



Volume XXV 2022

ISSUE no.1

MBNA Publishing House Constanta 2022



# Scientific Bulletin of Naval Academy

SBNA PAPER • **OPEN ACCESS**

## Optimization of the naval ventilation systems using experimental measurements and numerical simulations

To cite this article: O. N. Volintiru, E. Aleksey, I. C. Scurtu, F. Ioniță, G. C. Partene and G. Bălan, Scientific Bulletin of Naval Academy, Vol. XXV 2022, pg. 96-105.

Submitted: 24.03.2022

Revised: 28.07.2022

Accepted: 03.08.2022

Available online at [www.anmb.ro](http://www.anmb.ro)

ISSN: 2392-8956; ISSN-L: 1454-864X

doi: 10.21279/1454-864X-22-I1-011

SBNA© 2022. This work is licensed under the CC BY-NC-SA 4.0 License

# Optimization of the naval ventilation systems using experimental measurements and numerical simulations

Octavian Narcis Volintiru<sup>1</sup>, Epikhin Aleksey I<sup>2</sup>, Ionuț Cristian Scurtu<sup>1</sup>, Florin Ioniță<sup>3</sup>, George Cosmin Partene<sup>3</sup> and George Bălan<sup>4</sup>

<sup>1</sup>Naval Academy “Mircea cel Batran”, Constanta, Romania

<sup>2</sup>Admiral F. F. Ushakov State Maritime University, 93, Lenin Ave, Novorossiysk, 353918, Russia

<sup>3</sup>Romanian Navy, Constanta, Romania

<sup>4</sup>National Research and Development Institute for Gas Turbines COMOTI

Email: octaviannarcis@gmail.com

**Abstract.** In recent decades, a multitude of numerical methods has been developed to support analytical and experimental research, optimizing and adding value to them, through a low cost compared to the experiment and a short time to obtain results. The Ansys Fluent program allows the visualization of the various curves and the values of all the parameters that define the phenomenon studied in this paper, respectively the ventilation of the ship machinery compartment. The comparison of the results obtained in the simulation program with the results of the measurements carried out on a real ventilation system on board the ship leads to certain aspects regarding the optimization of this type of system. Thus, the air behaviour on the ventilation system of the machinery compartment, where the ventilation requirement is quite high, can be studied with numerical calculation methods using a computer modelling program and the results can be validated by a scale experiment 1:1 on board the ship. This leads to conclusive results on optimizing the entire system.

**Keywords:** air, ventilation, measurements, flow, machinery space

## 1. Introduction

The use of numerical simulations followed by experimental data is a certainty of the correctness of the results followed in research work. To carry out this work, It was executed the geometry of the ventilation system for the machinery room that corresponds to the real installation. According with nowadays standards, the arrangement of the ventilation sections must satisfy the following conditions for special ships:

- to ensure efficient air circulation inside the machinery room;
- to ensure efficient supply of air consumers;
- do not blow air directly into radiant areas such as exhaust manifolds of engines.

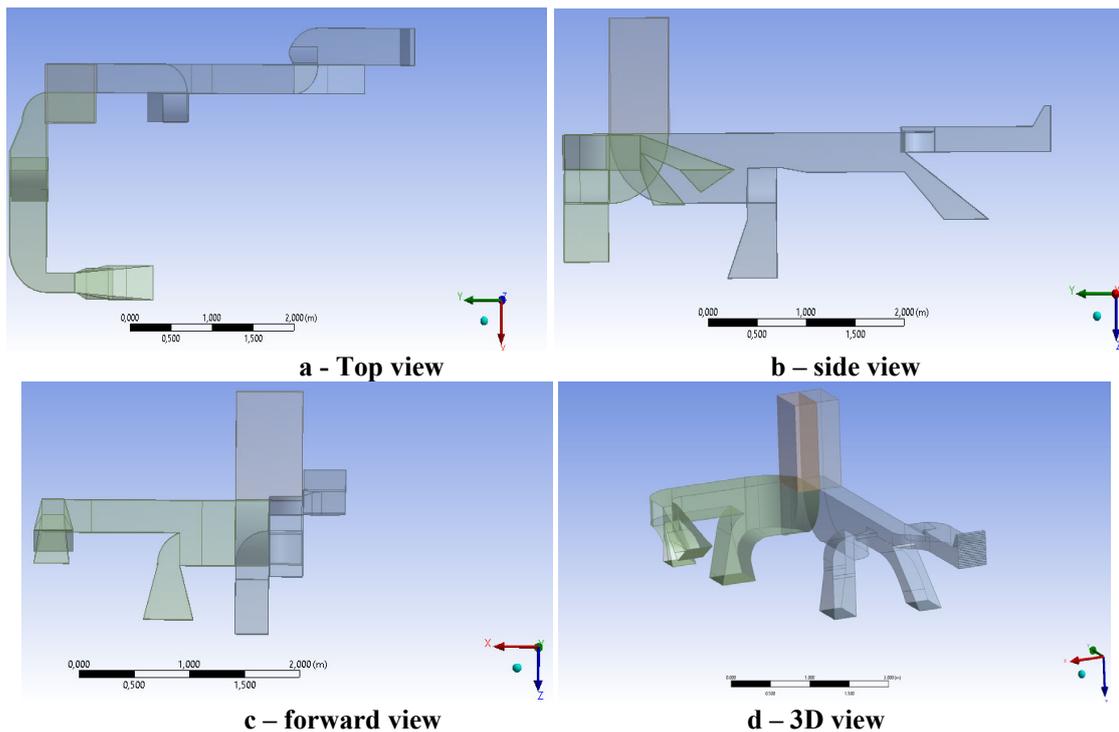
Flow and temperature distribution of air are governed by the laws of conservation of mass, momentum and energy. The finite element method was used in the discretization geometry. Also, the flow is considered constant, turbulent, incompressible and in three dimensions. The objective of the simulation and experimental work was to determine the speed of air in the ducts to improve the ventilation in the machinery room. [1][2][7]

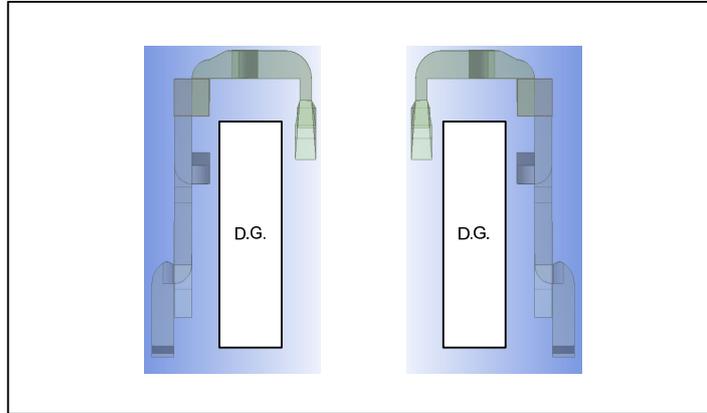


Figure 1. Special ship [3]

## 2. Characteristics of the ship auxiliary machinery room ventilation system

To accomplish this paper, it was taken as a reference a special ship auxiliary machinery room with 2 diesel generators inside of 1000 [kW] each. For the special ship, following classification societies, the flow rate of the exhaust fan is higher than the flow rate of the intake fan to ensure the extraction of gas and smoke from the compartment.[4] It was taken as a study the ventilation system configuration as shown in figure 2. For this machinery room, there are 2 fans of  $28746 \left[ \frac{\text{m}^3}{\text{h}} \right]$  intake air flow and 2 fans of  $32716 \left[ \frac{\text{m}^3}{\text{h}} \right]$  exhaust air flow. The main dimensions of machinery space is also shown in figure no. 3.





e – ventilation systems around diesel generators

Figure 2. Ventilation systems views. D.G. – diesel generator  
[author Visio drawing]

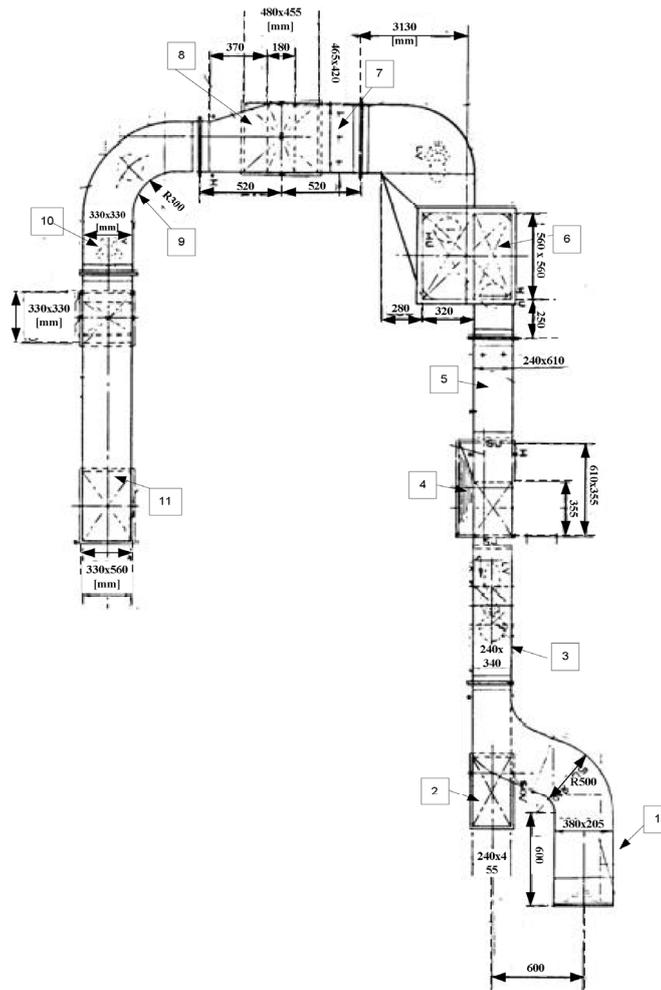


Figure 3. Main dimensions of ventilation system in machinery spaces [author Visio drawing]

**Table 1.** Dimensions of ventilation system as shown in figure 2

The part of ventilation system	Dimension L x h / R [mm]	Intake / outlet
1	380 x 205	intake
2	240 x 455	outlet
3	240 x 340	outlet
4	355 x 240	outlet
5	240 x 610	outlet
6	560 x 560	outlet
7	455 x 420	outlet
8	480 x 455	outlet
9	R 300	outlet
10	330 x 330	outlet
11	330 x 560	outlet

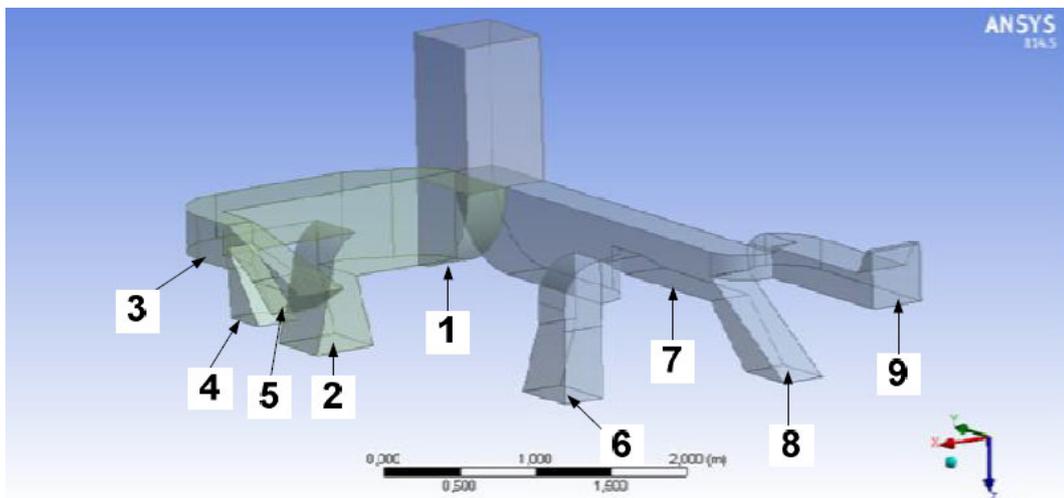
L – length [mm], h – high [mm], R – radius [mm]

### 3. Experimental monitoring of ventilation system parameters

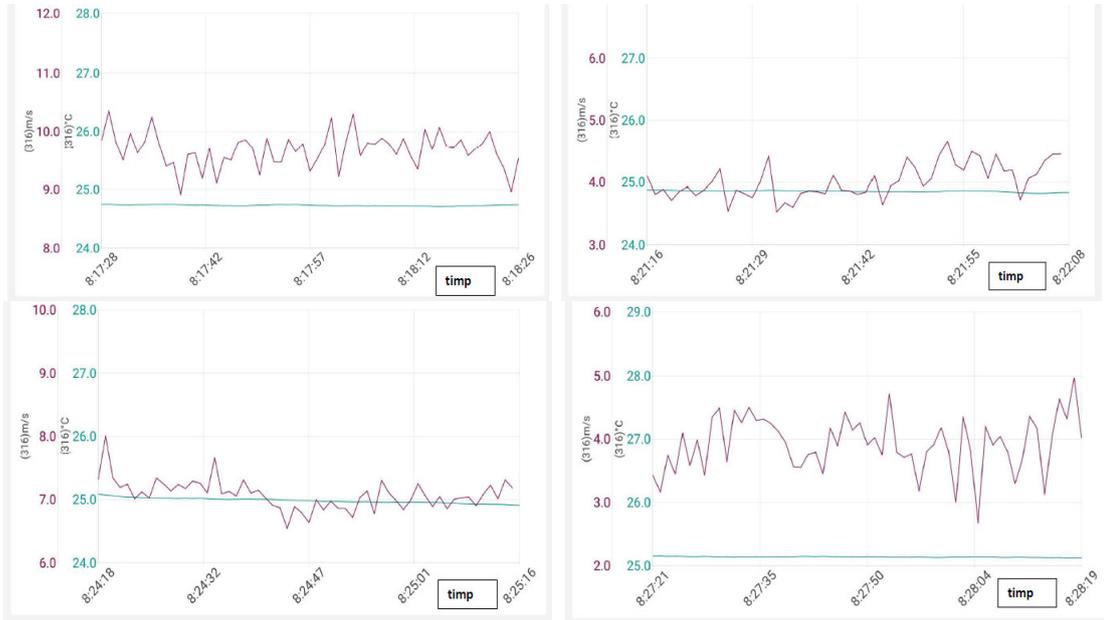
For the experimental research, the parameters monitored for the ventilation system of the machinery room were as follows:

- Air speed on the ventilation system;
- Air temperature on the ventilation system.

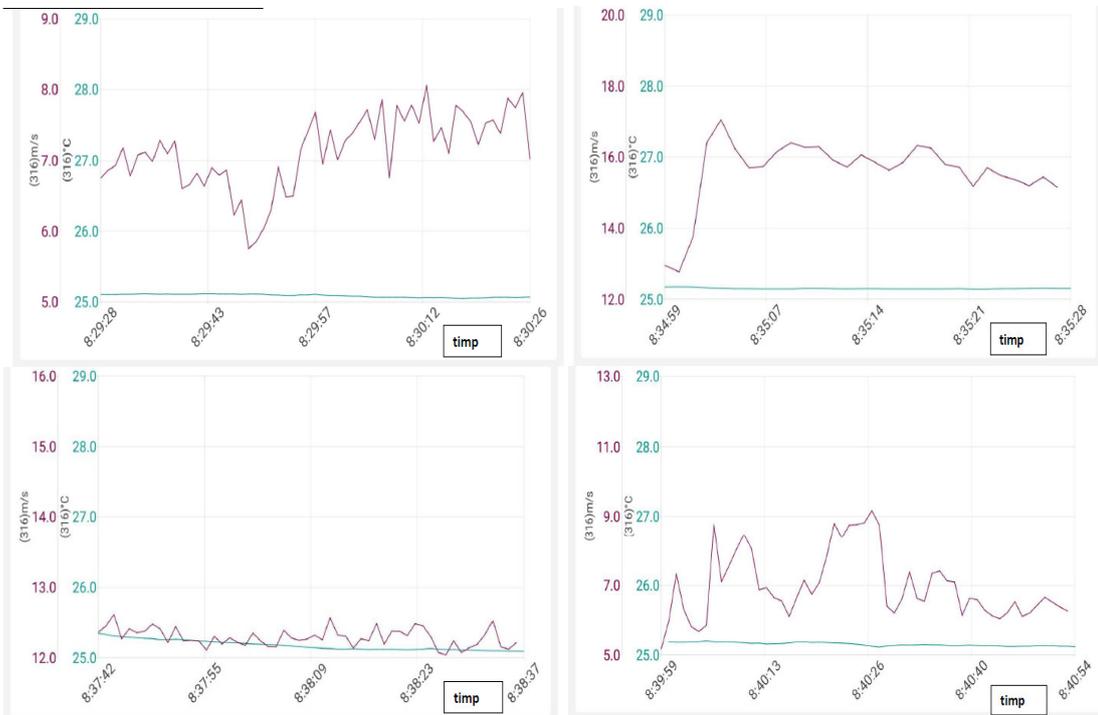
To carry out the measurements, 9 measuring points from the ventilation system were taken into account according to figure 4. The results and the graphs of speed and temperature obtained as a function of time were represented in figures 5, 6 and 7.



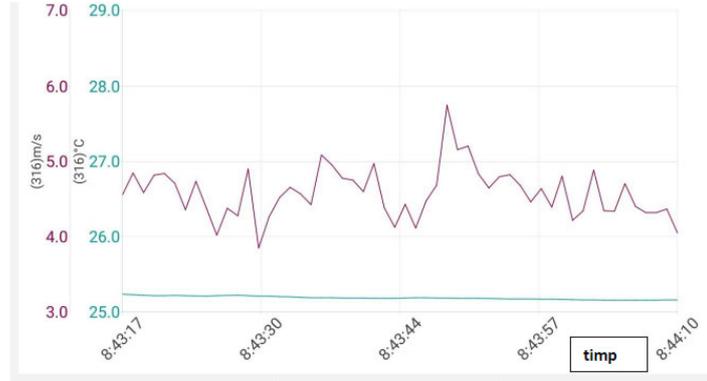
**Figure 4.** Diagram for measuring points [author Visio drawing]



**Figure 5.** Graph of measured values for point numbers 1, 2, 3 and 4 [graph generated by the Testo 405i measuring instrument]



**Figure 6.** Graph of measured values for point numbers 5, 6, 7 and 8 [graph generated by the Testo 405i measuring instrument]



**Figure 7.** Graph of measured values for point number 9 [graph generated by the Testo 405i measuring instrument]

#### 4. Simulation of the ventilation system and comparing results

The outcomes given by the computation and verification algorithm of the ventilation system were accomplished numerically with the assistance of the Fluent program created by Ansys. CFD represents an electronic mathematical procedure for solving research issues including liquid flow. This includes flow algorithms calculations, turbulence displaying model, time integration diagrams and more. So, CFD refers to settling and solving Navier-Stokes conditions. The Fluent program is utilized for a wide variety of designing applications, for example, displaying heat transfer and phase mixing through or around bodies in with various geometries, modelling fluid flow, etc.[5]

##### 4.1. Boundary conditions for the mathematical model

The air flow velocity distribution on the ventilation system is led by the laws of conservation of mass, momentum and energy. The finite element method was used to discretize the geometry system shown in figure 8 and the flow is considered constant, turbulent, incompressible and three-dimensional (3D). The standard k-ε model is used to model turbulent flow to highlight direction lines for velocity. The purpose of the paper is to determine the distribution of air velocity on the ventilation system to optimize the operation of the entire system. The standard model k-ε is based on the model of the equation of motion for kinetic energy (k) and its dissipation (ε). In the equations below we have mass conservation or continuity equation (1), moment equation (2), turbulent kinetic energy equation (3) and energy distribution (4). Limit conditions for the simulation are shown in table 2.[5][6]

$$\frac{d\rho}{dt} + \nabla x(\rho \vec{u}) = 0 \quad (1)$$

$\rho \left[ \frac{kg}{m^3} \right]$ – is density,  $t [s]$ – is time,  $u \left[ \frac{m}{s} \right]$ – is vector of fluid velocity;

$$\frac{d\rho}{dt} + \nabla x(\rho uu) = -\nabla p + \rho g + \nabla x(\mu \nabla u) - \nabla x \tau_t \quad (2)$$

$p [Pa]$ – is pressure,  $g \left[ \frac{m}{s^2} \right]$ – is gravitational acceleration,  $\mu [Pa \cdot s]$ – is dynamic fluid viscosity,  $\tau_t$ -is divergence of turbulence loading;

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u k)}{\partial x} = \frac{\partial}{\partial x} \left[ \mu + \frac{\mu_t}{Pr_k} \right] \frac{\partial k}{\partial x} + \mu_t G - \rho \varepsilon + S_{k,p} \quad (3)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho u \varepsilon)}{\partial x} = \frac{\partial}{\partial x} \left[ \mu + \frac{\mu_t}{Pr_\varepsilon} \right] \frac{\partial \varepsilon}{\partial x} + \frac{\varepsilon}{k} [C_1 \mu_t G - C_2 \rho \varepsilon] + S_{\varepsilon,p} \quad (4)$$

In equations no 4 and no 5:  $C_1$  and  $C_2$  – are empirical constants,  $Pr$  – is Prandtl number for kinetic energy,  $S$  – is user-defined source term,  $G$  – is turbulence of kinetic energy calculated in equation no 5.

$$G = \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \frac{1}{\rho^2} \frac{\partial \rho}{\partial x_j} \frac{\partial \rho}{\partial x_i} - \frac{2}{3} \left( \frac{\rho k}{\mu_t} + \frac{\partial u_i}{\partial x_j} \right) \frac{\partial u_j}{\partial x_i} \quad (5)$$

#### 4.2. Model geometry

The results provided by the calculation and verification algorithm of the ventilation system were achieved numerically in Ansys Fluent. With Design Modeler option offered by the Ansys, was made the geometric construction of the ventilation section as shown in figure 8. For this simulation, it was built the configuration of the ventilation system corresponding to the forward auxiliary machinery room for the special ship taken as reference according to figures 2 and 3.[5][7]

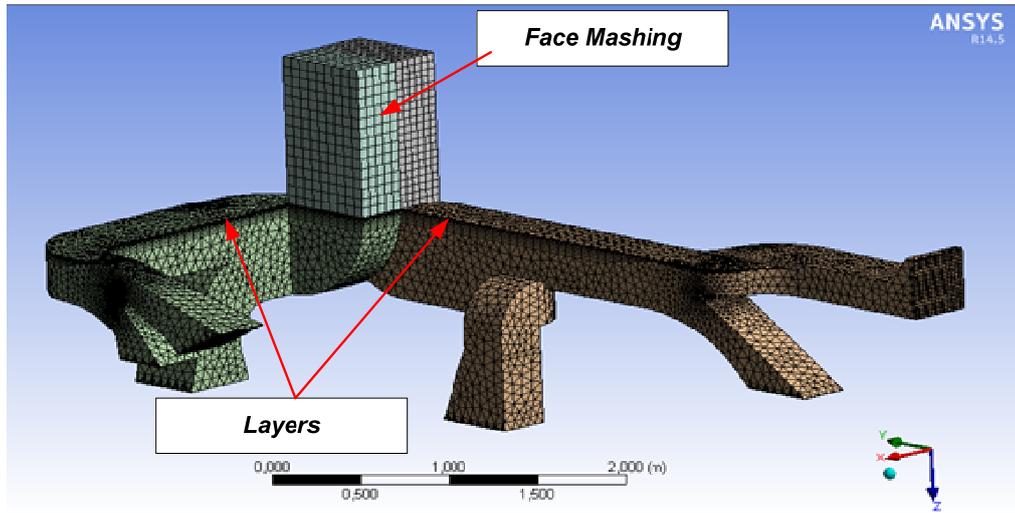


Figure 8. Geometry discretization

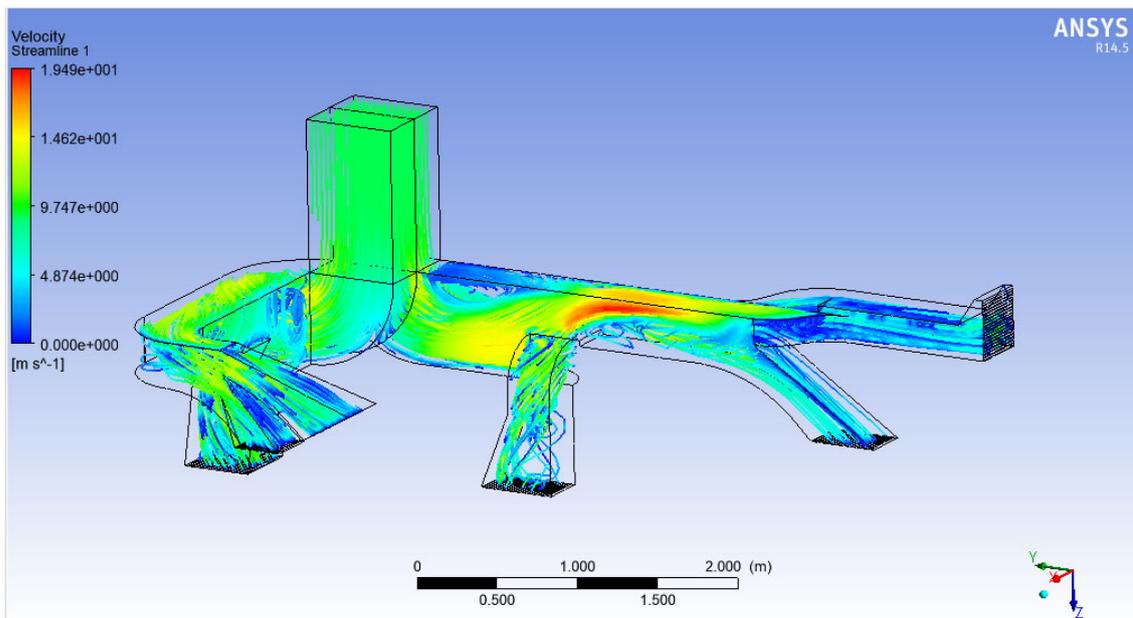
Table 2. Limit conditions for the simulation [5]

Nr.crt.	Limit condition	Value
<b>Inlet</b>		
1.	Velocity inlet	9 $\left[ \frac{m}{s} \right]$
2.	Specification method	Intensity and viscosity ratio
3.	Turbulent intensity	5 [%]
4.	Turbulent viscosity ratio	10
<b>Outlet</b>		
5.	Pressure outlet	-
6.	Specification method	Intensity and viscosity ratio

7.	Turbulent intensity	5 [%]
8.	Turbulent viscosity ratio	10
	<b>Wall</b>	
9.	Wall	

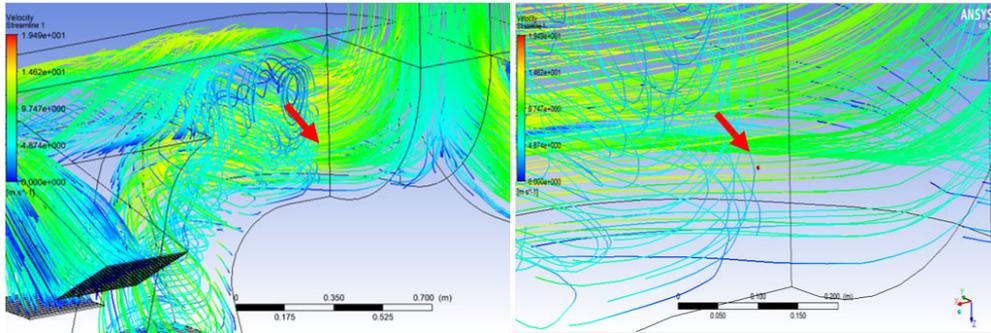
#### 4.3. Simulation and results for reference points

The variation of the air speed on the ventilation system can be seen in figures from 9 to 18. The results obtained have global values of speed in the range  $0 - 19.49 \left[ \frac{m}{s} \right]$ . Figure 9 shows quite explicitly how the air velocity is distributed on the ventilation system and figures from 10 to 18 show the values in studied points. Ansys-Fluent offers a fine variation of the working parameters of the air. Thus it can be concluded that the Ansys-Fluent numerical program provides a better diagram of air speed. It is observed that on the discharge sections the air speed has a value mainly in the range  $0 \dots 9.747 \left[ \frac{m}{s} \right]$ .

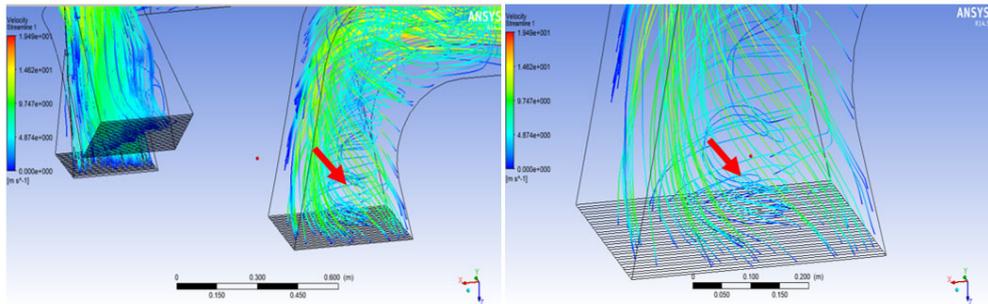


**Figure 9.** Velocity simulation

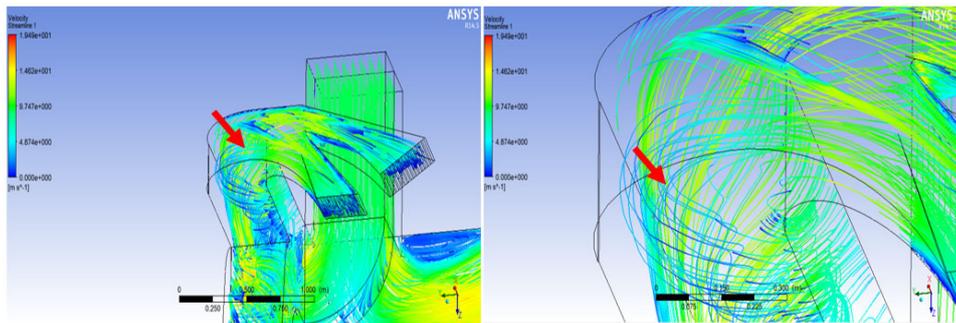
Figures from 10 to 18 show speed measurements using the Ansys program at the points where measurements were also made manually with the TESTO tool. The purpose of the measurements at the specified points is to compare the values from Ansys with the experimental ones for validation. Thus, the obtained were centralized in table 3.[8]



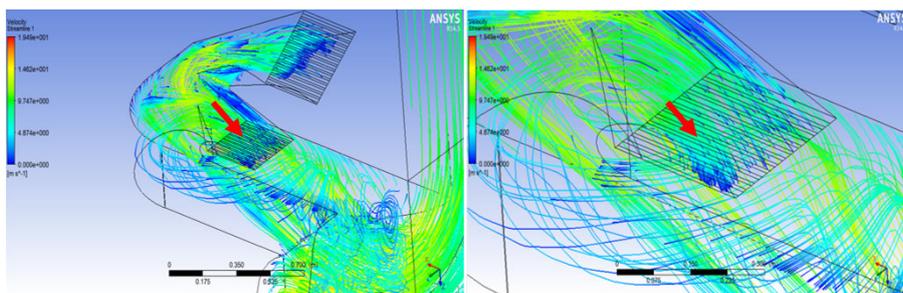
**Figure 10.** Air speed calculated by Ansys programme for point no 1



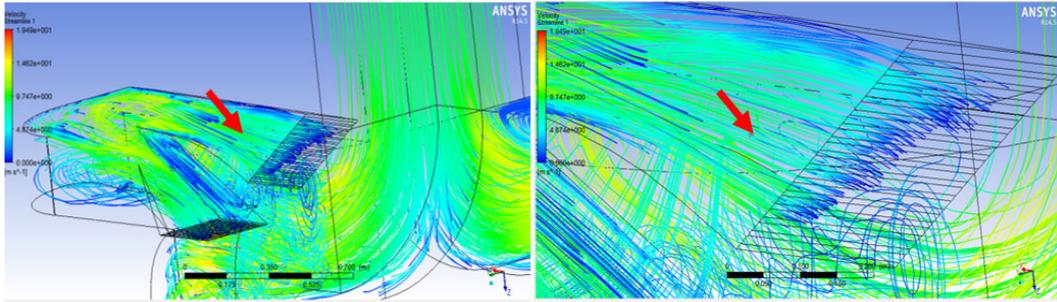
**Figure 11.** Air speed calculated by Ansys programme for point no 2



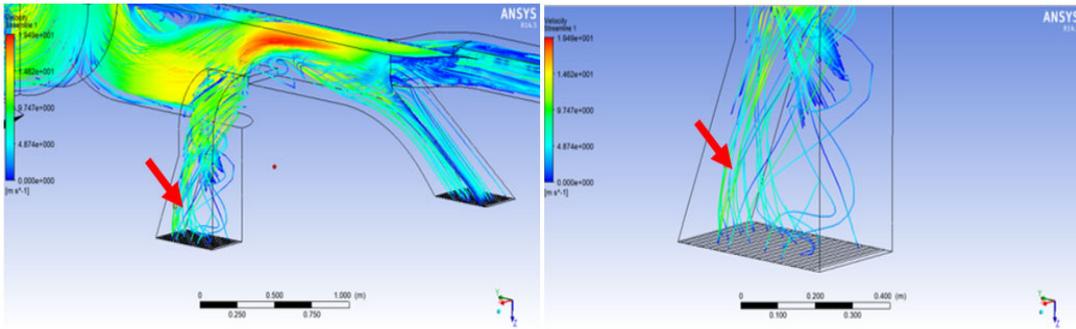
**Figure 12.** Air speed calculated by Ansys programme for point no 3



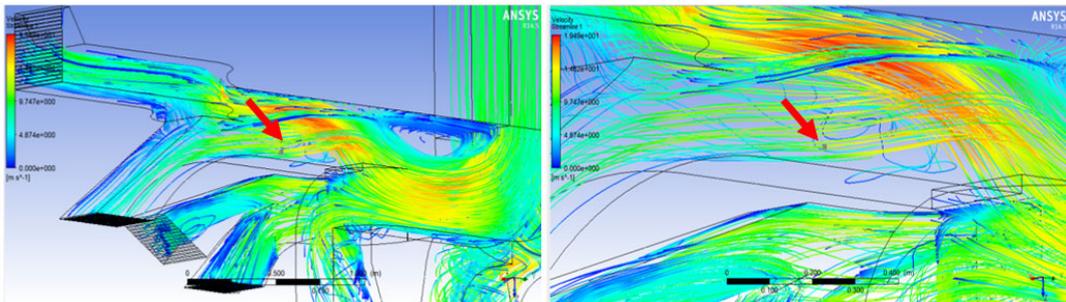
**Figure 13.** Air speed calculated by Ansys programme for point no 4



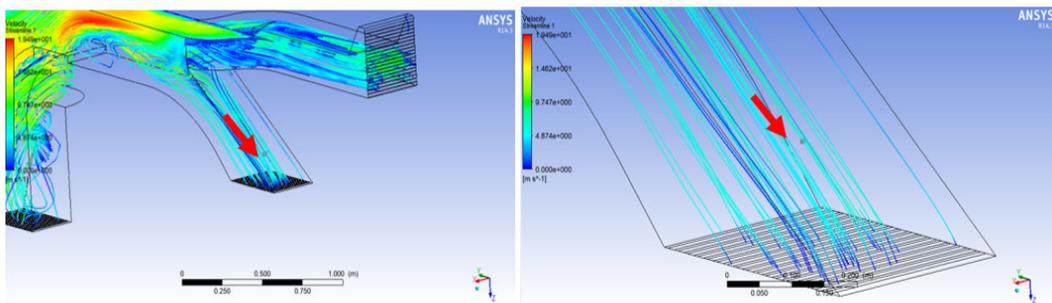
**Figure 14.** Air speed calculated by Ansys programme for point no 5



**Figure 15.** Air speed calculated by Ansys programme for point no 6



**Figure 16.** Air speed calculated by Ansys programme for point no 7



**Figure 17.** Air speed calculated by Ansys programme for point no 8

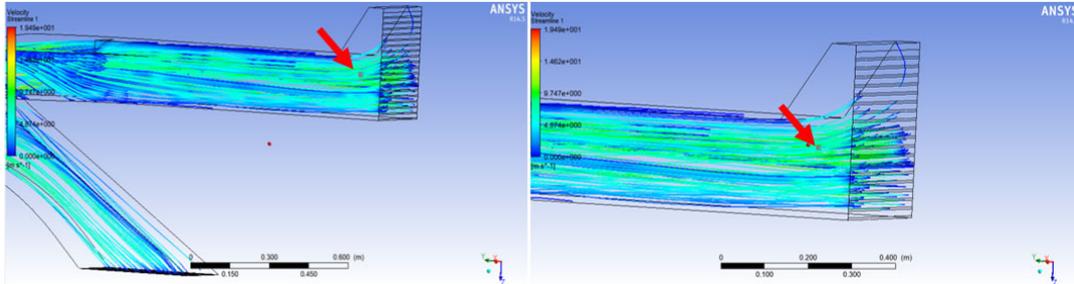


Figure 18. Air speed calculated by Ansys programme for point no 9

Table 3. Comparing results

Mesure point / ventilation inlet / outlet	Air speed measured by TESTO tool [m/s]	Air speed measured by Ansys [m/s]	Error [%]
1	9.706	9.620	0.88605
2	4.057586	4.112	1.341044
3	7.07871	7.056	0.320821
4	3.896667	3.976	2.03592
5	7.157297	7.013	2.016082
6	15.56118	11.892	23.57906
7	12.27	12.389	0.969845
8	6.995667	6.651	4.926864
9	4.554138	4.428	2.769745
<b>Medium error</b>			<b>3.863</b>

The measurements were performed for a short period, but before that, the ventilation system was allowed to operate for a period of 2 hours, long enough to reach the nominal operating state. The speed values were calculated at the measuring points and compared with the simulation results. A speed difference was observed at the measuring point number 6 where the measurements were higher than those obtained with the Ansys program and the error obtained was 23.57 [%]. This involves optimizing the ventilation sections in the mentioned areas both in terms of configuration and in terms of roughness.[9]

## 5. Conclusions

Ansys software displayed a series of information resulting from physical data and corresponding calculations following the geometry of the machinery space. Environmental conditions have repercussions on the airflow ventilating the machinery room. The simulation results in the Ansys program were very well correlated with the experimental measurements made on the real ventilation system.

The possibility to carry out, over a period of time at choice, experimental measurements on a 1:1 scale ventilation system located on board a special ship, allowed the validation of the results obtained with the numerical program. The validation was performed using a real machinery compartment ventilation system. The results were satisfactory, which confirms that the program created can also be used by students, PhD students, researchers and specialists within units of special ships to deepen the understanding of fluid flow and proper ventilation of ship's machinery compartments.

Simulation of air flow in the ventilation system takes into account both the atmospheric conditions and the operating conditions of the plants in the machinery compartment. The standard k-ε model was

used to demonstrate the effect of operating conditions on the ventilation system in the machinery compartment.

One of the first optimization directions, and the handiest, is to improve and optimize the calculation program through the variety of simulation conditions that may be required. Thus, simulations can be made for different working conditions of the ship (ship in operation, ship at anchor, ship in winter or summer conditions) [10], thus optimizing the entire system. Another direction of further research is the study of the elimination of heat flow from the machinery compartment. The heat flow or thermal footprint left by a special ship is vital in operation. For this reason, the optimization of the ventilation system in the sense of reducing the heat flow produced by the aggregates from machinery space is a very important point. The research can be extended to find the optimal variant of the ventilation system so that the intended purpose is achieved.

### References

- [1] Lloyd's Register – Military Design and Special Features, January 2015;
- [2] Lloyd's Register - Seagoing Ships, Ventilation, 2014;
- [3] \*\*\* <https://www.seaforces.org/marint/Royal-Navy/Frigate/Broadsword-Type-22-class.htm>
- [4] Standard 02-102 (NES 102), Requirements for air conditioning and ventilation, part 1 - (<ftp://ftp.iks-jena.de/mitarb/lutz/standards/dstan/02/102/01000200.pdf>), UK, September 2000;
- [5] \*\*\*[www.ansys.com](http://www.ansys.com) / simulation;
- [6] American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc., Ventilation for Acceptable Indoor Air Quality, ANSI/ASHRAE Standard 62.1-2007;
- [7] Naval Surface Warfare Center Carderock Division, 21 Century HVAC System for Future Naval Surface Combatants – Concept Development Report , USA, September 2007;
- [8] <https://www.testo.com/ro-RO/>;
- [9] Levent Ali, Beazit Ali and Daniel Mărășescu, Consideration on engine room ventilation, Proceedings of the 7th International Scientific Conference SEA-CONF 2021, issue no. 1, ISSN 2457-144X
- [10] NATO Standard ANEP-77 – *Naval Ship code* – Aprilie 2017 – NATO Standardisation office.