

Svetlana Zhukovetskaya
Computer Engineering Department
Odessa National Academy of Food Technologies
E-mail: szhukovetckaya@i.ua

Air flowing spatial modeling and simulation with SOLIDWORKS CAD

Keywords: SOLIDWORKS CAD, spatial modeling, air flowing simulation

Introduction

Air conditioning system provides a large number of air treatment processes, which could be used to meet various air requirements in confined spaces. Computer engineering comes up with the construction of a spatial model, to be used for simulation and analysis that leads to preliminary identification of potential problems in air-conditioning system.

Object models are simpler systems, with a clear structure and specified relationships between the component parts. That allows to analyse the properties of real objects and their behavior in various situations. Due to the rapid development of software and hardware solutions in computing machinery and the ability to conduct numerical spatial modeling of airflow, it became possible to use computer modeling as the initial method for studying the distribution of the airflow in the laboratory of the ONAFT.

The aim of the work is airflow movement simulation and spatial modeling in the laboratory of ONAFT based on the multi-zone air-conditioning system.

The Object of study. The objects of study are multi-zone air conditioning systems. Multizone system is one of possible solutions to the problem of air conditioning several spaces simultaneously. Multizone system consists of an outdoor unit and a variety of indoor units. Every indoor unit can be controlled centralized or controlled locally (Fig. 1).

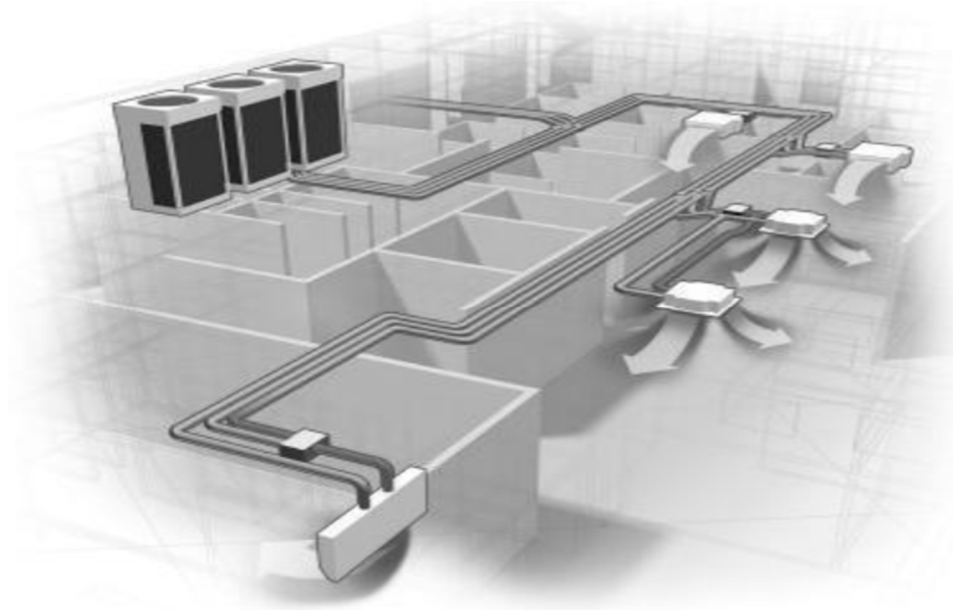


Fig. 1. Schematic representation of a multi-zone air conditioning system

Research methods. The most popular research methods are computer simulations with automatic design systems. SolidWorks had been chosen as a software solution for creating air movement model in the air conditioning laboratory.

SolidWorks is a system that allows you to create size-controlled solid models and can serve as the basis for solving many different engineering problems.

The extensive capabilities of the main module are combined with a large number of special-purpose applications, that makes SolidWorks a powerful software solution that can be flexibly configured to solve almost any design and production challenge [1].

1. Spatial model preparations

The algorithm for conducting a study includes the following steps:

1. The development of a spatial model of the object of study.
2. Determination of the area of calculation.
3. Setting the boundary conditions.
4. The calculation.
5. Visualization of the calculation results.

Floor plan of the laboratory and the drawings of the air conditioning units were used as the initial data for spatial modeling. The result is a spatial solid-state parametric assembly consisting of the parts shown in Figures 2-5.

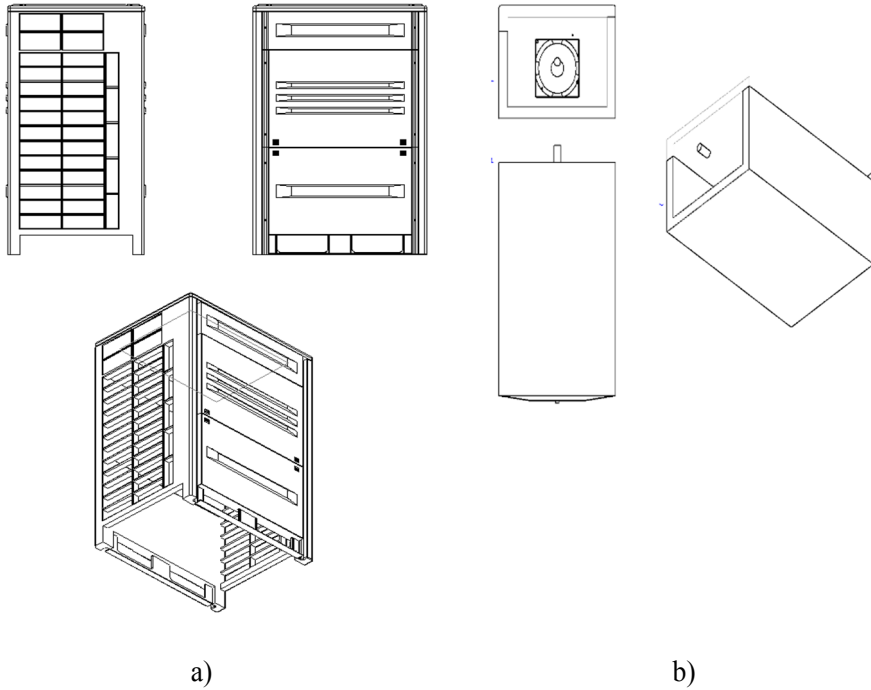
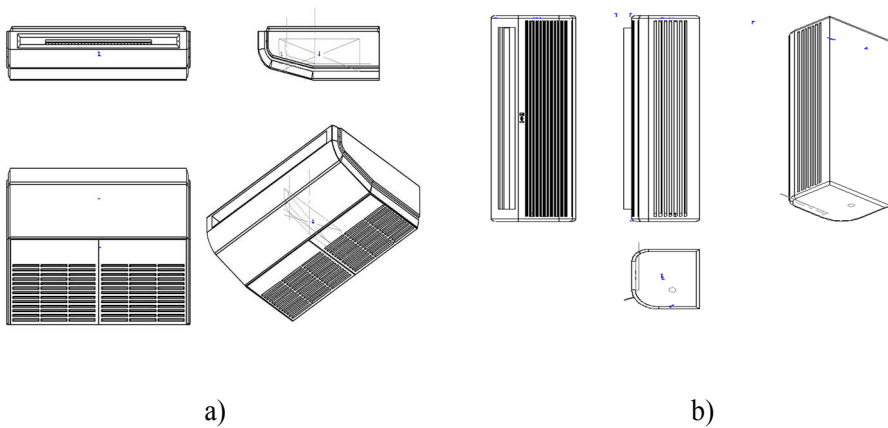


Fig. 2. Assembling the part of the outdoor unit: a) case, b) heat exchanger



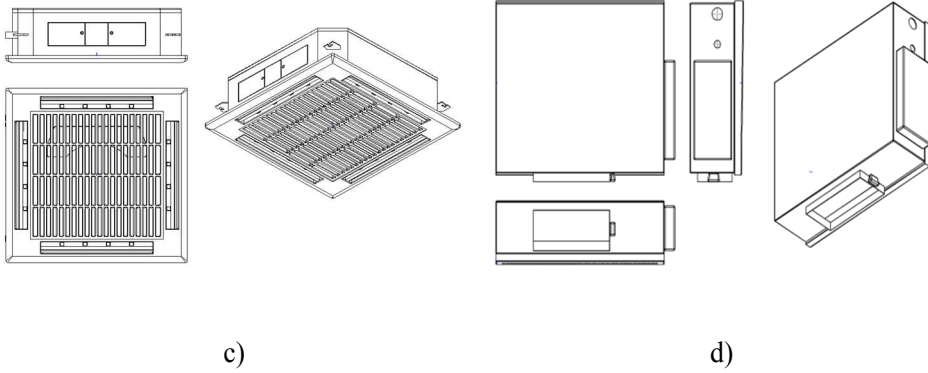


Fig. 3. Indoor unit parts: a) subceiling, b) wall mounted, c) cassette, d) channel

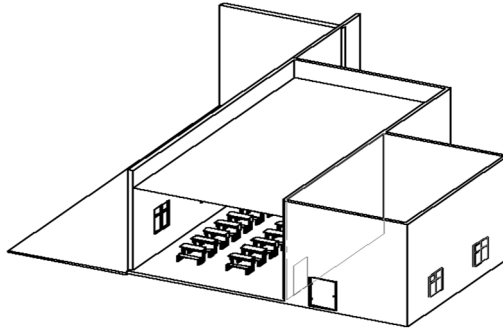


Fig. 4. Assembling the PartsRoom

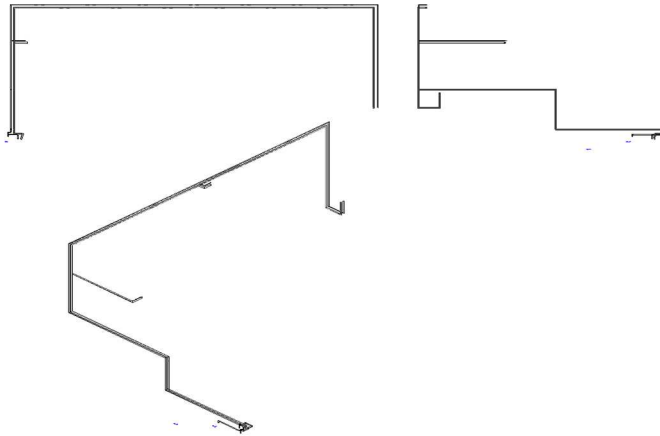


Fig. 5. The pipeline parts

The lighting and texturing of the model are specified for a visual representation. The visualization results are presented in Figures 6-7.

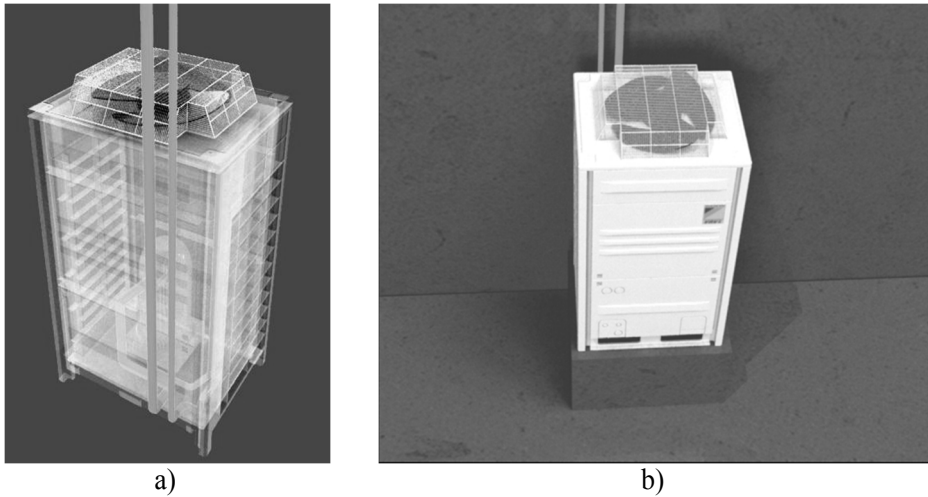


Fig. 6. Visualization of the outdoor unit a) transparent, b) non-transparent

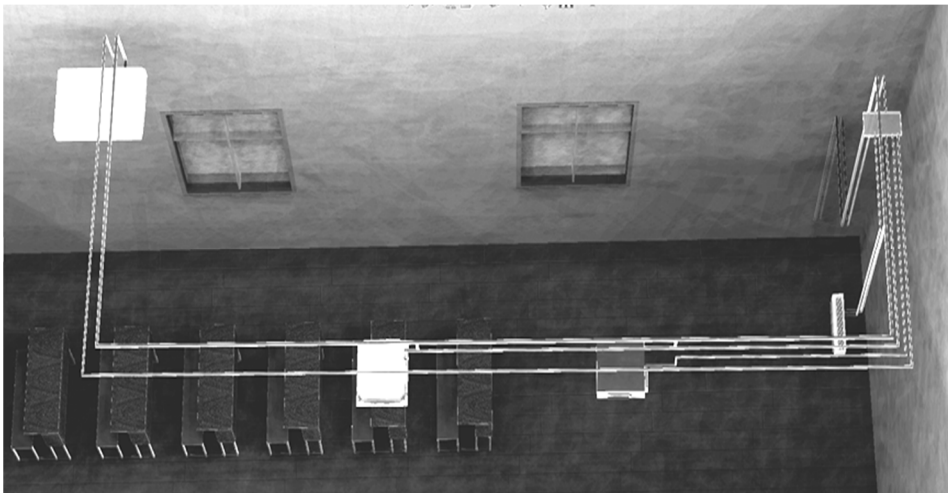


Fig. 7. The finalrender

2. Airflow Simulation

To analyze the movement of air masses, an additional SolidWorks engineering analysis module, Flow Simulation, was used. It allows you to simulate the flow of

gases, control the grid, perform complex thermal calculations, as well as evaluation of rotating objects, create gas-dynamic and thermal models of technical devices, etc.

In the Solidworks Flow Simulation, the motion and heat exchange of the fluid is calculated based on the Navier – Stokes equations. They simulate turbulent, laminar and transient flows [1]. The following initial conditions were defined for the aerodynamic calculations: air was selected as the fluid; ambient environment temperature - 20 °C; the remaining parameters are taken by default.

All calculations of flow and heat transfer in a solid body are performed within the computational domain. It is assumed that the system operates in recirculation mode, i.e. There is no external air intake into the room (Fig. 8).

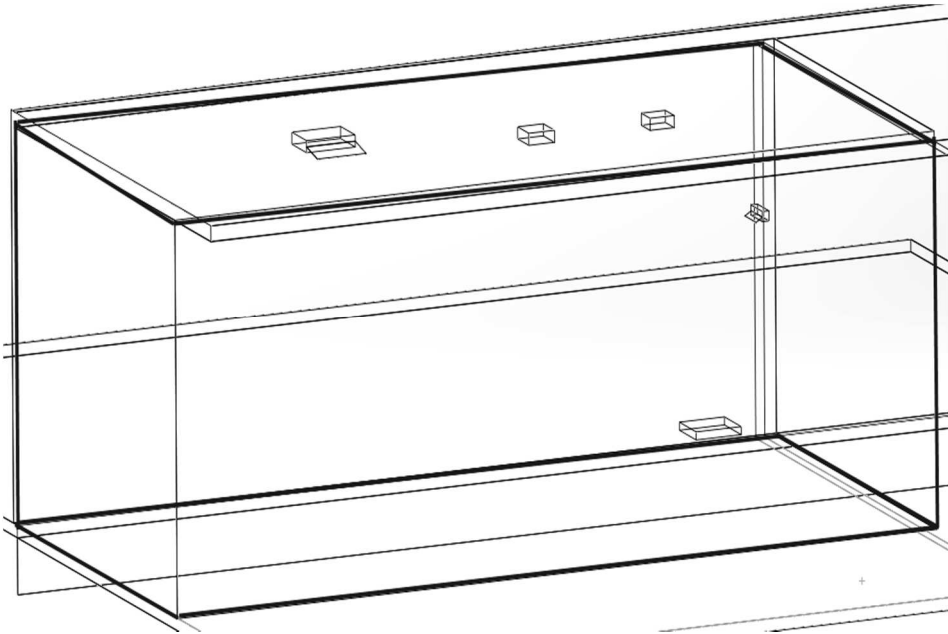


Fig. 8. The processing area

Apparently new objects had been added apart from simplifications. It is used to simulate air conditioners. It is represented by a simple geometry - a body without cavities, having two flat faces: one will imitate the input, the other - the output. It is irrelevant what is between these, therefore the profile of the bodies schematically reproduces the contours of air conditioners.

Creation of boundary conditions is one of the most important elements of the work. It is worth noting that it is necessary to set the boundary conditions for each element in the project, otherwise, the correct calculations are impossible. Following default thermodynamic parameters are used: pressure set equal to 101325 Pa,

temperature set equal to 293.2 K. Air is chosen as the fluid, humidity is not taken into account. The task is solved as an internal non-stationary.

The boundary condition of the SolidWorks “Fan” is considered as an ideal device that creates volumetric (or mass) consumption (flow rate can be set to spin) depending on the difference in static pressures at the inlet and outlet of this device. The volumetric consumption rate of the fluid through the SolidWorks “Internal Fan” is determined by the difference between the static pressures of the fluid on its inlet and outlet surfaces, that are calculated during modeling and averaged over these surfaces. Surfaces of the model on which “Fan” is set are selected. To set up “Internal Fan”, “Inlet” is set as the surface through which the fluid exits the fan, and the “Outlet” is set as the surface through which the fluid enters the fan [2].

The process of movement of the airflow was modeled after determining the conditions of the calculation.

3. Interpretation of the calculation results

One of the advantages of analysis tools integrated into CAD systems is the visualization of the calculation results directly in the graphics window of the modeling system. Flow Simulation has a complete visualization toolkit, that has already become the standard for that kind of applications.

It includes the shear and surface diagrams, the distribution of the results by the fluid flow (in our case it is air), etc. [3]. Spatial trajectories of air masses in an enclosed spaces were built as a result of the calculation (Fig. 9).

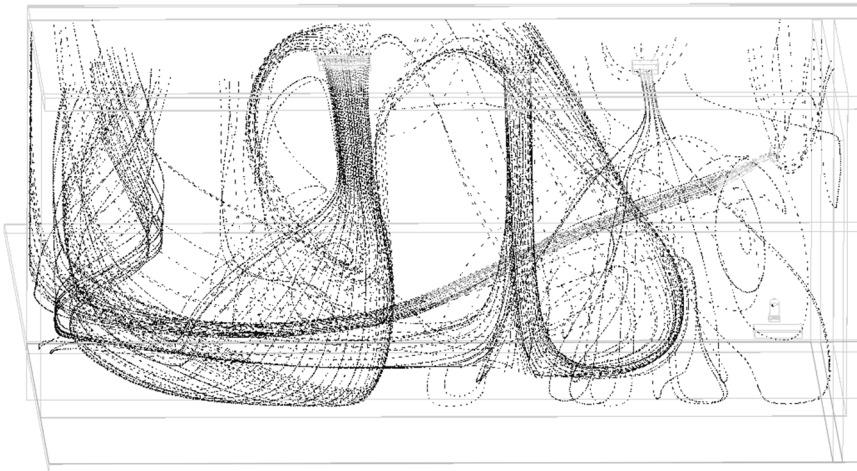


Fig. 9. Spatial trajectories of air masses

Current lines contain substantial information. Apparently almost any information can be extracted. The existing situation assessment and improvements proposals can be made based on that information. Main constraint is the large computational complexity of the problem.

In addition to the research results, demonstration results were obtained that represent the animation of the movement of air masses. Spatial geometry with textures and lighting has been added to the results described above. The frame of the animation is shown in Figure 10.

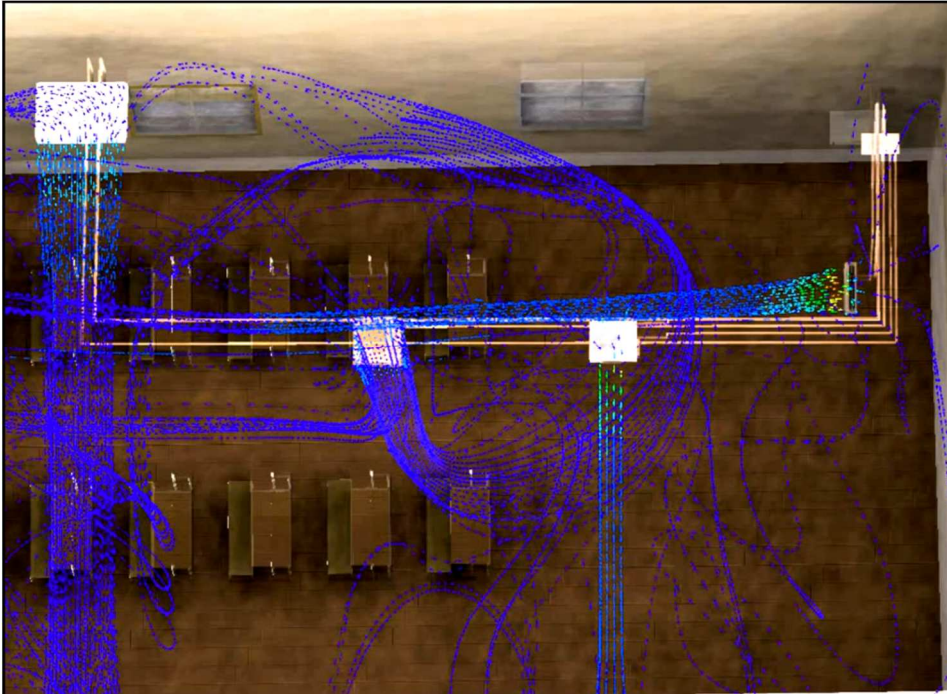


Fig. 10. An air mass animation frame

Conclusion

The paper contains overview of the results of solid modeling and assembly of the elements that composed the multi-zone air conditioning system. Generated a visual representation of results. Movement of air masses in enclosed space was studied based on assembly implementation. The results are presented in graphics and animations.

Simulation of the movement of air masses is carried out to obtain the following indicators: the direction of the air flow and the vectors of its movement; "dead zones"; airflow speed. Based on these indicators, experts will make reasonable

recommendations for improving the microclimate of the space, optimization of the air masses movement, placement and power of ventilation equipment, etc.

Computer Flow Simulation is faster than making a prototype or model, equipped with sensors, run a test cycle and get data suitable for further work. The resulting model can be used to compile and visually evaluate options for the planning of classrooms, as well as a demonstration material on the electronic resources of the school.

Literature

1. A.A. Alyamovsky. SolidWorks 2007/2008.Computer modeling in engineering practice / A.A. Alyamovsky, A.A. Sobachkin, E.V. Odintsov, A. I. Kharitonovich. –SPb .: BHV–Petersburg. 2008. – 1040 pp., Ill.
2. A. A. Alyamovsky. SolidWorks Simulation.How to solve practical problems.– SPb .: BHV– Petersburg.2012. – 448.
3. Flow Simulation 2009 Tutorial. [Electronic resource].– Access mode https://learn.ztu.edu.ua/pluginfile.php/29485/mod_resource/content/1/solidworks_flow_flow_simulation_2009_tutorial.pdf

Abstract

The technologies spatial visualization and simulation of the operation of technological equipment have become particularly relevant due to the fact that they provide an opportunity to obtain and analyze information about the operation of equipment before its installation.

The article provides an example of solid-state spatial modeling of a multizone conditioning system working in an chosen area – research laboratory. Based on the model, studies of the movement of air masses in a closed room were carried out.

Streszczenie

Technologie przestrzennej wizualizacji i symulacji działania urządzeń technologicznych stały się szczególnie istotne ze względu na fakt, że dają one możliwość uzyskania i analizy informacji o działaniu sprzętu przed jego zainstalowaniem.

Artykuł stanowi przykład modelowania przestrzennego wielostrefowego systemu kondycjonowania w przykładowym pomieszczeniu – laboratorium badawczym. W oparciu o model przeprowadzono badania ruchu mas powietrza w zamkniętym pomieszczeniu.

Słowa kluczowe: SOLIDWORKS CAD, modelowanie przestrzenne, symulacja przepływu powietrza