VIII International Conference on Textile Composites and Inflatable Structures
STRUCTURAL MEMBRANES 2017
K.-U.Bletzinger, E. Oñate and B. Kröplin (Eds)

NUMERICAL SIMULATION OF AN AIR-SUPPORTED STRUCTURE IN THE AIR FLOW

Nikolay A. Mokin[*], **Alexey A. Kustov**[†] **and Sergey I. Trushin**[††]

[*] Department of Structural and Theoretical Mechanics
Moscow State University of Civil Engineering (National Research University)
Yaroslavskoye shosse, 26, 129337 Moscow, Russian Federation
e-mail: mokiavelli@mail.ru, web page: http://www.mgsu.ru

[†] Department of Metal and Timber Structures
Moscow State University of Civil Engineering (National Research University)
Yaroslavskoye shosse, 26, 129337 Moscow, Russian Federation
e-mail: alexeykustov@outlook.com, web page: http://www.mgsu.ru

[††] Department of Structural and Theoretical Mechanics
Moscow State University of Civil Engineering (National Research University)
Yaroslavskoye shosse, 26, 129337 Moscow, Russian Federation
e-mail: trushin2006@yandex.ru, web page: http://www.mgsu.ru

Key words: air-supported structures, fluid-structure interaction, aeroelasticity, computational fluid dynamics, finite element method, ANSYS.

Summary. Wind load is often the critical load for air-supported structures. For example, high wind demolished an air-supported tennis court roof in Zhulebino district of Moscow on May 29, 2017. Aerodynamic instability (buffeting, flutter, etc.) can produce excessive slack regions and extremely large deflections. That is why the coupled analysis should be carried out to understanding aeroelastic behavior of air-supported structures under wind loads. The technique of the nonlinear numerical analysis of air-supported structures including fluid-structural interaction (FSI) has been described in the present paper. Numerical simulation of the tunnel test of large-scale air-supported model has been carried out as an example of using this technique. Wind tunnel study is described in the papers[1, 2]. Experimental deformed shape, pressure coefficients and aerodynamic forces were compared with the results of presented numerical simulation. Computations were carried out with using of commercial code ANSYS 15.0. Some difficulties appeared during simulation process are discussed. An applicability of the proposed technique to the considered problem was confirmed by a good agreement of the experimental and numerical results. Both methods showed that surface wind loads can increase due to deformation of a structure. These conclusions emphasize the importance of researches on wind interaction with air-supported structures.

1 INTRODUCTION

Air-supported structure – a membrane structure that encloses an occupied space and has a shape that is maintained by air pressure acting within the occupied space. Great opportunities of air-supported and inflatable structures were demonstrated at world's fair Expo '70 in Suita, Osaka, Japan. A lot of scientists worked on the problems of the air-supported structures. Most part of national standards has been developed. For example, in Russian Federation temporally instructions were written in 1977, and, unfortunately, have been used up to now without any changes.

Wind, especially in the form of an uplift, is regularly the critical load case for membrane and cable stresses in light-weight membrane structures[3].

High winds produce movement of the structure and deflections that may be quite significant. Large deflections are undesirable from structural and functional considerations and can be partially controlled by internal pressure of the structure.

Since for most air-structure shapes, the wind pressure is negative over the majority of the surface, the wind pressure plus maximum operating pressure in an appropriate load combination must not exceed the available resistance. The calculated forces in membrane and cables or webs should remain in tension so as to avoid structural instability and excessive motions, insofar as possible without violating the above in some other area of the structure.

There are some recommendations about determination of wind load in the Standard[4]: in case no reliable documentation pertaining to wind effects is available in the literature, experimental procedures are recommended for ascertaining wind loads. So, physical experimental approach is considered as basic research method of wind-structure interaction. Thus, a lot of tunnel tests on the air-supported structures have been carried out during the 1970-80th [1, 2, 5, 6, 7]. Experiments show that wind load on air supported structures is higher than on traditional rigid constructions of the same shape[1].

Tunnel tests with aeroelastic behavior of model are very sophisticated, and conditions of similarity can be met only partially. Amount of coupled multi-physical problems, which allowed to successfully solving, has been highly grown in last years. The reason is fast development of the computer technologies and software. Numerical simulation of strong coupled fluid-structure interaction (FSI) becomes “usual” nowadays. One can find a lot of test cases (benchmarks) in the paper[8] and its references.

There are some research works on the coupled numerical simulation of membrane structures under wind loads[3, 9], but not so much addressed to air-supported structures.

In this paper it is proposed to use commercial code ANSYS 15.0 for the numerical simulation of aeroelastic behavior of air-supported structure in the air flow. Results of fully coupled FSI numerical simulations can be used for improvement for design and load-analysis standards addressed to the air-supported structures.

For the treatment of nonlinear multi-physics problems, two general approaches can be identified[3]:

1) **Simultaneous** analysis (also referred to as monolithic analysis): the entire multiphysics problem, with all physical fields involved, is summarized in one set of equations, discretized, and solved as one.

2) **Partitioned** analysis: the physical fields are solved independently. The term ’partitioned’ refers to a spatial decomposition, which in the case of fluid-structure interaction means the decomposition into a structural and a fluid domain. The coupling of the individual fields is realized by the exchange of boundary conditions.

In present work we use the second strategy - partitioned analysis. ANSYS Fluent solves fluid dynamics, ANSYS Mechanical performs transient structural analysis, and the

Workbench System Coupling organizes data transfers between these solvers.

There are two general types of FSI simulation:

1-way FSI ("import load"). The results from solution of computational fluid dynamics (CFD) are transferred as pressure load to the transient structural analysis. Deformations of model doesn't influence on CFD solution.

2-way FSI (bidirectional). The solution of two-way fluid-structure interaction requires co-simulation between computational fluid dynamics and structural mechanics. There are two (or more) data transfers. Exchanging data is performed after every coupling iteration during time step. In our case, deformations of the air-supported shell from ANSYS Mechanical are imported to the ANSYS Fluent as a dynamic mesh motion (solution often need to include remeshing of the fluid domain to the process). Then CFD part of problem is solved on the new mesh and resulting pressure distribution forms new load case for the transient structural analysis.

2-way FSI simulation is more accurate, but, of course, more expensive (in point of view of solution setup, computer resources and calculation time) and less stable.

2 EXPERIMENTAL INVESTIGATION

This work is based on the experiment data obtained by V.P. Polyakov et al in 1970th. Wind tunnel study is described in the papers [1, 2]. It was a tunnel test of an air-supported 3/4 spherical shell with diameter of D = 4.2 m and height of H = 3.36 m. One can see that the sizes of an experimental model are comparable to the sizes of real constructions.

Wind tunnel T-101 (TsAGI, Zhukovsky, Moscow Region) is a subsonic continuous-operation, closed-layout wind tunnel with two reverse channels and an open test section. Area of its elliptical cross section is 264 m^2.

Experiments were conducted at several tunnel speeds from 12 to 45 m/s, which comparable with real wind speed during storms and hurricanes. Value of the Reynolds number based on the tunnel flow speed V = 40 m/s and the initial diameter of the structure D = 4.2 m was $Re = VD/\nu = 1.15\cdot10^7$.

Besides the great sizes of model, it should be noted the fact that the data set for comparison is presented rather fully: flow speed V and internal pressure p in different combinations, deformed shape of the structure, which was measured by stereophotogrammetry method and surface distribution of pressure coefficient C_p.

There were data of two series of tests:

1) Internal pressure was constant p = 1000 Pa, flow speed V was changed in such range: 12; 20; 30; 40; 45 m/s.

2) Flow speed pressure was constant V = 40 m/s, internal pressure was changed in range: 500; 1000; 1500; 2000 Pa.

In present work, we consider the first case of constant pressure p = 1000 Pa.

We enter the designation (1):

$$\psi = \frac{p}{q} \tag{1}$$

where q, Pa, is dynamic pressure of air flow, calculating by formula (2):

$$q = \frac{\rho V^2}{2} \tag{2}$$

where ρ is air density at 15°C, $\rho = 1.225$ kg/m^3.

The value of parameter ψ influences significantly on a membrane shape in the air flow. When $\psi < 1.0$ the dent is formed in front of shell. This case corresponds to the greatest distortion of an initial form of a shell, and formation of the dent can be referred to local loss of stability. This scenario presents the greatest difficulties for numerical modeling, that's why we focused on the value of $\psi \approx 1.0$. It can be observed when internal pressure $p = 1000$ Pa, and air speed $V = 40$ m/s ($q = 980$ Pa). For this case, there are two experimental curves for C_p along main meridian from two series of experiments (see Figure 4).

The material of membrane is rubber-coated kapron fabric (art. 51-060) 0.6 mm thick. As known, coated fabric has nonlinear mechanical properties. Unfortunately, only two moduli of elasticity were presented in the paper[1] – in warp and fill direction. There is no information about level of stress corresponded to this values. Also, there is no information about shear modulus and Poisson's coefficients. That's why mechanical properties, used in simulation, were adjusted during trial numerical calculations so that the deformed shape of numerical model membrane was close to the experimental one.

The air-supported model was mounted on the horizontal round plate. But, we have no geometrical parameters (height from floor, diameter, and thickness) of the plate. The lack of the specified information results in additional errors in results of numerical modeling.

3 NUMERICAL SIMULATION

3.1 On the simulation of fluid-structure interaction (ANSYS System Coupling)

The Workbench System Coupling component system is an easy-to-use, all-purpose infrastructure that facilitates comprehensive multidisciplinary simulations between coupling participants.

In this paper we have been carried out calculation using ANSYS Workbench platform. Project Schematic of the 2-way FSI analysis is shown on Figure 1. One can see different modules used in analysis and data links between them.

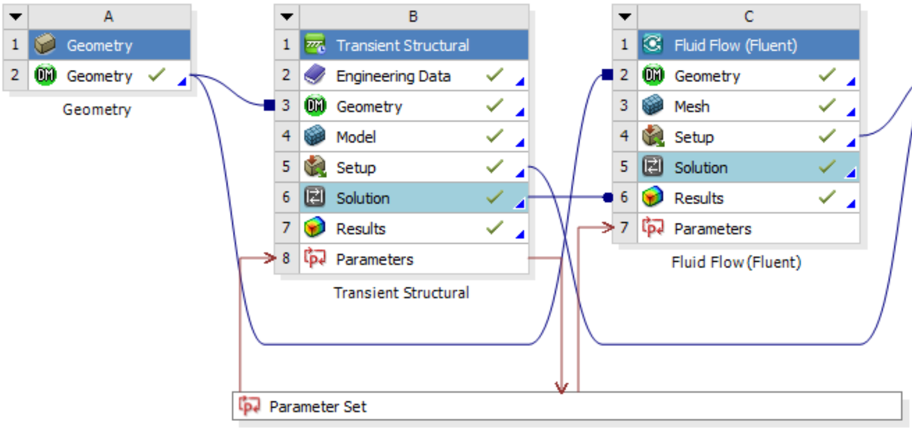

Figure 1: Project Schematic of the 2-way FSI analysis

3.2 Fluid model (ANSYS Fluent)

The fluid solver software used in this work is ANSYS Fluent based on the finite-volume method (FVM). In the finite volume method, volume integrals in a partial differential equation that contain a divergence term are converted to surface integrals, using the divergence theorem. Shell deformations results in need of the transient solution. The second order implicit transient formulation was used in this work. “Coupled” pressure-velocity coupling scheme were selected.

The carried-out calculations were directed to approbation and verification of a calculation procedure. Present results have been obtained on the rough unstructured tetrahedral mesh with rather small number of nodes (about 400000). This mesh was created by automatic procedures of ANSYS Meshing. Fluid domain has been a rectangular box with following dimensions: length – 25D, width – 10D, height – 8D (where $D = 4.2$ m is diameter of shell).

We have used diffusion-based smoothing and dynamic remeshing after every coupling step. These actions have prevented emergence of errors.

Turbulent intensity has been adopted as 1%, because it is usually low in tunnel tests. Air velocity at the inlet has been adopted constant. Gauge pressure at the outlet has been retained equal to the default value of 0 Pa. Top and lateral sides of fluid domain has been accepted as symmetry boundary conditions. Shell surface has been accepted as no-slip wall with 1 mm uniform roughness.

The boundary layer near the shell has been meshed into prismatic cells (Figure 2, a). Hexahedral structured has been already created, but it will be used in future work.

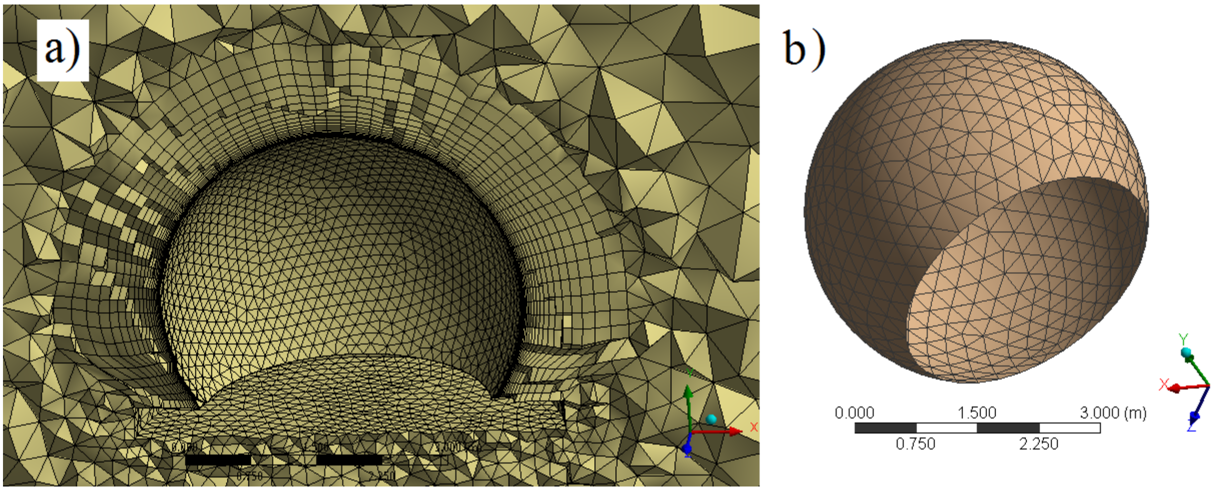

Figure 2: Computational models used in different modules:
ANSYS Fluent (a), ANSYS Mechanical (b)

We have decided to find out what model of turbulence within RANS approach[10] will provide the best coincidence of numerical and experimental results. Such approach the most economic from the point of view of requirements to mesh and time discretization and also to the computer resources.

Three RANS-based eddy-viscosity turbulence models have been considered in the fluid simulations:

1) One-equation The Spalart–Allmaras model[11]
2) Two-equation k-ε turbulence model[12]
3) Two-equation k-ω SST (Menter's Shear Stress Transport) turbulence model[13]

The last one model combines the k-ω turbulence model and k-ε turbulence model such that the k-ω is used in the inner region of the boundary layer and switches to the k-ε in the free shear flow. Thus advantages of both models are used in the respective regions.

3.3 Structural Model (ANSYS Mechanical)

ANSYS Mechanical is based on finite-element method (FEM). The structural dynamic solution has been got by Newton-Raphson method with taking into account large deflections (geometrical nonlinearity). Internal pressure has been simulated as a constant follower distributed force, changing of its value in the experiment has been neglected.

In this paper, we use Shell181 element type for the structural analysis. Shell181 is a three- or four-nodes element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes (if the membrane option is used, the element has translational degrees of freedom only). Shell181 accounts for follower (load stiffness) effects of distributed pressures. The triangle form is generally more robust when using the membrane option with large deflections[14]. Finite element mesh is shown on the Figure 2, b. Mesh size has been accepted as 0.3 m.

Shell181 has been chosen, because it has a "membrane" option. Elements with this option activated allow deformation in the plane of the surface only (that is, stresses do not vary through the thickness and all stress components with respect to the thickness direction are zero). Only membrane stiffness is accounted for. Shell bending and transverse shear stiffness are excluded; therefore, only translational degrees of freedom are retained[14].

During the modeling process we had to apply pressure load onto inner face of shell and assign fluid-solid interface to the outer face. In ANSYS Mechanical there is no difference between outer and inner faces of surface. That's why pressure load is applied to the fluid-solid interface. But all loads applied onto the fluid-solid interface are ignored (it is restriction of using the Workbench System Coupling). We had to separate membrane into two parts of half thickness (0.3 mm, see Table 1). Linear bonded contact (MPC formulation) has been applied on these membranes.

For the numerical model, a simple linear orthotropic plane stress material model is used. This simple approach is allowed[15], because real behavior of fabric is very sophisticated, but information about material used in the experiment, was scanty. That's why we had to perform several trial simulations that helped to determine the mechanical properties of fabric. Besides, the simple material model reduces time of carrying out calculation.

The formula (3) describes the stress-strain relationship:

$$\begin{bmatrix} \varepsilon_w \\ \varepsilon_f \\ 2\gamma_{wf} \end{bmatrix} = \begin{bmatrix} \frac{1}{E_w} & \frac{-\nu_{fw}}{E_f} & 0 \\ \frac{-\nu_{fw}}{E_f} & \frac{1}{E_f} & 0 \\ 0 & 0 & \frac{1}{G_{wf}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_w \\ \sigma_f \\ \tau_{wf} \end{bmatrix} \tag{3}$$

where E_w and E_f are elastic moduli in respectively warp and fill (weft) direction, MPa;
ν_{wf} and ν_{fw} are the Poisson's coefficients;
G_{wf} is the shear stiffness, MPa;
ε_{wf} and ε_{fw} are the normal strains;
σ_{wf} and σ_{fw} are the normal stresses, MPa.

A local spherical coordinate system has been used to orient the local axes of elements for the correct modeling of the meridian patterning.

As the stiffness matrix (eq.(3)) is symmetric, the membrane properties have to comply with the following reciprocal relationship:

$$\frac{\nu_{wf}}{E_w} = \frac{\nu_{fw}}{E_f} \tag{4}$$

Table 1 contains material properties, which were used in the calculations.

Table 1: Used reference material parameters

E_w, MPa	E_f, MPa	ν_{wf}	ν_{fw}	G_{wf}, MPa	t, mm
200	185	0.1	0.0925	0.1	0.6 (0.3+0.3)

4 RESULTS & DISCUSSION

On the Figure 3 one can see the contours of the pressure coefficient C_p in the plane of main meridian as results of 1-way FSI analysis (a) and 2-way FSI analysis (b). The minimum values of C_p (-1.45 and -1.22) differ significantly. Thus wind load (depending on pressure coefficient) is higher in case of 2-way FSI.

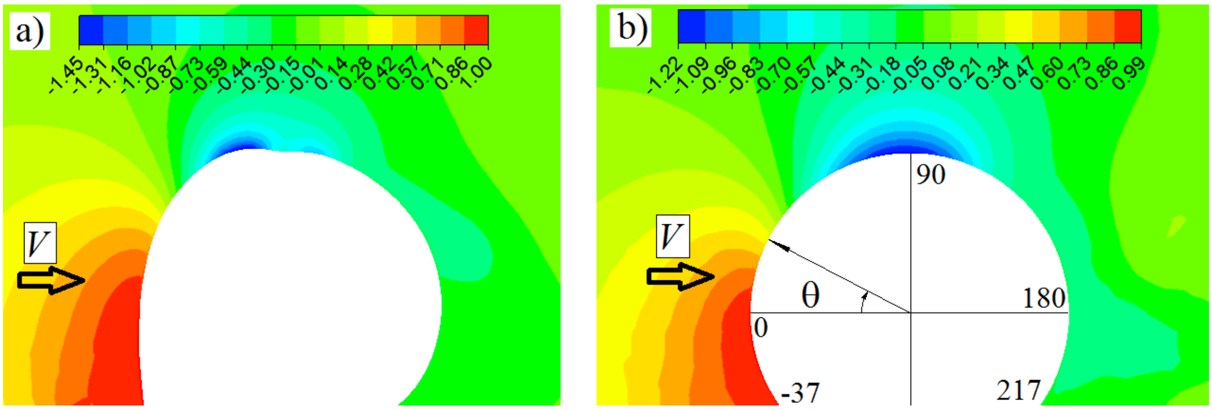

Figure 3: The distributions of pressure coefficient C_p in the main meridian plane: 2-way coupled FSI analysis (left) and 1-way FSI analysis (right). In both cases $V = 40$ m/s

Charts of pressure coefficient C_p for different turbulent models are shown on the Figure 4 and Figure 5. X-axis is the latitude angle θ (reference direction is shown on the Figure 3, b). There are two minimum peaks on the Figure 5. The second one arises because of emergence of deepening on the top of shell.

The k-ω SST turbulence model demonstrates better agreement of the experimental and numerical results than other turbulence models used in this work (k-ε and Spalart–Allmaras). It should be noted, that there are two curves (similar, but not identical) for the experimental results from two series of tests (in case of $V = 40$ m/s).

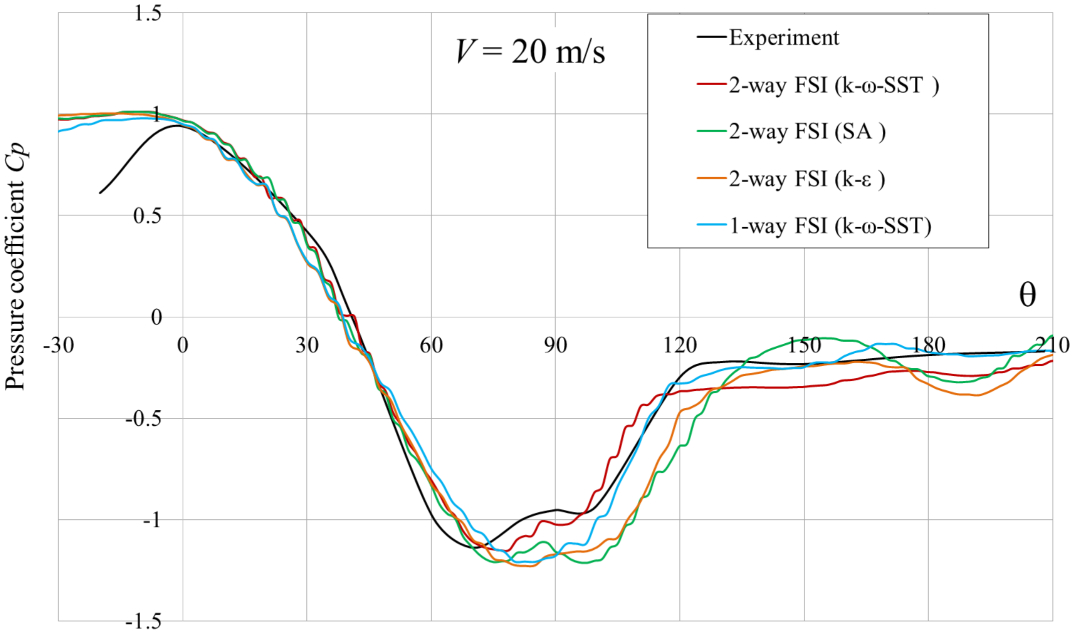

Figure 4: Results of experiments and numerical calculations of pressure coefficient C_p for different turbulent models on the line of main meridian. Internal pressure p = 1000 Pa, air speed V = 20 m/s, ψ = 4.08

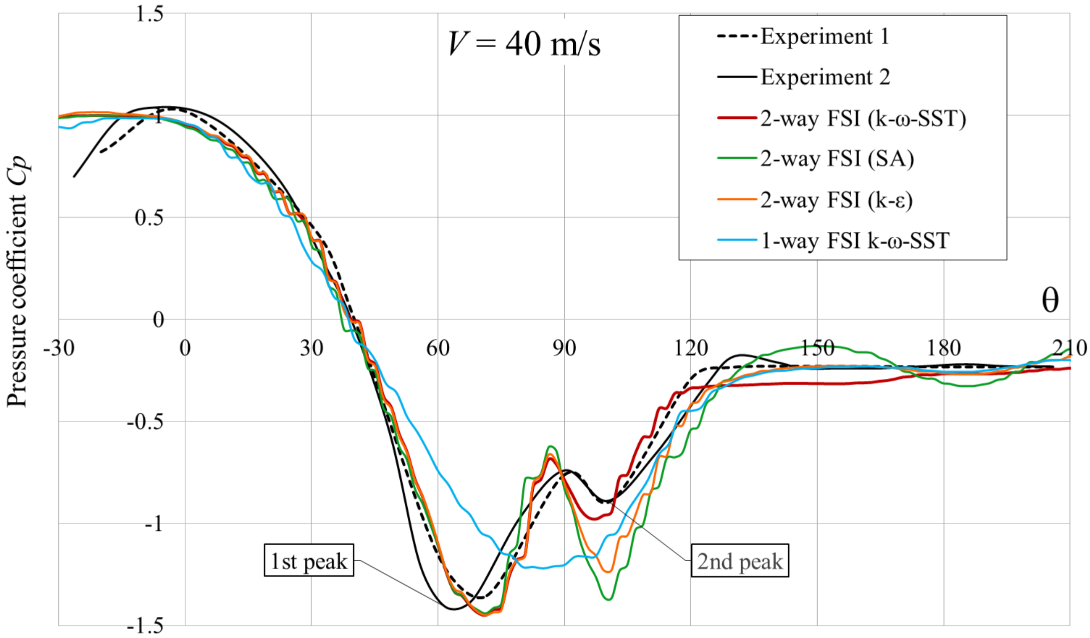

Figure 5: Results of experiments and numerical calculations of pressure coefficient C_p for different turbulent models on the line of main meridian. Internal pressure p = 1000 Pa, air speed V = 40 m/s, ψ = 1.02

The deformations of the shell are shown in the real scale on the Figure 3, and on the Figure 6. One can notice small deepening on the top of the shell. This deepening is appeared only when orthotropic material model with higher meridian modulus is used.

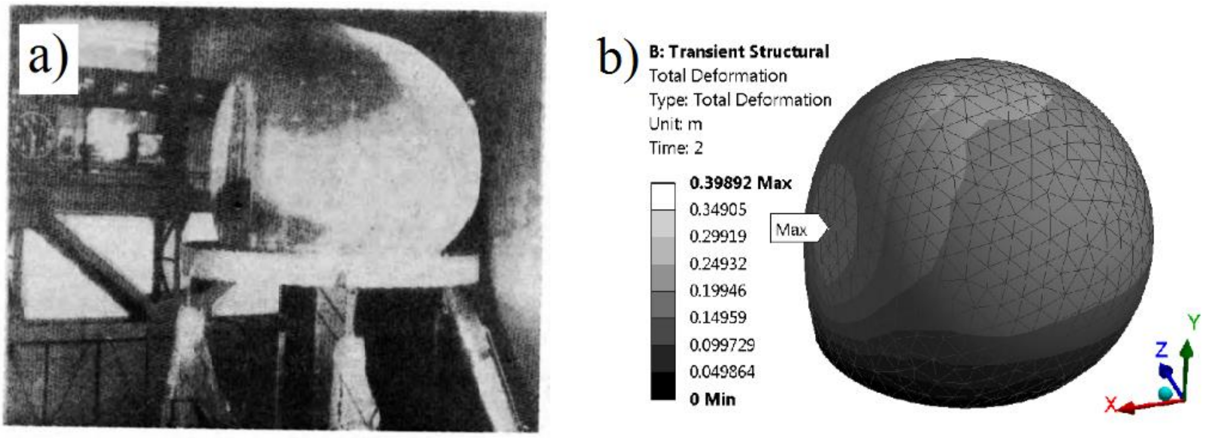

Figure 6: The deformed shape of the air-supported structure in the air flow. Photo of experiment (a), numerical results (b). Internal pressure p = 1000 Pa, air speed V = 40 m/s, ψ = 1.02

5 CONCLUSIONS

The fully coupled FSI simulation of air-supported structure in the air-flow is presented and discussed in this work. The comparison of the numerical and experimental results allows drawing the following conclusions.

Experimental and numerical methods showed that surface wind loads can increase due to deformation of a structure. These conclusions emphasize the importance of researches on wind interaction with air-supported structures.

An applicability of the proposed technique to the considered problem was confirmed by a good agreement of the experimental and numerical results.

The k-ω SST turbulence model demonstrates the best agreement of the experimental and numerical results than other turbulence models used in this work (k-ε and Spalart–Allmaras). That's why the k-ω SST model is planned to be used in further calculations.

When values of parameter ψ are near or less than 1.0, the initial spherical shape of the membrane becomes highly distorted. Therefore, location of the flow separation point is easier to define. It explains good coincidence of results at high speeds of a stream even using coarse mesh.

The orthotropic mechanical properties of membrane material significantly influence to the deformed shape. It is shown, that deformations, in turn, affects to the distribution and values of pressure coefficient C_p (the second peak doesn't appear and the first is smaller).

Using of the linear orthotropic model for the membrane material is reasonable compromise of accuracy and speed.

Periodic separation of vortices (like described at paper[16]) wasn't observed, because of using the coarse tetrahedral mesh and RANS approach. Hexahedral detailed mesh combined with using of DES approach solves this problem.

Received results are the basis for future work. The following steps are planned:

- correct simulation of transient response of vibrating air-supported structure including vortex shedding (by using detached eddy simulation approach and detailed hexahedral mesh);
- damping factors need to be clarified;

- fully coupled FSI numerical simulation of the real scale air-supported structure in the more natural turbulent wind flow.

6 REFERENCES

[1] D.A. Beylin, V.P. Polyakov, V.N. Shmyreva, V.A. Yakovlev, *"Using of stereophotogrammetry for the investigation of the stress-strain state of soft spherical envelope in the air-flow" (Russian)* Uchenye zapiski TsAGI [TsAGI Science Journal]. (1982) **6**:66-76. Available: https://cyberleninka.ru/article/n/ispolzovanie-stereofotogrammetricheskogo-metoda-dlya-issledovaniya-napryazhenno-deformirovannogo-sostoyaniya-myagkoy-obolochki

[2] V.P. Polyakov, D.A. Beylin. An experimental study of the wind pressure on the soft orthotropic spherical shell. (Russian) *Soobshcheniya DVVIMU po sudovym myagkim obolochkam [DVVIMU reports on the ship soft shells]*. (1977) **35**:215-227.

[3] A.M. Kupzok, *Modeling the interaction of wind and membrane structures by numerical simulation* (Dr.-Ing. Thesis), Technical University of Munich, 2009

[4] ASCE 17-96, *Air-supported structures,* American Society of Civil Engineers, 1997

[5] Newman, B.G., and Goland, D. Two-dimensional inflated buildings in a cross wind. *Journal of Fluid Mechanics* (1982) **117**:507-530.

[6] Newman, B.G., Ganguli, U. and Shrivastava S.C. Flow over spherical inflated buildings. *Journal of wind engineering and industrial aerodynamics* (1984) **17.3**:305-327.

[7] Kawamura, S. and Tatsuhiko, K. An experimental study of a one-membrane type pneumatic structure - wind load and response. *Journal of Wind Engineering and Industrial Aerodynamics* (1986) **23**:127-140.

[8] Bathe, K.-J. and Ledezma, G.A. Benchmark problems for incompressible fluid flows with structural interactions. *Computers & structures* (2007) **85.11**:628-644.

[9] Michalski, A., Kermel, P.D., Haug, E., Löhner, R., Wüchner, R., and Bletzinger, K.U. Validation of the computational fluid-structure interaction simulation at real-scale tests of a flexible 29m umbrella in natural wind flow. *Journal of Wind Engineering and Industrial Aerodynamics* (2011) **99.4**:400-413.

[10] D.C. Wilcox, *Turbulence Modeling for CFD,* DCW Industries, Inc., 3rd edition, 2006

[11] Spalart, P.R. and Allmaras, S.R. A one-equation turbulence model for aerodynamic flows. *AIAA Journal* (1992) **94**:5-21.

[12] Launder, B.E., Spalding, D.B. The numerical computation of turbulent flows, *Computer Methods in Applied Mechanics and Engineering*, 1974, **3.2**:269-289.

[13] Menter, F.R. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA journal* (1994) **32.8**:1598-1605.

[14] *ANSYS Mechanical APDL Element Reference. Release 15.0,* SAS IP, Inc., 2013

[15] ASCE/SEI 55-10, *Tensile membrane structures,* American Society of Civil Engineers, 2010

[16] Taneda, S. Visual observations of the flow past a sphere at Reynolds numbers between 10^4 and 10^6 *Journal of Fluid Mechanics* (1978) **85.1**:187-192.