Conference Article

Air Quality Optimization in Excavator Cabinets Using CFD

Ekin Can KAYAR[1*], Hasan ÖZCAN[2]

[1] Ankara Yıldırım Beyazıt University, Hidromek A.Ş., Orcid ID: https://orcid.org/0009-0004-4191-587X,
E-mail: ekin.kayar@hidromek.com.tr
[2] Ankara Yıldırım Beyazıt University, Orcid ID: https://orcid.org/0000-0002-0135-8093,
E-mail: h.ozcan@aybu.edu.tr
[*] Correspondence: ekin.kayar@hidromek.com.tr

Abstract

Air conditioning systems are vital to the operation of a vehicle. For this reason, earth-moving machinery manufacturers act very sensitively to provide their consumers with a comfortable journey and clean indoor air. In this study, the heating problem of an excavator operator cabinet is discussed. A climatic test chamber was used to perform the heating test. This allows detailed observation of airflow and temperature distributions within the cabinet. The cleaning of the geometric model and the creation of the flow model were carried out in the SpaceClaim program. The numerical mesh was created using ANSYS Fluent's meshing module. The results were obtained with the Computational Fluid Dynamics (CFD) program Fluent. The necessary data were collected during the acclimatization test while the excavator was running, and the obtained data was used at boundary conditions. The geometric model was optimized and CFD analysis was performed. Temperature measurements surrounding the operator were taken from the specified points in the ISO 10263-4 standard. CFD results show that it can replace expensive testing for excavator air conditioning applications.

Keywords: HVAC, Computational Fluid Dynamics (CFD), Optimization.

1. Introduction

Until the 1990s, driver's compartment air conditioning was mostly neglected in favor of vehicle aerodynamics or engine research. Thermal comfort was not considered a necessity in excavators. Thermal convenience as a factor in customer satisfaction has gained importance over time. The in-cabinet air conditioning system is very important for safety as well as thermal comfort.

Thermal comfort is becoming important as cabinets are now standard on modern excavators. Ambient temperature is one of the main factors that negatively affects the working conditions of excavator operators. In particular, it is difficult to provide comfortable air conditioning in excavator cabinets since weather conditions are unpredictable. As a result, before installing the air conditioning system in the cabinet, the performance of the air flow distribution in the excavator cabinet can be calculated and optimization studies can be easily repeated. Studies on system optimization are important in many respects. The design that provides optimum conditions for the system is completed faster, saving both money and time. In addition, air conditioning optimization research is carried out by determining the excavator air conditioning need according to the planned load.

In addition to computers, cabinet air conditioning systems in excavators are also developing. When the literature research is examined, studies on this subject can be seen. Bandi et al. investigated the effect of various climatic conditions on the thermal environment in an automobile cabinet exposed to direct sunlight. Three-dimensional analyzes of cabinet flow and heat interactions were carried out with the ANSYS Fluent program. The impact of each external climate parameter on the thermal environment inside the cabinet was evaluated and reported. As a result, automobile air conditioning systems are better controlled and optimized with CFD analyzes [1].

Zhang et al. created a heat transfer model in the tractor cabinet using computational fluid dynamics and confirmed the effectiveness of the numerical model with field measurement. The distribution of air flow, temperature and equilibrium temperature of the cabinet under different air-conditioning conditions are discussed in depth. This study provides a theoretical basis for adjusting air conditioning conditions from different aspects such as cooling effect and human thermal comfort [2].

Kayar Ekin Can, performed a windshield defrosting analysis study in the earth-moving machinery cabinet with the CFD approach. An earth-moving machinery cabinet and air conditioning system were used. When defrosting times in analysis and test data were compared, similar defrosting distributions were observed. As a result, this study serves as a model for using the CFD method instead of tests in air conditioning research [3].

2. Materials and Methods

In this section, the theory in the study, important details and test conditions in the process of creating the model that will be input to the CFD analysis are discussed.

2.1. Fluid Dynamics

Fluid mechanics is the branch of physics that studies the behavior of fluids and the forces acting on them. Fluid mechanics generally consists of fluid statics, fluid dynamics and fluid kinematics. Fluid statics deals with the mechanics of fluids at rest. Fluid kinematics is the section that studies the velocities and flow lines of fluids. In fluid dynamics, the relationships between the velocities and accelerations of fluids and the forces acting on fluids in motion are examined. One of the fundamental equations that describe the motion of fluids is the Navier-Stokes equations. These equations; It defines the speed of a fluid depending on pressure and viscosity and is used to analysis the behavior of fluids. A system is a constituency that has a fixed and definable mass and can be separated from its environment by boundaries. The control volume is an arbitrarily selected region in space into which the flow streams [4].

Conservation of Mass

The law of conservation of mass is that the rate of change of fluid mass within the boundaries of a control volume is equal to the rate of net mass flow entering the control volume [5].

$$\frac{\partial}{\partial t} \iiint \rho dV = - \oiint \rho u \cdot dS$$

Above, ρ represents the density of the fluid, u represents the flow velocity vector and t represents time.

Conservation of Momentum

Conservation of momentum is that the external forces acting on the fluid are equivalent to the rate of change in the linear momentum of the object. In this equation, f_{body} represents body forces and F_{surf} represents tensions created by surface forces [6].

$$\frac{\partial}{\partial t} \iiint \rho u dV = - \oiint (\rho u \cdot dS) u - \oiint p \cdot dS + \iiint \rho f_{body} dV + F_{surf}$$

Conservation of Energy

The law of conservation of energy is that the change in energy of a fluid particle over time is equivalent to the energy released by heat, work, and mass per unit time [7].

$$\rho \frac{Dh}{Dt} = \frac{Dp}{Dt} + \nabla \cdot (k \nabla T) + \Phi$$

2.2.   Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a field of fluid dynamics in which numerical calculations and solutions are used to solve complex fluid situations analytically. With the development of computers, CFD has become a rapidly growing tool in recent years. CFD works on the tests performed and the gains obtained. The outputs of CFD analyzes should be thoroughly examined and validated. Solutions are made based on assumptions and approaches. An important benefit of CFD is that time and money can be saved in experiments.

2.3.   Creating the CAD Model

In order to develop a numerical model and make calculations in analysis programs, the internal flow volume of the model is needed. The current model was evaluated taking into account the operator cabinet, air ducts and air conditioning system. SpaceClaim program was used to edit 3D geometric data. After the solid model was simplified in accordance with the analysis, the meshing process was started.
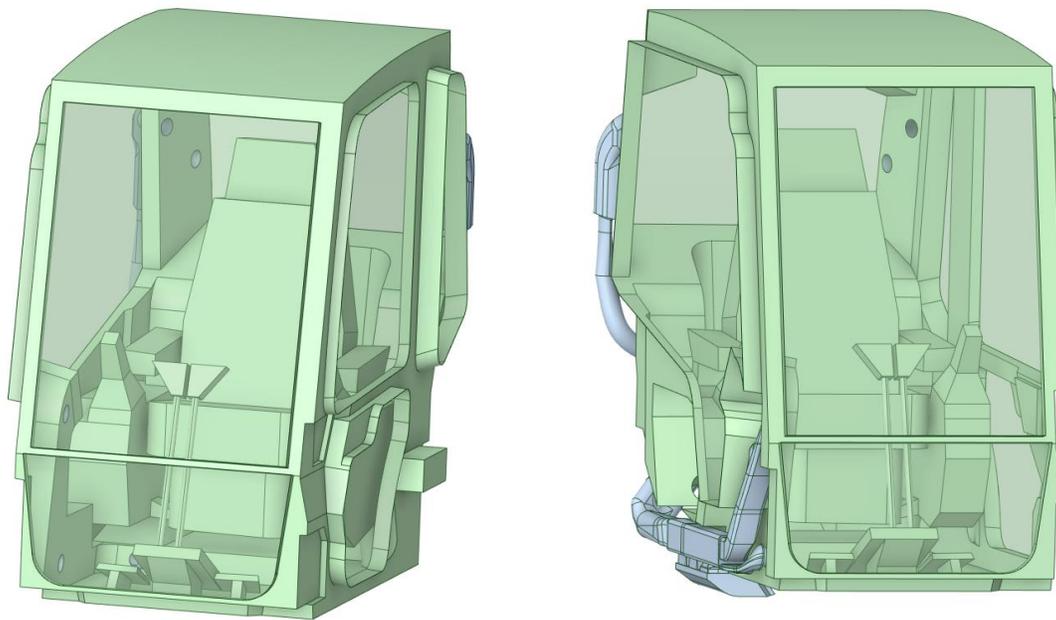
Figure 1: First model CAD data

Figure 1 shows the first model arranged to form the flow volume. It is the model used in the first analysis and verified by testing.
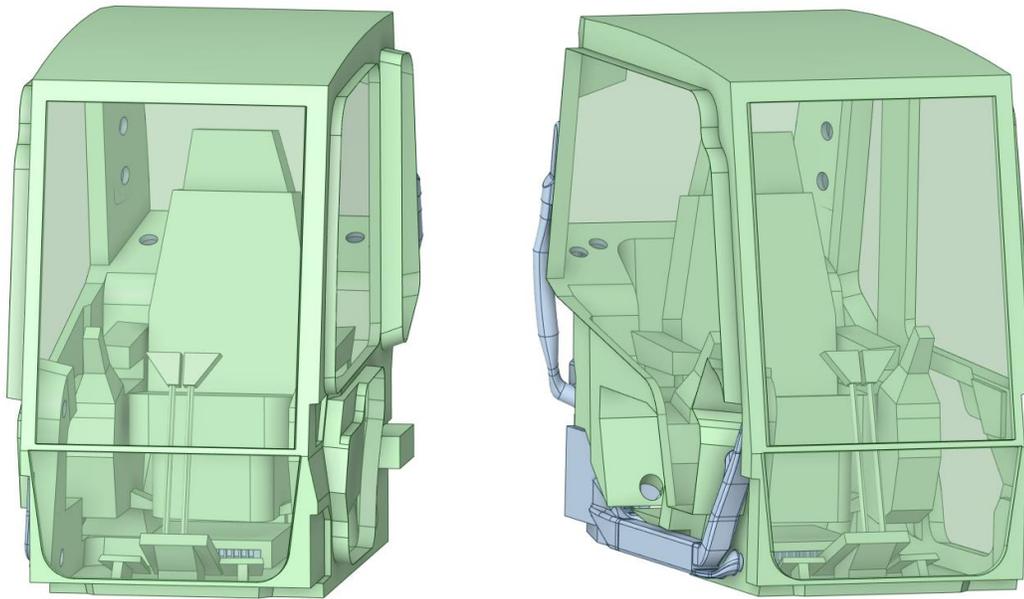
Figure 2: Optimized model CAD data

Figure 2 shows the optimized model arranged to create the flow volume. It is the model that has been optimized and compared with the first analysis.

The numerical mesh was created on the edited geometric data in the ANSYS Fluent Meshing module. Calculations were performed for various mesh sizes to improve calculation times and provide findings independent of mesh size. Comparisons are needed to determine which mesh size is suitable for the solution. In this study, 13 million elements were used to provide accurate findings and at the same time minimize the calculation time.


Figure 3: Mesh view of excavator cabinet

2.4.   Numerical Setup

The numerical models in question were created in the ANSYS Fluent tool. Analyzes were solved as time dependent (Transient). k-epsilon Realizable has been selected as the turbulence model and the energy tab has been activated. The air surfaces in the cabinet are directed towards the operator and the windows and are determined as inlet boundary conditions. Temperature and velocity values of air surfaces were measured during the test and defined as boundary conditions for the analyzes.

Initially, cabinet air conditioning tests were carried out in the climatic test room. CFD analysis results were examined and verified by comparison with test data. A second CFD analysis study was carried out by optimizing the solid model. The aim of this study is to create a healthy CFD method in which air conditioning studies can be carried out. Analysis studies were solved using a supercomputer.
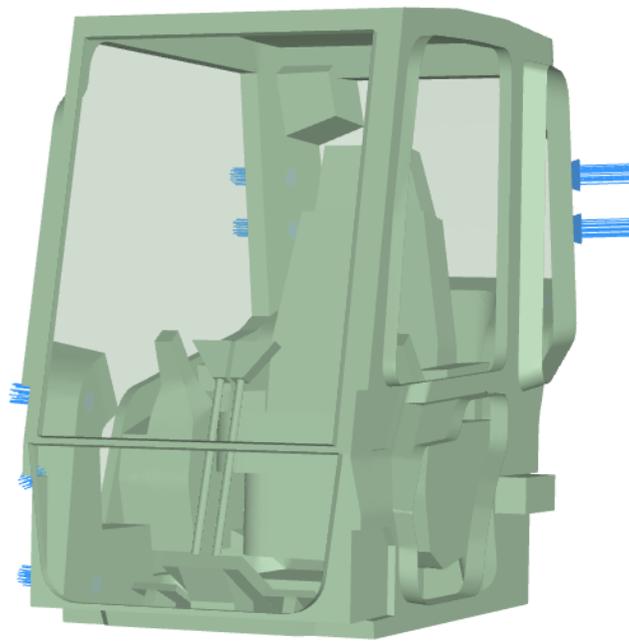
Figure 4: Vents in the excavator cabinet

2.5. Test Conditions

Excavator cabinet test conditions and equipment are described in this section. Excavator cabinet heating test is carried out in accordance with ISO 10263-4 standards. A climatic test chamber was used to evaluate the excavator driver's cabinet. The climatic test chamber was set at -15°C. The data obtained in the conditioning test were used as boundary conditions in analyzes. Air flow meter and thermocouple were used as test equipment.

10263-International Standart

There are international standards regarding the minimum performance levels of air conditioning systems in the earthmoving machinery equipment industry. The relevant standard specifies the method for measuring the contribution of the heating system to the operator's machine ambient temperature and explains the required heating level [5].
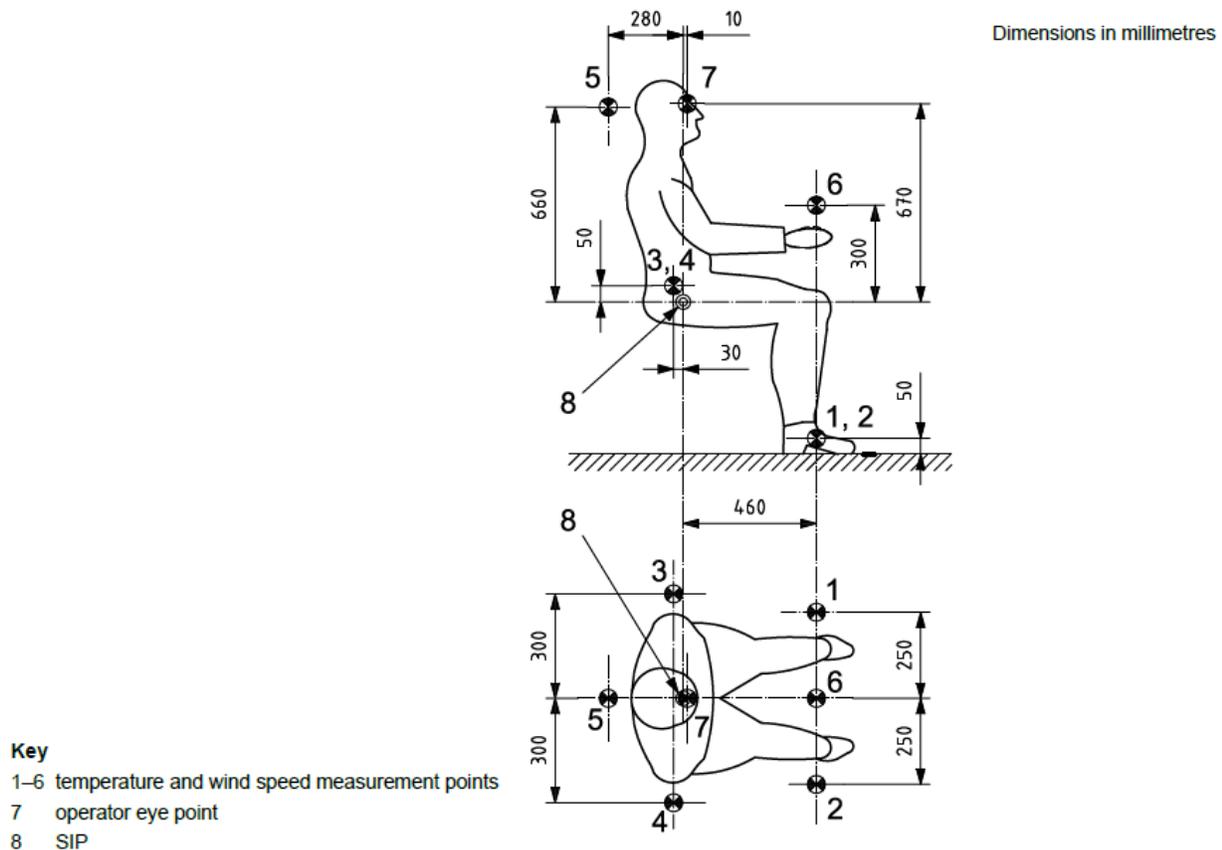

Figure 5: Measurement point locations

Figure 5 shows the locations of the measurement points in the operator cabinet relative to the SIP point. 1: Left foot location, 2: Right foot location, 3: Left arm location, 4: Right arm location, 5: Head location, 6: Steering location.

### 3.	Result

In this section, analysis verification and optimization studies were carried out. The analysis results of the excavator cabinet are given with graphs and figures.

### 3.1.	First Model CFD Analysis

Temperature measurements were made at certain points around the operator, taking the international standard as a reference. Model validation was done by comparison with data collected during testing. Validation studies were carried out by comparing temperature values and the results obtained from analyzes were evaluated.
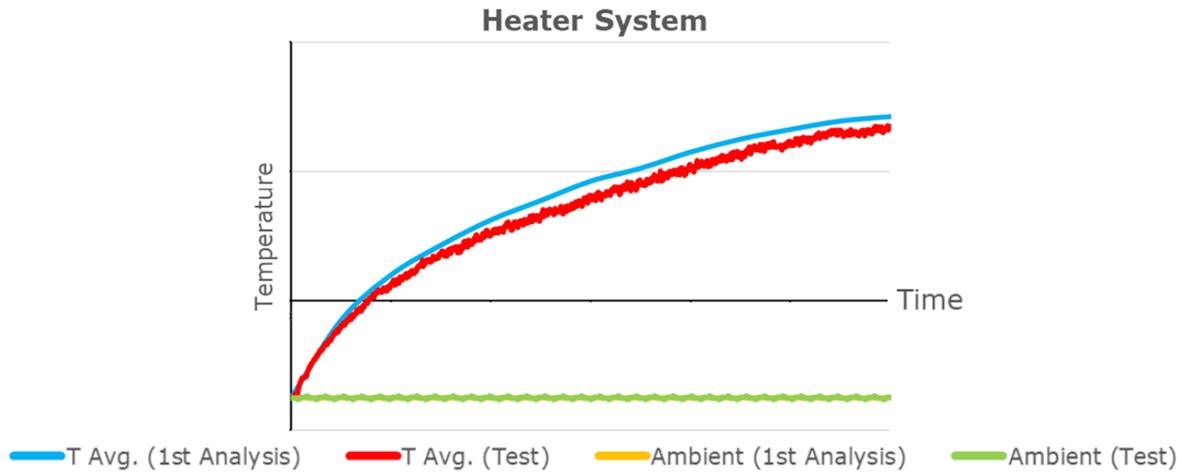

Figure 6: Graph of ambient and cabinet temperature values over time throughout testing and analysis

Figure 6 shows the time-dependent graph of ambient and cabinet temperature values throughout the test and analysis. It is seen that the changes in average ambient temperatures obtained as a result of test and analysis studies over time are similar.
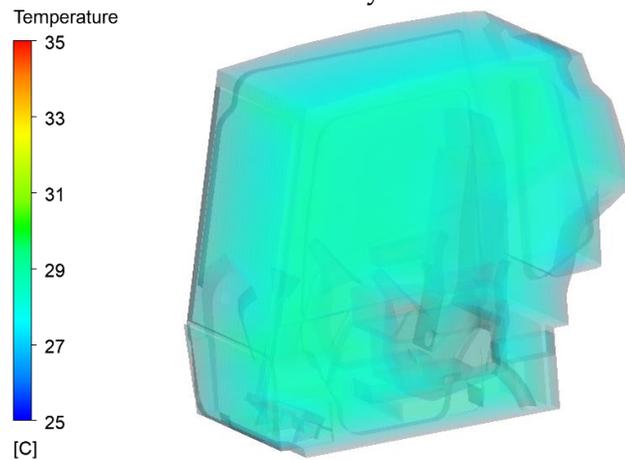

Figure 7: General display of temperature distribution inside the cabinet

As a result of analysis, the temperature distribution in the excavator cabinet is shown.

3.2. Optimized Model CFD Analysis

The second analysis was performed using the optimized model. Temperature values were compared with the first analysis and the results obtained from analyzes were evaluated.
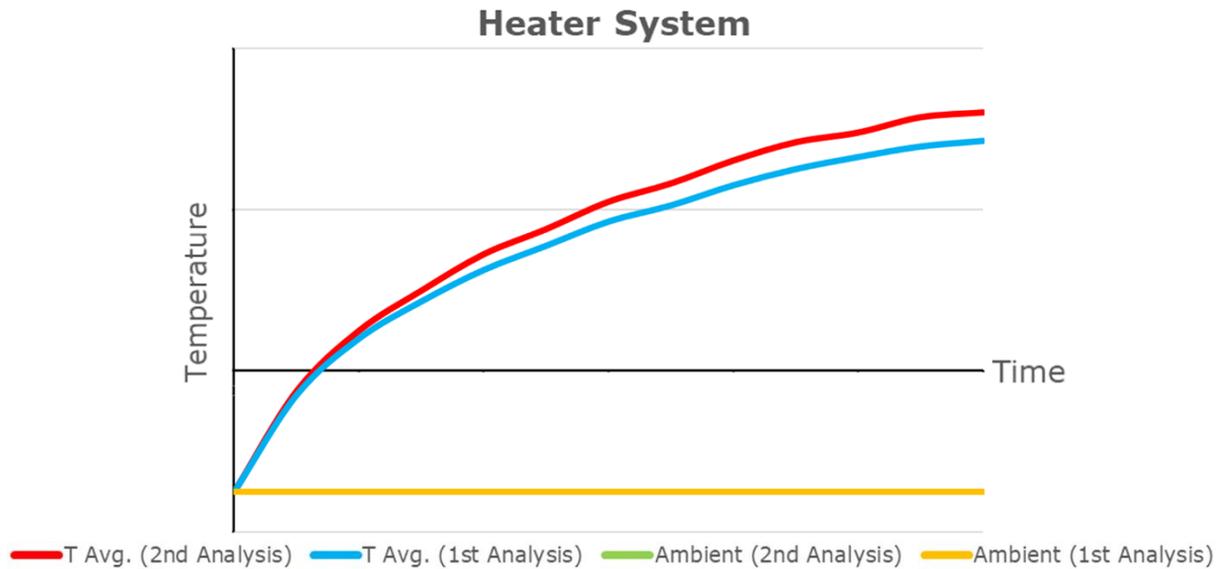
Heater System

Figure 8: Graph of ambient and cabinet temperature values over time during the second and first analysis

Figure 8 shows the time-dependent graph of ambient and cabinet temperature values throughout the optimized model and first analysis. The optimized model showed better heating performance compared to the first analysis.
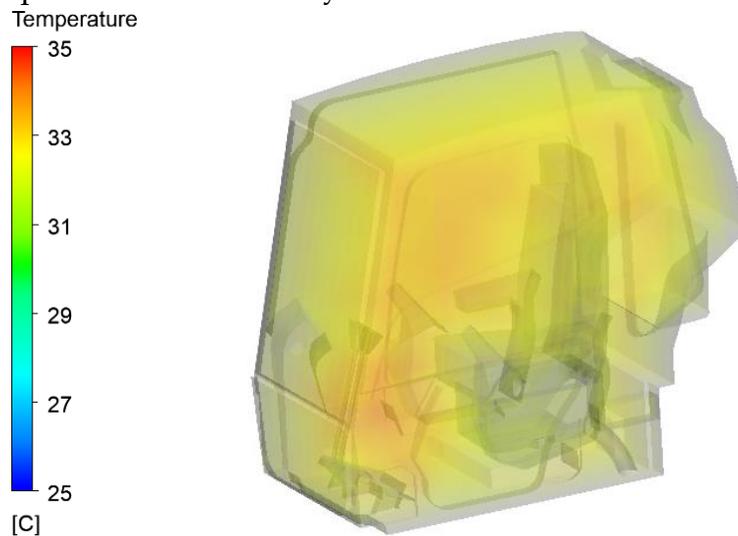
Figure 9: General display of temperature distribution inside the cabinet

As a result of analysis, the temperature distribution in the excavator cabinet is shown.

4.   Discussion and Conclusion

The excavator cabinet has a complex geometry due to the systems inside. Therefore, it is very difficult to develop an accurate numerical study. In this study, computational fluid dynamics analyzes and test studies regarding the heating performance of the excavator cabinet air conditioning system were examined. Excavator cabinet CAD data was used in the analyzes carried out via ANSYS Fluent, and boundary conditions were adapted according to the situation in the test run. CFD analysis and optimization studies were carried out and the results were evaluated.

To develop an accurate CFD model, validation with test data is necessary. In this study, analyzes were made using flow rate and temperature measurements of the vents in the excavator cabinet. CFD analysis results were compared with test outputs and confirmed with an acceptable difference of ± 3°C. Therefore, it was concluded that using the CFD method in cabinet interior heating performance is a good approximate method to mimic tests economically.

The air conditioning system consists of the excavator cabinet and the air conditioning air duct. The analysis was performed by optimizing the air conditioning air duct geometric model. CFD results of the optimized numerical model were compared with the first analysis. It was observed that the optimized model had better heating performance.

The result of this study shows that excavator cabinet air conditioning systems can be analyzed by computational fluid dynamics method. Using the CFD method in future studies has the potential to reduce the timeline and associated costs by significantly reducing the number of prototype parts and tests.

5. Acknowledge

References

[1] Bandi, Prateek, et al. "CFD driven prediction of mean radiant temperature inside an automobile cabin using machine learning." Thermal Science and Engineering Progress 37 (2023): 101619.

[2] Hou, Xiuning, Xuemin Zhang, and Yuancong Gong. "Effect of ventilation on airflow pattern and thermal environment inside tractor cabin in summer: A case study of numerical simulation." Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering (2023): 09544070231185189.

[3] Kayar, Ekin Can. "Analysis of Earth-Moving Machinery Cabin Windshield Defrosting Performance with Computational Fluid Dynamics Method and Verification by Testing." Orclever Proceedings of Research and Development 3.1 (2023): 45-63.

[4] Acheson, David J. Elementary fluid dynamics. Oxford University Press, 1990.

[5] Çengel, Yunus; Cimbala, John; Engin, Tahsin. Akışkanlar Mekaniği Temelleri ve Uygulamaları, 2015.

[6] Çengel, Y., & Cimbala, J.M. Fluid mechanics fundamentals, and applications, 2013.

[7] Doç. Dr. Bülent Yeşilata, Mühendislikte Temel Akışkanlar Mekaniği, 2007.

[8] ISO 10263-4 International Standard, Earth-moving machinery, Operator enclosure environment, Heating, ventilating and air conditioning (HVAC) test method and performance, 2009.