ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XI – SHIPBUILDING. ISSN 1221-4620
2015

PROPULSIVE PERFORMANCE IMPROVEMENT OF AN INLAND PUSHER

Florin Păcuraru
"Dunarea de Jos" University of Galati,
Faculty of Naval Architecture, Galati,
47 Domneasca Street, 800008, Romania,
E-mail:florin.pacuraru@ugal.ro

Adrian Presură
"Dunarea de Jos" University of Galati,
Faculty of Naval Architecture, Galati,
47 Domneasca Street, 800008, Romania,
E-mail: a.presura@shipdesigngroup.eu

Ştefan Iorga
"Dunarea de Jos" University of Galati,
Faculty of Naval Architecture, Galati,
47 Domneasca Street, 800008, Romania,
E-mail: stefan.iorga@gmail.com

ABSTRACT

The paper proposes a numerical investigation based on RANS computation for solving the free surface viscous flow around an inland pusher in order to provide a detailed insight into the critical flow regions for studying the hydrodynamic interactions between various components of the propulsion system, for predicting the hydrodynamic performance of the propulsion system and for determining the optimum propulsion arrangement.

Keywords: pusher, propulsion, numerical simulation, RANSE.

1. INTRODUCTION

The inland waterway transport industry on the Danube is confronted with particular challenges due to geographical related restrictions. Moreover, ship efficiency and emissions regulations (EEDI) are gradually increasing the need for more holistic design approaches in terms of energy efficiency, cost efficiency and environmental impact. From an environmental point of view, especially vessels for inland navigation should preferably be adapted to the natural waterway, and not rivers.

For inland ship design, each particular project typically requires an adapted design for the vessel and fairway. Important factors are depth, width, currents, stretch, harbour and the regime of the river. The depth and the width of the channel in the initial design stage are important for maximum beam, draft and length dimensions of the vessels. Self propelled vessels normally have their maximum draught at stern, where the most vulnerable parts are situated (screws and rudders).

In an attempt to meet the requirements mentioned above, the present paper introduces a numerical procedure to improve the propulsive performance of a pusher for inland navigation. The availability of the robust commercial CFD software as well as of the high performance computers have led to the increasing use of CFD for ship design solutions. CFD based design tools can thus provide a rather accurate solution to most of the problems, assuming that the flow solvers are able to deal with realistic geometries as well as to take into account complex physical phenomena, such as turbulence and free surface. Viscous free-surface flow calculations may provide a detailed insight into the critical flow regions, allowing the naval architect to scientifically improve hydrodynamic performances of the ship.

The paper is the first part of an exhaustive study which proposes a numerical investigation on the propulsive improvement of a typical pusher boat which navigates in restricted water conditions. Navigation in shallow water conditions has two consequences: for barge-pusher transportation, the barges need to be loaded at smaller draught, and in order to keep the same deadweight many barges are necessary and the pusher need to operate at law draughts, with increased propulsion power because of bigger convoys. The rational solution for the above mentioned problem is to use a propulsion system with three or four propellers. The aim of the research study is to improve the propulsion efficiency for a three propeller low draft pusher. In order to achieve the scope of the research, two investigation directions have been approached: the hull shape form and the propulsion arrangement. Due to the large required power and the low draught (max. 1.7 m), the propulsion solution and the ship shape in propeller area become to be critical for propulsion efficiency. Considering the specific stern shape characteristics of pusher boats, three main stern hull form concepts have been identified:

a. single tunnel stern (Figure 1a);
b. single tunnel with aft gondola (Figure 1b);
c. three tunnels stern (Figure 1c).

The investigation of the propulsion arrangement is focused on finding the optimum configurations of the propellers in terms of propulsive efficiency.

In the present paper, the commercial code NUMECA Fine/Marine 4.1 has been employed to evaluate the propulsive performance of a 4500HP pusher. Pusher hull is characterized by sledge bow, rounded bilge, arch tunnels aft, large aft gondola (for CL shaft line), as one can see in Figure 1. The main particulars of the ship are tabulated in Table 1.

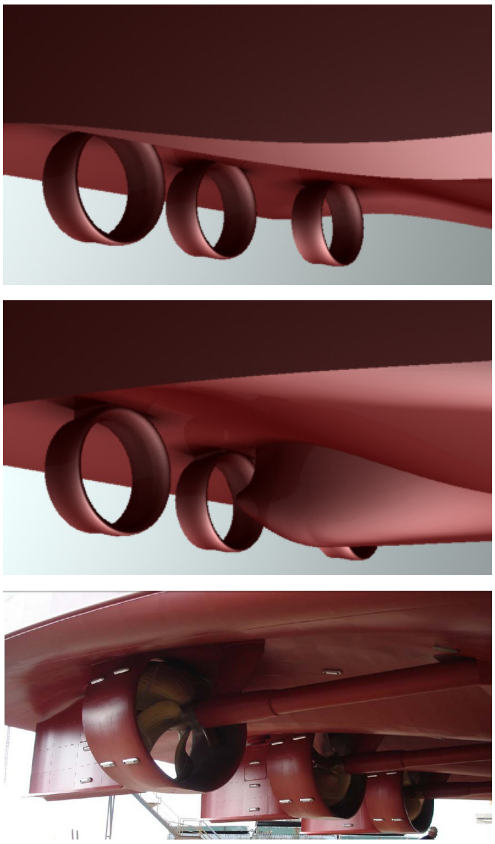

Fig. 1. Pusher stern shape concepts

Table 1. Main particulars of the container vessel

Main particulars	Symbol	Dim.	Units
Waterline length	L_{WL}	33.67	[m]
Breadth moulded	B	12.40	[m]
Draught	T	3.2	[m]
Depth moulded	D	3.30	[m]
Displacement	∇	502	[m^3]
LCB from AP	X_B	18.77	[m]
WSA (bare hull)	S	461	[m^2]
Speed	v	11	[Km/h]
Bloc coef.	C_B	0.67	
Waterline coef.	C_w	0.71	

A set of RANSE computations has been performed to better understand the influences exerted by different configurations on the wake structure in the pusher case. For practical reasons, the presence of the propeller is taken into account through a simplified model based on the actuator disk theory, according to which body forces are distributed in the flow field within a disk of finite thickness.

2. NUMERICAL SIMULATION

2.1. Computational Strategy

In the present work, the RANS equations are solved for the primitive variables to describe the 3D turbulent flow using the NUMECA commercial software. The ISIS-CFD flow solver uses the incompressible unsteady Reynolds-averaged Navier Stokes equations (RANSE). The solver is based on the finite volume method to build the spatial discretization of the transport equations. The face-based method is generalized to two-dimensional, rotationally-symmetric, or three-dimensional unstructured meshes for which non-overlapping control volumes are bounded by an arbitrary number of constitutive faces. The velocity field is obtained from momentum conservation equations and pressure field is extracted from mass conservation constraint transformed into pressure equation. In the case of turbulent flows, additional transport equations for modelled variables are solved in a form similar to that of the momentum equations and they can be discretised and solved using pressure-velocity coupling. The solution is obtained through a Rhie and Chow SIMPLE type method: in each time step, the velocity updates come from the momentum equations and the pressure is given by the mass conservation law, transformed into a pressure equation using the same principles. In the present work, closure to the turbulence is attained through the use of the k-ω SST model. Freesurface flow is simulated with an interface capturing approach. Both non-miscible flow phases are modelled using a conservation equation for a volume fraction of phase. The free-surface location corresponds to the isosurface with volume fraction 0.5. A detailed description of the solver is given by [1], [2], [3].

In the ISIS-CFD solver the momentum equations include a body-force term, used to model the effects of a propeller as an active disk. The viscous flow model includes the propeller action by applying the body force method.

2.2. Grid Generation

In the present particular case of the pusher appended hull, a mono-block unstructured grid has been generated to cover the computational domain around the bare hull. In the present paper all grids have been generated by the hexahedral unstructured grid generator HEXPRESS. It generates non-conformal body-fitted full hexahedral unstructured meshes. With appropriate refinement near the bow, the stern, the free-surface, the grid contains about 2.5 million cells. The surface grid of pusher with nozzle is depicted in Figure 2. Finer grids are generated in the areas of interest that may require better resolution. Such nozzles are presented in Figure 2. The minimum spacing (initial spacing normal to the body surfaces) is calculated based on a value of Y^+=50.

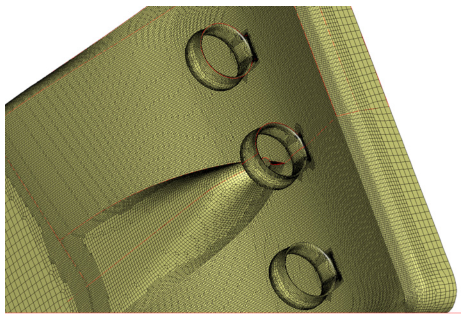

Fig. 2. Surface mesh on the pusher hull

The size of the computational domain is three times the ship length upstream, three lengths downstream, 1.5 lengths wide and two on height, as one can see in Figure 4.

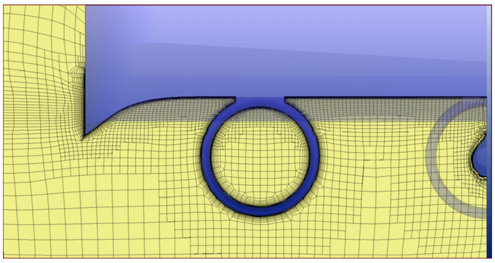

Fig. 3. Mesh refinement around hull and nozzle

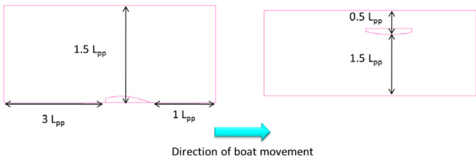

Fig. 4. Dimensions of computational domain

3. RESULTS AND DISCUSSIONS

In the present chapter the results obtained by the numerical simulation of the free-surface flow around full-scale inland pusher ship advancing in calm water under steady conditions are presented. A set of RANSE computations with and without active disk has been performed to better understand the influences exerted by different positions of the propulsors on the wake structure and propulsive efficiency. All the numerical simulations were performed for the design speed 12.5 Km/h (3.5 m/s).

For the configuration with no propellers effect considered, the most challenging task is to capture the vertical structured developed in the aft part of the ship. The difficulty arises whenever an accurate prediction of this phenomenon is required by advanced turbulence models. In the present study, the solution of the RANS equations is computed based on the use of the k-ω SST turbulence model.

The first step supposes pilot computations on initial configuration of the propulsors in order to analyse the flow field characterises in the stern area. The magnitude of the axial velocity component computed at four cross-sections at X=1.5, 2.75, 5.25 and 6.15m is shown in Figure 5. It reveals two vertical structures that are developed in the stern region. As it was expected, downstream gondola, the axial velocity contours indicate a characteristic "hook" shape in the central part of the wake correlated with the core of the longitudinal vortex, as the cross-sections plotted at X=2.75 and 1.50m show. On the other hand, the strong changes in hull geometry in the bilge area determine the flow development of a second vortex highlighted in Figure 5, which seems to be triggered at about 6 m from the transom stern. In future studies, hull shape modifications in condole and bilge area have to be considered.

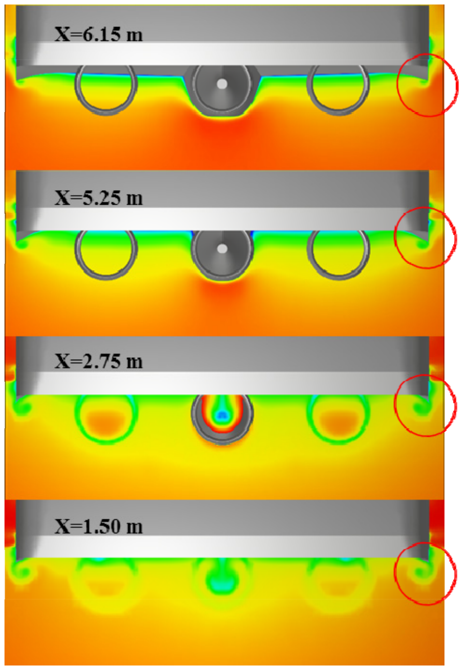

Fig. 5. Axial velocity computed at four cross-sections

Figure 6 shows the streamlines patterns computed around the stern hull. The streamlines plot has been necessary to follow the fluid particle path. As the flow develops along the hull, the hull geometry gradually forces the boundary layer to pack in an area whose girth wise dimension decreases, implying a progressive convergence of the streamlines in some regions of the hull.

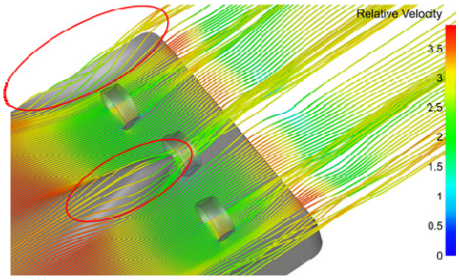

Fig. 6. Streamlines computed around the hull stern

The present research aims to investigate the propulsive performances developed by the operating propeller fitted in the nozzle. The propeller diameter is 1.54 m and the nozzle diameter of 1.925 m. The K-type propeller has 5 blades.

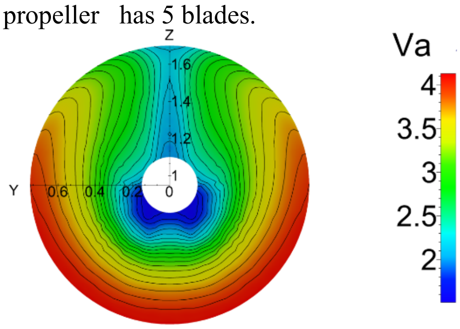

Fig. 7. Axial velocity contours in the CL propeller disk for the case without propeller

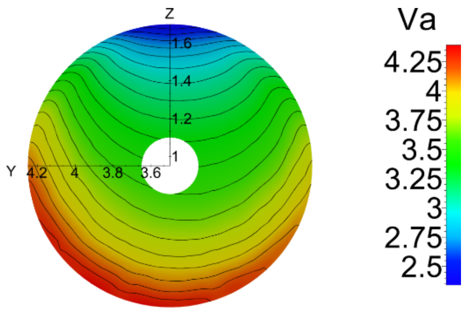

Fig. 8. Axial velocity contours in the SIDE propeller disk for the case without propeller

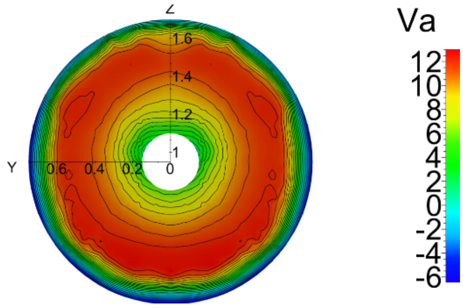

Fig. 9. Axial velocity contours in the CL propeller disk for the case with propeller

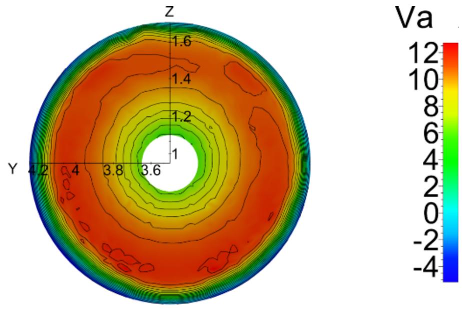

Fig. 10. Axial velocity contours in the SIDE propeller disk for the case with propeller

In order to qualitatively evaluate the flow in the propeller disk and the effect of active disc introduced in the computations, in Figures 7-10 the axial velocity components are plotted for the centre line propeller (CL) and side propellers (SIDE) with and without active disc.

Various simulations were carried out to compute the flow around different propulsors positions, in order to study the influence of each configuration on the wake flow structure and propulsion efficiency. The wake velocity is associated with the disturbed flow around the hull and it consists of a significant variation in magnitude and direction. An actuator disk is included in the RANS code, where the body forces are computed interactively with the RANS solver.

Four different positions of side propulsors have been investigated. The position of the CL propulsor has not been changed because this would imply the mofication of gondola position. The position of CL propulsor is defined by X=2.55 m, Y=0 and Z=0.95m. The positions of SIDE propulsors are presented in Table 2 and Figure 10. It should be mentioned that the position of the propulsors hase not been modified on the Z direction.

Table 2. SIDE propeller positions

Configuration	X[m]	Y[m]
V1 initial	3.25	3.5
V2	3.25	4.1
V3	2.35	3.5
V4	3.70	3.5

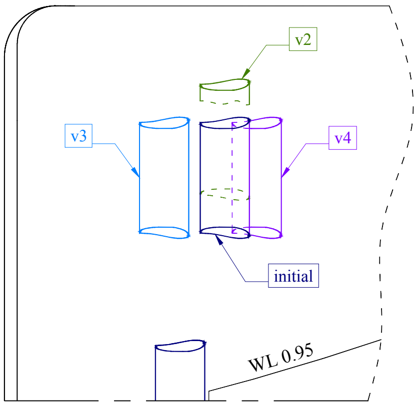

Fig. 10. Axial velocity contours in the SIDE propeller disk for the case with propeller

Table 3. Flow rate results

Configuration	**V1**	**V2**	**V3**	**V4**
CL, no AD				
Q[%]	0	-0.37	-0.74	-0.37
CL, AD				
Q[%]	0	0.28	1.13	-1.20
SIDE, no AD				
Q[%]	0	-0.33	1.31	-4.43
SIDE, AD				
Q[%]	0	1.63	5.42	5.22

In order to quantitatively analyse the effect of the hull shape and propulsion arrangement on the propulsion efficiency, the flow rate in the propeller disc based on computational simulation of each case has been compared. Flow rate results are presented in Table 3. Analysing the results, one can conclude that the optimum arrangement seems to be configuration V3.

4. CONCLUDING REMARKS

The paper describes a general numerical method to predict the propulsive performance of an inland pusher. Self-propulsion tests have been performed in order to quantify the effect of different propulsions.

Acknowledgements

The present research has been performed in collaboration with Ship Design Group.

REFERENCES

[1]. *"Environmentally Friendly Inland Waterway Ship Design for the Danube River", WWF- DCP, Project Danube Navigation, Raport , 2009*

[2]. **Queutey, P, Visonneau, M.,** *"An interface capturing method for free-surface hydrodynamic flows"*, Computers & Fluids, Vol. 36, No. 9, pp.1481-1510, 2007.

[3]. **Duvigneau, R., Visonneau, M.,** "*On the role-played by turbulence closures in hull shape optimization at model and full scale*" J. Mar. Sci. Technol., Vol. 8, pp. 11–25, 2003.

[4]. **Deng, G.B., Queutey, P., Visonneau, M.,** "A Code Verification Exercise for the Unstructured Finite-Volume CFD solver ISIS-CFD*"*, European Conference on Computational Fluid Dynamics ECCOMAS CFD 2006, pp.1-18.

Paper received on December 31st, 2015