IOP Conf. Series: Materials Science and Engineering **400** (2018) 082018 doi:10.1088/1757-899X/400/8/082018

New innovative backflow marine propeller optimisation study by CFD

LC Stan[1], I Calimanescu[2] and V Popa[3]
[1,3]Faculty of Naval Electromechanics, Department Electromechanics, Constanta Maritime University, 104 Mircea cel Batran Blvd, 900663, Constanta, Romania
[2]Nuclear Power Plant, Cernavoda, Romania

E-mail: liviustan14@yahoo.com

Abstract. There were many solutions to increase the marine propellers efficiency, like Propeller Boss Cap Fins, Tip Fins, Surface Piercing Propellers, Pre and Post-Swirl Devices, Ducted Propulsors etc. All of them are supposing sometimes intricate devices involving high implementation costs and/or maintenance.
The present paper is proposing a new innovative Backflow Propeller where the core element is the backflow shield (or screen) which is inbuilt in the phase of fabrication stage of the propeller claiming zero maintenance costs.
This paper is coming as a logical continuation of the article where the Author demonstrated the viability of the proposed solution with a simulation very near to the real case.
It is visible here that the suction pressure on the blade back is diminished, the whirlpools formed on the blade back are diverted, the jets on the backflow shield are there and working. All these are positive effects increasing the propeller efficiency. On the other hand, the leading edge of the propeller is increased, the opposing component of the jet is there and is taking its toll on the efficiency of the propeller. All these conclusions still are in need of experimental data.

1. Introduction

One of the targets of any ship Owner is, nowadays, to decrease the operation costs. One solution is to use of so-called power saving devices like Propeller Boss Cap Fins, Tip Fins, Surface Piercing Propellers, Pre and Post-Swirl Devices, Ducted Propulsors etc.; these are stationary devices positioned near the propeller that improve the overall propulsion efficiency.

This paper introduces a novel approach, a Backflow Propeller. This power-saving device consists of a classical propeller to which was added a backflow screen with jet holes on the aft side as seen in the figure 1.

The device essentially reduces the rotational losses in the resulting propeller suction pressure on the blade back and increases the efficiency by creating a backflow jet.

There were many solutions to increase the marine propellers efficiency, like Propeller Boss Cap Fins, Tip Fins, Surface Piercing Propellers, Pre and Post-Swirl Devices, Ducted Propulsors etc. All of them are supposing sometimes intricate devices involving high implementation costs and/or maintenance.

The present paper is proposing a new innovative backflow propeller where the core element is the backflow shield (or screen) which is inbuilt in the phase of fabrication stage of the propeller claiming zero maintenance costs.

The model is simplified as that used in the article [1] but is still meaningful and comprehensive. Whether one can figure out a certain device to diminish the suction pressure, then the propeller efficiency is, by all means, improved.

Such a device is described in the following.

2. Materials and methods

2.1. Invention description

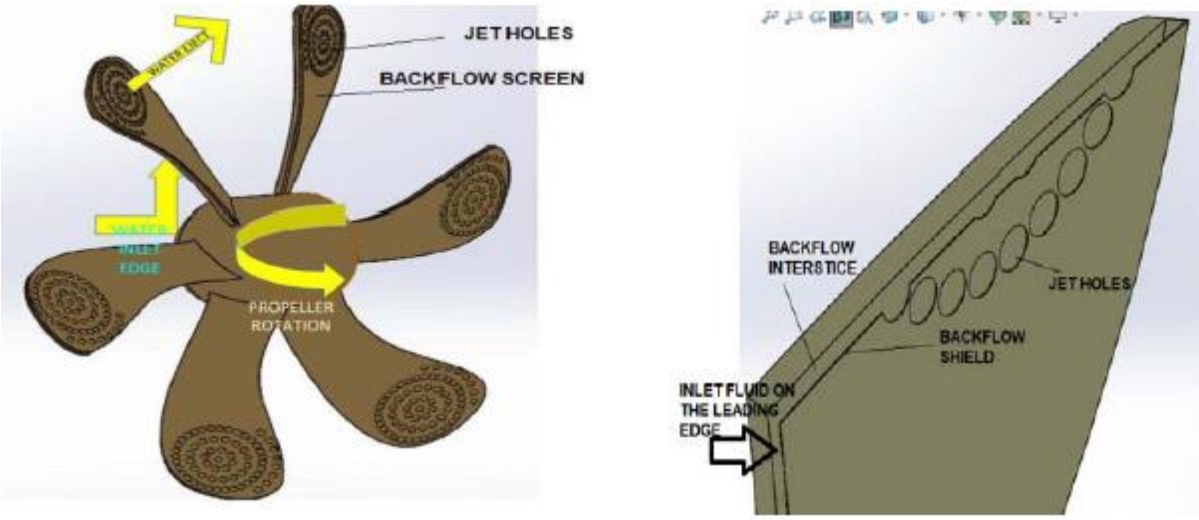


Figure 1. The backflow propeller.

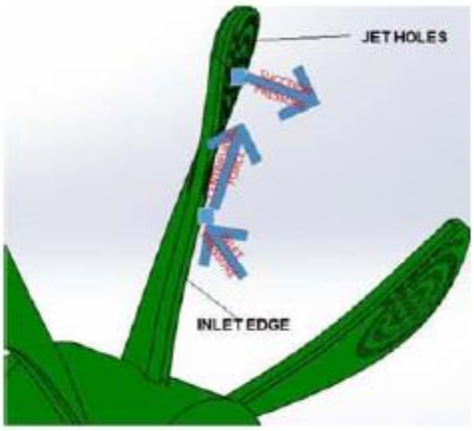


Figure 2. Forces acting on a fluid particle.

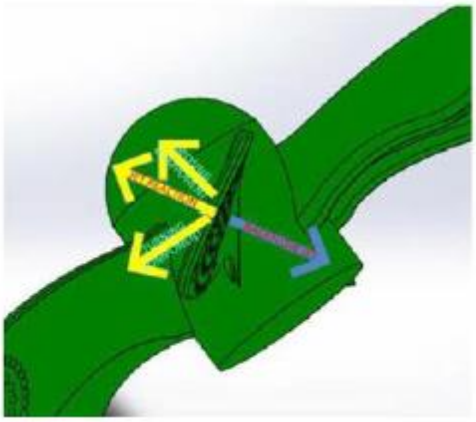


Figure 3. Jet reaction components.

Since it was already demonstrated that the suction pressure exerted on the aft side of the propeller blade is the main contributor in decreasing the propeller efficiency, the new proposed backflow propeller is simply a classical propeller to which was added a backflow screen (or shield) with jet holes on the aft side as seen in the figure 1. The backflow screen (or shield) is doubling the aft face of the blade leaving in between a backflow interstice. The backflow shield is closed in all directions leaving just one opening on the leading edge for the incoming fluid pushed here by the propeller rotation. In very plain words, the backflow interstice/screen (shield) is collecting the fluid from the leading-edge due to the rotation motion, and is directing it toward the jet holes in order to obtain the backflow jet.

The inlet edge of the backflow screen is placed as to allow a small quantity of fluid to enter inside the space between the aft side of the propeller and the backflow screen (backflow interstice). Once there, the fluid particle is subjected to a series of forces which will push the particle to form a jet inside the jet holes.

On the inlet edge of the backflow screen the rotation motion of the propeller if forcing the fluid particle to develop a certain pressure. There the kinetic pressure is translated into a static pressure. The fluid particle is then forced to follow the blade rotation hence a centrifugal force will be developed here to push further the particle toward the jet holes (figure 2).

Once arrived in the jet holes' region the suction pressure naturally existing there will suck the particle to form a backflow jet. This fluid jet is diverting the whirlpools mainly responsible for the suction pressure and the suction pressure itself is fed decreasing its intensity and increasing the propeller efficiency.

The backflow jet will develop a jet reaction force as seen in the figure 3, with two components: the Opposing Component, which will tend to oppose to the ship motion and the Turning Component, which will help the propeller to rotate.

In theory, the jet holes can be placed wherever the designer may deem appropriate on the backflow shield. The placement of the jet holes on the tip of the blade is following two rationales: firstly, the tip of the blade is fostering the biggest suction pressure as we'll see in the followings, and, secondly, the bigger is the force arm between the propeller axis and the jet holes' region, the bigger the turning component moment will be [2].

2.2. The CAD Models of the classical and innovative propellers

In this paper, we will develop two series of numerical experiments: firstly, the classical propeller Computer Fluid Dynamics analysis as opposed to the new innovative backflow propeller. In order to have meaningful results, the two propellers have the same dimensions and geometry and the Finite Volumes Elements Analysis (FVEA) involving Ansys 16 CFX, will have exactly the same parameters [3].

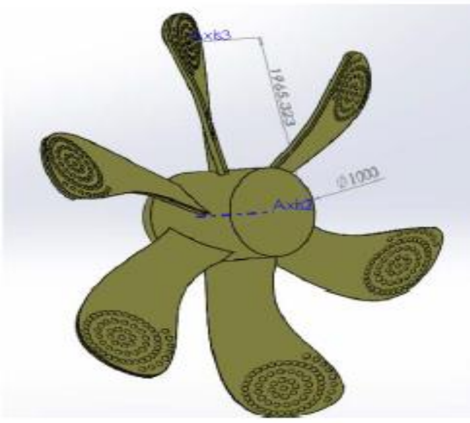


Figure 4. The innovative backflow propeller.

In the figure 4, it is to be seen the innovative backflow propeller with six blades and the hub diameter of 1000 mm. The central jet hole axis is placed at 1965 mm from the propeller axis. The propeller was designed using SolidWorks 2016.

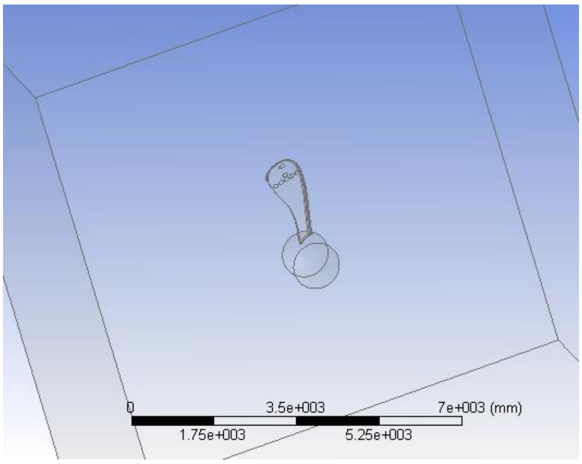


Figure 5. The fluid domain of the simulation.

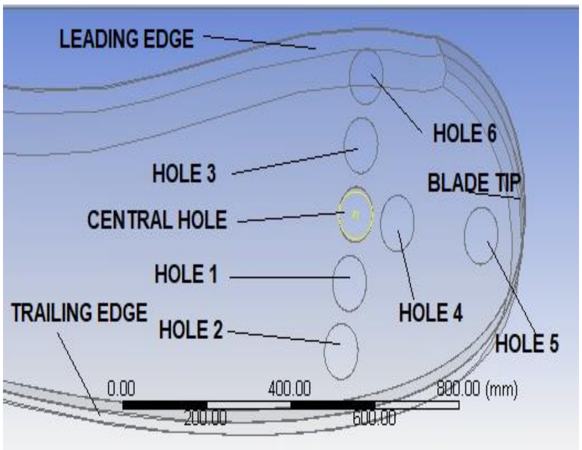


Figure 6. The jet holes assignment for the optimisation process.

The fluid domain as exported inside Ansys 16 CFX is comprising one single blade over which the fluid is passing in a linear direction as can be noticed as follow. This is the first simplification step: the fluid domain is no longer rotating around the propeller and this is deemed to disregard the centrifugal forces acting upon a fluid particle. The propeller blade is evolving in the manner of a flying hydrofoil.

Since one of the main targets of this study is to establish the influence of the jet holes' position and dimension, there were established 7 jet holes on the backflow shield with the diameter of 80 mm each. The diameter of this holes will be variated with ±20% in the optimisation process and response surfaces will be drawn measuring each hole influence over the pressure of the backflow shield. Hole 6 is the nearest to the leading edge whereas Hole 2 is nearest to the trailing edge. Hole 6 is placed near the blade tip.

2.3. The FVEA Model

The FVEA model was developed inside Ansys 16 CFX by importing the 3D model inside the Design Modeller.

The meshing is made out of finite volume elements. The meshing comprises 191033 finite volume elements with 35098 nodes.

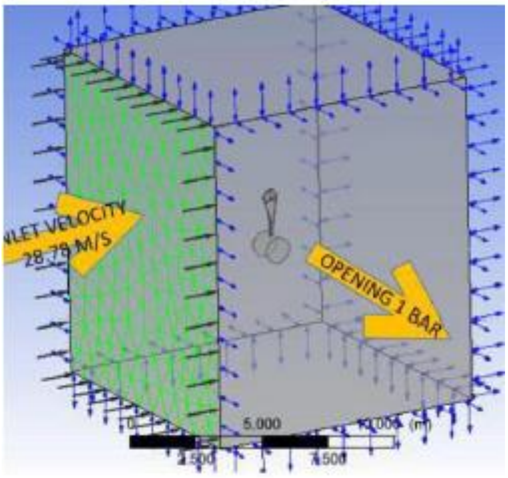


Figure 7. The boundary conditions.

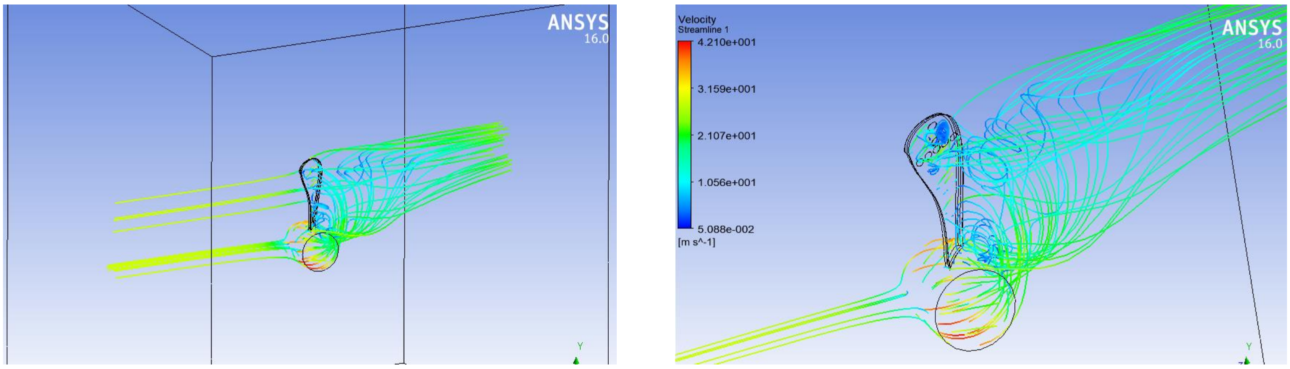
The boundary conditions imposed to the model are the Inlet fluid velocity 28.78 m/s to mimic the propeller rotational motion and the Opening boundary with relative pressure 1 bar as seen in figure 7.

The fluid is water with the well-known properties.

The turbulence will be modelled with the classical k-epsilon model.

3. Results and discussion

3.1. The Streamlines

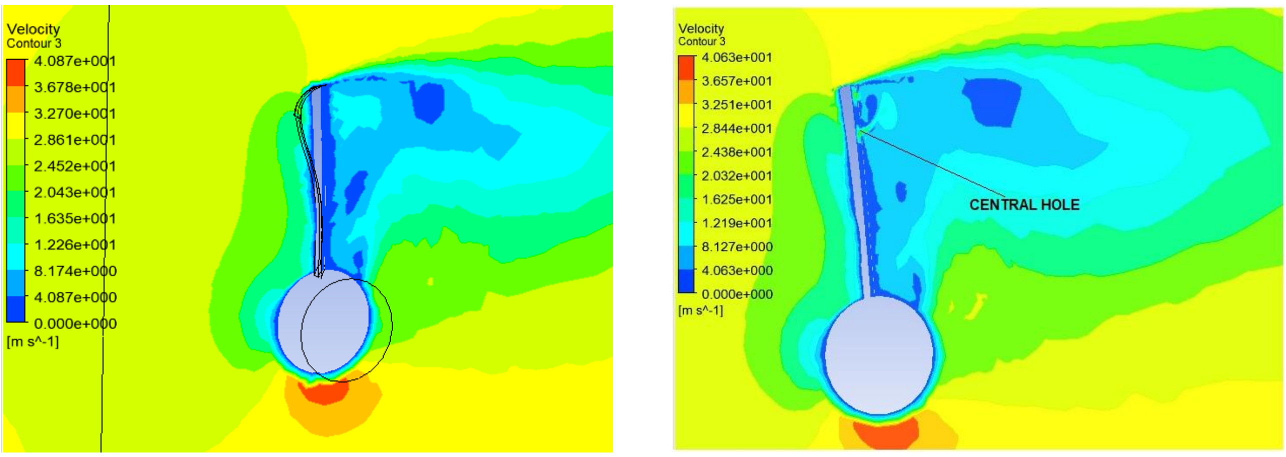


a-Normal propeller b-Backflow propeller

Figure 8. Streamlines for the two models.

By analysing the streamlines for the two models it is obvious that the Jet Holes are diverting the streamlines in an oblique direction as seen in figure 8. The first conclusion is that the backflow shield is working since the stream lines are affected by the presence of jet holes on the backflow shield.

3.2. The velocities fields on a vertical plane (transversal through the propeller axis)



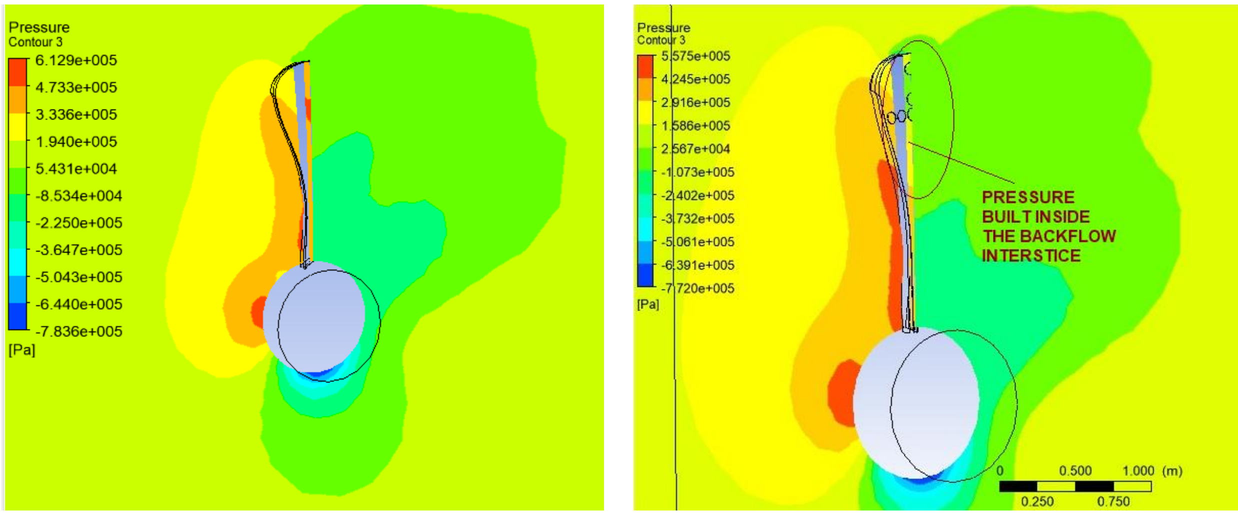
a-Normal propeller b-Backflow propeller

Figure 9. Velocities fields in vertical plane for the two models.

The velocities calculated for the normal propeller are reaching a maximum of 40 m/s at the inferior side of the hub as seen in the figure 9. The backflow once again is seen as working and the backflow jet from the central hole is quite visible with a calculated value of 15 m/s.

3.3. The pressures fields on a vertical plane (transversal through the propeller axis)

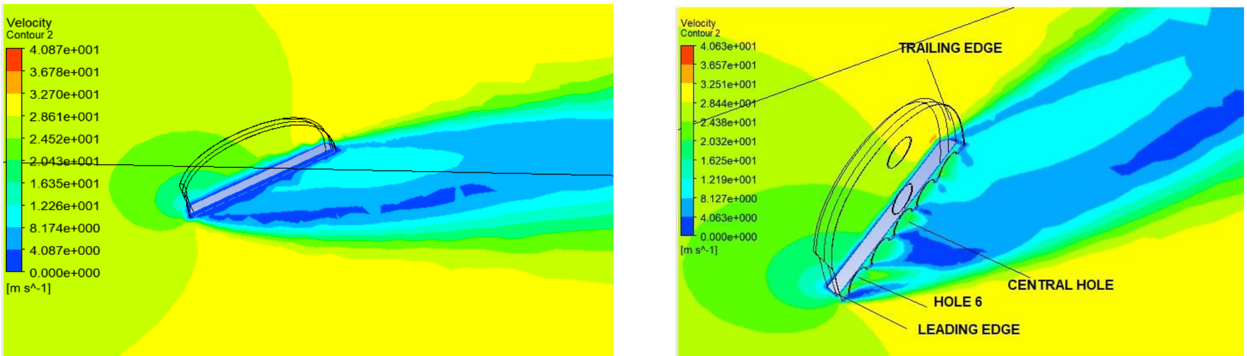
On the figure 10, it is obvious that inside the backflow interstice, the pressure is built with a calculated maximum of about 334000 Pa.



a-Normal propeller b-Backflow propeller

Figure 10. Pressure fields in vertical plane for the two models.

3.4. The velocities fields on a horizontal plane going through the jet holes (parallel to the propeller axis)



a-Normal propeller b-Backflow propeller

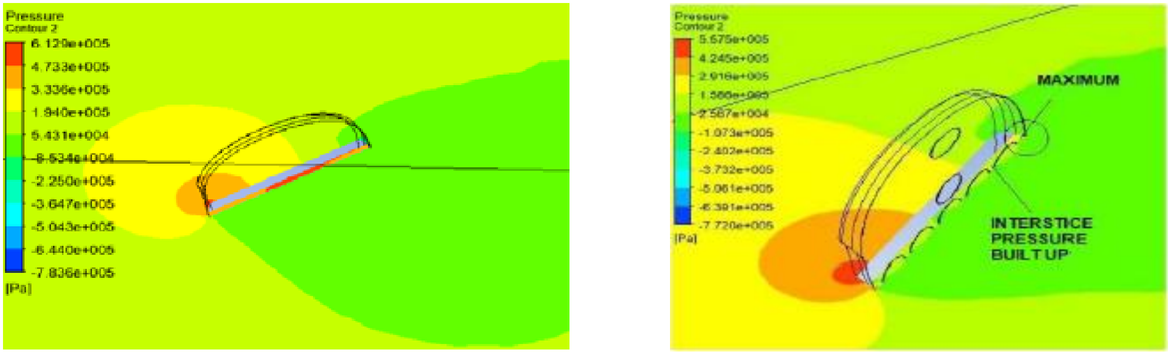
Figure 11. Velocity fields in a horizontal plane going through the jet holes for the two models.

The normal propeller has no backflow shield (screen) and jet holes.

By analysing the figure 11, it is obvious that the fluid inside the backflow interstice is migrating through the jet holes to form the backflow jet. The biggest velocity (21 m/s) is computed for the holes near the leading edge (Hole 6), the shape of the velocities filed of the fluid near this edge being altered by the presence and effects of the jet holes.

3.5 The pressure fields on a horizontal plane going through the jet holes (parallel to the propeller axis)

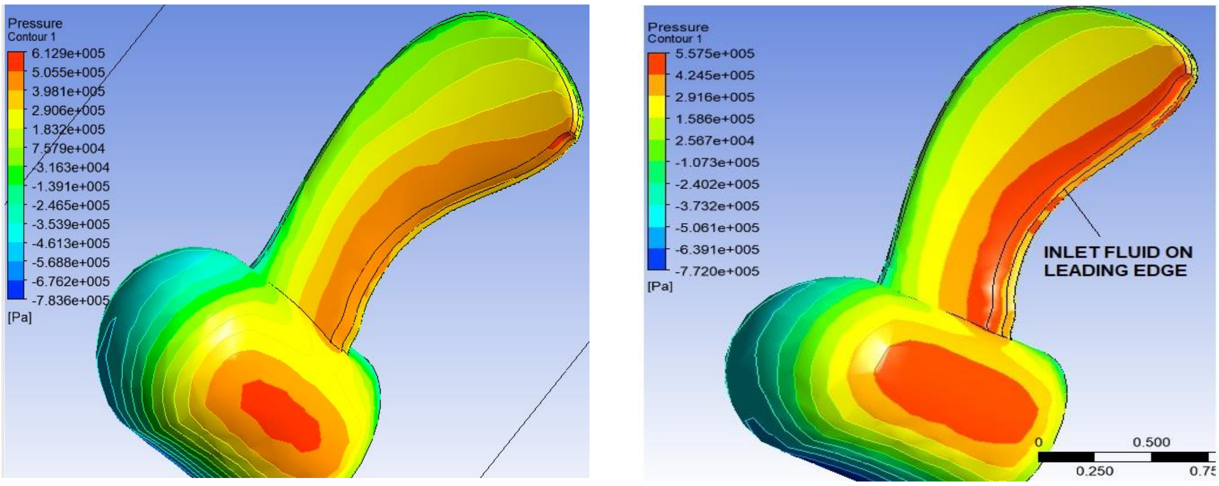
The shape of the pressure fields for the backflow propeller as compared to the normal propeller is obviously altered (figure 12). The backflow interstice is built up the fluid pressure with a calculated maximum of about 190000 Pa near the trailing edge.



a-Normal propeller b-Backflow propeller

Figure 12. Pressure fields in a horizontal plane going through the jet holes for the two models.

3.6 The pressure fields on the blade face



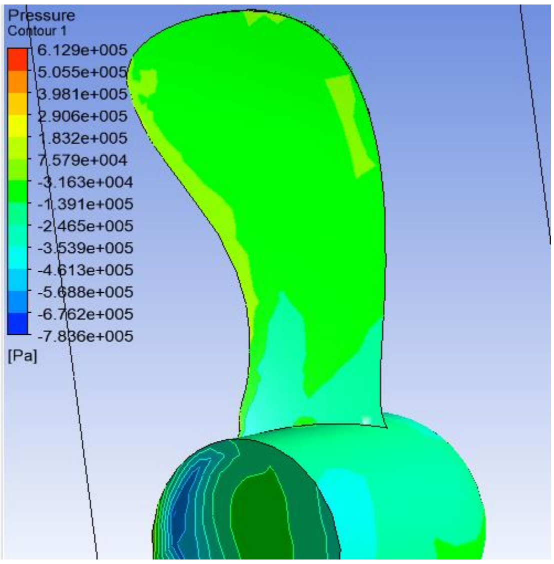
a-Normal propeller b-Backflow propeller

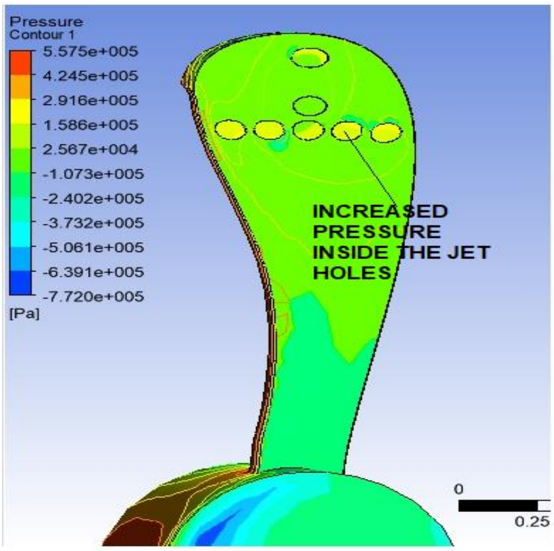
Figure 13. Pressure fields on the blade faces.

The shapes of the pressure distribution fields are similar for the two cases with the difference that the effect of the inlet zone for the fluid on the leading edge influence is visible, there the pressure is smaller (about 360000 Pa as compared to 480000 Pa of the surrounding zone).

3.7 The pressure fields on the blade back

By looking to the figure 14, we can see beyond any doubt that, the invention is working: the jet holes are contributing to the increasing of the suction pressure on the back face of the blade.

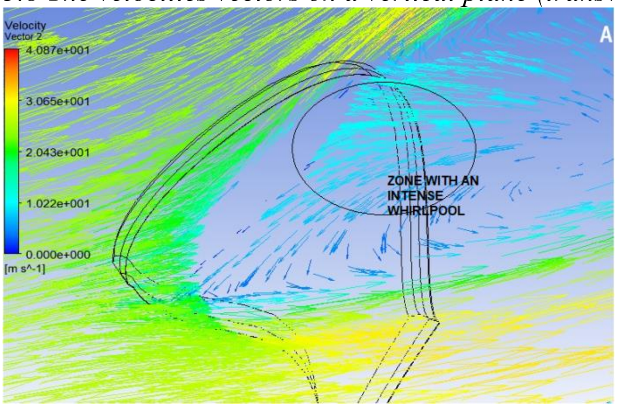


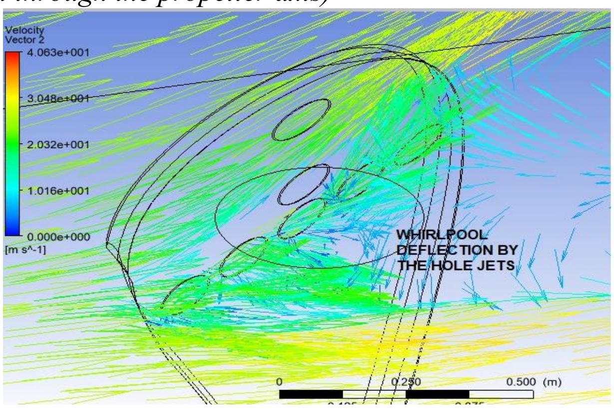


a-Normal propeller b-Backflow propeller

Figure 14. Pressure fields on the blade back.

3.8 The velocities vectors on a vertical plane (transversal through the propeller axis)





a-Normal propeller b-Backflow propeller

Figure 15. Whirlpool deflection on the blade back by the hole jets.

In the figure 15 above, it is demonstrated the original idea of the Author for this invention: how to divert the whirlpools on the back of the blade which are the main culprit for the suction pressure which by all means is lowering the marine propeller efficiency. The jet holes now are obvious that are pushing forward these whirlpools and their negative influence is diminished.

3.9 Optimization study

The optimisation study is attempting to answer to the following question: out of the 7 jet holes placed on the blade back, which one is more influential?

There were established 79 design points where the diameter of the holes is variated with ±20% and response surfaces were drawn.

IOP Conf. Series: Materials Science and Engineering **400** (2018) 082018 doi:10.1088/1757-899X/400/8/082018

4. Conclusions

This paper is coming as a logical continuation of the article [1] where the Author demonstrated the viability of the proposed solution with a simulation very near to the real case. The current approach is somehow simplified, instead of simulating the rotation of the propeller blade, the fluid domain was supposed to evolve in straight line. This time the results are more clear and visible.

It is visible here that the suction pressure on the blade back is diminished, the whirlpools formed on the blade back are diverted, the jets on the backflow shield are there and working. All these are positive effects increasing the propeller efficiency.

On the other hand, the leading edge of the propeller is increased, the opposing component of the jet is there and is taking its toll on the efficiency of the propeller.

To demonstrate and strike gold to make happy the ship Owners, all these conclusions still are in need of experimental data. The Author is positive in evaluating that a figure of 8-9% of increasing the efficiency sounds realistic, but again, this has to be demonstrated.

5. References

[1] Stan LC and Calimanescu I 2016 Computer Fluid Dynamics (CFD) Study of New Innovative Backflow Marine Propeller, *Proceedings of SPIE 0277-786X, V.*, Volume 10010, Proceedings of SPIE - Advanced Topics in Optoelectronics, Microelectronics, and Nanotechnologies VIII, Section Modeling, design, and simulation, Constanta, pp 21-30, DOI: 10.1117/12.2257302

[2] Stan LC and Buzbuchi N 2010 Operation Factors Influence on the Dynamics Behavior of Marine Propulsion Systems, *2nd International Conference on Manufacturing Engineering, Quality and Production Systems*, Constanta, pp 29-33

[3] Stan LC 2010 Engineering advanced research method for the study of marine propulsion systems, *6th International Seminar on the Quality Management in Higher Education*, Tulcea, volume 2, pp 643-646

[4] ANSYS 16 - Users'manual 2013 and soft

[5] Calimanescu I and Stan LC 2012 Elice de propulsie acvatică cu pale cu ajutaje laterale şi dorsale cu rezistenţă hidrodinamică diminuată, *Patent no. A 2012 00667 /20.09.2012*, published in the Official Industrial Property Bulletin - Inventions Section No. 3, Bucharest

[6] Scurtu IC, Clinci C and Popa A 2018 Water interference effect on ship due to square shaped object shielding, IOP Conference Series: Earth and Environmental Science, 172 (1), art. no. 012030, DOI: 10.1088/1755-1315/172/1/012030

[7] Scurtu IC 2015 Manufacturing and design of the offshore structure Froude scale model related to basin restrictions, IOP Conference Series: Materials Science and Engineering, 95 (1), art. no. 012068, DOI: 10.1088/1757-899X/95/1/012068