Using advanced fluid flow simulation of centrifugal pump for the purpose of education of maritime professionals

line 1: 1st Cvetelina Velkova
line 2: *Department of Mechatroncis*
line 3: *Naval Academy N.Vaptsarov,*
Varna, Bulgaria
line 4: Varna, Bulgaria
line 5: cvvelkova@nvna.eu

Abstract. **The purpose of the study is to suggest the advanced fluid flow simulation of centrifugal pump that will be implemented in training of maritime specialists. It has been developed numerical simulation using full-cloud CAE simulation software. The study describes how to use rotating zones to run incompressible fluid flow simulation on a centrifugal pump component of turbo equipment. The complexity of this use case results from the requirement of modeling a rotating region. Simulations including spinning regions necessitate additional processes for meshing and modeling the flow, which are discussed in this current study. The main value of that study is that here the task like modeling of the flow behavior of any complex turbo machinery like centrifugal pump is extremely difficult and all this is overcome by clarifying incompressible flow field around Centrifugal pump. Another main advantage of this study is that the results obtained to clarify the flow behavior through the centrifugal pump are applied in the training of students in the disciplines of marine machinery and mechanisms and hydraulics. With the results obtained in this way, which are implemented in the teaching methodology, a visual example is given to the future specialists of ship mechanics by clarifying a practical problem in a complex turbomachinery element such as the centrifugal pump using the methods of modern computing programs.**

Keywords: centrifugal pump, flow simulation, CAE, education, training simulation

I. Introduction

According to source [2] centrifugal pumps are one of the most prevalent components found in fluid systems.

Understanding the head and flow relationships for a centrifugal pump is necessary to comprehend how a fluid system with a centrifugal pump works. A centrifugal pump, according to source [1,] is a mechanical device that moves a fluid by transferring rotational energy from one or more driven rotors known as impellers.

When fluid enters a centrifugal pump, it is instantly directed to the low-pressure area at the centre or eye of the impeller. As they rotate, the impeller and blading impart momentum to the incoming fluid. A momentum transfer to a moving fluid raises its velocity. A fluid's kinetic energy increases as its velocity increases. Fluid with high kinetic energy is driven out of the impeller area and into the volute.

The volute is a region with a rising cross-sectional area designed to transfer fluid kinetic energy into fluid pressure. This energy conversion mechanism is the same as that for subsonic flow through a nozzle's diverging portion. The general energy equation, the continuity equation, and the equation connecting the internal attributes of a system are used to mathematically analyse the flow through the volute. The important parameters influencing energy conversion are the expanding cross-sectional area of the volute, the increased system back pressure at the volute discharge, and the incompressible, subsonic flow of the fluid. The fluid flow in the volute is analogous to subsonic flow in a pipe because both characteristics are interconnected.

The impeller is the most critical component of a centrifugal pump, according to [1]. It consists of a series of curved vanes. These are often sandwiched between two discs (an enclosed impeller). For fluids containing entrained particulates, an open or semi-open impeller (supported by a single disc) is ideal. Fluid enters the impeller via its axis (the 'eye') and escapes via the

circumferential between the vanes. The impeller is attached to a motor through a drive shaft and turns rapidly on the opposite side of the eye (usually 500-5000rpm). The spinning motion of the impeller vanes accelerates the fluid out of the pump casing.

There are two types of pump casings: volute and diffuser. Both designs aim to transform fluid flow into a pressure-controlled discharge, as shown in [1].

In a volute casing, the impeller is offset, resulting in a curved funnel with increasing cross-sectional area approaching the pump output. Because of this design, the fluid pressure rises towards the outflow, as seen in Fig. 1, [1].
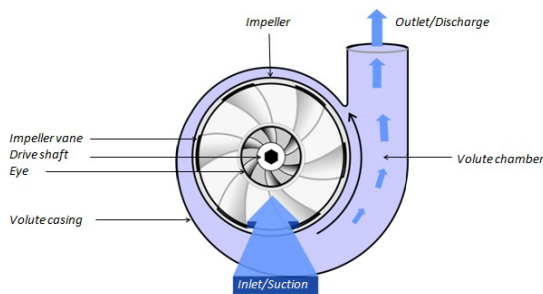

Fig. 1 Volute case design, [1]

In this study is suggesting the advanced fluid flow simulation of centrifugal pump that will be implementing in training of maritime specialists. We expect to obtain results to be a good visual example in the training of the future specialists of ship machinery and mechanisms to clarify a practical problem in a complex turbomachinery element such as the centrifugal pump using the methods of modern computing programs.

According to [2] a centrifugal pump typically produces a relatively low pressure increase in the fluid. Across a centrifugal pump with a single stage impeller, this pressure increase can range from a few dozen to several hundred psid. PSID (Pounds Force Per Square Inch Differential) is the same as P. It is the pressure difference between a pump's suction and discharge in this context. PSID can also refer to a pressure drop across a system component (strainers, filters, heat exchangers, valves, demineralizers, etc.). When a centrifugal pump is running at a constant speed, an increase in the system back pressure on the flowing stream reduces the magnitude of volumetric flow rate that the pump can maintain [2].

The relationship between the volumetric flow rate ($\dot{V}$) that a centrifugal pump can maintain and the pressure differential across the pump (Pump) is analyzed using various physical properties of the pump and the system fluid. The pump efficiency, the power supplied to the pump, the rotational speed, the diameter of the impeller and blading, the fluid density, and the fluid viscosity are all variables that design engineers consider when determining this relationship. The graph in Figure 7 in source [2] depicts the outcome of this complex analysis for a typical centrifugal pump operating at a single speed.

According to [2,] cavitation can be a significant problem for centrifugal pumps. Some pumps can be engineered to have very little cavitation. Most centrifugal

pumps are harmed by impeller erosion, vibration, or another cavitation-related problem and cannot survive extended cavitation. Cavitation can be avoided during pump operation by monitoring the pump's net positive suction head. The net positive suction head (NPSH) of a pump is the difference between the suction pressure and the saturation pressure of the fluid being pumped. [2] The NPSH value is used to calculate how close a fluid is to saturation. The net positive suction head available for a pump can be calculated using Equation 3-19 in source [2]. NPSH is measured in feet of water. Cavitation can be avoided by keeping the available NPSH higher than the NPSH required by the pump manufacturer, [2].

We can use source [1] to answer the question, "What are the limitations of a centrifugal pump?" According to [1,] the efficient operation of a centrifugal pump is dependent on the impeller's consistent, high-speed rotation. With high viscosity feeds, centrifugal pumps become progressively inefficient: there is greater resistance, and a higher pressure is required to maintain a specific flow rate. Centrifugal pumps are thus well adapted to low pressure, high-capacity pumping applications of liquids ranging in viscosity from 0.1 to 200 cP. Unlike a positive displacement pump, a centrifugal pump cannot create suction while it is dry; it must first be primed with the pushed fluid. Centrifugal pumps are consequently unsuitable for any application with an intermittent power supply. Furthermore, a centrifugal pump provides variable flow as the input pressure changes, whereas a positive displacement pump is insensitive to changing pressures and delivers a constant output. In applications requiring accurate dosage, a positive displacement pump is therefore chosen [1].

The applications of centrifugal pumps are presented in tabular form in source [1]. According to [1,] centrifugal pumps are commonly used to pump water, solvents, organics, oils, acids, bases, and any 'thin' liquid in industrial, agricultural, and home applications. In fact, a centrifugal pump design exists that is suitable for practically any application requiring low viscosity fluids.

II. CENTRIFUGAL PUMP- PROBLEM SPECIFICATION

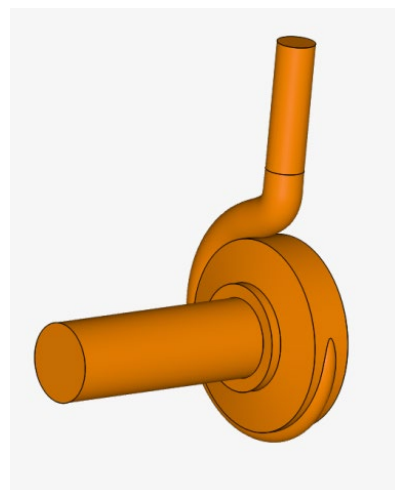Figure 2 shows the imported centrifugal pump into Sim Scale.

Fig. 2 Geometry of centrifugal pump

331

The study deals with conducting incompressible fluid flow simulation through centrifugal pump using rotating zone. The geometry presented in Fig. 2 contains the flow region, as well as a volume representing the rotating zone. The steps to make the original geometry CFD ready are given at source [5], [6].

After importing the geometry for this study, it is ready for CFD simulation using Sim Scale.

A. Numerical model of Centrifugal pump

The developed numerical model of centrifugal pump under this study is based on the relevant models of Sim Scale web, [3].

B. Steady state incompressible fluid flow simulation

Citing [3] k-omega SST turbulence model is commonly used in turbo machinery applications. Within the pump blades, the flow experiences separation, which is effectively captured by k-omega SST turbulence model.

C. Assigning the material and boundary conditions.

Using [3] Fig. 3 shows an overview of the boundary conditions.

Citing [3] velocity inlet and pressure outlet is a very common combination used in CFD simulations as it often results in good stability. This combination permits flow to adjust aiming to assure mass continuity.
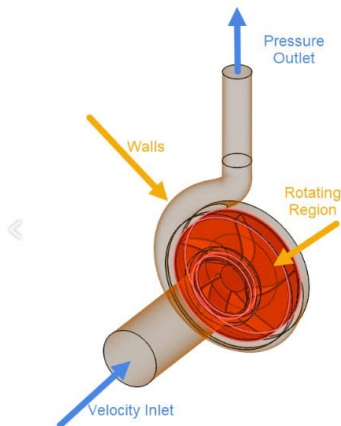

Fig. 3 Overview of the boundary conditions for centrifugal pump simulation, [3].

In the study the performed CFD simulation uses water as a material. After making the proper settings for the velocity inlet, a volumetric flow rate of '8.5e-3 m^3/s' enters the domain through the inlet. In the pressure outlet we must be sure that the gauge pressure is set to 'Mean value=0'. Next boundary condition that is assigned is wall, and all solid walls should receive a no-slip condition.

After assigning all boundary conditions it has been created a rotating zone. The rotating zone was created according to x axis. All entities within this rotating zone are rotating at 350 rad/s. According to [3] MRF Rotating zones are chosen in this simulation because we are running a steady-state simulation. If we were calculating a transient problem, we would need to choose AMI Rotating Zone.

D. Assigning the material and boundary conditions.

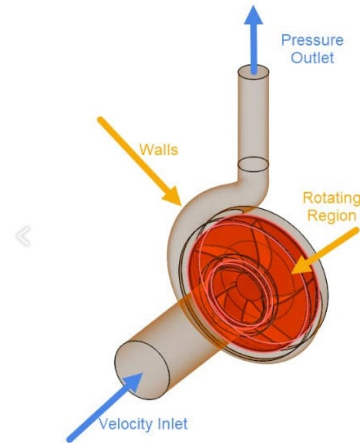Using [3] Fig. 3 shows an overview of the boundary conditions.

Fig. 3 Overview of the boundary conditions for centrifugal pump

simulation, [3].

Citing [3] velocity inlet and pressure outlet is a very common combination used in CFD simulations as it often results in good stability. This combination permits flow to adjust aiming to assure mass continuity.

In the study the performed CFD simulation uses water as a material. After making the proper settings for the velocity inlet, a volumetric flow rate of '8.5e-3 m^3/s' enters the domain through the inlet. In the pressure outlet we must be sure that the gauge pressure is set to 'Mean value=0'. Next boundary condition that is assigned is wall, and all solid walls should receive a no-slip condition.

After assigning all boundary conditions it has been created a rotating zone. The rotating zone was created according to x axis. All entities within this rotating zone are rotating at 350 rad/s. According to [3] MRF Rotating zones are chosen in this simulation because we are running a steady-state simulation. If we were calculating a transient problem, we would need to choose AMI Rotating Zone.

E. Numerical and simulation control.

Using the algorithm [3] under the Simulation control setting it must be chosen 'Potential flow initialization' which enhances the stability for velocity-driven flows, especially in early iterations. The maximum runtime was 30000 seconds, and the writing interval is 600 iterations.

F. Data control

Citing [3] outcome control allows you to see the convergence behavior during the calculation process at specified points in the model. As a result, it is a crucial indication for assessing the quality and dependability of the outcomes.
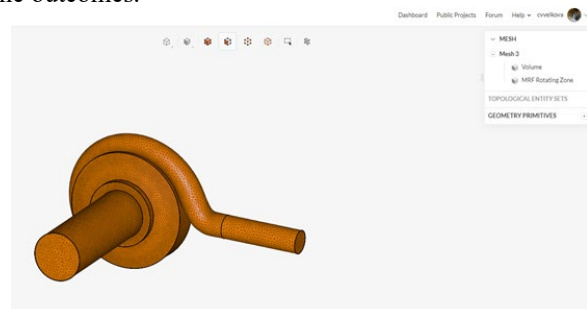

Fig. 4 View of mesh geometry around Centrifugal pump.

In this investigation, the global simulation settings were left at their default, while local refinement was applied to the regions of interest. As stated in [3,] defining cell zones is required if we want to apply a specific property, such as a rotating motion, to a subset of cells. When Physics-based meshing is enabled, the conventional Mesher automatically produces the required cell zones. Because physics-based meshing is used in this case study, the algorithm will handle cell zone definition.

For this study, it was made mesh refinement. Citing [3] the maximum edge length should be '7e-4' meters, applied to the Local element size refinement topological entity set. With this refinement we have better control over the cell size in the regions of interest.

The 'Force and Moments' control to the impeller was chosen for the simulation under this study. In addition, an 'Area average' for the inlet and another for the outlet were produced.

G. Mesh

According to [3,] the Standard algorithm is recommended for creating the mesh, which is a solid choice in general because it is quite automated and produces decent results for most complex geometries, such as the centrifugal pump in our instance. Figure 4 depicts the geometry of a centrifugal pump with a produced mesh.

Citing [3] MRF rotating zones are chosen in this simulation because we are running a steady-state simulation. If we are calculating a transient problem, we would need to choose AMI Rotating zone.

III. RESULTS AND ANALYSIS

Convergence is crucial for the fluid flow simulation. As the iterations go on, key parameters are expected to stop changing. At this point, the simulation is considered converged. For the flow simulation through centrifugal pump the key parameters for convergence that must be monitored are the velocity at the outlet, the pressure at the inlet, and forces onto the impeller. Fig. 5 shows velocity at the outlet.
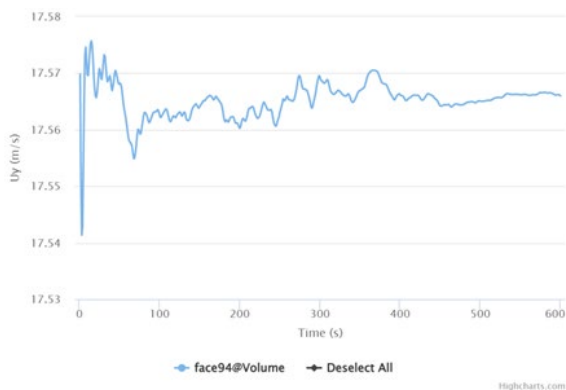

Fig. 5 Velocity at the outlet control.

From Fig. 5 we can notice that after 600 iterations the results become stable. Also, we must make sure to check the convergence plots to see residual levels. Smaller residuals indicate a more tightly converged solution.

Citing [3] Sim Scale has a built-in post-processing environment, which can be accessed by clicking on 'Solution Fields' or 'Post-process results. First, the simulation gives the opportunity to visualize the pressure. Fig. 6 shows the pressure visualization on the blades, and the bottom.
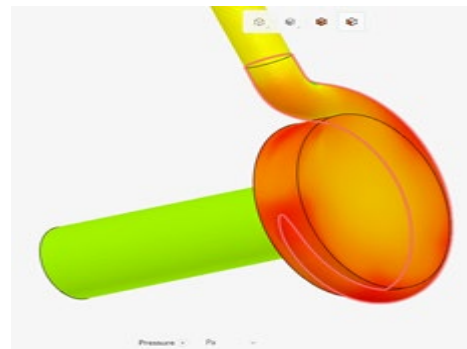

Fig. 6 The pressure visualization on the impeller from centrifugal pump.

The surface of the impeller reveals a high distribution on the edge towards the exit of the pump, near the exit of the fluid.

To get a better understanding of what is going on inside the pump, as a result it can be proceeded to add a 'Cutting Plane' filter. Fig. 7 shows the cutting plane normal to X-axis that shows the behavior of the flow when exiting the model.
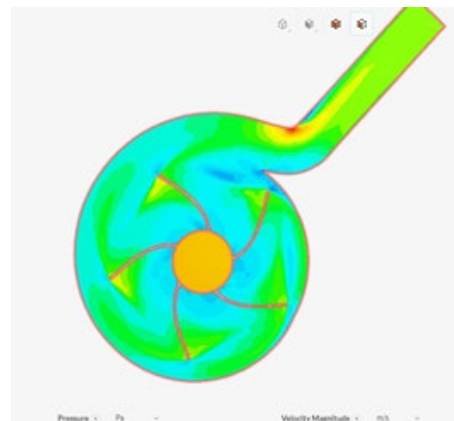

Fig. 7 Cutting plane filter for velocity magnitude

Fig. 8 shows the velocity contours with velocity vectors plotted. It can be observed that in areas where the fluid accelerates, such as the tips of the blades, and the entrance to the outlet, the vectors are also enlarged.

From Fig. 8 we can conclude that near the edges of the blades, the acceleration of the flow can be distinguished due to the representation with a warmer color compared to its surroundings.

To obtain the additional information we can look at other configurations for the cutting plane, for instance, we can change the orientation to Y axis as is shown in Fig. 9.

With the configuration given in Fig. 9 you can observe the flow pattern around the blades from a different perspective.
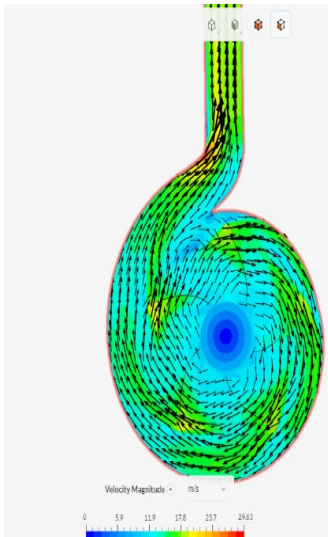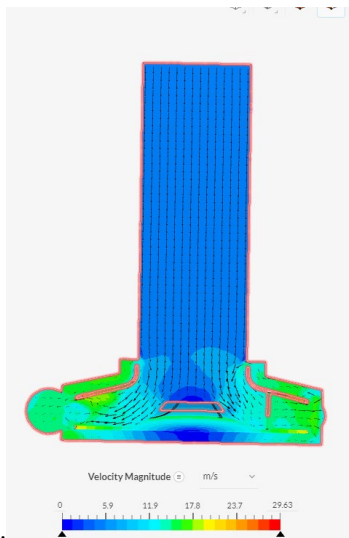
Fig. 8 Velocity contours with velocity vectors.

Fig. 9 Velocity vectors to Y axis

IV. Conclusions

Centrifugal pump was performed for the purpose of education of maritime students. All calculations were made with CAE software Sim Scale. It was used $k-\varepsilon$ turbulence model for flow simulation around Centrifugal pump. The flow domain has been modeled, meshed, and solved, and post-processing visualization options have been used for better understanding, investigation, and evaluation of the flow field around the pump.

The analysis is used to investigate the flow behavior around Centrifugal pump and to be applied as a tool part of educational training of the students. Pressure distribution and velocity magnitude are displayed. Using the analysis, it is getting velocity at the outlet and pressure of the pump.

The results of this study's efforts to clarify the behaviour of the flow via the centrifugal pump are used to instruct students in the fields of hydraulics and marine machinery and mechanisms. This is another major benefit of the study. By using the outcomes of this research, which are incorporated into the teaching methodology, future ship mechanic specialists are given a visual example by clarifying a practical issue in a complex turbomachinery component like the centrifugal pump using the techniques of contemporary computing programs.

Furthermore, this investigation is a guideline for obtaining quick and correct results for flow filed around Centrifugal pump.

Furthermore, the validity of the obtained results is supported by the fact that the study was carried out according to a well-established methodology approach, [3] that predicts certain flow characteristics around the centrifugal pump. The study's results match those expected from the used approach. Such conformation strengthens the reliability of the analysis and makes it suitable for educational purposes. It provides students of hydraulics and marine machinery and mechanisms with a real example for understanding complex flows into the turbomachinery.

REFERENCES

[1] Michael Smith Engineers Ltd. (2023). Useful information on centrifugal pumps. Retrieved from https://www.michael-smith-engineers.co.uk/resources/useful-info/centrifugal-pumps

[2] DOE Fundamentals Handbook:Thermodynamics, Heat Transfer, and Fluid Flow.DOE-HDBK-1012/3-92,U.S. Department of Energy, June 1992,https://engineeringlibrary.org/reference/centrifugal-pumps-fluid-flow-doe-handbook

[3] SimScale. (2022, August 30). Advanced Tutorial: Fluid Flow Simulation through a Centrifugal Pump. B SimScale Docs.
URL: https://www.simscale.com/docs/tutorials/

[4] Rana, M. S., Upadhyay, S. H., Harsha, S. P. (2014). "CFD Analysis of Centrifugal Pump: A Review." International Journal of Research in Advent Technology, 2(7), July 2014.

[5] Harikrishnan, T., Sankaranarayanasamy, K. (2014). "Numerical Investigation of the Performance Characteristics of a Centrifugal Pump Using CFD." International Journal of Mechanical Engineering and Robotics Research, 3(3), July 2014.

[6] Gautam, M., Deshpande, S. M., Rana, M. S. (2013). "Computational Fluid Dynamics Analysis of a Centrifugal Pump." International Journal of Engineering and Innovative Technology, 3(2), August 2013.

[7] Lobanoff, V. S., Ross, R. R. (1992). Centrifugal Pumps: Design and Application.

[8] Tu, J., Yeoh, G. H., Liu, C.,Butterworth-Heinemann, 2012 г. Computational Fluid Dynamics: A Practical Approach.

[9] Durran, D. R., Springer (2010). Numerical Methods for Fluid Dynamics: With Applications to Geophysics.