ANALYZING EFFICIENCY OF VENTILATION ELEMENTS USED IN PROTECTIVE CLOTHING WITH SIMPLIFIED MODEL

Sanjay-Rajni Vejanand, Alexander Janushevskis, Ivo Vaicis
Riga Technical University, Latvia
sanjay.vejanand@rtu.lv, aleksandrs.janusevskis@rtu.lv

Abstract. Applications of Computational Fluid Dynamics (CFD) extend to a wide range of academic disciplines, research studies, and industries. The importance of CFD validation and verification is growing as ventilation and fluid flow interaction problems are more complex and difficult to solve. Such problems usually take large computational time to provide solution depending on complexity of the model. Some problem depending on its definition can be simplified to reduce the high computation time. This study involves the simulation of two different models, referred as case 1 and case 2, in order to evaluate the efficiency of two distinct ventilation elements, E1 and E2, with different shapes. The first model is a simplified elliptical shape of the human body with a protective jacket that is 500 millimetres in height and consists of ten outlets and a single inlet. In the second case, this model is simplified by having a considerably smaller area; it is composed of two square plates with dimensions of 40×40 mm. In this case, one plate is a representation of a jacket surface that has a single inlet, and the other plate represents a human body. SolidWorks Flow Simulation is used to simulate both models individually to compare efficiency of ventilation elements E1 and E2. The flow simulation results in the first case do not provide sensitive values for the comparison, while in the second case sensitivity of results increases and it shows that the element E1 provides better cooling efficiency than E2. The results also indicate that computational time in the second case is reduced 15 times.

Keywords: CFD, ventilation elements, SolidWorks, protective clothing, computation time.

Introduction

The foundational work of researchers like Richardson [1] and Courant, Friedrichs, and Lewy [2] laid the groundwork for computational fluid dynamics (CFD). Their pursuit of understanding fluid motion led to the creation of robust numerical methods that have improved the numerical representation of various forms of fluid flow [3]. CFD is rapidly becoming into an influential and prevalent tool across numerous sectors; every solution embodies a complex web of mathematical physics, numerical methodologies, user interfaces, and cutting-edge visualization approaches [4]. Fluid flow problems are more often solved with computational fluid dynamics (CFD) as they are more conventional, analytical modelling and experimental approaches. The high expenses and time consumption of experiments have frequently prevented the pursuit of efficient, in-depth results, making the current acceptance of CFD both progressive and unavoidable [5].

For both experimental and computational fluid dynamics (CFD) purposes, it is necessary to precisely simulate the properties of the atmospheric boundary layer (ABL) in order to accurately anticipate the wind effect on built objects [6]. Thermal modelling of structures will also be affected by the flow characteristics [7]. The best approach for accurately representing turbulence is to maintain the spectral characteristics of the input flow [8]. It is ideal to match velocity spectra with those obtained at full-scale, including the complete frequency range. Yet, accomplishing this goal through small-scale experiments and simulations is challenging. One reason for inaccurate pressure predictions in reduced scale experiments is the occurrence of small-scale turbulence that causes the shedding of immature vortices. These experiments are both economically expensive and difficult to handle [9]. On the other hand, full-scale computational fluid dynamics (CFD) simulations need a significant amount of processing time and resources [10].

The primary objective of this study is to evaluate the effectiveness of ventilation elements E1 and E2 while minimizing the computation time. These ventilation elements are intended to be used in protective clothing for efficient cooling and to restrict direct access of insects, rain, and dust to human body [11]. Thus, the choice of the right element is very important. It is important to use fine mesh in simulation to obtain precise results but with fine mesh, computation time increases intensively, while using coarse mesh may not give precise values but it might work in comparison while using the same set of parameters in predicting efficiency. However, in most of simulations using fine mesh is important to achieve accurate results. In this work, ventilation element E1 and E2 are compared in two different cases to identify the most efficient element with respect to optimized computational time. In the first

DOI: 10.22616/ERDev.2024.23.TF036

scenario, a simplified elliptical model of the human body and jacket is utilized. In the second case, the model is further simplified into two square plates, with one plate representing the jacket surface and the other representing the body. SolidWorks 3D flow simulation is used in this study to obtain results at the inlet air velocity of 2 m·s^{-1}. The detail model dimensions and boundary conditions are described in the next section.

Model components and boundary conditions

Fig. 1 shows a schematic diagram of the model used in the first scenario. All the mentioned dimensions are in millimeters. The jacket has a single inlet with a diameter of 4.4 mm at the front, and ten outlets with a diameter of 4 mm at the back. Only a single ventilation element is attached at the inlet ventilation hole in a gap between the body and jacket, shown in Detail View D (3.4mm) in Fig. 1. The ventilation elements E1 and E2 are positioned individually in a concentric manner with respect to the inlet ventilation hole shown in the front view of Fig. 1, to obtain simulation results. These results are then compared to analyze efficiency of the elements. These ventilation elements are shown in Fig. 2 and 3.

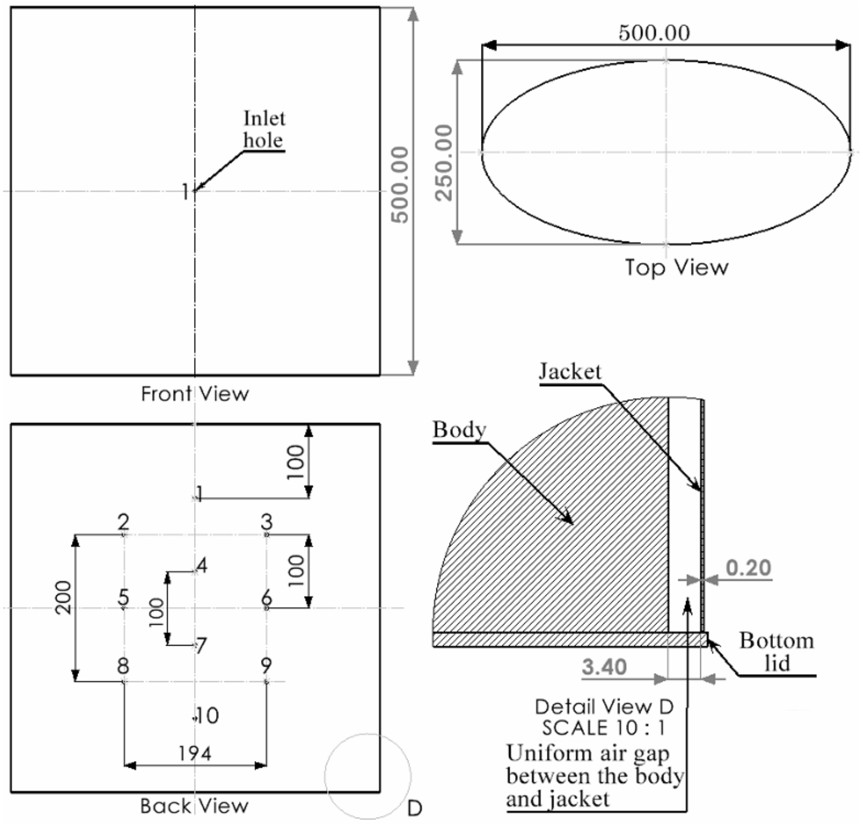
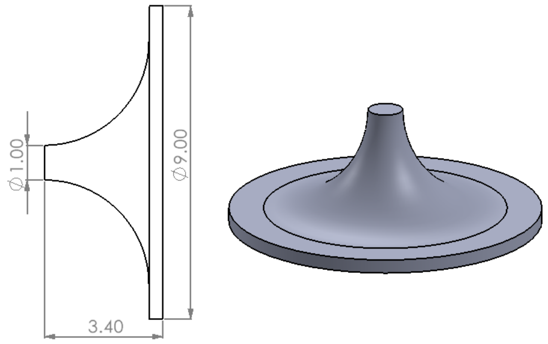


Fig. 1. **Case 1: Simplified elliptical model of human body and jacket [12]**



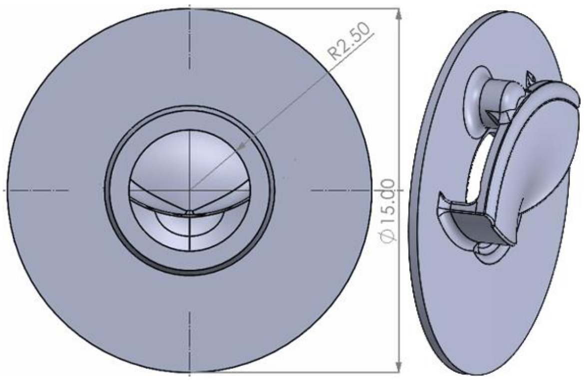


Fig. 2. **Ventilation element E1**

Fig. 3. **Ventilation element E2**

The inlet air temperature is set at 20 ºC and the air pressure is set at 101325 Pa for the 3D flow simulation study. This study is made at the inlet air velocity of 2 $m \cdot s^{-1}$ and the physical time of the study is 10 seconds. Different materials with specific qualities are initially assigned to the body and jacket in the simulation. Table 1 shows these material properties. We consider a normal human body temperature of 36.5 ºC and a heat generation rate of 200 W while normal walking conditions [13].

Table 1

Material Properties [14, 15]

Material property	Human body	Jacket
Average density, $kg \cdot m^{-3}$	985	1420
Specific heat, $J \cdot kg^{-1} \cdot K^{-1}$	3500	1140
Thermal conductivity, $W \cdot m^{-1} \cdot K^{-1}$	0.21	0.261

Some of considerations in the Flow Simulation study:

- Top and bottom part of the jacket sealed off (means no air passes through it), to investigate the effectiveness of the ventilation.
- The analysis does not take radiation into account because the amount of heat lost due to radiation is the same for all scenarios.
- Heat is transferred from the body to the jacket and then to the atmosphere by convection and conduction.

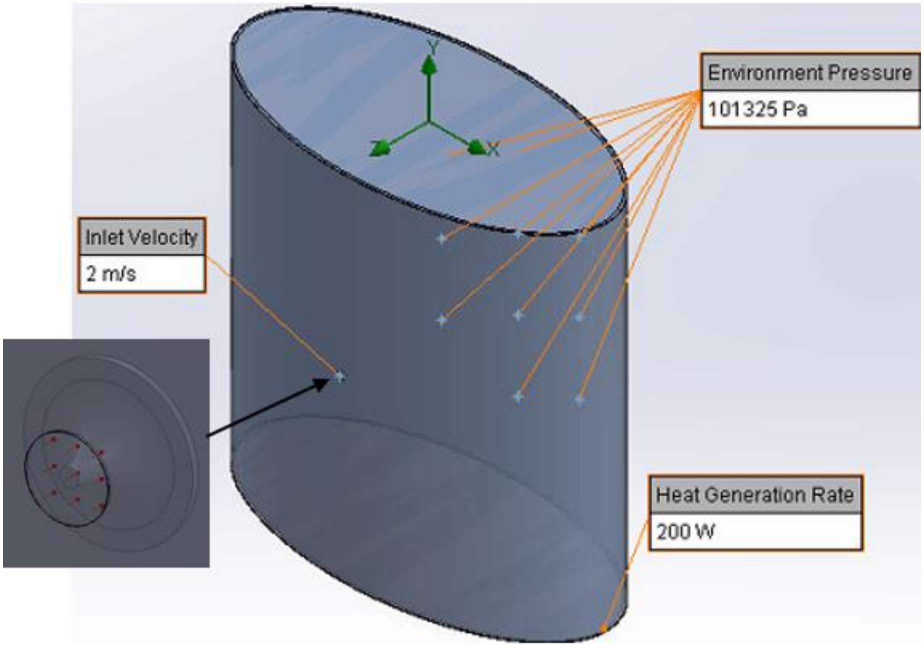


Fig. 4. **Boundary conditions in CFD study for case 1**

The study boundary conditions are depicted in Fig. 4. The ventilation element is positioned concentric to the ventilation hole, so that the front face of the element is directly aligned with the hole. Fig. 4 illustrates this configuration, with the enlarged view of the ventilation region indicated by an arrow sign. The dark circle in the enlarged view indicates the position of the ventilation hole. The inlet velocity of 2 $m \cdot s^{-1}$ is used as the inlet boundary condition. The airflow is perpendicular to the front surface of the model. The 10 outlets on the backside of the model are set to environmental pressure as their output boundary conditions. In the present study, SolidWorks Flow Simulation is used for obtaining results, which employ transport equations for the turbulent kinetic energy and its dissipation rate, using the modified k-E turbulence model with damping functions proposed by Lam and Bremhorst [16].

In case 2, the model design is simplified to two square plates with reduced area, where one plate represents the jacket and other represents the body. This model design in shown in Fig. 5. The same set of parameters are used in the flow simulation study as described above. The only difference in this case is that external flow simulation is considered here with enclosed boundary to the model dimensions. The length of the computational domain is 73.40 mm. Fig. 6 shows the computational domain, where the red arrow in front shows the inlet velocity and the blue arrow shows the environmental condition.

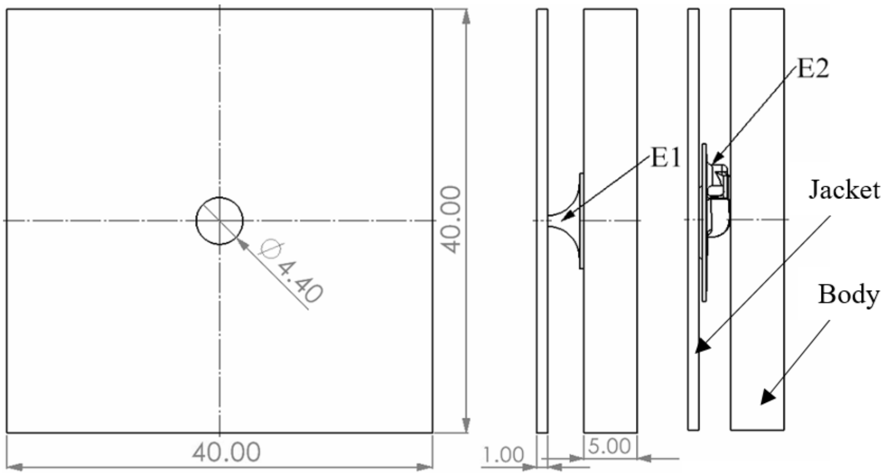


Fig. 5. **Case 2: Simplified model with two square plates**

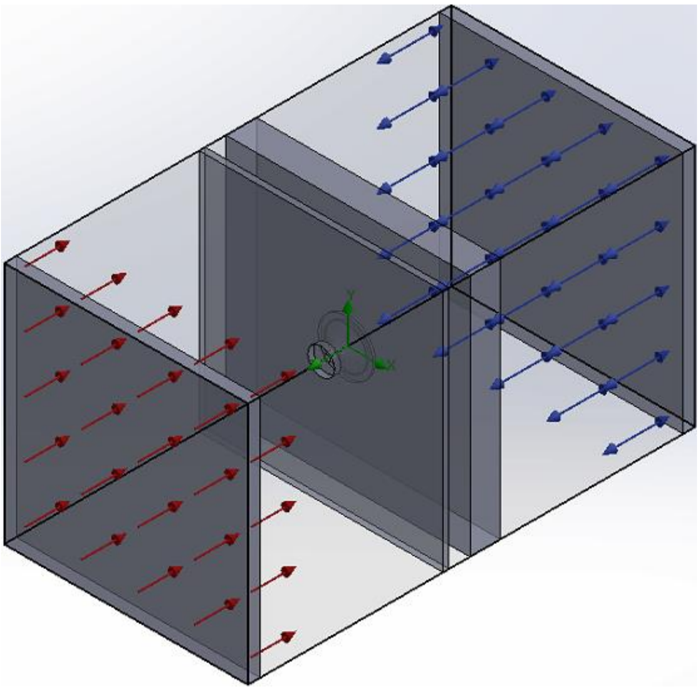


Fig. 6. **Case 2: Computational domain**

Here are the criteria that were considered in the analysis of the flow simulation study:

1. H.T.R. – Heat Transfer Rate, W;
2. H.F. (avg.) – Surface Heat Flux (average), W·m^{-2};
3. dP – Flow Pressure Difference, Pa;
4. dT - Surface Temperature Difference (body), ºC;
5. T. avg. – Average Surface Temperature (body), ºC.

Results and discussion

All the results mentioned here are simulated for the physical time of 10 seconds and since this study is transient in nature, the physical time of the study has a great influence on the results and the computational time. Additionally, the size of the mesh significantly affects both the outcomes and the time required to complete the calculations. Incorporating a fine mesh in the study enhances the precision of the results, significantly lengthening the processing time required for the study. To ensure the accuracy of results, a fine mesh is employed in this study.

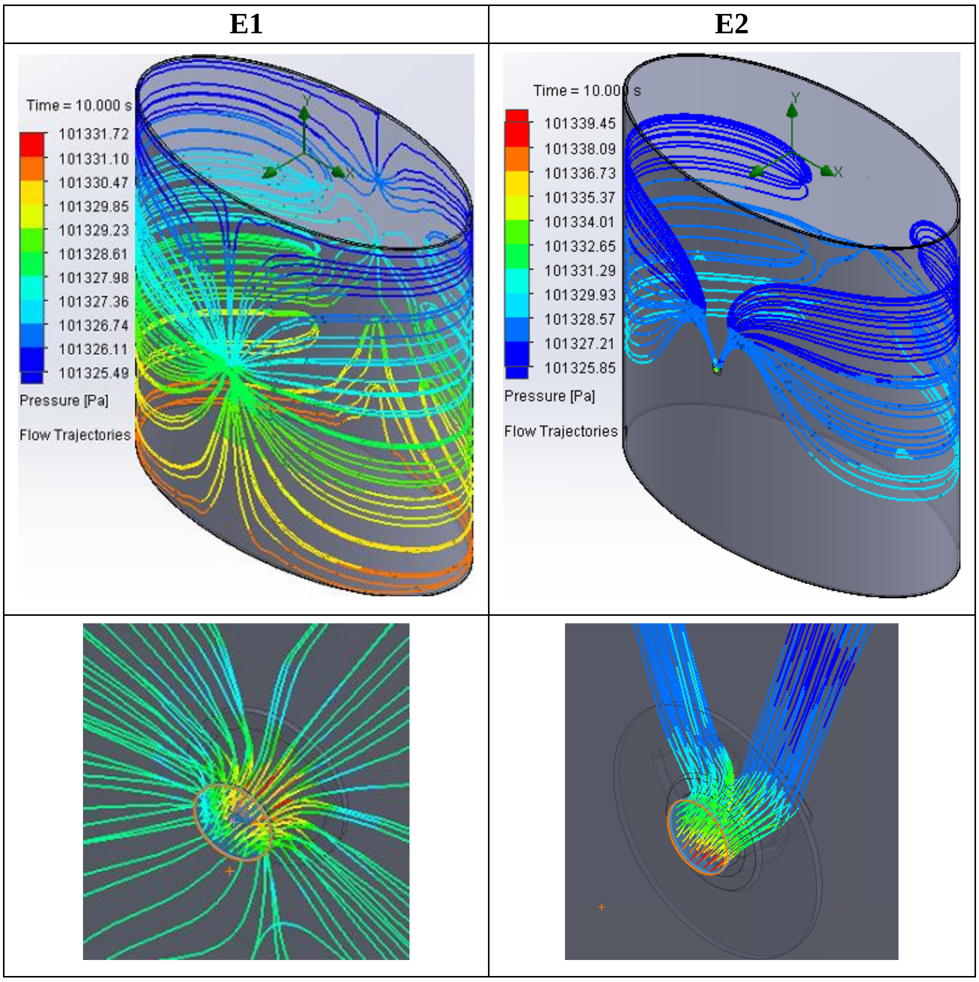


Fig. 7. **Flow pressure plots**

In Fig. 7, the first row shows the pressure distribution over the entire model and the second row shows a zoom view of the pressure variation near the inlet flow channel of the ventilation element E1 and E2 respectively. If we have a look on the flow trajectories over the entire model then it is clearly visible that element E1 shows better flow distribution than E2 as E1 provides flow distribution on all sides while in E2 most of the air flows upward and sideways leaving the bottom part of the model. Moreover, the second row images clearly demonstrate that the pressure variation in E1 is more gradual compared to E2. The second picture in E2, displays dark blue lines indicating a sharp drop in pressure, which signifies higher-pressure fluctuation.

In Fig. 8, the first row shows the surface temperature of the body over the entire model and the second row shows a zoom view of the temperature variation near ventilation. Table 2 displays more detailed values of the investigated parameters.

Table 2

Numerical values of the results for case 1

Parameters	**E1**	**E2**	**Difference in values between E1 and E2**
H.T.R., W	8.739	8.702	0.037
H.F. (avg.), W·m^{-2}	12.154	12.065	0.089
dP, Pa	6.23	13.60	7.37
dT, ºC	4.75	4.70	0.05
T.(avg.), ºC	36.44	36.44	0

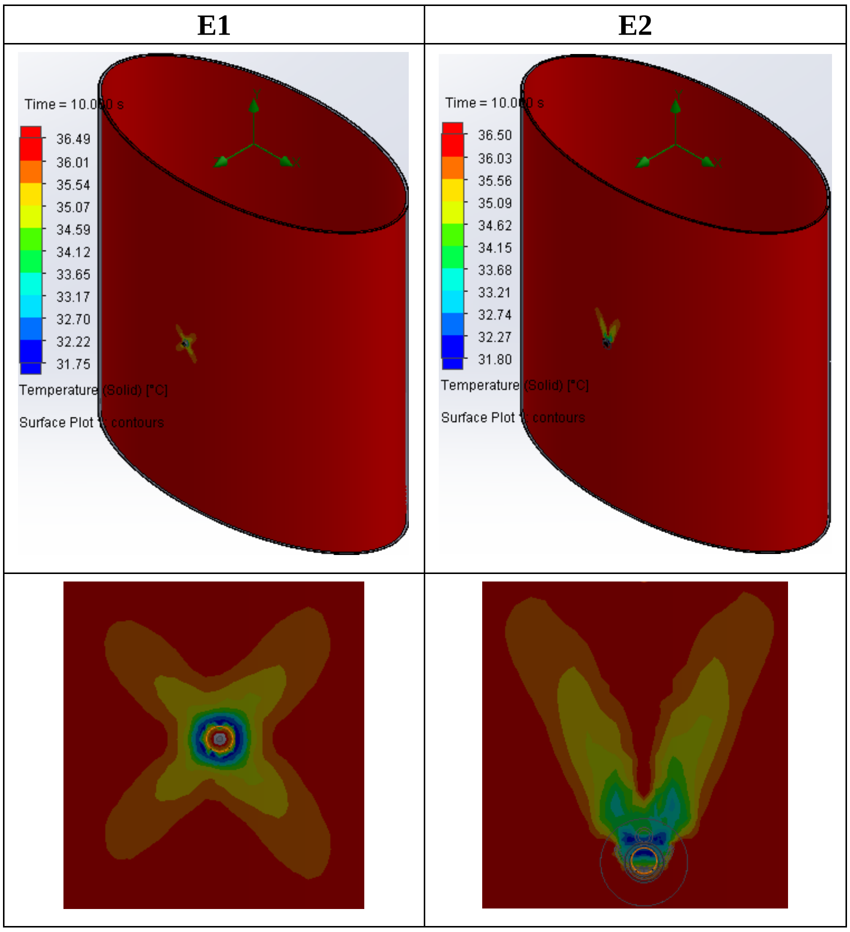


Fig. 8. **Surface temperature plots**

As it is visible from Table 2, the values of the results obtained for E1 and E2 have a relatively low sensitivity in predicting the efficiency of the elements. This is because the area of the model is higher, and we have only a single inlet ventilation. The value of dP shows a sensible difference in the results for E1 and E2, but this difference is not big enough to provide a precise conclusion. Therefore, it is important to evaluate the reliability of the results by including more than one criterion prior to reaching the conclusion. One option to obtain comparable results is to increase the number of ventilation units but it will increase significant amount computational time, hence it is not an ideal option. The other option is to reduce the area of the model proportional to the ventilation element that could provide sensible results and significantly reduce the computation time. This option we have considered for the model used in case 2 (Fig. 5). The same set of criteria was considered in case 2, and detailed values of the obtained results are mentioned in Table 3.

Table 3

Numerical values of the results for case 2

Parameters	**E1**	**E2**	**Difference in values between E1 and E2**
H.T.R., W	0.328	0.196	0.132
H.F. (avg.), W·m^{-2}	207.097	124.507	82.59
dP, Pa	3.28	4.20	0.92
dT, ºC	2.91	4.03	1.12
T.(avg.), ºC	35.34	35.90	0.56

Table 3 shows that the sensitivity of the surface heat flux (H.F.) and the average body temperature {T. (avg.)} increases in case 2. Element E1 provides a higher value of heat flux (H.F.) compared to E2, indicating that the heat transfer rate of E1 is better. A higher heat transfer rate refers to a higher cooling efficiency. Therefore, the average body temperature for E1 (35.34 ºC) is lower than that of E2 (35.90 ºC).

Based on these two figures, it can be concluded that element E1 offers better cooling efficiency compared to E2. The final step is comparing the time required to complete the simulation process in both the scenarios. In the first case, the computation process took about 16 hours, however, in the second case, it took only 1 hour. This shows effective time optimization as the computation time in case 2 is reduced by 15 times.

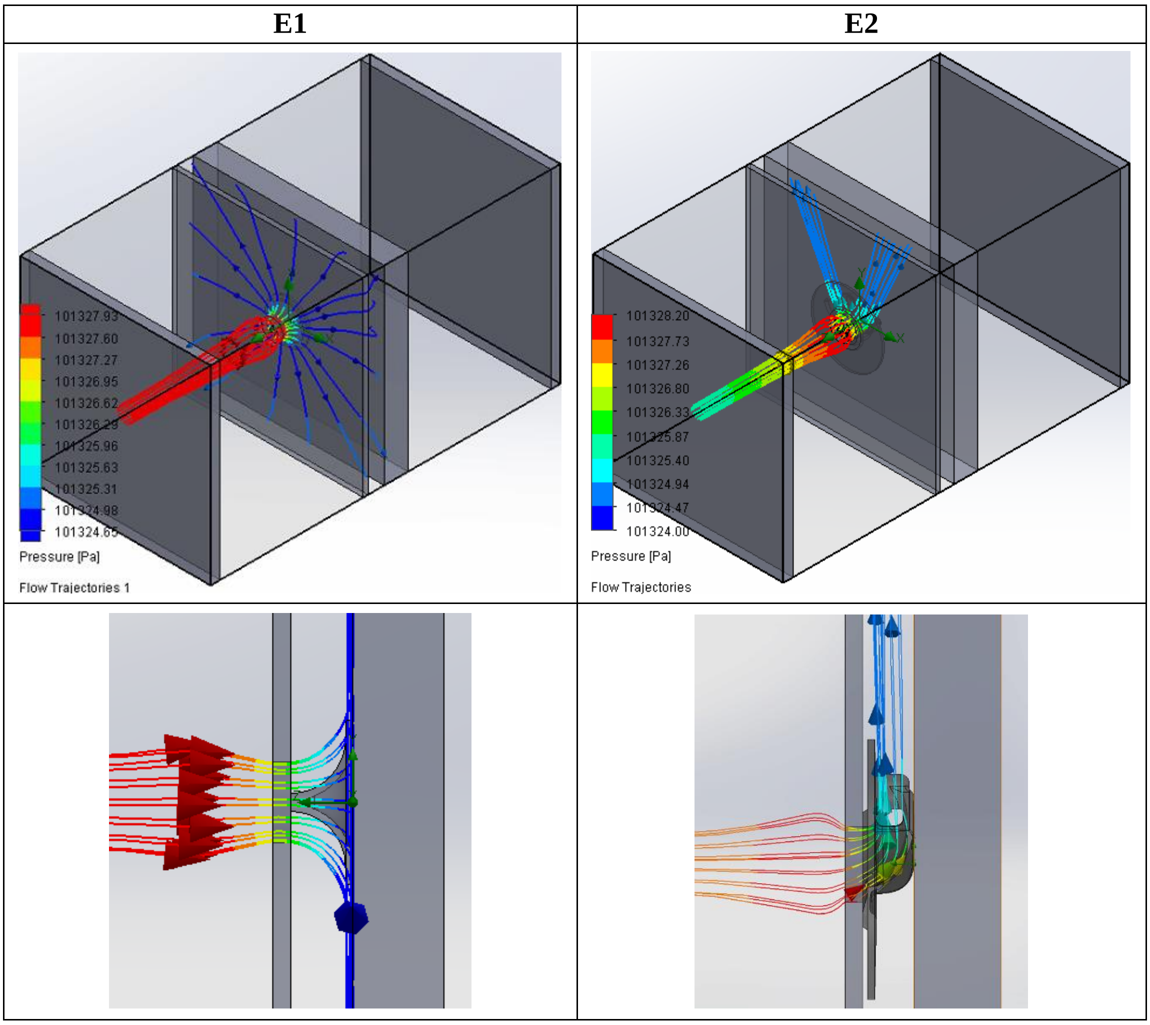


Fig. 9. **Flow trajectories (Case 2)**

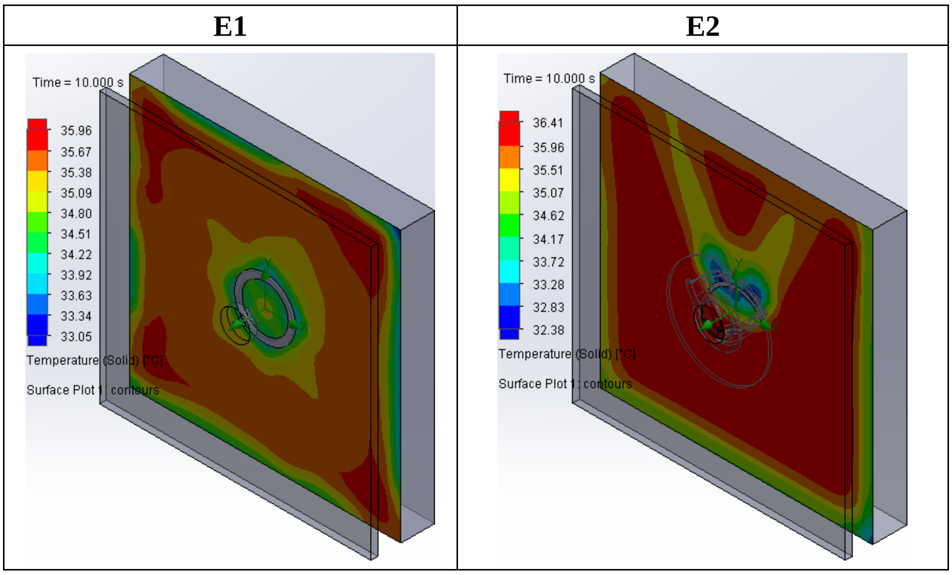


Fig. 10. **Surface temperature (Case 2)**

These calculations were done on Core i7, 8-core processor computer. The parameters like the processing power of the computer, physical time of the study and the final output required can highly influence the computation time of the study.

Conclusions

From the result analysis it can be concluded that the results in the first case do not provide sensitive values for predicting the efficiency of element E1 and E2. This is due to the larger surface area of the model in comparison to ventilation. In order to solve this issue, the model is simplified into two square plates with a significantly less surface area in the second case. As a result, the sensitivity of the obtained results is enhanced in this case. The most significant criterion for comparison is the heat flux (H.F.) and the average body temperature (T. avg.), indicating that element E1 offers better cooling efficiency compared to E2. Moreover, the results also indicate a significant reduction in the computing time, specifically by an amount of 15 times lower than case one. This study clearly indicates that employing a simplified model with a smaller area can provide results that are more sensible and enhance the computational efficiency when estimating the effectiveness of items with smaller dimensions, such as ventilation elements in the current study. It allows in future to efficiently predict the cooling efficiency of different shape ventilation elements with optimal computation time.

Acknowledgments

This research/publication was supported by the Riga Technical University Doctoral Grant programme.

Author contributions

Conceptualization, I.V. and S.R.V.; methodology, S.R.V. and I.V.; software, S.R.V.; validation, S.R.V. and A.J.; formal analysis, S.R.V.; investigation, S.R.V.; data curation, S.R.V.; writing – original draft preparation, S.R.V.; writing – review and editing, I.V.; S.R.V. and A.J.; visualization, S.R.V. and A.J; project administration, S.R.V.; funding acquisition, S.R.V., A.J., and I.V. All authors have read and agreed to the published version of the manuscript.

References

[1] Richardson L.F. The approximate arithmetical solution by finite differences of physical problems involving differential equations, with an application to the stresses in a masonry dam. Philosophical Transactions of the Royal Society of London, Series A, 210 (1910), pp. 307-357.

[2] Courant R., Friedrichs K., Lewy H. Die partiellen differenzengleichungen der mathematischen Physik. Mathematische Annalen (Historical Archive), 100 (1928), pp. 32-74.

[3] Shang J.S. Three decades of accomplishments in computational fluid dynamics. Progress in Aerospace Sciences, 40 (2004), pp. 173-197.

[4] Xia B., Sun D.-W. Applications of computational fluid dynamics (CFD) in the food industry: a review. Computers and Electronics in Agriculture, 34 (2002), pp. 5-24.

[5] Norton T., Sun D.-W. Computational fluid dynamics (CFD) – an effective and efficient design and analysis tool for the food industry: A review. Trends in Food Science & Technology. Volume 17, Issue 11, November 2006, pp. 600-620.

[6] Davenport A.G. How can we simplify and generalize wind loads? Journal of Wind Engineering and Industrial Aerodynamics. Volumes 54–55, February 1995, pp. 657-669.

[7] Pedroso M., Flores-Colen I., Silvestre J.D., Gomes M.G., Silva L., Sequeira P., de Brito J. Characterisation of a multilayer external wall thermal insulation system. Application in a Mediterranean climate. Journal of Building Engineering. Volume 30, July 2020, 101265.

[8] Gol-Zaroudi H., Aly A.M.. Open-jet boundary-layer processes for aerodynamic testing of low-rise buildings. Wind and Structures, Vol. 25, No. 3(2017), pp. 233-259 DOI: 10.12989/was.2017.25.3.233

[9] Liu F.F., Cai Q., Yang G., Leng J.. Field measurements of wind pressure on an open roof during Typhoons HaiKui and SuLi. Wind and Structures, Vol. 26, No. 1 (2018), pp. 11-24 DOI: 10.12989/was.2018.26.1.011

[10] Khaled M.F., Aly A.M., Elshaer A. Computational efficiency of CFD modeling for building engineering: An empty domain study. Journal of Building Engineering. Volume 42, October 2021, 102792.

[11] Janushevskis A., Vejanand S. R., Gulevskis A. Analysis of Different Shape Ventilation Elements for Protective Clothing. WSEAS Transactions on Fluid Mechanics, Vol.17, ISSN: 1790-5087, 2022. pp. 140-146.

[12] Janushevskis A., Vejanand S. R., Gulevskis A.. Comparative Analysis of Different Shape Ventilation Elements for Protective Clothing. 21st International Scientific Conference Engineering for Rural Development. Proceedings. ISSN 1691-5976. Vol.21, Jelgava, 2022, pp. 179-186.

[13] Rajesh Kumar R.K. Aggarwal J.D. Sharma and Sunil Pathania. Predicting Energy Requirement for Cooling the Building Using Artificial Network. Journal of Technology Innovations in Renewable Energy, 2012, 1, pp. 113-121.

[14] Giering K., Lamprecht I., Minet O. Specific heat capacities of human and animal tissues. Proceedings of SPIE – The International Society for Optical Engineering, 2624, January 1996, pp. 188-197.

[15] Janushevskis A., Vejanand S. R., Gulevskis A.. Shape Optimization of Ventilation Elements for Protective Clothing by Using Metamodeling Approach. 22th International Scientific Conference Engineering for Rural Development. Proceedings. ISSN 1691-5976, Vol.22, Jelgava, 2023, pp. 164-172

[16] Lam C.K.G., Bremhorst K.A. Modified Form of Model for Predicting Wall Turbulence. ASME Journal of Fluid Engineering, 103, 1981, pp. 456-460.