST model is the most commonly used turbulence simulation model due to the simultaneous combination of the strengths of the k - ω (description of the flow near walls) and k - ϵ (description of the free flow of fluids) models [7]. The advantage of the SST model over the above models is its high accuracy for flows with unfavorable pressure gradients and separation, [6] which offers improved predictions of the turbulent shear stress in regions with high shear and adverse pressure gradients [4], but flows on curved paths are still not well simulated [3].

In addition to stationary calculations, if it is necessary to describe complex flows whose parameters are difficult to average over time, transient modeling is used. In this case, the following models' groups and hybrids models are used:

- *Unsteady Reynolds Averaged Navier-Stokes (URANS)*, a time-averaged approach that solves the Reynolds-averaged Navier-Stokes equations, listed above.
- *Scale-Resolving Simulation (SRS)* is a CFD approach, i.e. an intermediate approach between URANS simulations and Direct Numerical Simulation DNS, which aims to describe both the large-scale and small-scale turbulent flow structures. Although the URANS models perform well in calculating the flow confined by walls, it should still be noted that the use of such models is important in solving complex multi-physics problems of turbomachinery, such as thermal calculations in zones of non-stationary mixing of flows at different temperatures, when simulating vortex cavitation, which is caused by a non-stationary turbulent pressure field, modeling acoustics in hydraulic systems when turbulence creates noise sources, etc [8-14]. These models include Large Eddy Simulation (LES), a model that decomposes the flow into large-scale vortex structures and models the small-scale turbulent eddies and its hybrids with RANS models. The above are extremely sensitive to mesh density and therefore cannot be used in the optimization process.

We can see that the

optimization process described in figure 1 is a multi-iteration one, the calculation time problem is a decisive one. It can be mentioned that the description of the geometry of the pumps parts can be done in dozens of parameters, which leads to the increase of the number of calculation cycles to hundreds of iterations. Therefore, we cannot use computationally expensive turbulence models. Figure 8 shows the study of the effect of two different RANS models on the accuracy of measuring the pump head, taking into account the maximum size of the final mesh element. When comparing, it can be seen that the *SST model* gives a more accurate result in comparison with the *k-ε model*, both at nominal feed, and at minimum and maximum feed.

As already mentioned, three-dimensional simulation can be performed in two approaches: Steady-state and Transient. Steady simulations compute a solution with time-averaged values, and dynamic simulation computes instantaneous values in each time unit for each finite volume. Transient simulation considerably increases the resources required for simulation, but provides a complete picture of the processes taking place in the working organs of the pump.

Both approaches were compared and as a result, the difference between the researched parameters, under nominal flow conditions, with a discretization grid having a relatively small finite volume size, is not considerable. Although URANS transient simulations provide a more accurate picture, they are more time consuming than steady state simulations. So, we can state that in the final phase of the process optimization it is favorable to apply steady-state simulations to a fine discretization network.

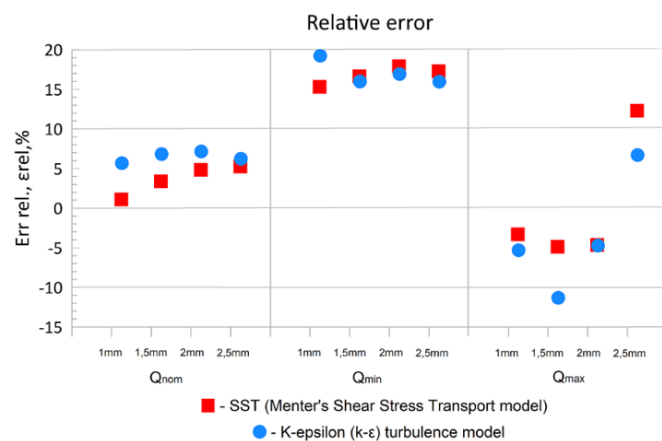


Figure 8. Comparison of the precision of calculating the head of the pump CH 6,3/32-2,2-2 using k-ε and SST turbulence models.

It should be noted that the geometric model consists of three domains (Figure 3), so it is necessary to apply the transition model between the domains: the Stage (mixing plane) model – averaging the velocities on the interface surface and the Frozen rotor model – which better models' interaction between different domains. The Frozen rotor model additionally adds the Navier-Stokes equations, written for the mobile domain to the rotation tensor, thus virtually simulating its relative rotation. The GGI (General Grid Interface) method is used as the connection method of the finite volumes.

The comparison between the Stage and Frozen rotor models was performed. We can see that when simulating the flow through a centrifugal pump, the Frozen rotor model provides true results compared to the Stage (mixing plane) model, in relation to the size of the finite volume, with approx. 2-5% relative error (calculated as a function of pumps head).

The mass transfer model represents the cavitation model, defined by the Rayleigh-Plesset equation, which is used to model the cavitation process [7]:

The Rayleigh-Plesset equation (5) derives from the Navier-Stokes equations and represents a differential equation that governs the dynamics of a spherical bubble (formed as a result of the cavitation process inside the pump) in an incompressible fluid.

$$R \frac{d^2 R}{dt^2} + \frac{3}{2} \left(\frac{dR}{dt} \right)^2 + \frac{4\nu}{R} \frac{dR}{dt} + \frac{2\gamma}{\rho_L R} + \frac{\Delta P(t)}{\rho_L} = 0 \quad (5)$$

where: ρ_L is the density of the liquid, $R(t)$ - radius of the bubble, ν – kinematic viscosity of the liquid, γ being the surface tension between the bubble and the liquid, $\Delta P(t) = P_\infty(t) - P_B(t)$, where $P_B(t)$ - pressure in bubble, and $P_\infty(t)$ - liquid pressure.

The cavitation model used is the Zwart, Gerber and Belamri model based on the Rayleigh-Plesset equation [4]:

$$\dot{S}_{lv} = \begin{cases} F_{vap} \frac{3r_{nuc}(1-r_v)\rho_v}{R_B} \sqrt{\frac{2P_v - P}{3\rho_l}}, & P < P_v \\ F_{cond} \frac{3r_v\rho_v}{R_B} \sqrt{\frac{2P - P_v}{3\rho_l}}, & P > P_v \end{cases}$$

The model assumes that the overall rate of multiphase mass transfer per unit volume \dot{S}_{lv} is calculated using the nucleation site volume fraction r_{nuc} , assuming all bubbles in the system are the same size [9].

In present research, the nucleation site radius R_B is $1 \cdot 10^{-6}$, the vapor saturation pressure P_v is chosen to be 3170 Pa.

5. Establishing initial and boundary conditions

The following initial conditions were applied in the study (Figure 9):

- The impeller area rotates at a speed of 2950 min⁻¹.
- Zero reference pressure ($P_{ref} = 0$ atm).
- Isothermal flow ($t = 25^\circ\text{C}$).
- Minimal inlet turbulence (1%).
- In the initial state, the fluid consists entirely of water in liquid form.

- Walls type boundary conditions are applied to the walls with the "no slip" specification - as a wall that does not allow mass or energy transfer, the velocity on these surfaces is considered zero, which allows the simulation of flow in the area near the wall.

It should be noted that the model does not take into account the gravitational forces and the roughness of the contact surfaces.

The simulation is based on the biphasic continuous fluid model, consisting of:

- water in liquid form at 25° C;
- water vapor at 25° C.

The choice of the time step size is made due to the use of a static calculation model. The Timescale Control function allows you to adjust the size of the time step during simulation to ensure calculation convergence and accuracy. As the Timescale Control model, we selected the *Physical Timescale* option.

Since the main movement under the study is the rotational movement of the impeller at a constant speed, in the search for the optimal step, we tie it to the angular velocity. The following cases were investigated: $2/\omega$, $1/\omega$, ..., $1/8\omega$, where: $\omega=2\pi n$ - angular velocity. The optimal time step was equal to $1/\omega = 0.003237$ s, decreasing the step below this level does not bring tangible results.

At the *Inlet* in the flow area, Total Pressure (Stable), $P_{inlet} = 10^6$ Pa is indicated, the flow direction being normal. The character of the domain is indicated – Subsonic and the state of low turbulence (1%). It is assumed that at the entrance to the system the flow consists entirely of water in liquid form.

At the *Outlet* from the flow area, the Flow (Bulk Mass Flow Rate) is indicated, for a nominal flow rate of 1.75 kg/s. At the system output, the flow characteristics related to turbulence and cavitation are calculated by the solver (Zero gradient).

6. Establishing the calculation parameters

Calculation parameters are also important as they allow to ensure numerical convergence, optimization of computational resources, such as allocated memory and execution time.

For the Stage and Frozen rotor models, the final number of calculation iterations of 400 was selected, also the calculation is completed when the residual error tolerance of 10^{-5} is reached during the simulations.

To monitor the convergence, the following indicators were selected: static and absolute pressure at the inlet and outlet, as well as the torsional moment relative to the axis of rotation applied to the pump impeller and domain imbalance.

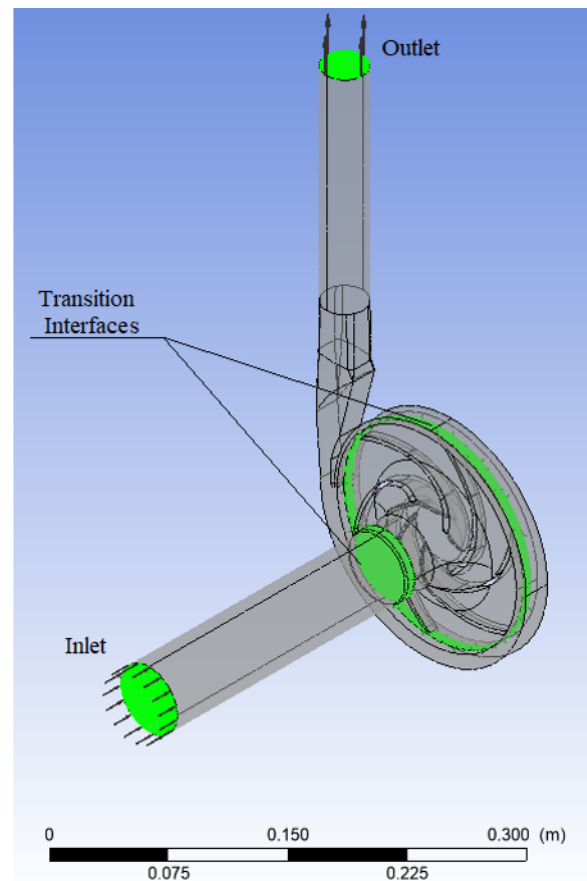


Figure 9. Application of boundary and initial conditions in ANSYS CFX pre.

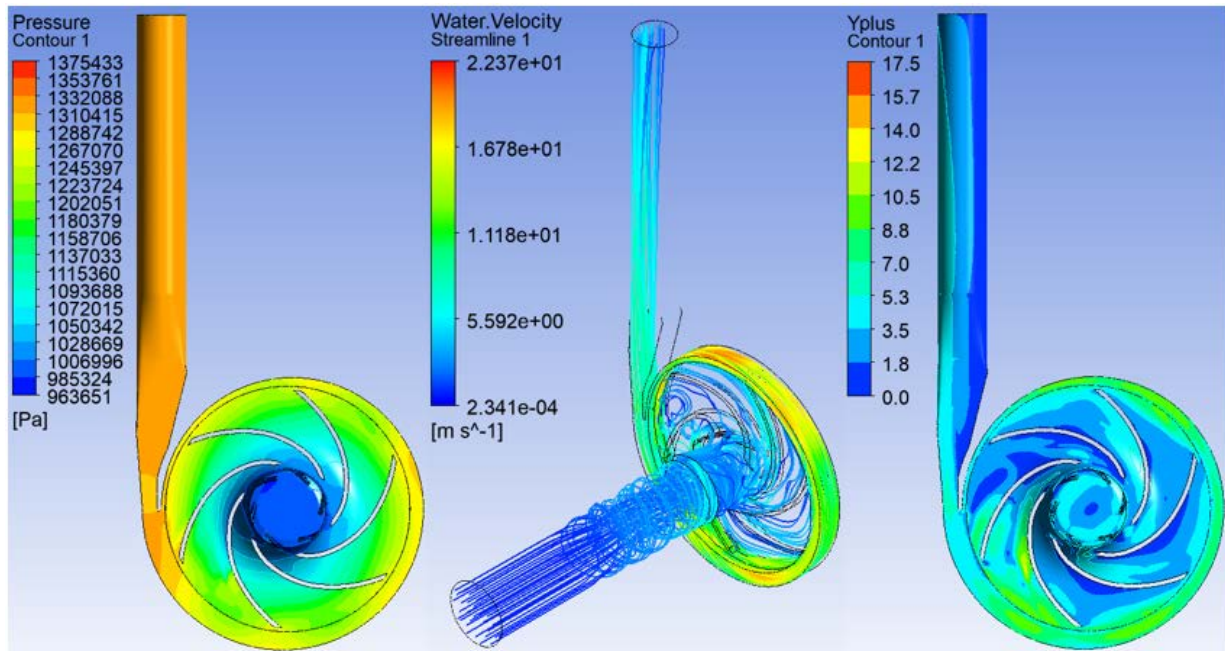


Figure 9, a. Representation of the pressure and velocity field, as well as the distribution of the y+ parameter in ANSYS CFX post.

7. The results of the simulations

Throughout this study, 128 simulations were performed on different discretization grids, with the size of the finite volume $1 \div 2.5$ mm and with $5 \div 20$ layers of inflation, at different flow regimes: minimum flow ($Q_{\min} = 2$ m³/h), Best Efficiency Point nominal flow ($Q_{\text{nom}} = 6.3$ m³/h) and at maximum flow ($Q_{\max} = 9.5$ m³/h), also the k-ε and SST turbulence models at nominal flow were compared.

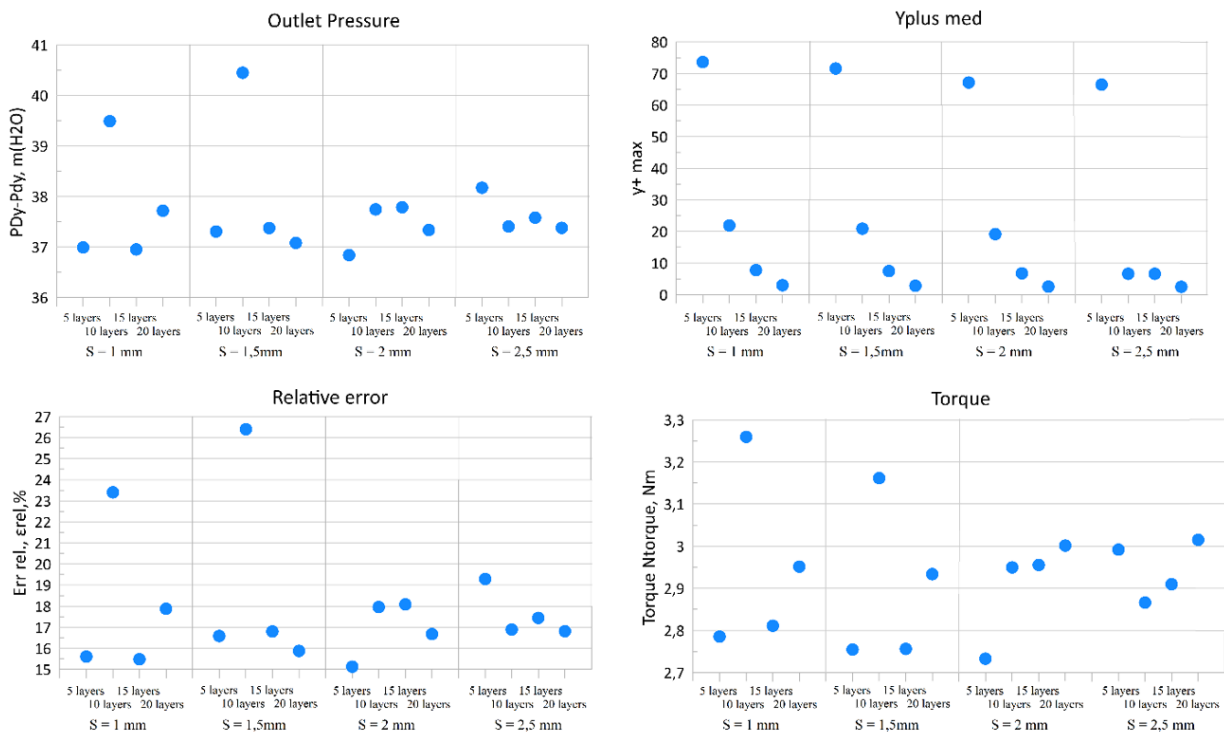


Figure 10. Simulation results (pump: CH-6,3-32; $Q_{\min} = 2$ m³/h, Frozen rotor, SST model).

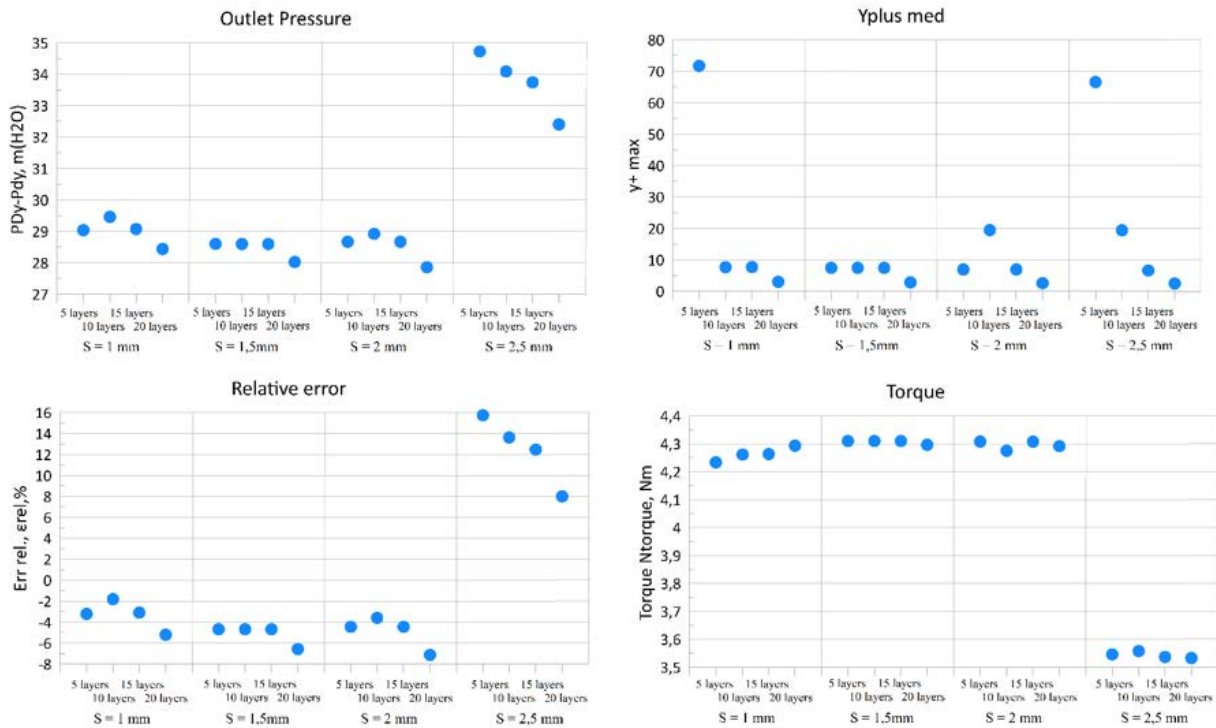


Figure 11. Simulation results (pump: CH-6,3-32; $Q_{nom} = 6,3 \text{ m}^3/\text{h}$, Frozen rotor, SST model).

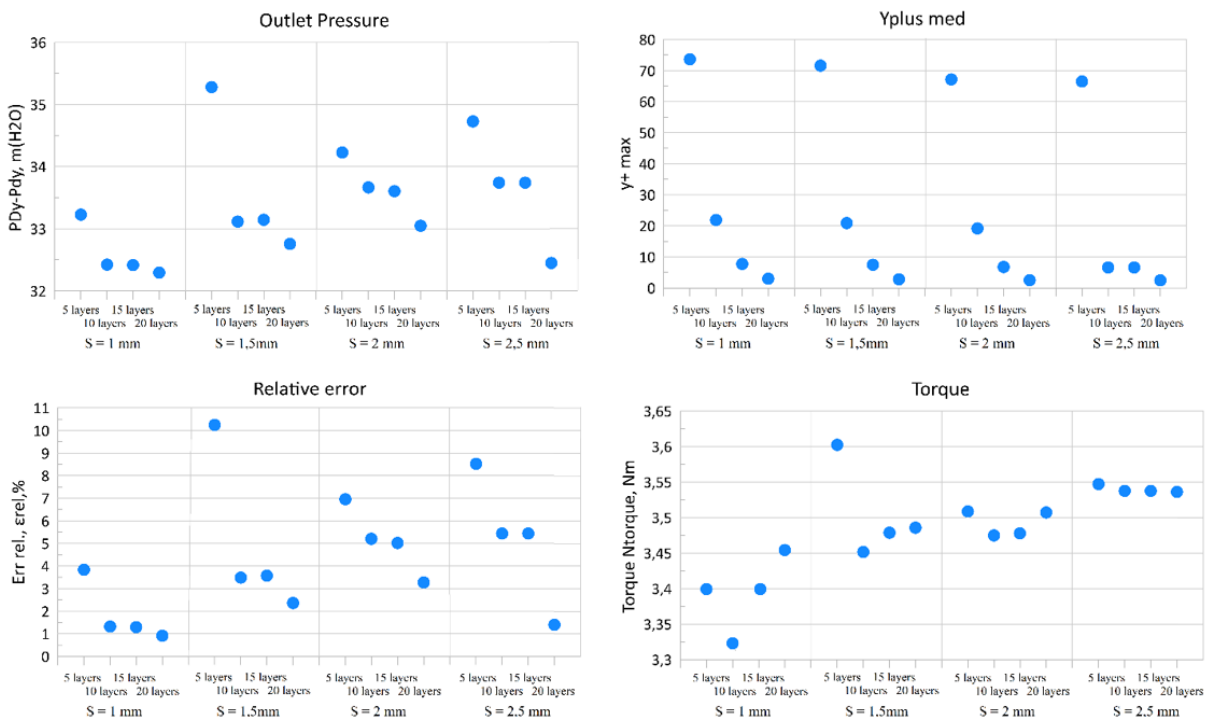


Figure 12. Simulation results (pump: CH-6,3-32; $Q_{max} = 9,5 \text{ m}^3/\text{h}$, Frozen rotor, SST model).

8. Conclusions

A series of simulations was carried out on different grids, with different sizes of the finite element ($S = 1 \div 2,5 \text{ mm}$), with a different number of pumping layers ($5 \div 20$ layers). Unstructured and structured meshes were also compared, when carrying out process optimization. It is recommended to use an unstructured mesh obtained in ANSYS Mesher and

a structured mesh obtained in ANSYS TurboGrid, due to the high degree of automation of mesh generation.

The accuracy of the calculation increases proportionally to the decrease in size of the fine volume and the increase in the number of layers of inflation describing the boundary layer. The best option is to choose the size of the final volume of the order of 1.5 mm and the order of 15-20 layers of inflation, because the difference in the number of final volumes between the mesh of the volume size of the order of 1.5 mm and 1 mm is more than 2 times (Fig. 5), and the available computing resources are limited.

The numerical convergence of the results is obtained. As the size of the final volume decreases, the parameter y^+ drops to the region of units.

It can be noted that the accuracy of the SST model is higher, and is optimal for the optimization procedure of parts of centrifugal pumps. This fact is due to the simultaneous combination of the strengths of $k-\omega$ and $k-\epsilon$.

The results of applying the SST RANS and URANS models were also compared. It was found out that the use of transient calculation significantly increases the consumed computing resources, which is unacceptable for building process optimization.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Petco, A. Numerical simulation of liquid flow in the working organs of the centrifugal pump by means of Ansys CFX. In: Technical-Scientific Conference of Students, Masters and Doctoral Students, Technical University of Moldova, Chisinau, 23-25 March, 2021. [in Romanian].
2. Bostan, V. Mathematical models in engineering: contact problems: modeling and numerical simulations in aero-hydrodynamics. Chisinau, 2014, 456 p. [in Romanian].
3. Gülich, J.F. Centrifugal Pumps. Springer International Publishing, Cham, 2020, 966 p.
4. Ansys CFX Solver Theory Guide. Release 2021 R2. ANSYS, Inc., 2021.
5. Ansys CFX-Solver Modeling Guide. Release 2021 R3. ANSYS, Inc., 2021.
6. Menter, F.R., Sechner, R., Matyushenko, A. Best Practice: RANS Turbulence Modeling in Ansys CFD. Available online: <https://semiengineering.com/best-practice-rans-turbulence-modeling-in-ansys-cfd/> (accessed on 12.03.2023).
7. Petco, A. Constructive-functional development of centrifugal pumps through multiparametric optimization and CFD simulations. In: Technical-Scientific Conference of Students, Masters and Doctoral Students, Technical University of Moldova, Chisinau, 26-29 March, 2019.
8. Best Practice: Scale-Resolving Simulations in Ansys CFD. Available online: <https://www.ansys.com/resource-center/technical-paper/best-practice-rans-turbulence-modeling-in-ansys-cfd> (accessed on 12.03.2023).
9. Zwart, P.; Gerber, A.G.; Belamri, T. A two-phase flow model for predicting cavitation dynamics. In: Fifth International Conference on Multiphase Flow, 2004.
10. Matlakala M.E.; Daramy V. K. Optimization of the Pumping Capacity of Centrifugal Pumps Based on System Analysis. MATEC Web of Conferences 347, 00024 (2021) SACAM2020.
11. Lomakin, V.O. ; Chaburko, P.S.; Kuleshova, M.S. Multi-criteria Optimization of the Flow of a Centrifugal Pump on Energy and Vibroacoustic Characteristics, Procedia Engineering, Volume 176, 2017, 476-482.
12. Zhenwang L.; James D.; Shen K.; Man J. Performance Optimization of High Specific Speed Centrifugal Pump Based on Orthogonal Experiment Design Method. October 2019, Processes 7(10):728
13. Pouria H.; Shahriyar M. Modeling and optimization of centrifugal pumps using ansys fluent® and genetic algorithm analysis. International Transaction Journal of Engineering, Management, & Applied Sciences & Technologies, 2019, p. 903-911.
14. Ling Z.; Weidong S.; Suqing Wu. Performance Optimization in a Centrifugal Pump Impeller by Orthogonal Experiment and Numerical Simulation. Advances in Mechanical Engineering 2013, p.1-7.

Citation: Bostan, V.; Petco, A. Determining optimal simulation settings for the centrifugal pump parts optimization process. *Journal of Engineering Science* 2023, 30 (2), pp. 8-22. [https://doi.org/10.52326/jes.utm.2023.30\(2\).01](https://doi.org/10.52326/jes.utm.2023.30(2).01).

Publisher's Note: JES stays neutral with regard to jurisdictional claims in published maps and institutional affiliations.



Copyright:© 2023 by the authors. Submitted for possible open access publication under the terms and conditions of the Creative Commons Attribution (CC BY) license (<https://creativecommons.org/licenses/by/4.0/>).

Submission of manuscripts:

jes@meridian.utm.md