STERN FLOW IMPROVEMENT BY USING SOME NEW, ORIGINAL CONCEPTS AND IDEAS, IN THE WORLD OF SHIP HYDRODYNAMICS

Horațiu TĂNĂSESCU[1], Ovidiu TĂNĂSESCU[2]

Lucrarea încearcă să atragă atenția şi se concentrează, pe scurt, asupra curgerilor din pupa navelor în lumina a doua concepte (idei) noi de cercetare fundamentală, originale în domeniul hidrodinamicii navale: 1-un nou concept hidrodinamic de forma pupa (NSHC), cu secțiuni crenelate-rotunjite radial, tip distribuitor dinamico - gravitațional ; 2-folosirea efectului piezoelectric invers [(curent electric→generator de putere de înaltă frecventa→driver piezoelectric confecționat dintr-un material ceramic special, pentru inducerea unei mişcări vibratorii eliptice (cu frecventa mai mare de 20 KHz), in tablele elastice ale învelişului carenei (groase de 15 mm) in direcția liniilor de curent (a apei care curge in exterior)], capabil să reducă rezistenta totala la înaintare.

The paper tries to draw attention and briefly focuses on ships hull's stern flows in the light of two new fundamental research concepts (ideas), original in the world of ship hydrodynamics: 1-a new stern hydrodynamic concept (NSHC), with radial crenellated-corrugated sections, dynamical - gravitational distributor type; 2-using of an inverse piezoelectric effect [(electric current→high-frequency power generator→piezoelectric driver made of certain ceramic material, which induces an elliptical vibratory movement (high frequency over 20 kHz), into the elastic side plates (15 mm thickness) in the streamlines direction (of the external flowing water)], able to reduce the total forward resistance.

Keywords: hydrodynamics, stern turbulent boundary layer, flow separation, nominal wake, inverse piezoelectric effect, model, physical and numerical tests

1. Introduction

Present-day tendency in maritime transportation industry is represented by designing and building of bigger, faster, more energy-efficient and stable ships but simultaneously having stricter noise and vibration levels for stern hull structure. A modern ships hull lines are designed to minimize forward resistance, to reduce propeller cavitation, to improve propulsion performance and to increase global hydrodynamic stability. As a general recently accepted opinion, the ships of the future will be designed and built only on the basis of some **new original**

[1]Researcher, S.C. ICEPRONAV S.A., Galati, Romania
[2] PhD. Student, University "Lower Danube" of Galati, Romania

devised concepts. It is well known that the stern flow problem is rich in complexity and poses many challenges. Ships hull's stern flows have received much attention these last years, in particular with regard to their modeling and design principles. As **a state of the art** in the field, the most recently known industrial achievements, focused on flows improvement in the stern region, which consist in symmetrically flattening of the stern lateral surfaces towards the central plane [1]. This concept has resulted in a huge amount of inconveniences almost in all practical applications to real ships (unsuitable placing of equipments, lack of necessary spaces for inspections, repairs, etc.). For a long time, the present proposal's authors thought how to redesign the two systems - hydroframe system and propulsion system – very important (critical) for a ship, so that the hydroframe may meet the propulsion and the propulsion may meet the hydroframe in an optimal way.

2. Investigation's scientific objectives

- Total forward resistance reduction, propulsive efficiency increasing (for minimum energetic consumptions obtaining);
- Propeller cavitation reduction (for level of noise and vibration induced on board and in the stern structure decreasing);
- Development of a numerical parameterized model;
- Design sensitivity analysis of fields generated;
- Optimizations;
- Original concepts and ideas validation;
- Professional methodologies establishing.

It is hoped that the successful solving of the above mentioned objectives will contribute to the top new knowledge accumulation and progress, in a very important, actual and complex scientific field as contemporary ship hydrodynamics is.

3. Interdisciplinary degree:

The main disciplines from which the present project is taking its sap are:
- physical-mathematics (partial differential equations; integral equations);
- materials physics;
- physics applications; technical physics;
- modeling and simulation;
- hydraulics and fluid mechanics;
- ship hydrodynamics.

4. Some basic theoretical aspects [2]

In real conditions, a propeller is fitted behind the ships (models) hull's stern, working in a non-uniform water stream, which has been disturbed by the ships hull during its forward motion. The ship's moving hull carries with it a certain mass of the surrounding water forming a region in which there is a rapid change in velocity well-known under the name of **boundary layer**. The propeller being placed behind the ship's hull stern, there is in the ship's body trail. As a consequence (even considering the average velocity), the velocity of water particles relative to the propeller disk is no longer (both neither in magnitude and nor in direction), equal to the velocity of advance of the propeller relative to still water. This trail, in which there is a difference between the ship speed and the speed of the water particles relative to the ship is also termed **wake**. Generally speaking, the wake is a zone not investigable theoretically (analytically), due to very complex, aleatory flow character within it. In ship's propeller theory, a distinctive importance is having only the incipient part of the trail (wake), located immediately in the front of the propeller disk plane. The movement from this zone is called wake movement or simply wake. The wake movement can be investigated or in the presence or in the absence of the propeller, when is taking the attribute of the effective wake or the nominal wake, respectively. However, the wake movement of interest is only that from the plane where the propeller follows to be situated. The flow's average velocity from that plane is termed wake speed V_W, and is in general smaller then ship's speed V_S, relative to infinite upstream water. If the water is moving in the same direction as the ship, the wake is said to be positive. Then

$$Wake = V_S - V_W \tag{1}$$

For adimensionalization, the precedent relation can be divided by either V_W or V_S leading to two wake factors

- *Froude wake factor* $= w_F = (V_S - V_W) / V_W;$ (2)
- *Taylor wake factor* $= w = (V_S - V_W) / V_S;$ (3)

Besides, this general effect of the ship's hull, there will be local perturbations due to the shaft, shaft bossings or shaft brackets and other appendages. These effects combined lead to the so called relative rotative efficiency (*RRE*), defined by:

$$RRE = \eta_R =$$

efficiency of propeller behind the ship hull / efficiency of propeller in open water (at speed V_W).

5. A new stern hydrodynamic concept (NSHC)

First, the one-dimensional flow was considered and analysed [2] from four different sections of a stream tube (Fig. 1) which includes:

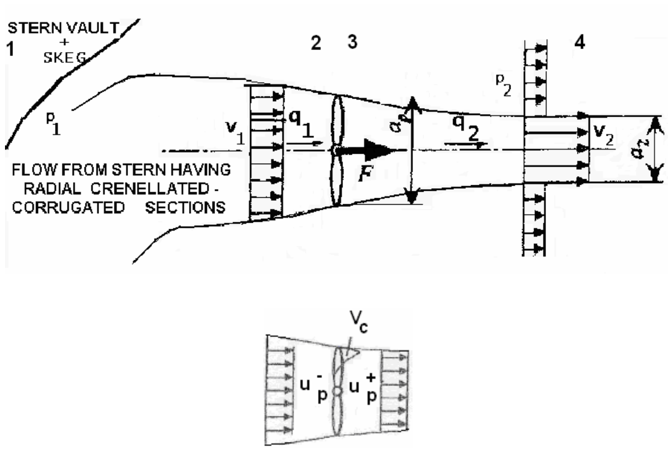


Fig. 1. Stream tube theory and Bernoulli effect application
as a supplementary background for the new stern hydrodynamic concept devising.

1 – stern (far upstream);
2 – the disk situated immediately (upstream) in front of the propeller;
3 - the disk immediately behind (downstream) the propeller;
4 - a disk downstream (far downstream);

Then, based on experience and the fundamental hydrodynamics laws [4], [5], [6], (mentioned above), a new stern hydrodynamic concept (NSHC - [7], [8], [9], [10]) of the streamline tube dynamical-gravitational distributor type (having quasi-cylindrical decreasing sections), which starts from behind last hull right cylindrical section and stretches until front of the propeller disk, was proposed intuitively (Fig. 2a and 2b).

Practically the new stern hydrodynamic concept, consists of a special (distinct) construction, complementary to classical stern sections (with or without bulb), realizable behind the last hull right cylindrical section, symmetrically in both boards, between the central plane and bilge, having as main target the hull hydrodynamical performances improving.

The number of crenellated-corrugated "teeth" and their heights will be optimized by direct numerical experiments. For each section the "teeth" step size (the distance between two consecutive crests or troughs) decreases on girth, from the centerline plane towards the boards. The maximum heights (amplitudes) of the crenellated-corrugated “teeth” will be progressively reached longitudinally in front of the propeller and transversally in the centerline plane, respectively.

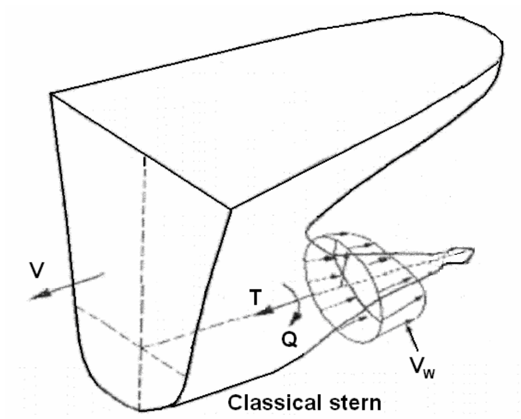


Fig. 2a. The classical stern, nominal wake.

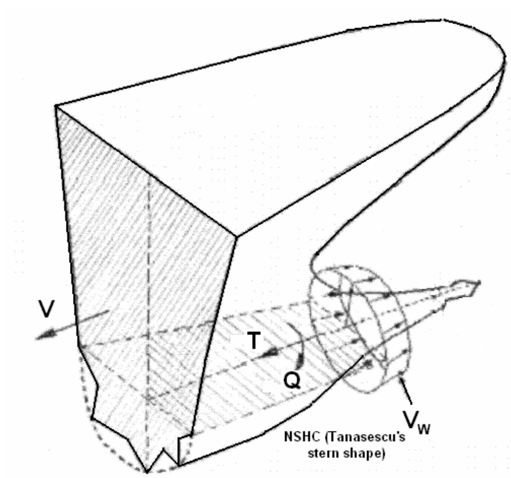


Fig. 2b. The new stern hydrodynamic concept (NSHC), nominal wake.

The directions of the crenellated-corrugated sections “teeth” crests and troughs forming some longitudinal curved grooves, will be those of the stern natural streamlines (which can be experimentally established in a flow visualization test), for vortices turning up avoiding and for a minimum forward resistance obtaining (Fig. 3).

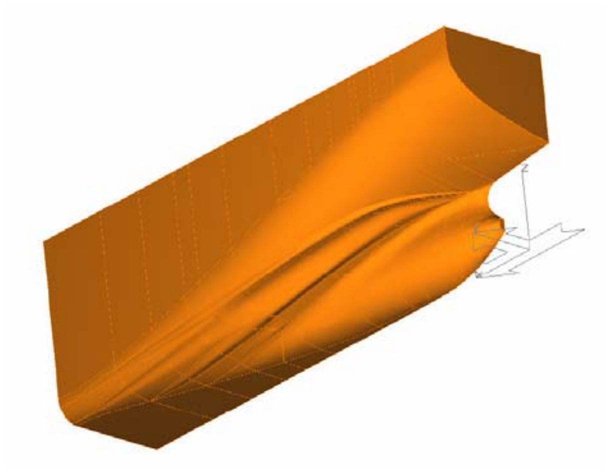


Fig.3. The new stern hydrodynamic concept (NSHC).

6. Practical application to a 7000 tdw tanker – Numerical (Fluent 6.3) & Experimental (ICEPRONAV's laboratories) results

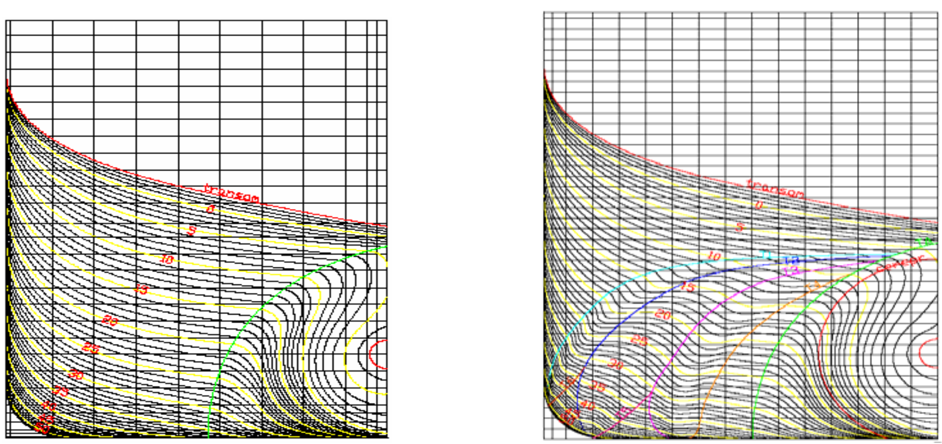


Fig. 4. Body plan (stern part) for the original – left side;
Body plan (stern part) in conformity with the new stern concept – right side.

For numerical modelling and simulation [11], [12], [13], exists two possible approaches: 1- the trimmed hull (a simplified approach, in which the free surface is not directly computed, but reduced to a flat surface numerically approximated with a symmetry boundary condition); is far less computationally intensive, but the free surface form is generating errors, more significantly with the increasing of the Froude number of the flow; 2- the full hull (a physically correct approach, in which the free surface is take into account using VOF & Open Channels models); is valid for all possible Froude numbers, but it imposes more carefully meshing (the free surface resolution is highly dependent on the mesh distribution and density) and higher computational effort (bigger meshes and lower numerical stability).

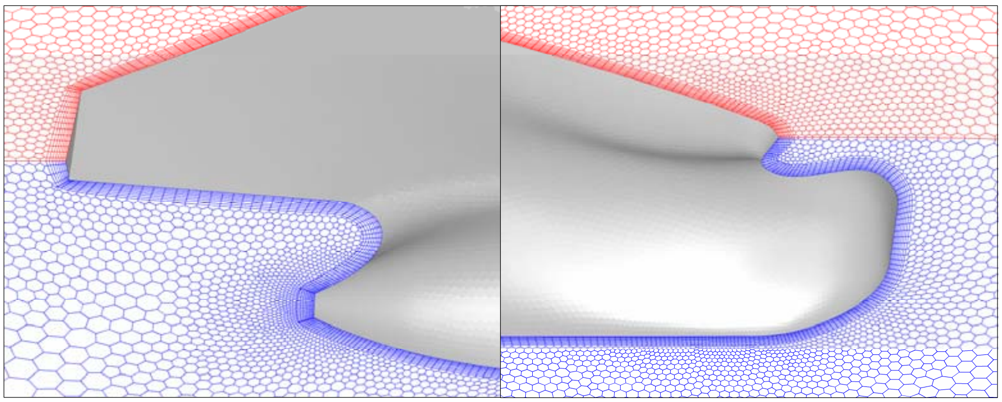


Fig. 5. Bulb and stern regions of the polyhedral grid used for the unmodified hull (boundary-layer type grid is visible); cells: 737,959; nodes: 2,544,318.

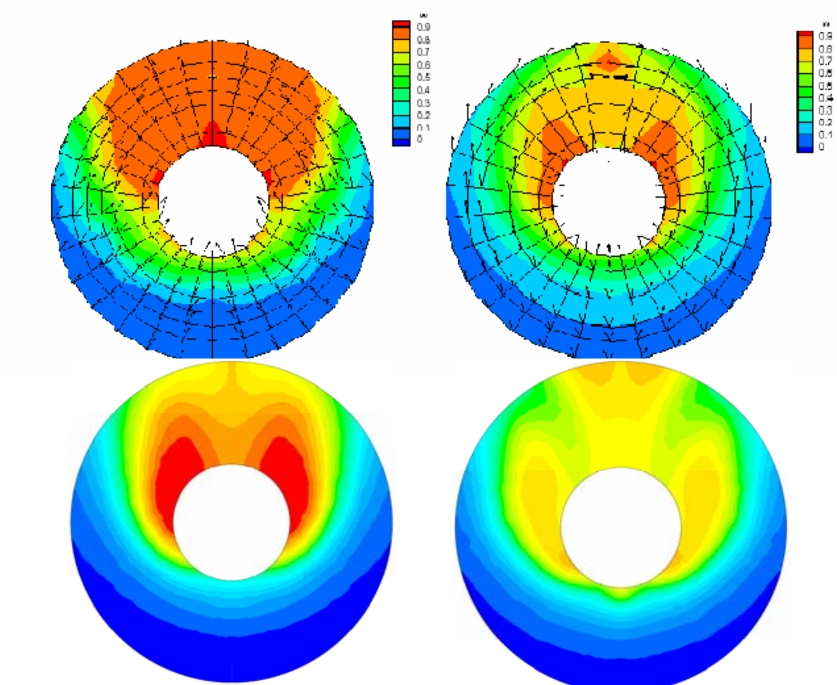
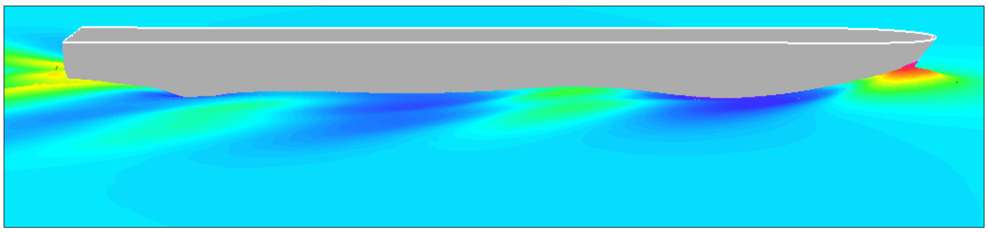


Fig. 6. Comparison between the experimentally measured nominal wake maps (upper row), and the numerically computed nominal wake maps (lower row).

Experimental model is free to sink



Numerical model is fixed

Fig. 7. Visual comparison between experimental and numerical model hull wave profile.

The most important result until now demonstrated, is the reducing of propeller cavitation (working in the simulated nominal wake of the hull using the new stern hydrodynamic concept with radial crenellated - corrugated sections, practically to zero (Fig.3).

Fig. 8. Simulated nominal wake testing, in 850x850 mm section of the cavitation tunnel at 25 rps rotative speed (it can be remarked lack of cavitation).

Vs[Nd]	R-v0[Kg]	R-v4.3[kg]
14,5	29433	27486
15	35995	32344
15,5	39939	36862
16	42847	39585
16,5	46205	42136

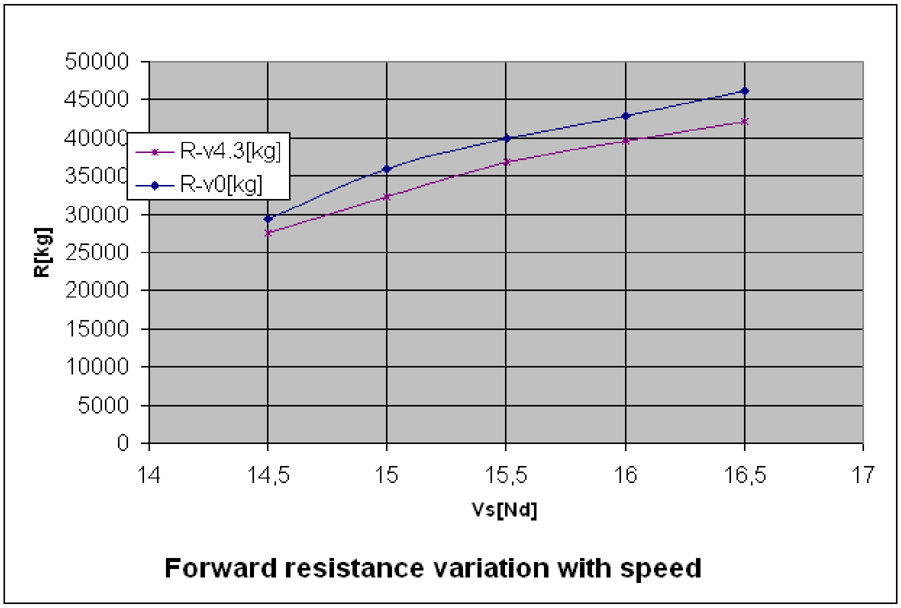


Fig. 9. Another very important result is represented by forward resistance reduction with approximately 7.7 %, as is shown into the above diagrams;
V0 – reference model; V4.3 – NSHC model.

7. Future developments

In the near future, as a continuation of the researches presented above it is considered imperatively the realization of: 1-a parameterized geometrical model streamline tube type, (including the effects, of new stern design having quasi-cylindrical increasing radial crenellated-corrugated sections, on inside propeller flow); 2-a systematic design sensitivity analysis of the new stern fields generated (using FLUENT and FEMAP well known tools).

In these cases different geometries (as grooves form, width and depth, along hull distances, for flow separation avoiding), should be studied theoretically, numerically and experimentally. Design sensitivity analysis [14], consists in determining derivatives of a system's response with respect to its design parameters x_i. In the context of design optimization (of the new hydrodynamic stern concept proposed), the response is expressed in terms of objective and constraint functions, and accordingly the overall aim of design sensitivity analysis is to find the gradients of these functions. However, since any such problem function depends explicitly on the dependent variables φ of the considered problem, sensitivity formulations in essence aim at the calculation of the derivatives $\partial\varphi/\partial x_i$. In other words, the changes in flow field φ resulting from a given change in design must be predicted. After the determination of these flow field sensitivities, it is a matter of straightforward calculus to compute the design sensitivities of any problem function.

$$\nabla f = \frac{df}{dx_i} = \underbrace{\frac{\partial f}{\partial x_i} + \frac{\partial f}{\partial \gamma}\left[\frac{\partial \gamma}{\partial x_i}\right]}_{\text{grid sensitivity}} + \underbrace{\frac{\partial f}{\partial \varphi}\left[\frac{\partial \varphi}{\partial x_i}\right]}_{\text{flow sensitivity}} \tag{4}$$

where $i = 1 \ldots n_{dv}$ and:

$f(x_i)$ – problem function (typically identical with objective and constraints function);

x_i – design parameters;

$\gamma(x_i)$ – geometrical quantities;

$\varphi(x_i)$ – vector containing unknown flow variables (velocities, static pressure, possibly turbulence modelling quantities), determined by the governing equations);

dv – design variables.

Obviously, both geometry and flow are implicitly controlled by the design parameters through surface parameterization, mesh generation and flow analysis.

On the other hand, unfortunately, in the results previous obtained, the cavitation decreasing is accompanied by initiation of some multiple increased vortices (Fig. 10), resulted from the separation (although a lower one – Fig. 11) of the boundary layer.

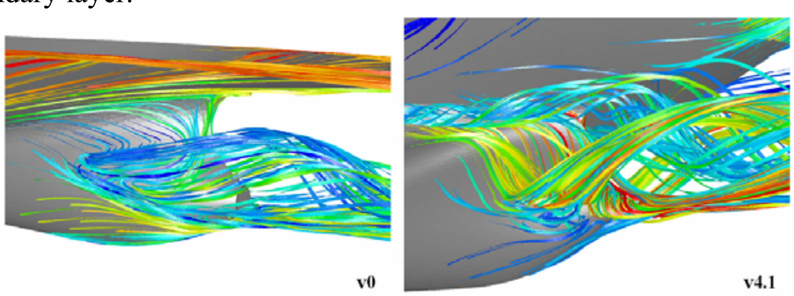


Fig. 10. Vortex initiation and separation – FLUENT 6.3
(left - the model with initial stern shape design - simple vortex; right- the model with modified stern shape in conformity with the new concept design- multiple vortices).

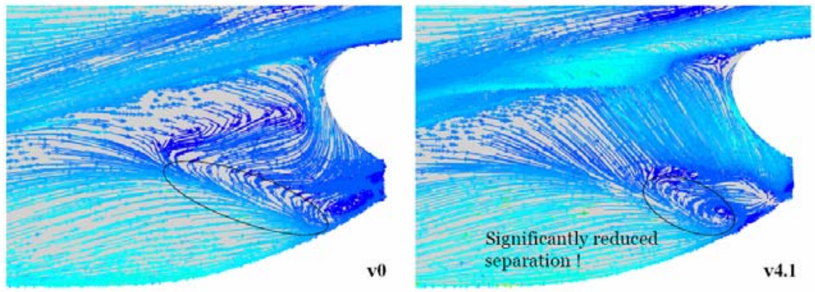


Fig. 11. Limit streamlines on stern surface (left side - the model with initial stern shape design; right side - the model with modified stern shape in conformity with the new concept design).

Therefore, it would be necessary (a much more) reducing or even complete separation and multiple vortices phenomena (within the turbulent boundary layer) avoiding. In this direction, especially the paper's second author thought [15], that we should try to use the inverse piezoelectric effect (electric current → high-frequency generator → piezoelectric driver made of certain ceramic material – Fig. 12), which induces an elliptical vibratory movement (high frequency over 20 kHz), into the elastic side plates (15 mm thickness) in the streamlines direction (of the stern external flowing water).

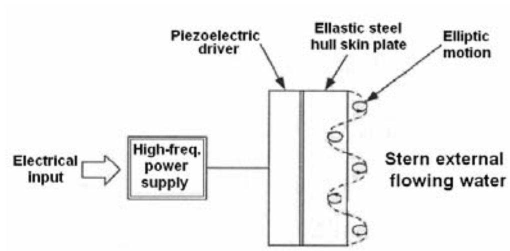


Fig. 12. Principle scheme of an ultrasonic vibrator

It is a known fact that certain special piezoelectric ceramic materials can be used to convert electrical energy into mechanical energy in the form of vibrations of an elastic body. The idea consists in inducing a vibratory movement into ship hull (stern region), so that the surface points have a linear elliptic motion in the streamlines direction of the external flowing water. Water particles, within generated turbulent boundary layer on curved steel surface which vibrates, are expected to reduce skin friction drag coefficient. It is estimated that such a combination of devices can reduce ship forward resistance due to hull skin-water friction reduction by controlling the inside turbulent boundary layer flow characteristics.

8. Conclusions

Based on those provided above, the following conclusions and recommendations can be put forward: - the research creates a new concept of ship aft end form, of so-called crenellated-corrugated type; - the concept was designed and tested in the ICEPRONAV Galati, hydrodynamics laboratories, and modeled and simulated numerically with FLUENT software; - the results of the research showed an improved flow around the hull, a wake which is advantageous from a propulsion point of view, and a reduced ship total forward resistance - the new concepts and ideas proposed seems to be very viable and need be patented; - at the present state of the art, FLUENT can be regarded as a reliable design tool; - ICEPRONAV is proposing further research on the encouraging initial results obtained, with a view to their soon practical application to real ships.

R E F E R E N C E S

[1] *J. Webster, L. Mutnick*, "Future Surface Combatant", Proceedings 25th ATTC Conference, Iowa City, IA, Sept. 1998.
[2] *K. J. Rawson, E. C. Tupper,* "Basic Ship Theory", Longman Scientific & Technical, Fourth Edition, 1994-1995.
[3] *J. P. Ghose, R. P. Gokarn,* "Basic Ship Propulsion", Allied Publishers Pvt. Limited, New Delhi, Mumbai, Kolkata, Lucknow, Chennai, Nagpur, Bangalore, Hyderabad, Ahmedabad, 2004;
[4] *H. C. Tanasescu, Al. A. Vasilescu,* "Mecanica Fluidelor", Curs pentru ingineri, Universitatea "Dunarea De Jos" din Galati, 2000.
[5] *G. K. Batchelor,F. R. S.,* "An Introduction to Fluid Dynamics", *Cambridge University Press, 1991.*
[6] *Al. A. Vasilescu,* "Hydromechanics Course", University of Galati, 1962.
[7] *H. C. Tanasescu,* "New Radial Crenellated-Corrugated Stern Sections (Tanasescu's Stern Shape) Obtained by Progressive Numerical Carving, coupled with an Inverse Problem for Optimizing the Expected Wake", 5th Osaka Colloquium on ADVANCED RESEARCH on Ship Viscous Flow and Hull Form Design by EFD and CFD Approaches, Osaka Prefecture University (OPU), March, 14-15, 2005, Japan.
[8] *H. E. Saunders,* "Hydrodynamics in ship design", The Society of Naval Architects and Marine Engineers 74 Trinity Place, New York 6, N.Y. 1957.
[9] S*ir Horace Lamb,* "Hydrodynamics", Cambridge University, 1952.
[10] *J. N. Newman,* "Marine Hydrodynamics", Massachusetts Institute of Technology, 1982.
[11] *R. Mahu, H. C. Tanasescu,* "A New Stern Concept – Modeling and Simulation using FLUENT 6.3", UK UGM, CHESTER, Oct. 4-5 2007.
[12] *FLUENT 6.3,* "User's Manual", Fluent Incorporated, Lebanon, NH, 2006.
[13] *C. A. J. Fletcher,* "Computational Techniques for Fluid Dynamics", Springer-Verlag, 1991.
[14] *H. C. Tanasescu,* "Numerical Analysis", Curs pentru masterat, Universitatea "Dunarea de Jos" din Galati, 2001.
[15] *J. Bird,* "Electrical and Electronic Principles and Technology", Publisher: Newnes, 2000, ISBN - 10:0750647485.