



UDC 004.94

DOI <https://doi.org/10.15673/>¹V. Petrov, PhD of Technical Science, Associate Professor, E-mail: 0673972002@ukr.net

ORCID: 0000-0003-3262-9463

¹D. Goncharenko, Master's degree, E-mail: dmitriy_goncharenko@ukr.net²Zhdanov O., PhD of Technical Science, Associate Professor, E-mail: ajdanow1945@gmail.com

ORCID: 0000-0002-3304-5729

¹Department of Technological Equipment of Grain Productions, Tel. 0673972002, Odesa National University of Technology, 112, Kanatna Str., Odesa, 65039, Ukraine²Одеська державна академія будівництва та архітектури, 4, Didrihsona Str., Odesa, 65029, Ukraine

AERODYNAMICS OF INDUSTRIAL BUILDINGS

Annotation

The design of metal silos and other structures (towers, silo galleries) of the grain receiving point under the influence of wind load is considered in the work. The positive and negative aspects of the structures of the silos under consideration have been identified. The method of researching wind load on silos in the ANSYS CFX package was applied. A three-dimensional model of industrial buildings was made and, taking into account the recommendations, the estimated volume around the research object was adopted. The volume for air movement around industrial buildings is also defined. Taking into account the method of finite elements, a grid was built on the layout of the research object. Assigned initial data and boundary conditions. The structure of six metal silos with a diameter of 15.2 m and a height of a cylindrical wall of 12 m of the Drokiivskiy HPP (Republic of Moldova) was taken as the object of the study. Wind pressure on cylindrical walls and conical roofs is estimated by the finite element method using the ANSYS CFX package for wind direction perpendicular to the line of silos at maximum speed. Pressure isobars are found in various auxiliary vertical and horizontal planes. Lines of equal air flow velocities and their vector analogues are constructed. Zones of maximum and minimum pressure on industrial facilities have been established. The correspondence of the general picture of the pressure distribution along the wall in the circumferential direction when the aerodynamic coefficient changes is established. Determined wind loads on structural elements of industrial buildings. Calculations were performed using the functional calculator built into this ANSYS CFX package. To verify the numerical consistency of the obtained results, additional research is required. Due to the movement (animation) of auxiliary vertical planes, it is possible to consider the dynamics of changes in pressure and velocity of the incoming air flow in the calculated volume.

Keywords: metal cylindrical silo, wind pressure, computational aerodynamics, finite element method, ANSYS CFX package.

Introduction

In the process of operation, silos are exposed to various climatic influences, including wind, which manifests itself in excessive external pressure and back pressure (diluting effect of wind) [1-3]. Strength assessment

under wind pressure is mandatory when designing similar structures and is regulated by DBN B.2.2-8-98 and DBN B.1.2-2:2006, however, the calculation method is far from perfect. This is due to the insufficient study of the features of the distribution of wind pressure along the wall, the covering of such grain storages, which are located both separately and grouped (to optimize the processes of grain loading and unloading). Fig. 1 shows the results of the storm load on the silos, which led to their destruction [1].

Statement of the problem

The task of this study was to evaluate the distribution of wind pressure on the cylindrical wall and conical roof of grouped cylindrical granaries.

Of the existing methods for calculating the distribution of wind pressure on engineering structures, the most informative and most recently used method of computational aerodynamics based on the ANSYS CFX complex [4], which uses the finite element method, is the most in-



Fig. 1. General view of silos after the hurricane, Ohio, USA



formative. Let's note some features of the use of this complex.

1. CFX software requires more temporary memory than Fluent software due to the ability to store values at grid points during simulation. Of course, the CFX software will have a more appropriate convergence than the Fluent software due to the use of the quasi-transient coupling method. Also, CFX software has more and wider chaotic models.

2. CFX software, like other computational fluid dynamics software, has the ability to write macros that can be used to apply various initial and boundary conditions in the software.

3. Another important feature of CFX software is the ability to get more graphical results and higher quality simulation videos.

4. The ability to analyze multiphase problems, fluid structure interaction with models, turbulent flow models, turbomachinery, heat transfer, viscous and inviscid flow studies, nanofluid analysis, etc. are just some of the known features of this software.

5. The speed, accuracy and efficiency of CFX software in simulating problems related to rotating equipment and turbomachinery (pumps, compressors, fans, gas and hydraulic turbines) are greater compared to other similar software such as Fluent.

6. However, it is less powerful than the Fluent software in problems that do not require special physical knowledge, such as reaction domain and dynamic boundary value problems.

7. Ansys CFX and Fluent software are often used by professionals when working with industrial objects.

8. The CFX software is divided into three stages: CFX-Pre, CFX-Solver, and CFD-POST, which are used to call the mesh and apply the solution settings and conditions, execute the solution process, and obtain the solution results (post-processing).

To solve the problem, you need to perform several steps.

First step: Create a solution domain.

In general, to solve a problem in Ansys software, we must first create and mesh the solution domain. Then, in the next step, this domain is transferred to the solver environment and the phenomenon conditions are applied to the solution domain. For example, imagine that the goal is to simulate the movement of air around industrial buildings. For this, at the first stage, it is necessary to prepare the plane and the volume around the industrial buildings.

Second step: network generated domain solution. In the next step, the projected area should be marked with a grid in one of the ANSYS programs. The grid means the division of the entire area into many elementary volumes (tetrahedra or other cells), whose equations are solved numerically in these units. Note that the method of communication has a great influence on the time of solving the problem, as well as on the quality of the final solution.

The third step: transfer the model to the solver and run the simulation.

In this step, the mesh domain is passed to the solver and the simulation is performed on it. A solver is

software that solves the physical equations for a generated domain. The most important solvers in Ansys are CFX and FLUENT.

CFX is mostly used to analyze systems based on different machines. In fact, this software can be used to optimize systems such as turbines, compressors, jet engines or pumps. It is possible to perform an analysis of unsteady flow inside a tank, analysis of two-phase flow inside a bent pipe with a half-open valve, and finally, the interaction of the structure with the fluid. First, you need to create the solution domain and mesh in CFX, then apply the boundary conditions, get the initial data, and check it.

The study of the model consists of the following stages:

- selection of the calculation module;
- specifying the physical properties of materials;
- construction of a geometric model;
- creation of a finite element grid;
- imposition of loads and fastenings;
- selection of initial data;
- conducting calculations;
- analysis of the obtained results.

Materials and methods

The results of the study of the effect of wind on a smooth vertical cylindrical tank, which is free-standing, with a detailed analysis and comparison of regulatory documents of Ukraine and the countries of the European Union, are given in modern literature. In DBN B.1.2-2:2006, wind loads and effects are given, as well as formulas for the task of wind pressure distribution when evaluating the strength of vertical cylindrical structures being designed, both for single and grouped ones, are given.

This study is designed to reveal some features of the wind load on the grouped metal cylindrical silos of the Drokiivsk HPP (Republic of Moldova).

The storage is a group of six silos arranged in a row with a working tower in the middle. The diameter of each silo is 15.2 m, the height of the cylindrical part is 12 meters, a conical roof with a slope of 30°. Silos with a smooth cylindrical wall erected by the rolling method, resting in the lower part on a ring reinforced concrete foundation protruding above the ground level by 800 mm. A conveyor loading gallery is located above the silos.

The minimum distance between the cylindrical walls of neighboring silos is 1 meter. The silos are located near the border of the HPP site, the axis of the group of silos is turned 30 degrees clockwise in relation to the west-east orientation, and there are no buildings up to 200 meters from the windward side, the height of rare vegetation does not exceed 1.5...2, 5 meters. On the leeward side, there are railway tracks, equipment for loading grain into wagons, grain warehouses for floor storage of grain with a height that does not exceed the height of metal silos. In fig. 2 shows an east side view of a group of metal cylindrical silos. The prevailing winds in Drochia in winter and spring blow from the north, north-east, almost perpendicular to the axis of the storage, reaching an average hourly speed of 16.5 km/h in March with short-term maximum values of 26 km/h.



Fig. 2. General view of the silos of the grain-receiving enterprise in the city of Drokiya

Research results

Stage 1. Mathematical formulation of the problem (choice of a mathematical model).

In ANSYS, this stage is carried out at the stage of selecting the calculation module.

Stage 2. Assignment of physical properties of the material.

The first step to conducting an analysis is to determine the properties of the materials to be analyzed.

The Engineering Data module serves this purpose. To start this module, you can either double-click on its name with the left mouse button, or click on the name of the module with the right mouse button and select the Edit command in the context menu. In the Physical Properties section, you can select such properties as density, coefficient of thermal expansion for an isotropic or anisotropic material, in the Linear Elastic tab you can set Young's modulus and Poisson's ratio for isotropic or anisotropic materials, etc. If the name of the material parameter is displayed in gray, it means that this parameter is involved in the current analysis.

Outline filter – allows you to select the source of available materials.

The Engineering Data line means that the "default" material is used or its properties are set (changed) by the user. The rest of the lines correspond to libraries of already known substances and materials.

Outline Schematic – a list that uses given materials or materials from the library.

No Date – list of parameters of the selected material. Each parameter, depending on the model used, can be either a numerical constant or determined by tabular dependence on other parameters, for example, on temperature. This dependency, if necessary, is specified in the Table Properties window.

Stage 3. Creation (import) of a geometric model of the object under study.

At this stage, it is necessary to decide on the calculation area, its dimensions and the method of creation or import into the calculation module. Some problems can have symmetry, which significantly simplifies the formulation of the problem and speeds up the calculation.

Moreover, at this stage, it is possible to significantly simplify the model, remove from it unimportant details that do not affect the modeling result.

Workbench uses the Geometry module for this, which allows you to call the Design Modeler or Space Claim Direct Modeler application.

Stage 4. Creation of a finite-element mesh.

Mesh – contains a list of all operations and tools used to build a calculation mesh. Component options are displayed by global mesh settings, or through the context menu of the available toolbar for local mesh setup tasks.

With automatic grid generation, the user can only visually assess the quality of the generated grid. If it is satisfactory, then you can proceed to the calculation stage. If the grid is unsuccessful from the user's point of view, then the grid parameters need to be adjusted.

Setting the mesh parameters is done in the details window of the Details of Mesh command.

In the Defaults section, the grid type and grid compression ratio are adjusted.

Section Sizing ("Dimensions") - allows you to choose the quality of the grid, its size, configure curved elements, etc.

Statistics - shows the number of nodes and the number of elements.

The Mesh Control command is used to adjust the geometric shape of elements, control the dimensions of elements, including local ones.

Stage 5. Setting of boundary conditions and loads.

Boundary conditions include boundary and initial conditions. The initial conditions are set in the case of solving a non-stationary problem.

To carry out the research, a model of granaries was first built according to the design data. The model included three dimensional elements: silos, tower, supersilo galleries made in natural scale (Fig. 3).

The estimated volume (Fig. 4) taking into account the shape of the granaries is chosen in the form of a parallelepiped with the following dimensions of the sides, adopted taking into account professional recommendations:

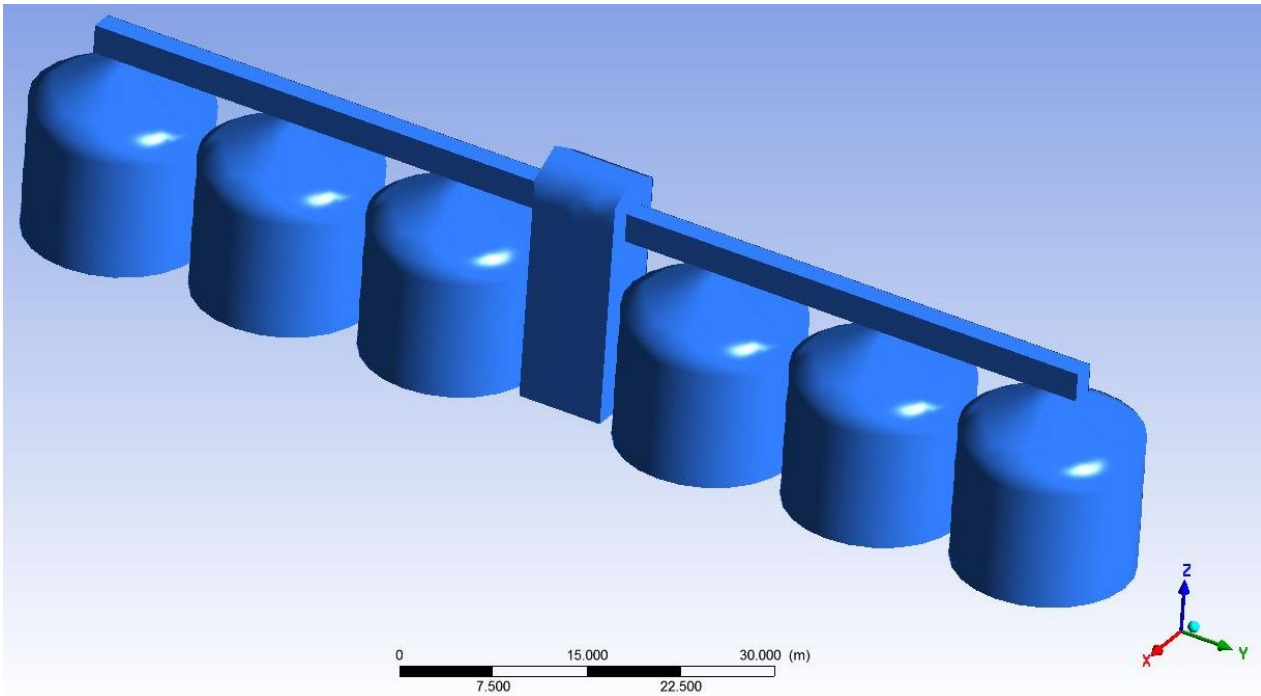


Fig. 3. Models of the tower, silos and supersilo galleries

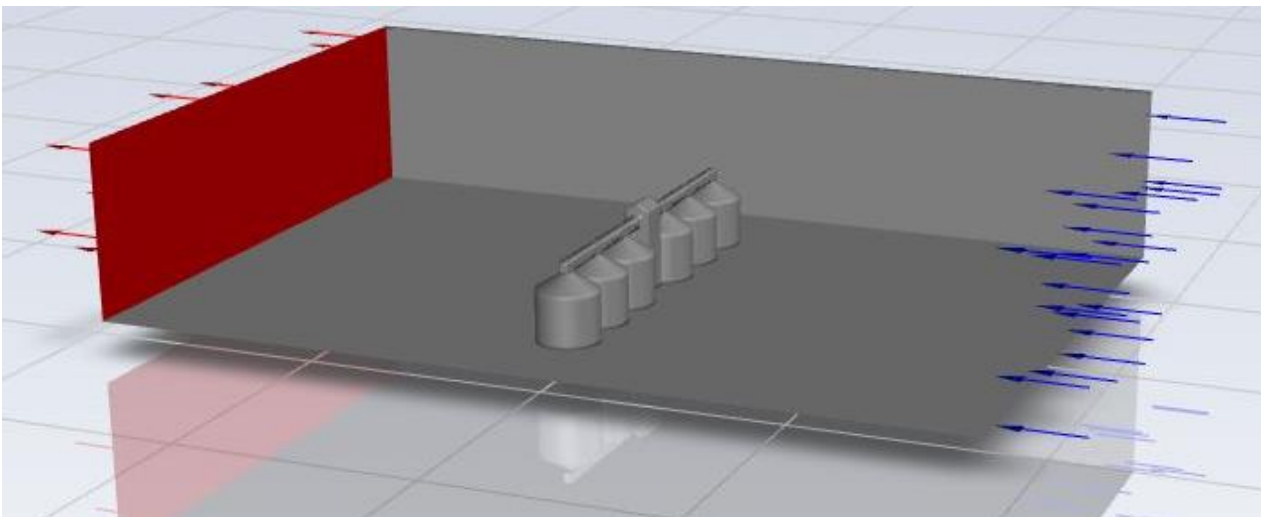


Fig. 4. Estimated volume for the study of aerodynamic parameters of buildings

- windward and leeward sides with a width of 300 m at a height of 120 m;
- side faces 420 m long and 120 m high;
- the length of the windward zone is 100 m, and the length of the leeward zone is 300 m.

The boundary conditions are adopted as follows: there is no friction on the side and top faces of the calculated volume.

Along the windward face, the velocity of the air flow is distributed uniformly with the velocity vector parallel to the longitudinal axis of the calculated volume. The speed of the incoming air flow is taken to be 20 m/s.

To model the trajectories of air movement, first of all, a Boolean operation was performed, the volume of buildings was subtracted from the total calculated volume (Fig. 4). This determined the volume in which the air moves, and allows to simulate the nature of its movement, for example, vortices. In addition, it provides an opportunity to find air pressure fields on industrial buildings.

The dimensions of the calculated volumes in the Ansys CFX software are assigned automatically, with a decrease in the size of the final volumes near the areas of contact of the air flow with the structures and an increase in the speed of the air flow.

Depending on their orientation with respect to the oncoming flow, the wind effect on the surface of buildings manifests itself in excessive pressure or back pressure (suction effect of the wind). When calculating wind loads, the average, pulsating and peak components of wind pressure are distinguished. At the first stage, the work takes into account the uniform laminar course of the air flow, which is included in the calculated volume.

First, let's find air pressure isobars on the model of our buildings (Fig. 5). As follows from the obtained result, the main wind loads fall on the lower cylindrical walls of the silos, the front wall of the tower and the front wall of the silo gallery. Isobars on the walls of the silos emphasize the nature of the received loads.

The pressure distribution on the front wall of the



tower is similar. At the same time, the supersilo gallery has its influence on the pressure distribution, part of the air passes over the roof of the silos and under the supersilo gallery, reducing the pressure on the upper part of the tower. By connecting the Function Calculator, we will calculate the force with which the air presses on the

model buildings. This force in the direction opposite to the X axis is equal to 365255 N.

Having received the initial data, we will introduce an auxiliary horizontal plane into our model space at a height of 1 meter from the ground, for a more characteristic determination of the distribution of pressures

TECHNOLOGICAL PROCESSES, EQUIPMENT

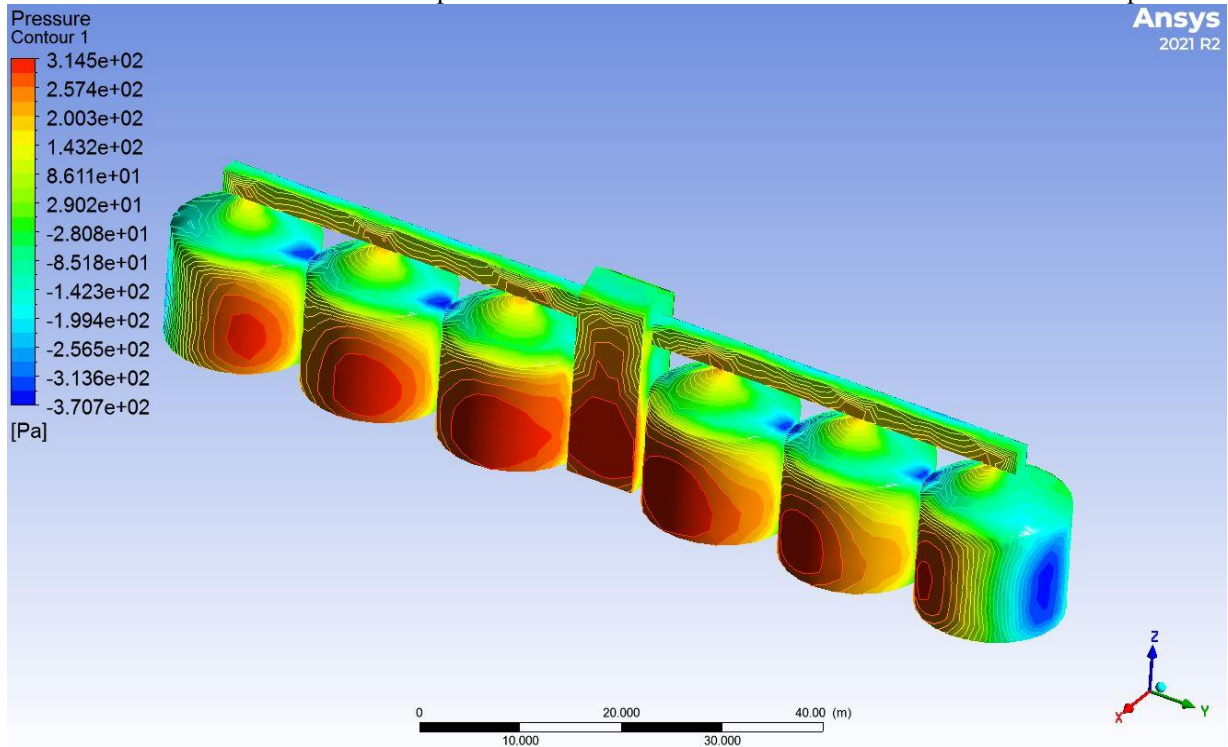


Fig. 5. Isobars of air pressure on the walls and roof of the silos and on the walls of the tower and the gallery above the silo

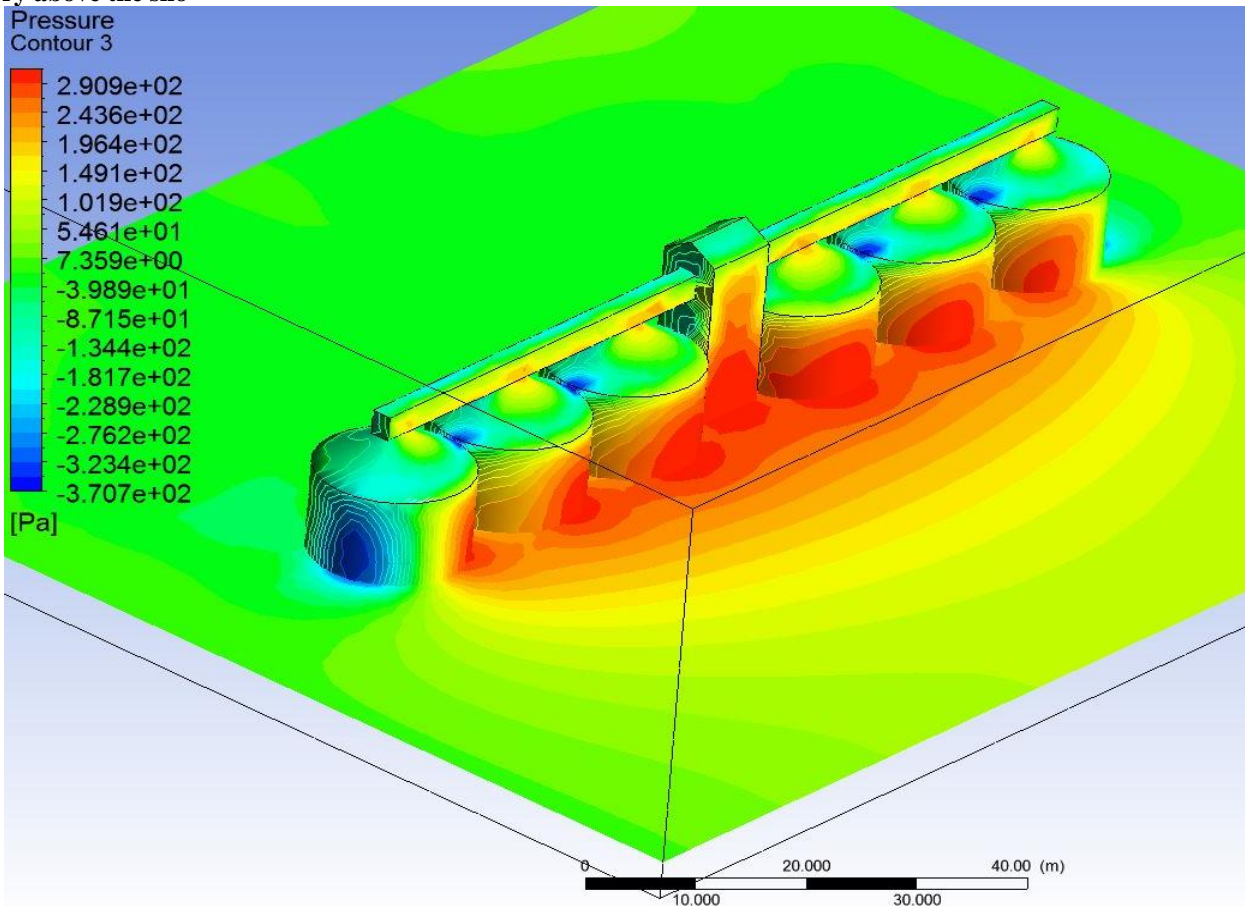


Fig. 6. General view of pressure distribution on structures in the city of Drokiya



on the model structures, without the influence of the boundary layer of the soil (Fig. 6). In the introduced plane, we will construct pressure isobars that characterize the most loaded zone of the simulated structures. The obtained isobars reflect a characteristic pattern of excess pressure distribution on the windward side of buildings. Rarefaction zones have been identified, which can be estimated using the presented pressure scale.

In fig. 7, two auxiliary planes are introduced into the granary model, one horizontal at a height of 1 meter from the ground, and the other vertical along the longitudinal area of symmetry of the calculated volume. The pressure that occurs in front of the silos and the tower

can be estimated from the simulation picture of the pressure distribution in the horizontal plane. In addition, the air pressure on the tower from the windward side and the nature of its distribution can be considered on the vertical plane. Behind the tower in the vertical plane, pressure isolines show zones of reduced air pressure.

The general nature of the movement of air flows can be traced along the air current lines, also shown in the calculation volume (Fig. 8). On the cylindrical surfaces of the silos and their roofs, on the surfaces of the supersilo gallery and the tower, the values of air pressure were reflected, which especially characterizes the distribution of pressure on the cylindrical wall of the silos. It

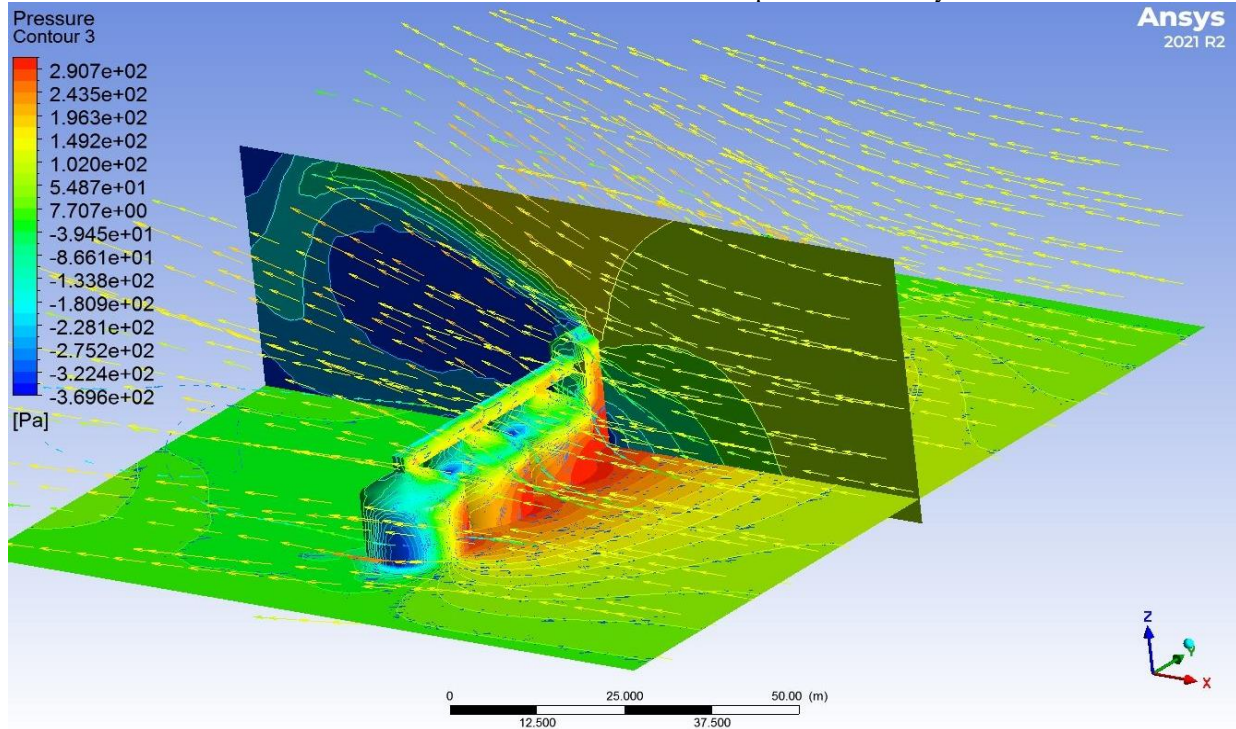


Fig. 7. Isobars and streamlines of air flows in a fragment of the calculated volume and on the surfaces of the silos (view from the windward side)

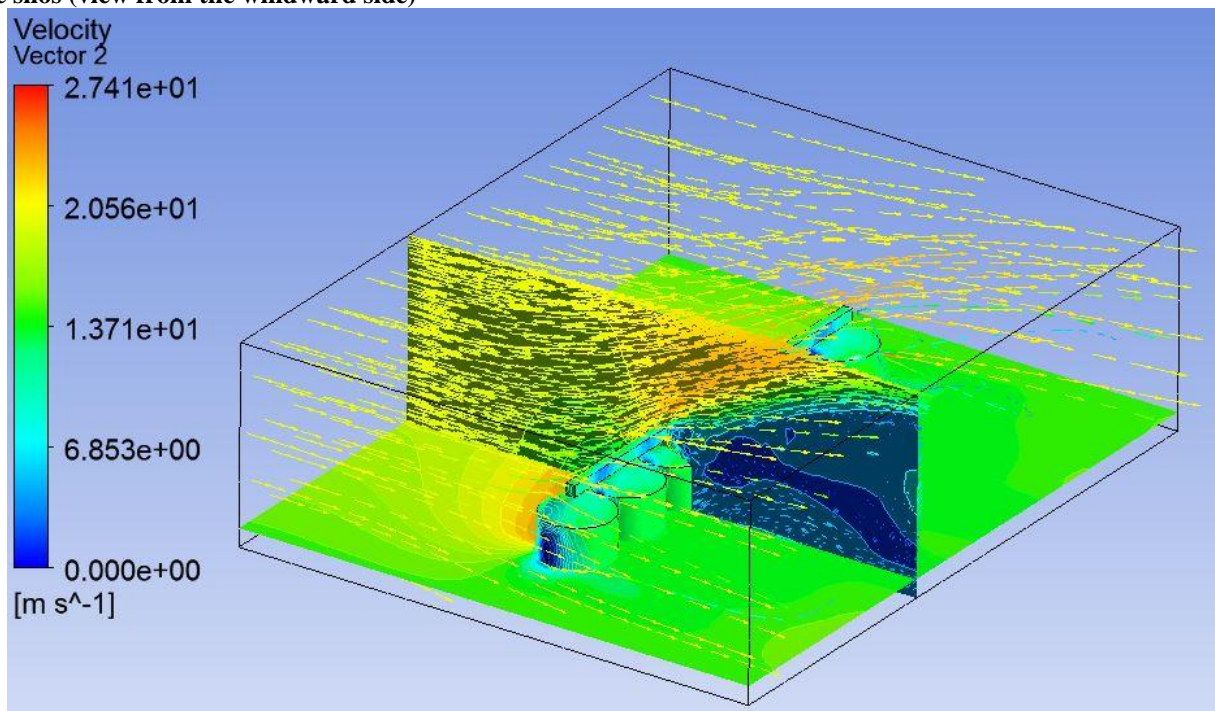


Fig. 8. General view of the distribution of air flow velocities in the vertical plane



also follows from the model that there are significant zones of low pressure (blue zones).

Having considered the pattern of air movement (streamlines) and the distribution of pressures in two perpendicular planes, let's move on to the general distribution of air flow velocities in the vertical plane (Figs. 8, 9). As follows from the obtained results, the air flow rises very sharply above the building models, leaving a rarefied space outside the buildings. It also follows from the obtained results that the height of the calculation space selected for these models is insufficient, it should be increased in further calculations.

To study the air pressure gradient in the vertical frontal plane, another auxiliary vertical plane was intro-

duced (Fig. 9). The obtained result makes it possible to estimate the distribution of pressure around the building models. The results of the calculations are presented graphically in fig. 10. Fragments of the animation of the transverse movement of the cutting plane with fixation of isobars and air current lines for the position of this plane on the axis of symmetry of the calculated volume (Fig. 3) and on the longitudinal face (Fig. 4) are presented in different views.

In fig. 10 presents isobars of air pressure on silos, tower and supersilo galleries (shades of red - external pressure and blue - discharge) and isobars of air pressure in the calculated volume for various auxiliary planes.

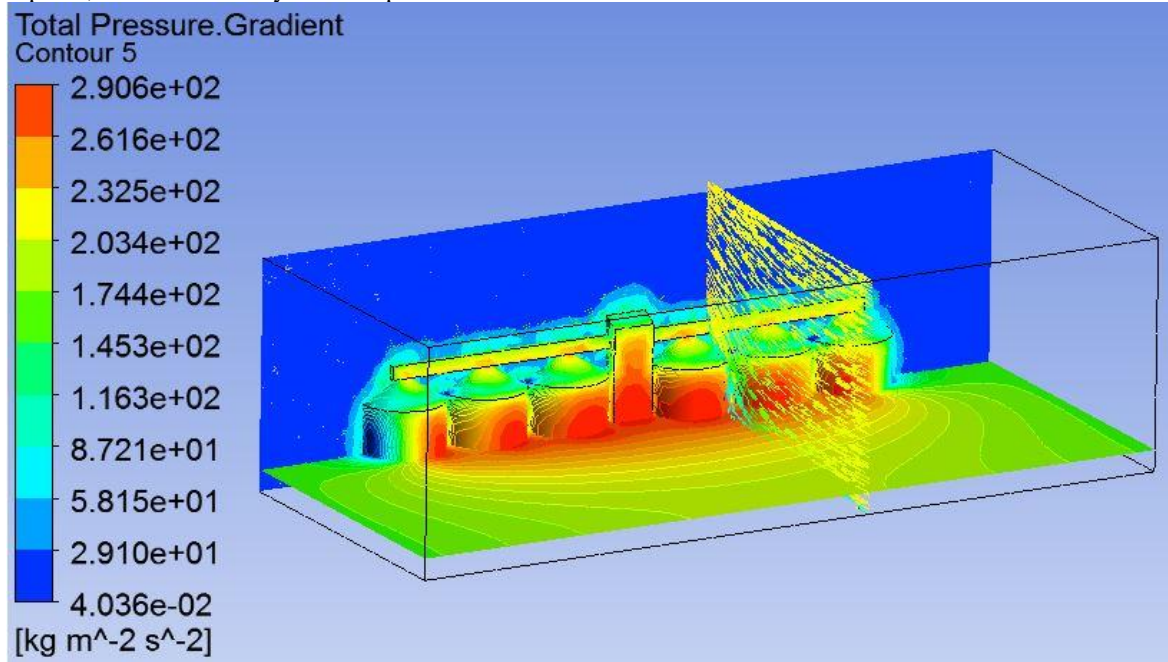


Fig. 9. Isolines of air pressure in the calculated volume, on the surface of the silos and air pressure gradient in the vertical frontal surface located along the axis of symmetry of the silos

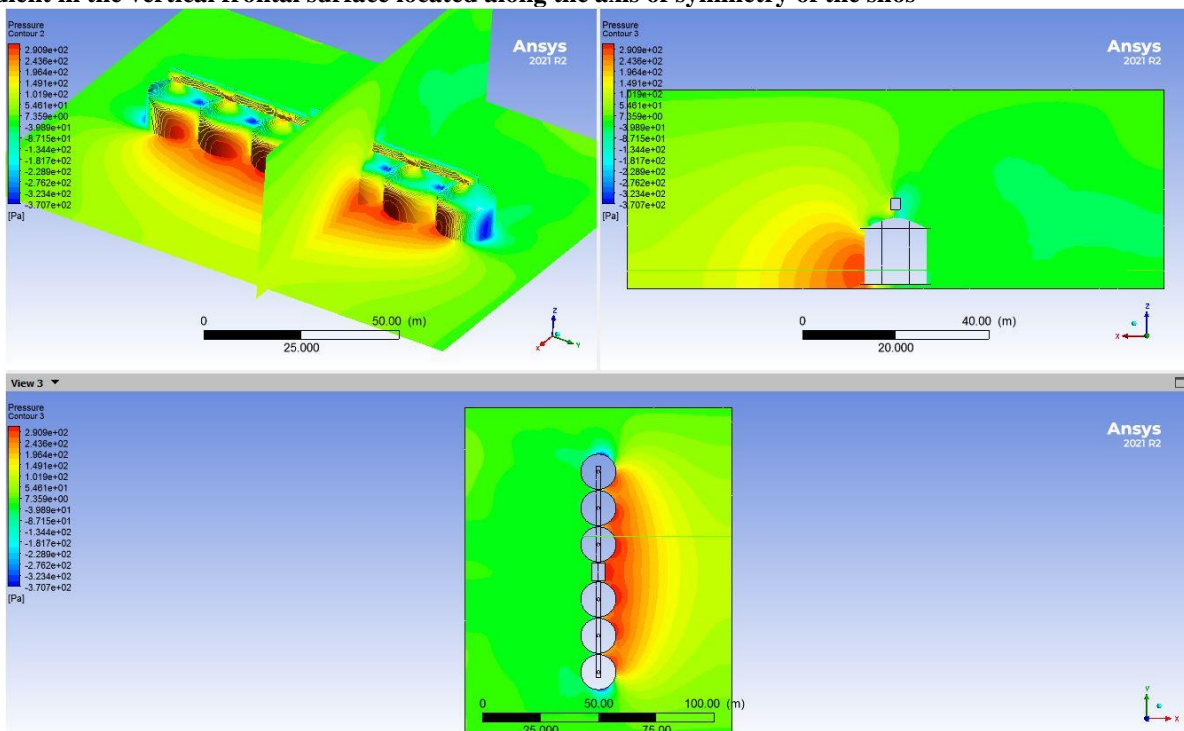


Fig. 10. Isobars of air pressure on the walls, roof of silos and in the calculated volume (views from different sides)



The application of computational aerodynamics based on the ANSYS CFX complex made it possible to simulate industrial buildings located on the territory of a grain-receiving enterprise, to set a calculation model of the interaction of building models and incoming air. This calculation model allows you to obtain various air flow parameters, as well as parameters of the interaction of air and building elements (pressure, effort, nature of load on various elements).

Conclusions

1. The qualitative picture of the distribution of normal air pressure on the cylindrical wall of the outer

extreme zones in a number of silos corresponds to the distribution of the aerodynamic coefficient given in modern literature.

2. The distribution of air pressure in other zones is different and requires clarification. The presence of rarefaction zones on the leeward side of the silos and the conical roof of the silos is characteristic.

3. The functional calculator built into ANSYS CFX allows you to calculate the wind load on the elements of industrial buildings. Data on the numerical compliance of calculation results with regulatory documents require additional research after specifying the calculation volume.

REFERENCE

1. Paying Attention. Electronic resource. Access mode: <https://disasterphilanthropy.org/blog/paying-attention/>
2. Rozirvalo banku, abo chomu ruynuyut'sya zernovi sylosy. Elektronnyy resurs. Rezhym dostupu; <https://agravery.com/uk/posts/show/rozirvalo-banku-abo-comu-ruynuutsa-zernovi-silosy>
3. Malynov'skyi B. 9 prychny avariyy sylosiv. // Propozytsiya. Elektronnyy resurs. Rezhym dostupu: <https://propozytsiya.com/ua/9-prychyn-avariy-sylosiv>
4. ANSYS Fluent User's Guide: ANSYS, Inc. Southpointe 275 Technology Drive Release 15.0 November 2013 Canonsburg, PA 15317.

УДК 004.94

¹В.М. Петров, канд. техн. наук, доцент, E-mail: 0673972002@ukr.net

¹Д.Л. Гончаренко, магістр E-mail: dmitriy_goncharenko@ukr.net

²О.О. Жданов, канд. техн. наук, доцент, E-mail: ajdanow1945@gmail.com

¹Кафедра технологічного обладнання зернових виробництв,

¹Одеський національний технологічний університет, вул. Канатна, 112, Одеса, 65039, Україна

²Одеська державна академія будівництва та архітектури, Дідріхсона 4, Одеса, Одеса, 65029, Україна

АЕРОДИНАМІКА ПРОМИСЛОВИХ СПОРУД

Анотація

В роботі розглянуті конструкції металевих силосів та інших споруд (башти, надсилосних галерей) хлібоприймального пункту під дією вітрового навантаження. Виявлено позитивні та негативні сторони конструкцій силосів, що розглядаються. Застосовано методику дослідження вітрового навантаження на силоси в пакеті ANSYS CFX. Виконана трьох вимірна модель промислових споруд та з урахуванням рекомендацій прийнятий розрахунковий об'єкт навколо об'єкту дослідження. Також визначений об'єкт для руху повітря навколо промислових споруд. Враховуючи метод кінцевих елементів побудована сітка на макеті об'єкта дослідження. Призначені початкові дані та граничні умови. В якості об'єкта дослідження взята конструкція із шести металевих силосів діаметром 15,2 м з висотою циліндричної стінки 12 м Дрокієвського ХПП (Республіка Молдова). Тиск вітру на циліндричні стіни та конічні дахи оцінено методом кінцевих елементів застосувавши пакет ANSYS CFX для направлення вітру яке перпендикулярне лінії силосів при максимальній швидкості. Знайдені ізобари тиску в різних допоміжних вертикальних та горизонтальних площинах. Побудовані лінії однакових швидкостей повітряних потоків та їх векторні аналоги. Встановлені зони максимального та мінімального тиску на промислові споруди. Встановлена відповідність загальної картини розподілу тиску по стінці в окружному напрямку при зміні аеродинамічного коефіцієнта. Визначені вітрові навантаження на елементи конструкцій промислових споруд. Розрахунки проводили використовуючи функціональний калькулятор, який вбудований в даний пакет ANSYS CFX. Для перевірки числової відповідності отриманих результатів потрібне виконання додаткових досліджень. За рахунок руху (анімації) допоміжних вертикальних площин можливо розглянути динаміку зміни тиску та швидкостей набігаючого потоку повітря в розрахунковому об'ємі.

Ключові слова: металевий циліндричний силос, вітровий тиск, обчислювальна аеродинаміка, метод кінцевих елементів, пакет ANSYS CFX.

Received 30.05.2024

Reviewed 13.06.2024

Revised 22.06.2024

Approved 25.06.2024



Cite as Vancouver Citation Style

Petrov V., Goncharenko D., Zhdanov O. Aerodynamics of industrial buildings. Grain Products and Mixed Fodder's, 2024, 24 (2, 94): 39-46. DOI <https://doi.org/10.15673/>

Cite as State Standard of Ukraine 8302:2015

Aerodynamics of industrial buildings. / Petrov V. et al. // Grain Products and Mixed Fodder's, 2024, Vol. 24, Issue 2 (94). P. 39-46. DOI <https://doi.org/10.15673/>

