

## EFFECTS OF DUCT SHAPE ON DUCTED PROPELLER THRUST PERFORMANCE

KRZYSZTOF SZAFRAN\*, OLEKSANDR SHCHERBONOS\*\*, DARIUSZ EJMOCKI\*

*\*Institute of Aviation, Al. Krakowska 110/114, 02-256 Warsaw, Poland*

*\*\*National Aviation University, Kosmonavta Komarova 1, 03058 Kyiv, Ukraine*

### Abstract

*The theory of cooperation with the propeller nozzle is currently being developed as applied to air drives. In the present study attempted to synthesize a new aerodynamic profile for powertrain hovercraft. We analyzed several well-known theoretical solutions aerodynamic airfoils. On the basis of the results, we designed new aerodynamic profile. The new airfoil, which has improved properties in the fields of small and medium speed.*

*The authors present the theoretical results of comparative tests of selected airfoils with the new profile ILot-HR. The new duct shows better thrust characteristics in medium and high advance ratio in comparison with 19A nozzle.*

*Keywords: duct, fan, thrust propeller, airfoils.*

### INTRODUCTION

It is well known that a duct offers significant advantages for the thrust performance of the propeller. Studies about nozzle and propeller shape have been done for a while and conventional 19A nozzles are used ordinarily for a ducted propeller (Gornicz T., 2013). But widely used 19A nozzle originally designed to meet requirements of maritime industry and does not fully meet the requirements of other vehicle types. This paper compares thrust performances of different nozzles shapes in order to use them in hovercraft (Pałowski, Z.T. & SZAFRAN, K. The Ecological Hovercraft - Dream or Reality. *Marine Navigation and Safety of Sea Transportation*, 2013) designed in Institute of Aviation. ISBN: 978-1-138-00105-3.

### OBJECT OF STUDY

Four geometries of the duct were investigated in the current work: Wageningen 19A – a conventional nozzle; NACA 73\_4212 – the duct geometry proposed in (Serdar Yilmaz, 2013); Wartsila-HR – high performance Rice Thrust Nozzle; Ilot-HR – duct designed in the current work (Fig. 1). All ducts used the same fan manufactured by Multi-Wing. The fan diameter is 1600 mm, duct length is 1200 mm. The propeller location is in 20% chord from the leading edge, as for the requirements of hovercraft design (in order to install reverse thrust system).

The calculations were performed for a constant fan revolution  $n = 1400$  [rpm] and a variable velocity  $V = 4 \div 36$  [m/s] to simulate advance ratio in the range  $J = 0.1 \div 1$ .

## CFD MODELS

### Computational domain

The research task did not include studies of a flow around fan blades, but the efficiency of the duct geometry. This makes it possible to reduce the problem to the axisymmetric formulation. The fan simulated by boundary conditions Fan-Pressure-Jump. Fan characteristics are calculated in a software product Multi-Wing Optimizer 7. Calculations were made in the domain of the cylindrical shape 40 duct chords long with radius of 20 duct chords. Convergence results on the size of the computational domain was carried out for three values of the calculated domain size: 30, 40 and 60 duct chords. The results are shown in Table 1.

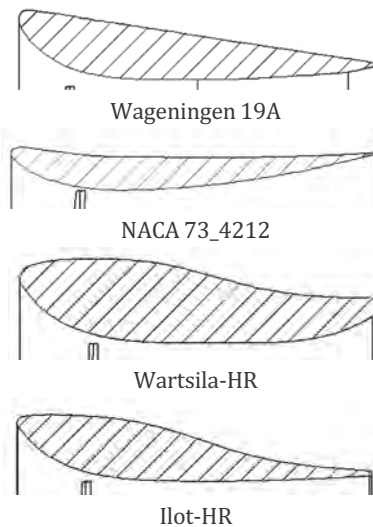


Figure 1. Studied duct airfoils.

The calculations used k- $\omega$  SST turbulence model (D.C. Wilcox, 2002). Unlike the standard  $k$ - $\epsilon$ -equation, the  $\omega$ -equation can be integrated through the viscous sublayer without the need for a two-layer approach (Ansys, 2013). This makes it possible to use a single computational grid for the entire range of calculated velocities. The maximum value of the dimensionless wall distance  $Y^+$  is in the range of 2.4 ( $J=1$ ) and 0.74 ( $J=0.2$ ). Dependence of the results of calculation of the value  $Y^+$  tested on four computational grid  $Y^+ \sim 0.5, 1, 5, 30$ . The results are shown in Table 2. Hybrid computational mesh contain about  $2e+5$  elements (Fig. 2).

Table 1. Convergence results on the size of the computational domain.

Domain size/duct chord	$C_d$	$\Delta C_d$ %
30	-0.5566	0.08
40	-0.557	-----
60	-0.55549	0.32

Table 2. Convergence results on the  $Y^+_{max}$ .

$Y^+_{max}$	$C_d$	$\Delta C_d \%$
0.565	-0.55964	0.47
1.15	-0.557	-----
5.79	-0.558	0.18
34.7	-0.56448	1.34

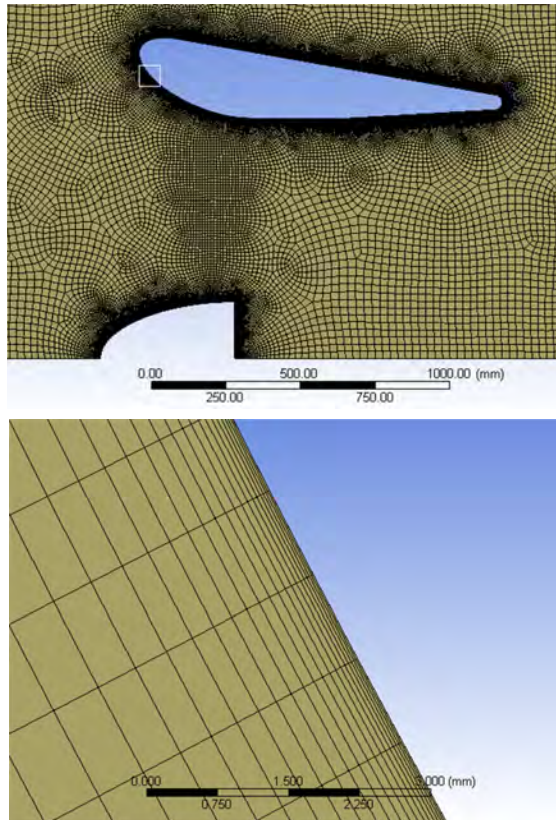


Figure 2. Hybrid computational mesh.

## Numerical methods

Calculations were done using CFD toolbox Ansys - Fluent. Fluent was validated many times and it was proven to be a reliable tool (Gornicz T., 2013), (Jiyuan Tu, 2008). It was used as pressure-based, a steady state, axisymmetric calculation model with SIMPLPE solution algorithm and the second order gradient schemes. The flow turbulence was calculated using  $k-\omega$  SST model (D.C. Wilcox, 2002), (Hellsten, 1998), (Mani M., 2004). Boundary conditions were defined in next way: *velocity-inlet* for inlet of domain, *pressure-outlet* for outlet, *slip wall* for outer border of domain, *wall* for duct and hub walls, *fan* for plane simulating pressure jump caused by fan and *periodic* for sides of computational domain. During calculation the mass flow rate through a duct stabilized after about 3000 iterations, with residuals less than  $10^{-4}$  for continuity,  $k$ ,  $\omega$  and  $10^{-6}$  for velocities.

## RESULTS

Duct thrust coefficients ( $C_{Df}$ ) are represented in Figure 3. The total thrust coefficients, which include thrust produced by the propeller, the duct and the hub, are represented in Figure 4.

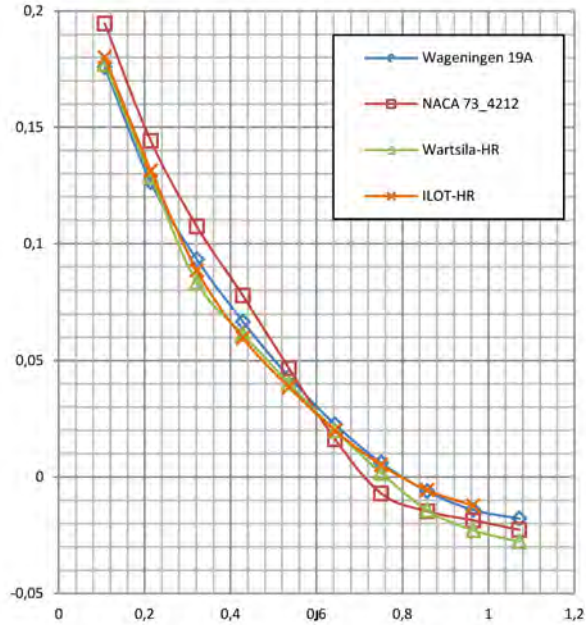


Figure 3. Duct thrust coefficients.

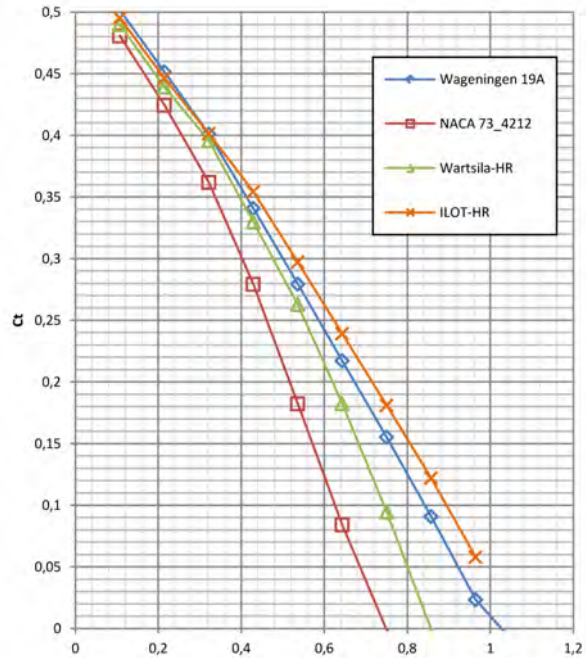


Figure 4. Total thrust coefficients.

The highest thrust was produced by duct with NACA-73\_4212 airfoil, but the total  $C_t$  of this nozzle was the worst. This can be explained by a significant, increasing flow velocities induced near the leading output of the airfoil, where the propeller is installed, and as a result a higher local advance ratio and lower thrust was created (Fig. 5).

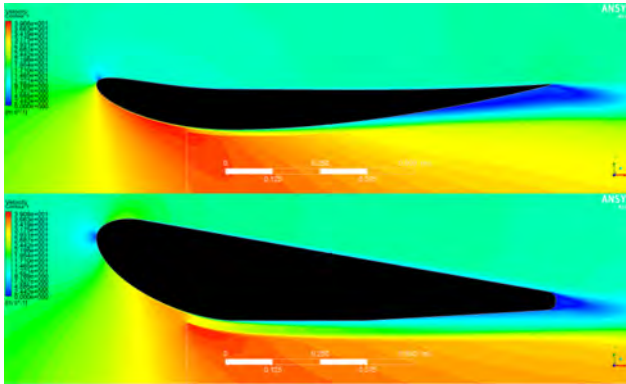


Figure 5. Velocities comparison of NACA-73\_4212 and 19A ducts.  $J=0.43$ .

The high performance rice thrust nozzle Wartsila-HR shows unexpectedly lower results in comparison with 19A. A big separation flow area near the trailing edge increase drag, and high velocities, induced by thick airfoil and degrade fan characteristics. This is probably caused by overdesigned conditions for the nozzle designed initially for maritime purposes. This case confirms the thesis that vehicles should used ducts designed especially for operating conditions.

To improve thrust characteristics of the ducted propeller in the medium and increase high  $J$  conditions, a new design concept of duct were proposed in the current work. Ilot-HR duct use deeply modified a geometry of Wartsila-HR nozzle. Goal of the design was to reduce the drag by removing flow separation regions of the original Wartsila-HR geometry (Fig. 6). Also the new design uses thinner airfoil with less induced circulation, which slightly reduces the duct thrust but improves the fan operation conditions. The results show an increase in the thrust coefficient in the range of  $J > 0.3$ . Figure 7 shows a comparison of pressure coefficients on the duct surfaces. Figure 8 represents a comparison of dimensionless velocities at the trailing edge of the duct.

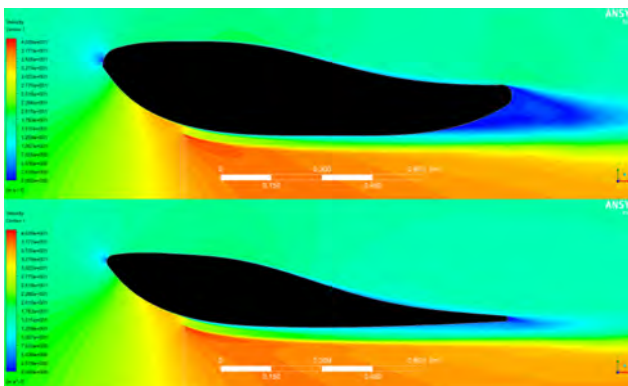


Figure 6. Velocities contours around Wartsila-HR and Ilot-HR nozzles.  $J=0.43$ .

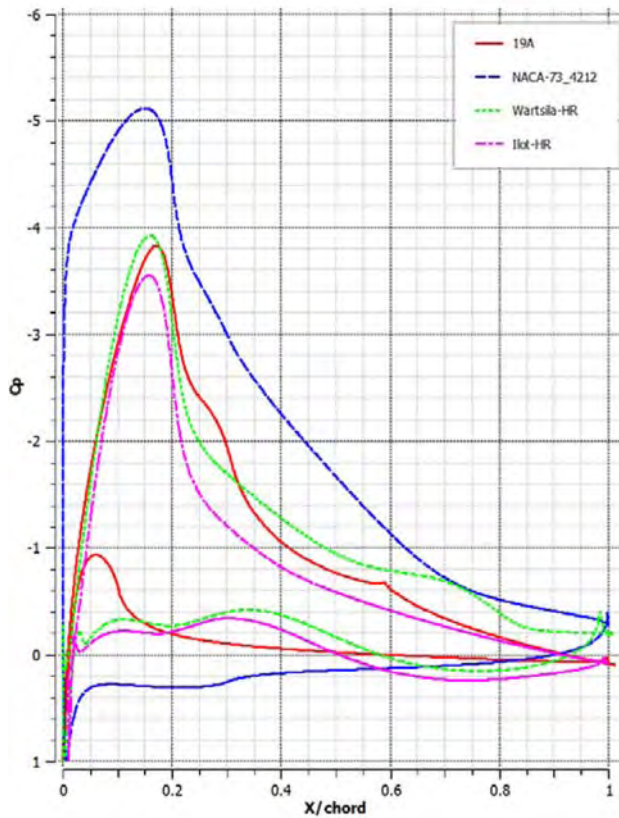


Figure 7. Pressure coefficients at ducts.  $J=0.43$ .

## CONCLUSIONS

The initial goal of the current work was to compare propulsion characteristics of the duct propeller with the conventional nozzle 19A and perspective nozzles NACA-73\_4212 and Wartsila-HR in hovercraft operation conditions. The results show a disadvantage of using this nozzle instead of the conventional 19A nozzle in the case of the hovercraft. But during this study a new geometry of the duct was designed as deeply modified geometry of Wartsila-HR nozzle. This duct shows better thrust characteristics in medium and high advance ratio in comparison with 19A nozzle.

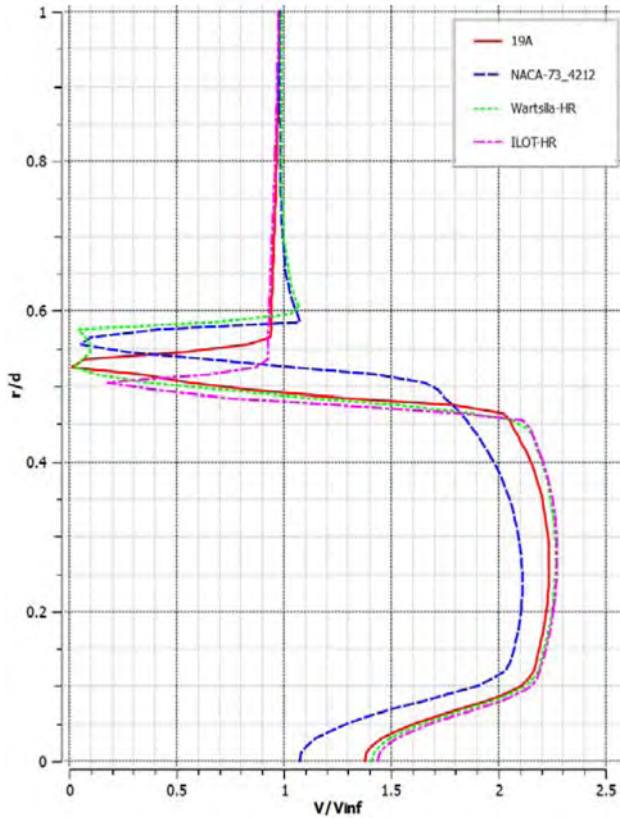


Figure 8. Dimensionless velocities at trailing edge.  $J=0.43$ .

## REFERENCES

- [1] Ansys. (2013). ANSYS FLUENT Theory Guide relies 14.5. ANSYS, Inc.
- [2] D.C.Wilcox. (2002). Turbulence Modeling for CFD. DCW Industries inc.
- [3] Gornicz T., K. J. (2013). The Assessment of the Application of CFD Package OpenFOAM to Simulation Flow Around the Propeller. Marine Navigation and Safety of Sea Transportation .
- [4] Hellsten. (1998, June). Some Improvements in Menter's k-omega SST Turbulence Model. AIAA Paper 98-2554 .
- [5] Jiyuan Tu, G. H. (2008). Computational Fluid Dynamics A practical Approach. Elsevier.
- [6] Mani M., L. J. (2004, No. 2). Rotation and Curvature Correction Assessment for One- and Two-Equation Turbulence Models. Journal of Aircraft , pp. 268-273.
- [7] Serdar Yılmaz, D. E. (2013). 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition. Effects of Duct Shape on a Ducted Propeller Performance.
- [8] Tamura, Y. (2010). ITS. Development of a High Performance Ducted Propeller (pp. Day 1, Paper No.7). ABR Company.
- [9] Pałowski Z.T. & SZAFRAN K. (2013) The Ecological Hovercraft - Dream or Reality. Marine Navigation and Safety of Sea Transportation. ISBN: 978-1-138-00105-3. DOI: 10.1201/b14960-7

## WPLYW KSZTAŁTU PROFILU OTUNELOWANIA ŚMIGŁA NA SIŁĘ CIĄGU

### *Streszczenie*

*Teoria współpracy z dyszy ze śmigłem lub wentylatorem jest obecnie szeroko rozwijana w zastosowaniu do napędów lotniczych. W niniejszym opracowaniu podjęto próbę syntezy nowego profilu aerodynamicznego dla zespołu napędowego poduszkowca. Przeanalizowaliśmy teoretycznie kilka dobrze znanych rozwiązań profili aerodynamicznych. Na podstawie tych wyników opracowaliśmy nowy profil aerodynamiczny. Nowy profil ma lepsze właściwości w zakresie małych i średnich prędkości.*

*Autorzy przedstawiają teoretyczne wyniki testów porównawczych wybranych profili z nowym profilem Ilot-HR. Nowy tunel wykazuje lepsze właściwości oporowe w porównaniu z dyszą 19A.*

Słowa kluczowe: tunel, wentylator, profil lotniczy, ciąg śmigła