Slavica Ristić
Leading Research Engineer
Military Technical Institute, Belgrade, SCG

Jovan Isaković
Military Technical Institute, Belgrade, SCG

Milesa Srećković
Professor
Faculty of Electrical Engineering

Dušan Matić
Military Technical Institute, Belgrade, SCG

Comparative Analysis of Experimental and Numerical Flow Visualization Methods

The results of flow visualization around various models in MTI's wind and water tunnels are presented in this paper. They were obtained by different techniques: holographic interferometry, surface oil emulsion, dye and air bubbles. Besides experimental results, the flow simulations by Fluent for the same experimental conditions are presented too. The obtained comparative analyses shows that it is helpful to use both methods, experimental and numerical, in complex flows investigations

Keywords: *wind tunnel, water tunnel, flow visualization, numerical simulation.*

1. INTRODUCTION

Comparative visualization of data from different sources provides useful presentations to highlight similarities or differences. Comparative visualization may be used to compare:

- data from different sources, such as two simulation codes for similar physical phenomena,
- results from numerical flow simulation and images taken during wind tunnel experiments,
- different visualization methods.

Comparison may be performed in different ways. Results of such comparison may be presented side-by-side as well as integrated in a single image by various techniques. It is important to point out that the visualization of the numerical simulation was not a simulation of the experimental visualization technique.

By varying data processing steps and the visualization parameters, the similarity between simulation and experimental visualization may be improved.

The basic fluids in experimental aerodynamics are air and water. They are colorless fluids, transparent media, whose movements around airplane or their models can't be directly observed and recorded. Because of that, the implicite methods and techniques are developed to make flow visible. These methods belong ito a specific scientific discipline of the experimental aerodynamics, to flow visualization [1-3].

The flow visualization methods are classified into three groups. The first group consists of all techniques by which seeding materials are added to the flow. The visualization is obtained indirectly because the motion of seeding material is observed, instead of the fluid itself. The second group are optical methods. The compressible flow can be made visible by optical methods, which are sensitive to changes of the refraction index in the fluid under investigation. The third group of visualization methods is the combination of the above mentioned methods. This group can be applied in the low density gas flows.

The innovation of the optical laser contributes to expanding the application of flow visualization methods in different areas of aerodynamics. Numerical methods, which have been developed during the last several years thanks to powerful computers, are used for numerical flow visualization and for comparison of numerical results with experimental flow visualization in wind tunnels [4-9]. Two goals are achieved: the experimental images are easier to interpret by aerodynamicists, conclusions about important mechanisms are made possible, as well as an insight into visualization techniques. The second goal is to test ability of the code and used turbulence model to predict the flow.

The examples of various flow visualization methods in subsonic, supersonic wind tunnel and water tunnel are presented in this paper. The results of experimental visualization are compared with the visualization of the numerical simulations [1-12].

2. SHORT REVIEW OF OIL SURFACE FLOW VISUALIZATION METHODS

The boundary layer flow visualization is important for description of flow around models and for studying flow around real aircraft and missiles [1-3,6,7]. There are a lot of surface flow visualization techniques. Depending on activated mechanism, these techniques can indicate the flow streams, pressure or temperature distribution. These methods can be applied in wind tunnel or in free flight tests.

Surface flow visualization techniques are based on smearing an emulsion with dye over the surface by the flow when the oil streaks have the forms of streamline. The oil emulsions for surface coatings are prepared with petroleum, kerosene, light transformer oil, light diesel oil, or silicon oil. Different pigments and additives are added. If the model surface is bright-coloured, graphite, lampblack, some paints, kaolin or some other black pigments are added into the mixture. If the model

Received: September 2005, Accepted: November 2005.
Correspondence to: Slavica Ristić
Military Technical Institute,
Ratka Risanovića 1, 11000 Belgrade 35, Serbia
E-mail: s1avce@yahoo.com

© Faculty of Mechanical Engineering, Belgrade. All rights reserved

surface is dark, the powder of titanium dioxide (TiO_2), aluminum trioxide (Al_2O_3), milk and fluorescent pigments are added to mixture. As additive, the oleic acid is often used to control the size of coagulating flocks in the emulsion.

Thickness of the applied layer of emulsion must not disturb the flow in the boundary layer. The emulsion is applied just before a wind tunnel run. It is very important to run the wind tunnel as long as to make possible the development of oil flow pattern. The emulsions can be very easily removed from the surface, they are not aggressive and do not damage the model. TV, cine or still camera are used for recording the visualization effects, during wind tunnel runs, or immediately after runs, to prevent emulsion smear.

2.1 Experiment

Visualization of the flow around axisymetrical model, made from aluminum alloy and steel, was performed in trisonic wind tunnel T – 38 in MTI. Scale of the model is 1:8, corresponding to real torpedo. Length of the model is 967 mm and its diameter is 66.6 mm. Model was fixed to the mechanism for change of angle of attack of the model through specially designed adapter and bent sting. It was possible to achieve desired pitch angle in the range from – 2° to + 18°[6, 7].

In this experiment, in order to ensure maximum visibility of flow visualization effects, satisfactory roughness of the whole surface was obtained and model surface was blackened. Flow visualization was performed on entire model surface. Because of model axisymmetry, one part (left) was covered with continual coating of emulsion, and the other (right) side was covered with dots of emulsion. After every wind tunnel run, the angle of attack was changed, the model was wiped and again prepared for next visualization test. The emulsion, petroleum with oleic acid and titanium dioxide (TiO_2), was deposed on model immediately before each wind tunnel run. Incidence angles of 0°-18° were chosen for recording flow visualization effects. Wind tunnel run time was 50 s.

Recording of flow visualization effects was made after the end of each wind tunnel run. The model was pulled out of the test section, and immediately photographed with Canon A70 digital still camera.

2.2 Numerical simulation

Fluent 6.1 was used for numerical simulations of the turbulent flow around the axisymmetrical body. Solver of the software is based on the finite volume method for discretization of governing equations of the flow. Originally, numerical simulations were performed for a wide range of Reynolds numbers and angles of attack of the body. Flows with seawater and air were examined. Only a part of results, which concerns the numerical simulations of the flow with air, for conditions which correspond to the flow conditions during the experiment in the wind tunnel, are shown in this article [6].

Unigraphics 18.0 was used for modeling of body torpedo without fins and control surfaces, in its real measures. After modeling, the geometry of the body was exported to Gambit 2.0, software for basic geometry modeling and computational grid generation. To achieve the size of the body used in experiments in the wind tunnel, whole geometry was scaled 1:8 [7]. Computational space was in a cylinder shape. Distance from the body to the symmetry boundary (envelope of the cylinder) of the computational space was bigger than 5 diameters of the body.

Tetrahedral hybrid unstructured computational grid with variable density was generated inside of the computational space for each angle of attack of the body separately. Appropriate hexahedral grid for boundary layer was generated in the vicinity of the body. Computational grid density gives good representation of the body geometry. Depending on the angle of attack of the body, number of elements in the generated computational grids varies in the range from 348 000 to 386 000. Average value of the local Reynolds number was in the range from 40 to 71, depending on angle of attack of the body.

Boundary conditions for numerical simulation of the flow are chosen to be the same as the conditions in the wind tunnel during the experiment. Converged solutions for integral quantities of interest were obtained after about 250 iterations. Solution convergence criterion for all calculations was based on the scaled sum of the mass residuals. Value of $1 \cdot 10^{-4}$ was enough to obtain convergence of all flow parameters of interest, primarily resistance coefficient and lift coefficient.

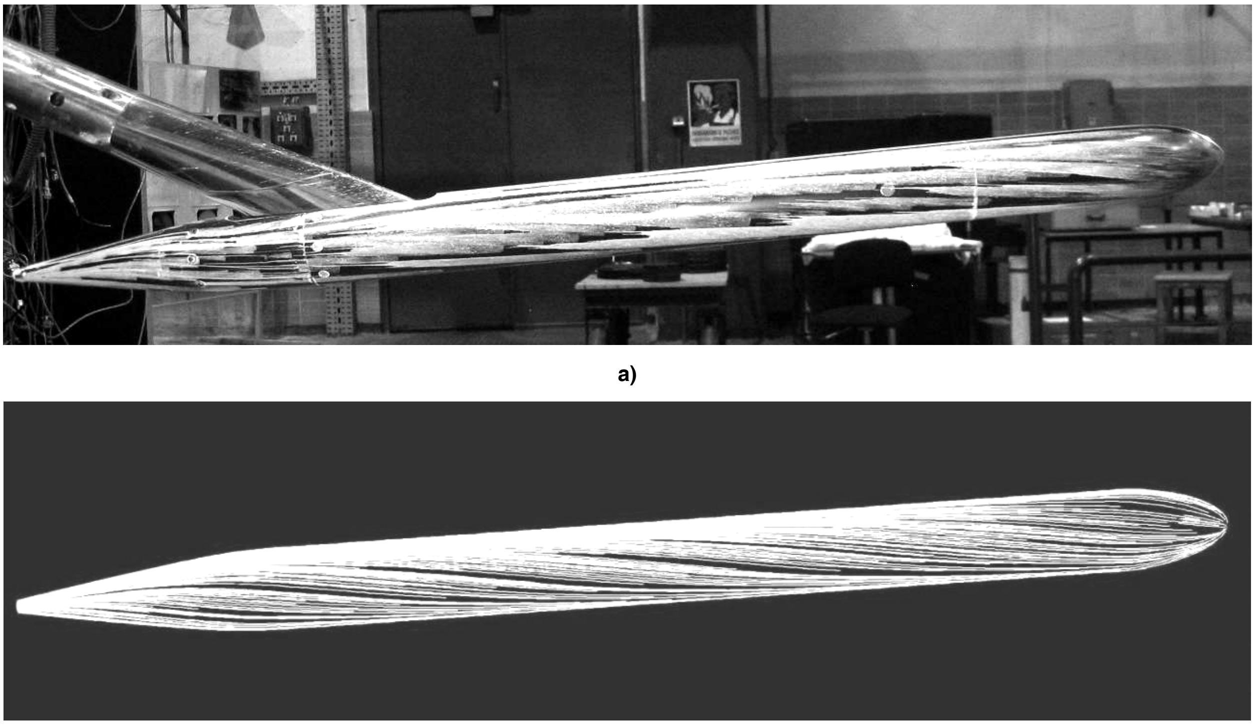
Analysis of photographs (Figs. 1 and 2) demonstrates an excellent agreement of flow patterns obtained by the experiment and the numerical simulations. Certain differences are visible in the area behind the model support sting and in its immediate vicinity because this sting is not included in numerical simulation. Those differences are small, which shows that model support sting consisted of a 15º adapter, and a sting bent through 15º, with diameter $D = 50$ mm, introduses smal disturbances in the flow.

Photographs of experimental flow patterns and numerical simulations flow patterns show that comparison of flow patterns is easier when oil emulsion is deposited in dots over the surface of the model [6,7].

3. HOLOGRAFIC INTERFEROMETRY

Optical methods make possible to visualize and determine aerodynamical parameters of compressible flow in total volume of wind tunnel test section: density, pressure, flow velocity, Mach number, show location of shock and expansion waves, nature and transformation of boundary layer, interaction of different effects in complex flow fields and so on [1-3].

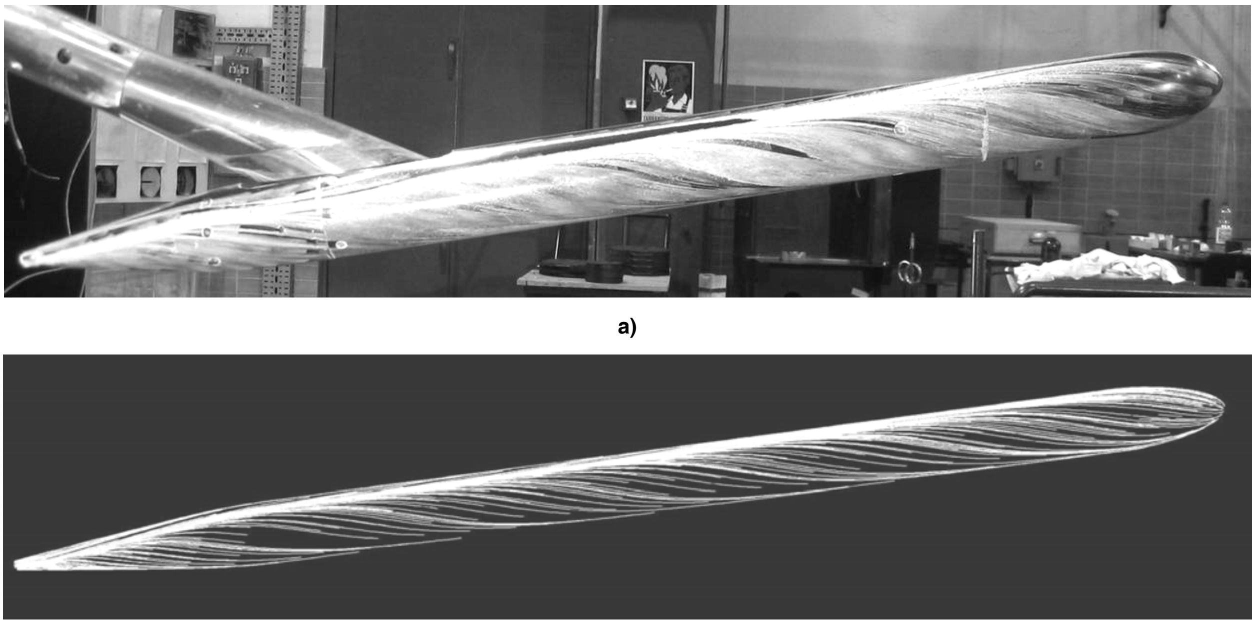
Optical methods make possible to visualize and determine aerodynamical parameters of compressible flow in total volume of wind tunnel test section: density, pressure, flow velocity, Mach number, show location of shock and expansion waves, nature and transformation of boundary layer, interaction of different effects in complex flow fields and so on [1-3].



a)

b)

Figure 1. Flow pattern on the model obtained by the experiment (a) and by the numerical simulation (b) of the flow for M_∞ = 0.3 and α = 4° (side view)



a)

b)

Figure 2. Flow pattern on the model obtained by the experiment visualization (a) and by the visualization of numerical flow simulation (b) for M_∞ = 0.3 and α = 14° (side view)

3.1 Experiment

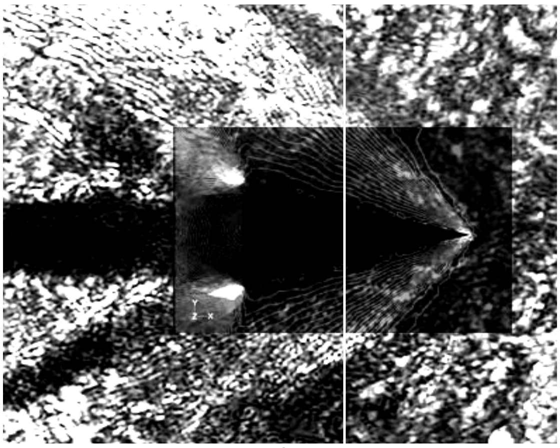
During the last twenty years, holographic interferometry is used for flow visualization in the MTI's labs of experimental aerodynamics. The holographic interferometer has been designed, made and tested for wind tunnel T-38. It is the base for various holographic interferometry applications. The description of the interferometer, the results of the number of experimental visualization tests are presented in the cited literature [8].

The flow around sphere and cone models can be calculated and simulated with different theoretical methods and methods of computational fluid dynamics (CFD) [4,5]. In recent times, flow visualization gives computational base for numerical methods (4,5,9-12) and because of that the flow visualization test by double exposition holographic interferometry has been performed in T-38.

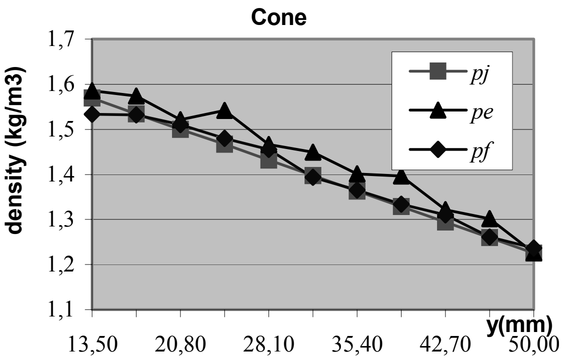
3.2 Analysis of results

The experimental recorded and theoretical simulated isodensity lines of conical flow are presented in Figs. 3 and 4. The upper layer in the holographic interferogram (Fig. 3a) shows calculated isodensity lines around model cone ($\theta_c = 30°$, $l_c = 93.5$ mm and base diameter $D_c = 50$ mm) in the flow with $M_\infty = 1.71$. The upper part of the fig 4a is the interferogram of flow around model cone-cylinder ($\theta_c = 15°$, $l_{cone} = 300$ mm base diameter $\Phi = 160$ mm, $l_{cylinder} = 160$ mm) for $M_\infty = 1.474$. The calculated flow isodensity lines for the experimental conditions and for the same model are presented on the lower part (Fig.4a).

The excellent agreement of theoretical [8-9] and experimental isodensity lines was obtainrd in the region between model surface and shock wave as well as in the region of expansion fans (Figs. 4 and 5). The diagram in figure 3b represents the values for air density and Mach number of conical flow between shock wave and model surface, for section with $x = 50$ mm, $y = 12.5 - 100$ mm, $M_\infty = 1.71$, $MRe = 37.84$ (white vertical line in Fig 3a).



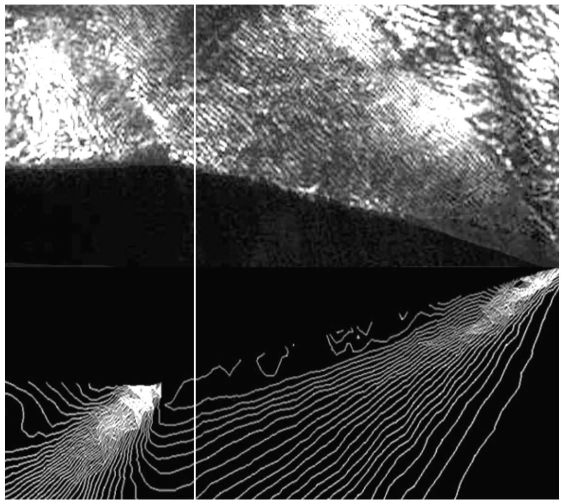
a)



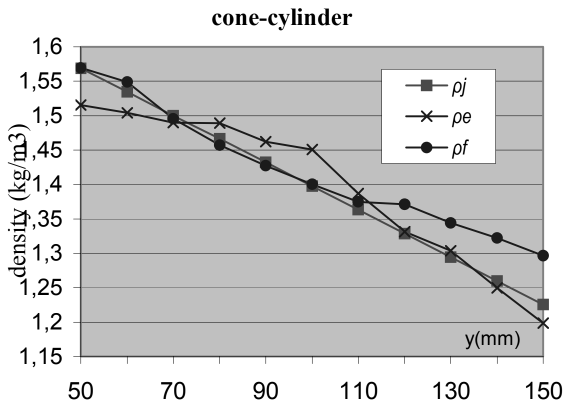
b)

Figure 3. Simultaneous presentation of experiemntal and numerical flow around cone $M_\infty = 1.71$ (a) and density distribution behind shock wave for $x = 50$ mm (b)

The same, for the model cone-cylinder is presented with a diagram in Fig. 4b. The index a (ρj) means theoretical values (AGARD 137, Jones [9]), e (ρ_e) means experimental and f (ρ_f) are the numerical values obtained by Fluent. Good agreement between experiments and theory was achieved. Fig. 4b represents the same results for model cone-cylinder. The density differences of flow behind the shock wave is: experimentally determined in regard to theoretical 2.4%, experimentally in regard to numerical one 3.6% [8].



a)



b)

Figure 4. Simultaneous presentation of experimental and numerical flow around cone-cylinder for $M_\infty = 1.474$ (a) and density distribution behind shock wave for $x = 150$ mm (b)

4. EXPERIMENTAL AND NUMERICAL VISUALIZATION OF THE HIDROFOIL FLOW

4.1 Visualization using dye and gas bubbles

For the purpose of flow visualization in the water tunnel, can be used: food coloring dyes, aniline or methylene dues, permanganate, potassium, ink or fluorescent dyes (fluorescent rhodamine), mixing in milk or alcohol. Very important is to have dyed solution with the same specific density as working fluid. This can be met by mixing the prepared dye with alcohol. The dye can be injected in a tested flow either from a small ejector tube placed at a desired position or from small orifices that are provided in the wall of a model. The dye can be generated in the flow too, but it must be without disturbing the flow [1,2,10-12].

Gas bubbles visualization is an example of application tracer method where tracer particles have low (in the water), or similar density (in the air) as the flow. The observation of such gaseous tracers in a gaseous flow requires the use of optical visualization methods.

The gas bubbles (air, hydrogen) can be injected in the flow or generated by electrolysis. The bubble motion can be photographed or recorded with a move camera. Time of bubble observation in the flow is limited by the dissolution of the bubbles in the fluid.

4.2 Experiment, LDA measurements and flow visualization

Two methods have been used for investigating the flow through straight profile grid (with three profiles), carried out in the water-cavitations tunnel in MTI.

The flow visualization was performed using aniline dyes and air bubbles. They were injected in the flow at the distance of about 1m in front of the model, with a special device. Fig. 5 shows the flow field visualized by air bubbles (a). Figure 5b is the numerical simulation of the filament lines. Results of comparison are presented as integrated, single image 5(c). Good agreement of flow patterns obtained by the experimental visualization and by numerical simulation of the flow has been obtained [12]. Figs. 6a, 6b and 6c shows the simulated flow for $V_\infty = 5.32$ m/s ; (a) $t = 0.04$ s , (b) $t = 0.1$ s and (c), $t = 0.15$ s).

The velocity distribution around central profile has been measured by laser Doppler anemometry. The flow velocity in the free stream has been 5.32 m/s.

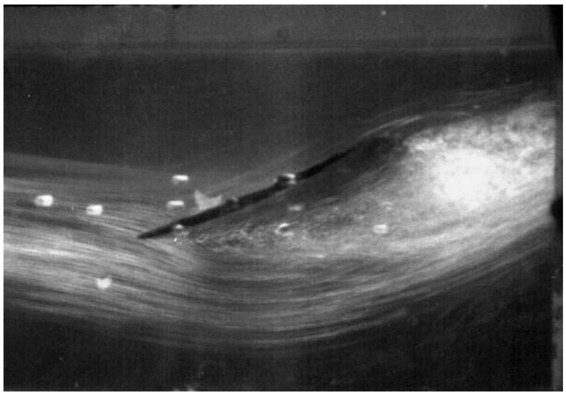
The Laser Doppler Anemometer, or LDA, is a widely accepted tool for fluid dynamic investigations in gases and liquids. It is a well-established technique that gives information about flow velocity. It requires tracer particles in the flow. The method's particular advantages are: non-intrusive measurement, high spatial and temporal resolution, the insensitivity to temperature and pressure variations, the ability to measure in reversing flows, no need for calibration and the ability to measure in reversing flows. These make LDA the first choice in many applications.

The results of 2D LDA measurements have been used as data for making the velocity diagrams shown in fig. 7a, and for definition of boundary conditions in numerical simulation by Fluent 6.1.

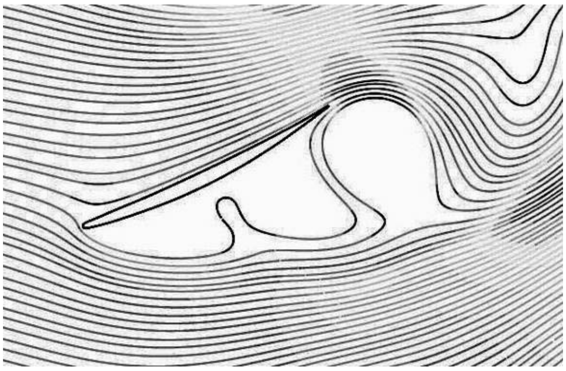
The turbulent flow around the straight profile grid model was simulated by Fluent 6.1 too. Numerical simulations of filament lines (streak line) were performed for $\alpha = 0°$, $12.5°$ and $25°$ angles of attack. The flow was considered as steady for $0°$ hydroprofile angle of attack, and as unsteady for $\alpha = 12.5°$ and $\alpha = 25°$.

The numerical simulation of the velocity vector distribution was made too. Since the velocity vectors are the tangent lines of the streamlines, the vector diagram indirectly makes possible the flow visualization.

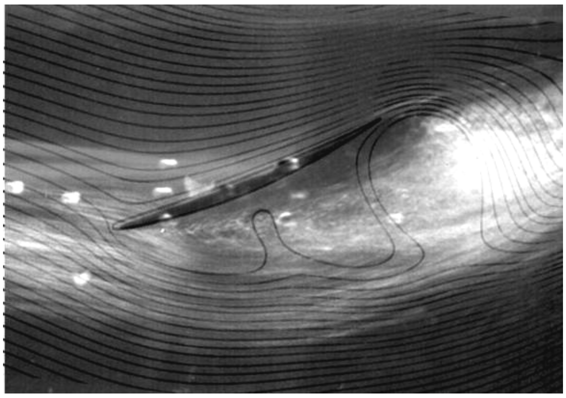
Boundary conditions for numerical simulation of the flow are chosen to be the same as the conditions in the water tunnel during the experiment. Very useful was the LDA measured turbulence in the test section, before the model.



a)



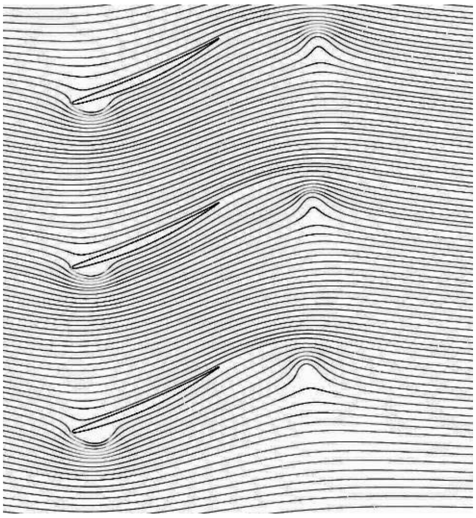
b)



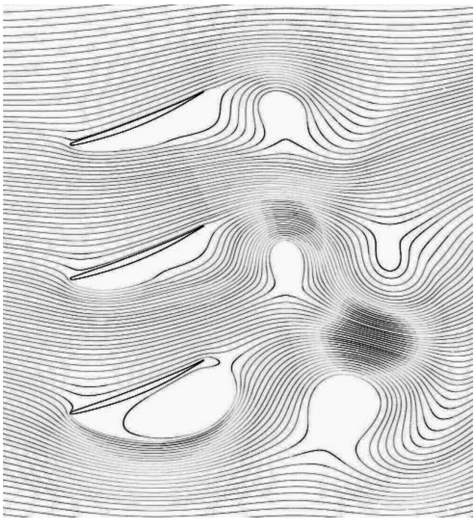
c)

Figure 5. Flow visualization around hydroprofile for V_∞ = 3 m/s ; (a) air bubbles, (b) visualization of numerical data, (c) composite visualization .

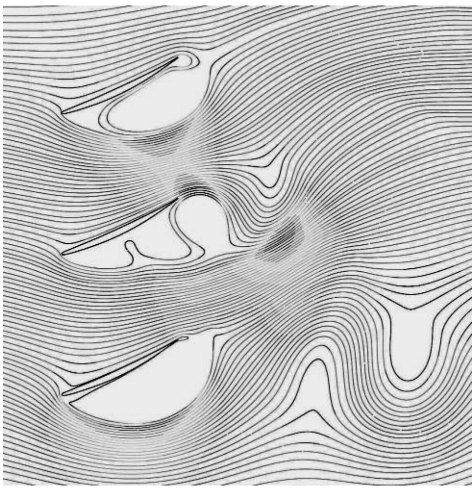
Convergent solutions for integral quantities of interest were obtained after about 550 iterations for stationary flow (angle of attack $0°$). Solution convergence criterion for these calculations was based on the scaled sum of the mass residuals less than $1 \cdot 10^5$. $k - \varepsilon$ RNG model for turbulent stresses was used. Value of $1 \cdot 10^{-4}$ was enough to obtain convergence of flow parameters of interest for cases of no stationary flow ($\alpha = 12.5°$ and $25°$). Number of iterations was about 20 for every time interval



a)



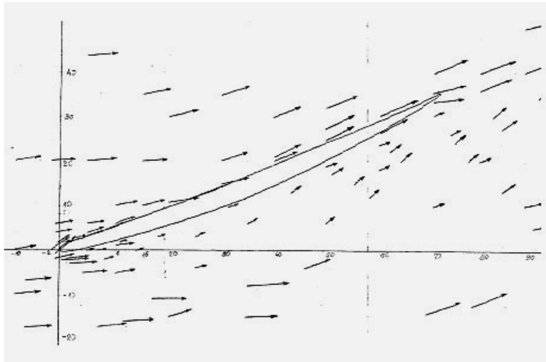
b)



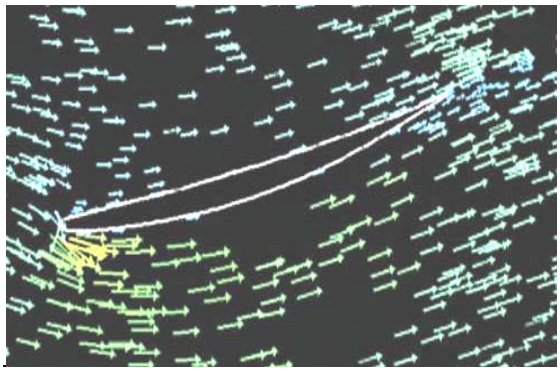
c)

Figure 6. Visualization of numerical flow lines through straight grid with three profiles for V_∞ = 5.32 m/s (a) t = 0.04 s , (b) t = 0.1 s and (c), t = 0.15 s)

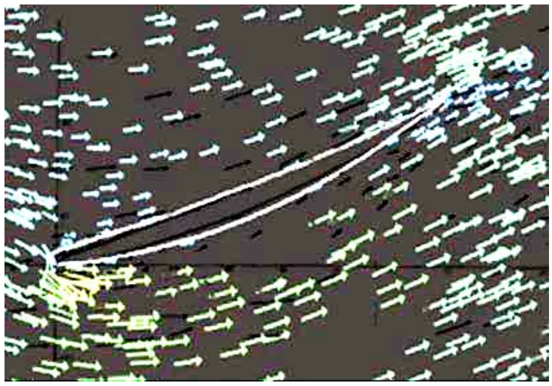
Figure 7 shows velocity vectors around hydrofoil for $\alpha = 25°$, $V_\infty = 5.32$ m/s ; 7a LDA measurement, 7b numerical simulated velocity vectors and 7c composite image.



a)



b)



c)

Figure 7. Velocity vectors around hydrofoil for V_∞ = 5.32 m/s , α = 25° , (a) LDA measurement, (b) numerical simulated velocity vectors, (c) composite image

5. CONCLUSION

The main goal of the test, comparative visualization of different flow in wind and water tunnel, has been successfully performed. Visualization of the flow in the boundary layer by the oil emulsion films for subsonic test, holographic interferometry for visualization of supersonic flow around the model and air bubbles visualization of the flow through profile grid in water tunnel appear as good choice for comparative analyses.

Good agreement of flow patterns obtained by the experiment and by numerical simulation of the flow confirms capability of used numerical technique. These experiments show that methods of visualization are very useful for aerodynamic experiments that were performed in wind and water tunnels and for uncertainty analysis of the numerical simulation results.

REFERENCES

[1] Marzkirich, W.: *Flow visualization*, Academic Press, New York, 1977.

[2] Ristić, S.: *Review of flow visualization methods in wind tunnels*, STI, MTI, Belgrade 1999

[3] Ristić, S.: Optical Methods In Flow Visualization, “Applied Physics in Serbia”, Belgrade, 27-29. may, proccedings, pp. 143-146, 2002

[4] Versteeg, N.K., Malalasekera, W. : *An Introduction to computational fluid dynamics-The finite volume method*, Longnman, London1996

[5] Ferziger, J.N., Perić, M.: *Computational Methods for Fluid Dynamics*, Springer, Berlin, 1997.

[6] Ristić, S., Vitić, A., Matić, D.: Visualization and Numerical Solution of the Flow Around Front Part of Torpedo Model, HIPNEF 2004, Vrnjačka banja, 19-21 may 2004, Proceedings, pp.267-273, 2004.

[7] Ristić, S., Vitić, A.., Anastasijević, Z., Vuković, Đ.: Investigation of Support Interaction Upon Aerodinamic Characteristics of a Torpedo Model in theT-38 Wind Tunnel, STR, Vol.1, No.1.pp. 1-12, 2004.

[8] Ristić, S.: Investigation of Supersonic Flow in Wind Tunnel T-38 by a Method of Holographic Interferometry, STR, vol.54, no.2, pp.3-10, 2005.

[9] Jones, D.J.: *Tables of Inviscid Supersonic Flow About Circular Cones at Incidence*, Agard 137, Editor: Rainbird, W.J., Technical Editing and Reproduction Ltd.,London, 1971

[10] Ristić, S., Majstorović, P.,: Experimental investigation relative flow through model of straight profile grid,, STR, Vol. 50, No. 6, pp. 29-36. 2000.

[11] Ristić, S., Kozić, M.: Investigation of the possibility to apply the LDA method for the determination of pressure coefficients on a high speed axial pump blade model, STR, Vol. 51, No. 5, pp. 25-36, 2001.

[12] Ristić, S., Matić, D.: Experimental and theoretical testing flow through model of high speed axial pump blade profile grid, Technical diagnostique, No. 2, pp.17-25, 2005.

КОМПАРАТИВНА АНАЛИЗА ЕКСПЕРИМЕНТАЛНИХ И НУМЕРИЧКИХ МЕТОДА ВИЗУАЛИЗАЦИЈЕ СТРУЈАЊА

Славица Ристић, Милеса Срећковић, Јован Исаковић, Душан Матић

У раду су приказани резултати визуализације струјања око различитих модела у аеро и воденом тунелу у Војнотехничком институту. Резултати су добијени различитим техникама: холографском интерферометријом, са премазима, бојама и мехурућима ваздуха. Поред експерименталних резултата дате су и симулиране струјне слике добијене Флуентом за услове експеримента. Урађена је упоредна анализа која показује да је неопходно паралелно користити експерименталне и нумеричке методе у испитивању сложених струјања.