MODELING OF WATER FLOW FOR WATERJET PROPULSION SYSTEM

Marina Cerpinska, Martins Irbe, Andrejs Pupurs, Kaspars Burbeckis
Riga Technical University, Latvia
marina.cerpinska@rtu.lv, martins.irbe@rtu.lv, andrejs.pupurs@rtu.lv,
kaspars.burbeckis@gmail.com

Abstract. The paper provides simulation results for water flow ratio in the waterjet attached to the SUP (Stand Up Paddle) board fin. The goal of the project is to attach the waterjet propulsion to the SUP board with maximum efficiency. To do it, the model of the person driving the board and waterjet was created in SolidWorks. In this paper the water flow rate depending on different nozzle and duct design was analysed. There are two conflicting requirements for the propulsion duct design. On the one hand, the duct must protect the propulsor from stones and water plants, because SUP boards travel on rivers, sometimes very close to stones. On the other hand, the efficiency of the waterjet requires maximum water flow rate available for a possibly greater Inlet Velocity Ratio. Finally, there should be enough place for the battery pack, which drives the waterjet, but it creates an additional barrier for the water flow. The flow rate at the outlet depends on the nozzle and is also important for the propulsor efficiency. Variable design options for the waterjet duct were proposed and tested in SolidWorks Flow simulation environment. At the beginning of the project there was an intention to create different ducts via 3D printing. The simulation showed that modification of the inlet duct results in minor (only 0.01%) increase of the flow rate, if the front panel remains closed due to the oblique stagnation point flow phenomenon, and 3D printing resources should not be wasted on this test.

Keywords: waterjet; SolidWorks Flow simulation; duct; oblique stagnation point flow.

Introduction

This study focuses on simulation of the flow for the waterjet duct attached to the SUP fin. The need for the study arose, because there were not enough data available from previously published simulations. The original design of the waterjet allowed only minor modifications on the waterjet duct wall, which has a cylinder shape. During literature review we learned that modelling of flow over bluff bodies including cylinders is common, because it allows the user selecting the best operation mode, avoiding unsafe modes by eliminating the vortex shedding effect [1]. Extensive research is available on bluff body wakes [2] and some of the research was done before by our colleagues [3]. Meanwhile, modelling flow through the cylinder with cuts or holes is developing [4]. The study available on the flow around perforated cylinders focuses on the control of vortical structures [5] and did not cover all modifications we were looking for yet. Thus, simulation was required to compare the different inlet design for the waterjet. Our previous research focused on the drag force experienced by the frontal area of the waterjet, and we concluded that at the frontal area the drag force is felt most intensively [6]. This lets us think that opening a small section at the front for the water inlet could increase the volume flow rate for the waterjet.

The highly complex geometries require detailed Computational fluid dynamics (CFD) analysis and should be preferably verified with the site measurement system, which is currently under development in research centres [7]. During the CFD analysis around the cylinder the following aspects must be considered: boundary layer separations, vortex structures and the effect of surface roughness [8]. Meanwhile, for the flow simulation inside the duct the boundary layer is still important because viscous forces and rotationality allow less fluid to enter the duct, but it is even more important to predict the flow field, plot velocity and pressure along the duct and rotation axis of the jet propeller, the location and value of the maximum velocity along the duct and predict which cuts on the duct would make the designed waterjet most effective.

Materials and methods

Analysis of the flow was done focusing on the inlet duct with differently shaped grooves as presented in Fig. 1. To intentionally increase the flow rate to the waterjet the following inlet designs for the duct pipe were proposed and tested via SolidWorks simulation tool. SolidWorks Flow simulation tool calculates the flow characteristics in the computational domain based on the solution of the Navier–Stokes equations [9] and compared to other existing simulation tools like OpenFOAM provides similar results [10]. For this study we used steady, incompressible and irrotational flow.

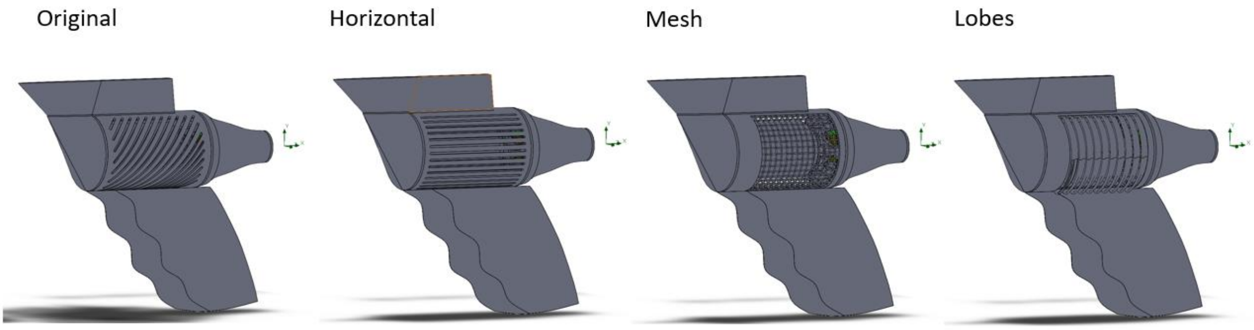

Fig. 1. **Inlet geometry configurations:** original; horizontal; mesh; lobes

An example of the experimental data plot can be found in Fig. 4.

We used the basic mesh for the model and adaptive mesh [11] with smaller cells for the critical areas, specifically improving the mesh at the holes.

The propulsion power of the waterjet P_T depends on the mass flow rate, therefore the volume flow rate and mass flow rate were set as simulation Goals (Global Goals) in SolidWorks Flow simulation environment. From Steady Flow Equations [6] calculating the flow rate Q:

$$v_A = \frac{Q}{A_A} \Rightarrow Q = v_A A_A, \tag{1}$$

where v_A – velocity at the specific inlet or outlet, m·s^{-1};
Q – flow rate, m^3·s^{-1};
A_A – area of the inlet or the outlet, m.

The propulsion power of the waterjet P_T depends on the mass flow rate as shown in the following equation (2) [12]:

$$P_T = TV_S = \dot{m}V_S\left(V_2 - V_1\right), \tag{2}$$

where T – thrust produced by the system;
V_S – speed of the vessel, m·s^{-1};
$\dot{m}$ – mass flow rate of the water $\rho A_2 V_2$;
V_1 – velocity of water that enters the system, m·s^{-1};
V_2 – velocity of water that leaves the system, m·s^{-1}.

Results and discussion

The simulation results showed that pressure was growing as the water approached the wall of the duct, highlighted with the yellow frame in Fig. 2, which does not provide information about the velocity distribution yet. Based on an irrotational flow approximation [9], the maximum speed was expected close to the duct original cuts, and the stagnation point was expected to occur directly before the fin, as presented in Fig.4. In this simulation we cannot see the effect of the boundary layer growth, but it is understood that there will be some change in the velocity profile, when water enters the cut due to the displacement thickness of the boundary layer.

Fig. 4 explains why the waterjet could not be designed with the wall at the front. We may think of a waterjet as a ducted propeller [12], and also as of a pump, because it is supposed to add energy to the fluid and to the system [9], in our task the waterjet is supposed to add velocity to the SUP board. Since pumps may only add energy to the fluid supplied, if the inlet flow is very weak, the energy added will not be significant, and this is demonstrated in Fig. 5.

When the inlet duct is designed with original grooves, the volume flow rate from the simulation is as shown in Fig.4, where the propulsion power of the waterjet $P_T = \dot{m}V_S(V_2 - V_1)$ decreased sharply as the flow meets the rudder and front panel of the jet. This conclusion agrees with our previous research, which showed that the front panel of the duct creates additional drag for the fin [6].

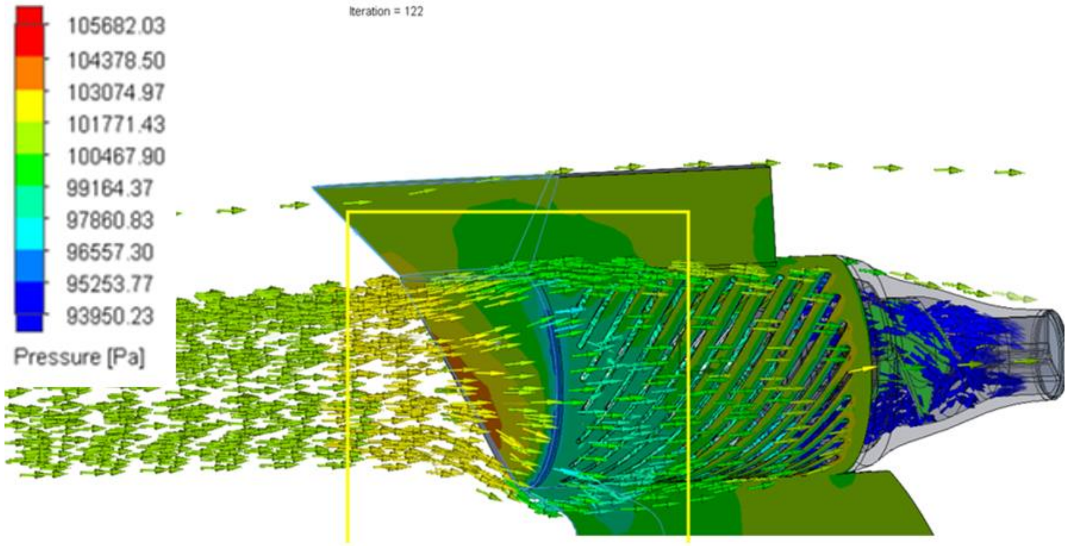

Fig. 2. **Pressure simulation example**

Since the pressure was the greatest at the front of the duct, the design with the cut at the front was proposed and tested as shown in Fig. 3.

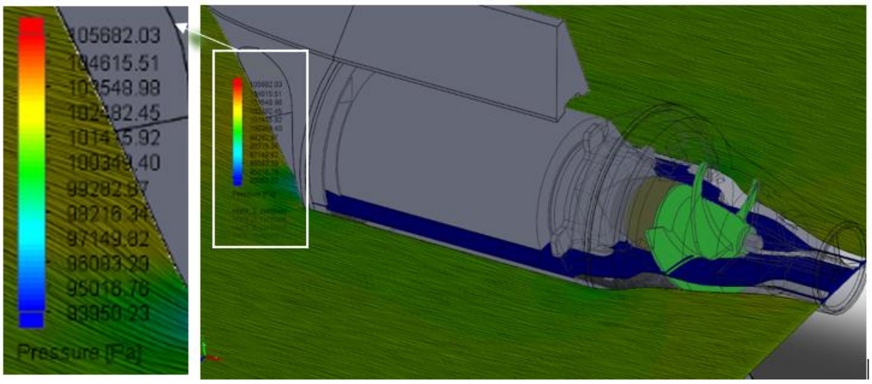

Fig. 3. **Inlet geometry configurations with cut at the front**

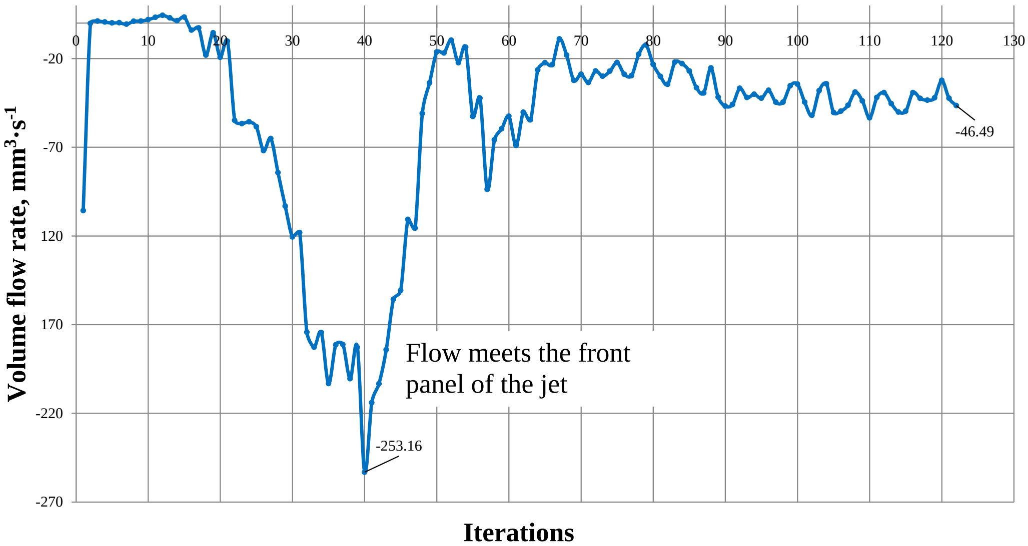

Fig. 4. **Volume Flow Rate from SolidWorks Simulation for the original design and stagnation point**

The modified design allows increasing the flow rate by 0.0006 kg·s^{-1} only, and the velocity at the outlet varies for 0.1 m·s^{-1}, therefore intentions to increase the flow through optimized duct design were not successful. As a result of the given simulations resources were saved on manufacturing different models. The results showed that there is no need to manufacture different duct models.

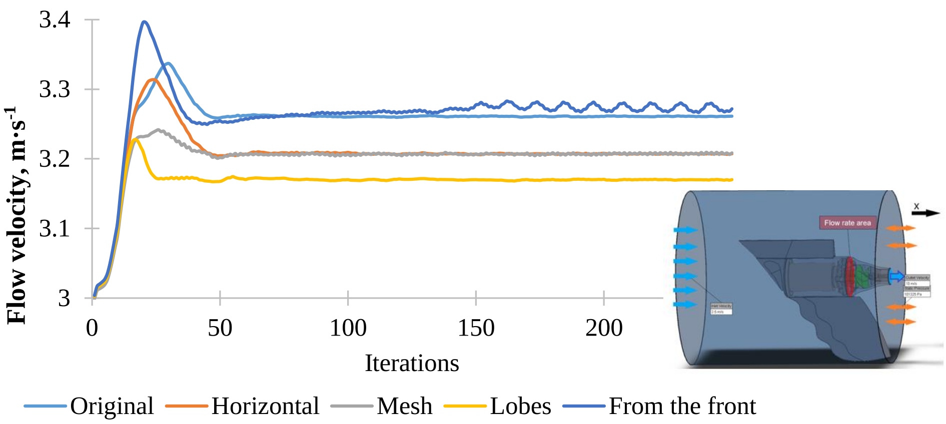

Fig. 5. **Flow velocity from SolidWorks Simulation for modified designs**

The greater the inlet area, the greater the flow rate. However, modification of the inlet duct resulted in minor increase of the flow rate. Following the ideas of competitors, who work with similar projects and general waterjet design, the flow rate might be increased more by placing the impeller in front of the fin [13] or below the battery pack. Furthermore, the ideas from design of the rocket engines with converging–diverging nozzles could be used to increase the thrust [9], meaning that instead of the cylinder with the closed front converging nozzle should be used.

It is understood that calculated and measured flow rates could vary by 15% to 25% [7]. This suggests that the actual flow rate for the design with inlet of the flow may be greater than the simulated value. Based on the simulation results, it appears that there is an oblique stagnation point flow at the wall of the duct. This phenomenon typically occurs when a fluid meets the surface of the wall obliquely [14]. For the waterjet duct design the oblique stagnation point flow made options proposed during the project (modifications of cuts in the duct) inefficient.

Conclusions

1. The equations of propulsion suggest that the greater the inlet area, the greater the flow rate. However, modification of the inlet duct resulted in minor (0.01%) increase of the flow rate, if the front panel remains closed due to the oblique stagnation point flow.
2. When the front panel cannot not be removed, the flow rate might be increased more by placing the impeller in front of the fin or below the battery pack.
3. As a result of the given simulations material resources were saved on manufacturing different models. The results showed that there is no need to manufacture different duct models.

Acknowledgements

The research has been performed within the research project "Development of technical solutions for water transport transmission system", grant number ZI-2020/3, Framework of Science and Innovation within research platforms at the Riga Technical university, funded by the Riga Technical university.

References

[1] Yoon H. S., Nam S. H., Kim M. Effect of the geometric features of the harbor seal vibrissa based biomimetic cylinder on the flow over a cylinder. Ocean Engineering, vol. 218, 2020, pp. 108-150. DOI: 10.1016/j.oceaneng.2020

[2] Derakhshandeh J.F., Mahbub Alam Md. A review of bluff body wakes. Ocean Engineering, vol. 182, 2019, pp. 475-488. DOI: 10.1016/j.oceaneng.2019.04.093

[3] Vutukuru S. K., Tipans I., Viba J., Irbe M. Form optimization and interaction analysis of plane symmetry prism in air. Proceedings of 19th International Scientific Conference “Engineering for Rural Development”, May 20-22, 2020, Jelgava, Latvia, pp. 739-746. DOI: 10.22616/ERDev2020.19.TF170
[4] Firat E., Ozkan G.M., Akilli H. Flow past a hollow cylinder with two spanwise rows of holes. Experiments in Fluids, vol. 60, 2019, article nr.163. DOI: 10.1007·s^{-1}00348-019-2814-2
[5] Pinar E. et al. Flow structure around perforated cylinders in shallow water. Journal of Fluids and Structures, vol. 55, 2015, pp. 52-63, DOI: 10.1016/j.jfluidstructs.2015.01.017
[6] Cerpinska M., Irbe M., Pupurs A., Burbeckis K. Modeling of drag force reduction for a waterjet propulsion system. Latvian journal of physics and technical sciences. Accepted for publication in 2020.
[7] Kotzé R., Wiklund J., Haldenwang R., Fester V. Measurement and analysis of flow behaviour in complex geometries using the Ultrasonic Velocity Profiling (UVP) technique. Flow Measurement and Instrumentation, vol. 22, 2011, pp. 110-119. DOI: 10.1016/j.flowmeasinst.2010.12.010
[8] Noureddine S., Mahfoudh C., Smail K., Toufik B. Numerical investigation of the surface roughness effects on the subsonic flow around a circular cone-cylinder. Aerospace Science and Technology, vol. 107, 2020, pp. 106271. DOI: 10.1016/j.ast.2020.106271
[9] Cenqel Y. A., Cimbala J. M. Fluid Mechanics: Fundamentals and Applications. 4th ed. New York: McGraw-Hill Education. 2017. 1056 p.
[10] Synylo K., Krupko A., Zaporozhets O., Makarenko R. CFD simulation of exhaust gases jet from aircraft engine. Energy, vol. 213, 2020, pp. 118610. DOI: 10.1016/j.energy.2020.118610
[11] Hawk Ridge Systems Engineering team. Adaptive meshing in Solidworks simulation. August 2, 2013. [online] [18.03.2021]. Available at: https://hawkridgesys.com/blog/simulation-adaptive-meshing
[12] Carlton J. Marine propellers and propulsion. USA: Butterworth-Heinemann, 2018. 544 p. DOI: 10.1016/B978-0-08-100366-4.00016-X
[13] Hibbeler, R. C., Fan, S. C. Engineering Mechanics: Dynamics. 12th edition. New York: Prentice Hall. 2009. 752 p.
[14] Sarkar S., Sahoo B. Analysis of oblique stagnation point flow over a rough surface. Journal of Mathematical Analysis and Applications, vol. 490 (1), 2020, pp. 124208 DOI: 10.1016/j.jmaa.2020.124208