Conference Article

Analysis of Earth-Moving Machinery Cabin Windshield Defrosting Performance with Computational Fluid Dynamics Method and Verification by Testing

Ekin Can KAYAR

HİDROMEK A.Ş., Orcid ID: https://orcid.org/0009-0004-4191-587X
Correspondence: ekin.kayar@hidromek.com.tr

Reference: Kayar, E., C. 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), 45-63.

Abstract

Driver and passenger safety and thermal comfort are seen as the most important parameters in the development of earth-moving machinery cabin air conditioning systems. The formation of a layer of ice and fog on the windshield restricts the driver's field of vision and distracts driving attention. For this reason, the earth-moving machinery air conditioning system has an important place. The ability of the defrosting and defogging system is measured by how quickly and effectively the ice layer formed on the glass is dissolved and to what extent the mist formed inside the glass is cleared. In this study, three-dimensional air conditioning numerical analysis was carried out using the Computational Fluid Dynamics (CFD) method in order to remove ice formed on the windshield of the earth-moving machinery cabin. The earth-moving machinery cabin was used as a mathematical model. Boundary and initial conditions were chosen to be the same as the test data to ensure appropriate validation. By solving the numerical analysis energy equation, time-dependent temperature distribution and defrosting time in the windshield area were analysed. Validation was provided by comparing the results with test data. As a result of the analysis, it was understood that the windshield defrosting time, which is of vital importance for driving safety due to visibility

restrictions, especially in regions with cold climates, can be optimized by analysing it with numerical analyses.

Keywords: CFD, Defrost, Air Conditioning, Heat Transfer

1. Introduction

Air conditioning systems of earth-moving machines play a major role in ensuring the safety and thermal comfort of passengers. In regions where air temperatures drop during the winter months, the layer of ice formed on the windows of vehicles blocks the driver's vision, and this is important for driving safety. The air conditioning unit (HVAC) is responsible for providing warm air into the cabin with the least resistance, and with this application, it removes ice and fog from the windshields, ensuring driving safety in cold weather conditions. The efficiency of the air conditioning system is extremely effective in removing ice from the windows. Computational fluid dynamics (CFD) method is used to develop air conditioning systems in the earth-moving machinery industry [1].

As a result of the developments made in the field of computational fluid dynamics in recent years, this method is actively used in the earth-moving machinery sector. While providing appropriate test conditions and saving time and money spent on prototype costs, the designed improvements can be applied to the models thanks to the variety of simulations that can be carried out with this method. The results obtained from different solution suggestions can be observed quickly and comparatively.

The CFD method can be summarized as the numerical simulation of complex engineering systems that include fluid motion analysis, using mathematical models and numerical solution methods. While in previous studies only analytical or test studies could be carried out, CFD analyses are progressing at an increasing pace with the developing computer and microprocessor technology, and commercial software companies are allocating more resources in this field. Numerical analyses performed with the CFD method are carried out together with test studies and support the test data [2].

Numerical analyses and test results performed with the computational fluid dynamics approach for defrosting were examined by scanning the literature. It has been determined through experiments that quite accurate results can be obtained using the computational fluid dynamics method. Many important scientific studies have been conducted on this subject, as scientists and construction equipment manufacturers have the opportunity to

easily measure the performance of air conditioning (HVAC) elements through better solvents with developing technology.

In some of the studies carried out in the literature, researchers examined the movement of air coming out of the vents and carried out analyses and experiments on the flow affecting the inner surface of the windshield. The main purpose of these researches is to clean the ice and mist on the windshield, to observe the movement of air in the cabin and to determine the temperature distribution on the windshield. In the light of these data, the defrosting phenomenon was defined and the performance criteria of air conditioning elements and designed systems were tried to be determined.

Nasr and AbdoulNour have conducted extensive studies, including experimental and numerical analysis, determining measurement methods for temperature distribution and air flow in the windshield. AbdoulNour performed air velocity measurements at the vent exit and in areas close to the windshield using a hot wire anemometer. It aimed to verify the test results by discussing the numerical simulation of the flow field of the deicing system using computational fluid dynamics [3].

The defrosting model and computational fluid dynamics methodology for the defrosting process were discussed by Farag and Huang, who compared CFD results with thermograph temperature distribution and wind tunnel tests [4]. Park et al. simulated phase changes during the defrosting process and performed numerical analyses of the defrosting phenomenon [5].

Willenborg et al. observed and documented the jet flow emerging from the vents and affecting the inner surface of the windshield by hot wire velocity measurement [6]. Ikeda et al. published both experimental and CFD analysis results of the air coming out of the defroster and affecting the windshield. To compare the experimental and numerical analysis results, the unstructured mesh structure of a commercial code was used in CFD analysis [7].

Sugano et al. documented a numerical method for deicing model prediction using three-dimensional air flow analysis that showed agreement with experimental data [8]. AbdoulNour and Fous performed numerical analysis of defroster and windshield air flow in order to obtain qualified data in terms of the direction and value of the flow field

[9]. Vivek Kumar, Kedar Tare and Sangeet Kapoor worked on air flow optimization on the windshield using a complete vehicle cabin geometry using the CFD method. In this way, they had the opportunity to evaluate the defrosting performance of air conditioning elements with less cost, time and labor [1].

As a theoretical framework, detailed information about computational fluid dynamics and the basics of numerical calculations performed during the analysis are explained. Computational fluid dynamics method is a numerical analysis method based on calculating fluid behaviour via computer with the help of mathematical models. In other words, systems involving fluid motion are analysed through computer-based simulations of heat transfer and related phenomena. The use of CFD analyses in the design cycle directly reduces the steps in the creation of the final design, providing shorter delivery times and improvements in prototype costs. From the first moment the mathematical model is formed, verification of whether there is a thermal or flow problem can be performed with CFD analysis. The suitability of data can be adjusted and optimized in the early stages of design rather than waiting for prototype production. This provides significant savings in terms of time and costs.

Discretization

A computer can only carry and process a limited amount of information, so solutions to partial differential equations must be represented by a limited amount of data. For this reason, the space-time continuum is divided into small cells known as a computational mesh structure. The data may be written as point samples at the node of the mesh, appropriate for finite differences, or it may be the average value of the inner surface of the mesh, appropriate for finite volumes. In this study, the behaviour of the solution within the mesh should have been predicted in detail by interpolation. On the other hand, in the finite element method, the solution weights in a mesh are defined in detail by the sum of independent data basis functions. The process of determining the result with a reduced data set with the help of partial differential equations is called discretization.

Differential Flow Analysis

In computational flow dynamics, flow is described by differential equations. The technique that gives the most accurate results in fluid analysis is experimental studies. However, different methods are used for experimental studies, considering the costs of the prototype, test equipment and time loss. Theoretical approaches come first among these methods. Differential analysis is also performed within the theoretical approach, and this is achieved with differential equations. In cases where these differential

equations cannot be solved analytically, numerical solutions are used. We can call these numerical solutions the computational fluid dynamics method. Numerical calculations were carried out for steady and transient state.

The continuity, momentum and energy equations can generally be written as shown.

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \vec{V}) = S_m$$

S_m in the continuity equation is expressed as the source term. It refers to the mass added to the system.

$$\frac{\partial(\rho u)}{\partial t} + \nabla(\rho u \vec{V}) = -\frac{\partial \rho}{\partial x} + \nabla(\mu \nabla v) + S_{M_x}$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla(\rho v \vec{V}) = -\frac{\partial \rho}{\partial y} + \nabla(\mu \nabla v) + S_{M_y}$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla(\rho w \vec{V}) = -\frac{\partial \rho}{\partial z} + \nabla(\mu \nabla w) + S_{M_z}$$

$$\frac{\partial(\rho e)}{\partial t} + \nabla(\rho e \vec{V}) = -p \nabla \vec{V} + \nabla(k \nabla T) + \Phi + S_e$$

In the equations expressed in the Cartesian coordinate system above; ρ represents the density, $\vec{V}$ represents the velocity vector, u, v and w represent the x, y and z components of the velocity vectors respectively, S_m represents the mass source term, S_M represents the momentum source term, p represents pressure, S_e represents the energy source term, and Φ represents the dissipation term. The term dissipation refers to the loss of energy. The standard k-ε turbulence model was chosen in numerical calculations. This turbulence model is used for such calculations due to the stability of numerical results in the literature [10].

Since differential equations are solved in the CFD method, these equations must be completed with certain boundary conditions and initial conditions.

Turbulence Models

Continuous and non-periodic turbulent movements of velocity components in three directions are called turbulence. Turbulent flow is characterized as a turbulent velocity field. These fluctuations are a mixture of transported quantities such as momentum, energy and concentration types, and due to their small scale and high frequency, they are computationally very difficult to simulate directly in practical engineering calculations. By making changes to the main equations instead of using the real main equations, the computational cost and the inability to solve them are reduced. However, these modified

equations will contain additional unknown variables and turbulence models will need to calculate these variables using known quantities [11].

Turbulence models used by Fluent software;
1. Spalart-Allmaras
2. k-ε models (two equations)
   Standard Renormalization-Group
   Realizable
3. k–ω models (two equations)
   Standard Shear Stress Transport (SST)
4. v²-f Model
5. Reynolds Stress Model (five equations)
6. Large Eddy Simulation Model (LES)

Since each flow has different initial and boundary conditions, it is not possible to obtain accurate results by using the same turbulence model for each flow. For this reason, the correct turbulence model should be selected by examining the initial and boundary conditions of the flow, simulation time, accuracy level, and physical environment of the flow [11].

Solidification and Melting Module

Earth-moving machinery windshield defrosting analyses are performed as a result of heat transfer on the glass and from the glass to the ice layer on the outer surface as the air coming out of the vents hits the windshield. ANSYS Fluent commercial software for CFD analyses uses conduction and convection modes of heat transfer. Fluent enthalpy-porosity technique is used for solidification and melting modulus. In this technique, the "β" variable defines the liquefaction rate in any cell. The liquid percentage "β" is associated with each cell in the ice sheet, and enthalpy balance calculations are performed in each cell to determine the liquid content of the cell. The enthalpy of the material is defined as follows.

$$H = h + \Delta H$$

h sensible enthalpy, ΔH=latent heat=βL, L latent heat of the material, β liquefaction rate.

The liquefaction rate "β" defined for each cell varies between 0 and 1 depending on the fluid content in the cell and is defined as follows [12].

$$\beta = \begin{cases} \dfrac{\Delta H}{L} = 0, & T < T_{solid} \\ \dfrac{\Delta H}{L} = 1, & T > T_{solid} \\ \dfrac{\Delta H}{L} = \dfrac{T - T_{solid}}{T_{liquid} - T_{solid}}, & T_{solid} < T < T_{liquid} \end{cases}$$

10263-5, International Standard

The defrosting activity, which is understood to be extremely important in terms of driving safety in the earth-moving machinery sector, has made it mandatory for earth-moving machinery manufacturers to have standards that include some restrictions on the subject. In the relevant standard, the windshield in front of the operator is divided into regions and there are percentage values that must be defrosted in the regions depending on the earth-moving machinery model.
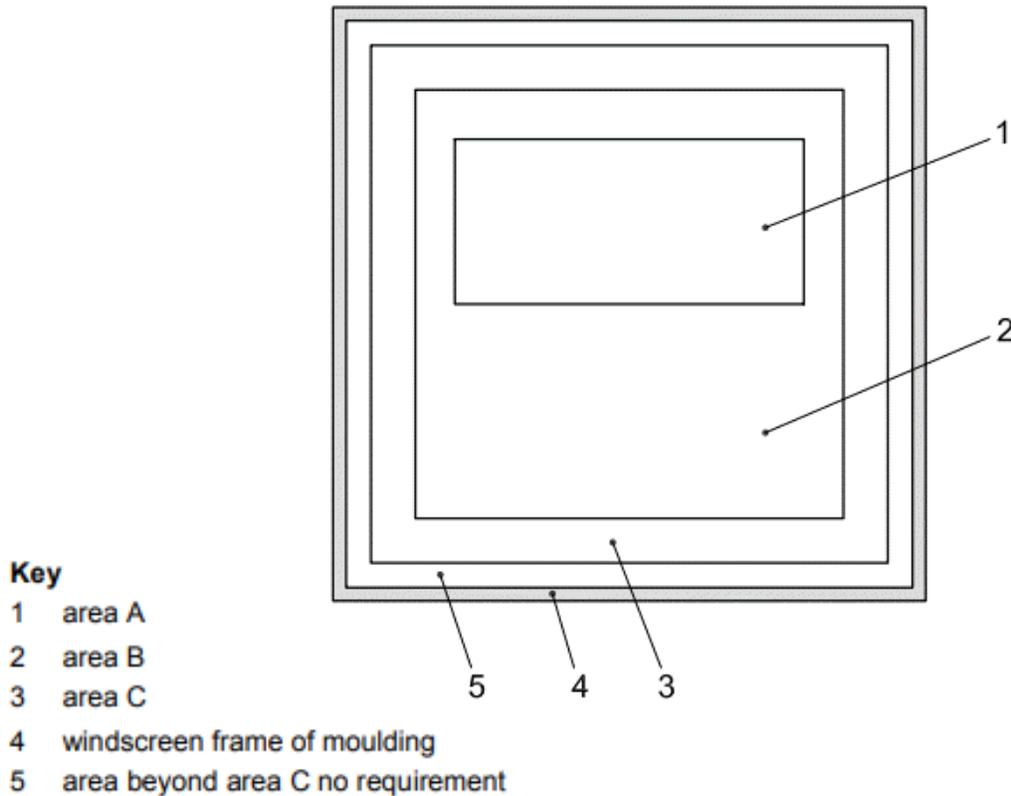

Key

1   area A
2   area B
3   area C
4   windscreen frame of moulding
5   area beyond area C no requirement

Figure 1: Windscreen areas

In this study, it is aimed to determine the accuracy of the analysis by performing windshield defrosting analysis on the earth-moving machinery cabin and comparing the obtained results with test data. In this analysis, the defrosting of the windshield occurred due to the effect of air conditioning. Taking these into consideration, the front part of the

earth-moving machinery cabin was modelled in exact dimensions by adding glass and ice layer. A mesh structure was created for defrosting analysis on this mathematical model. Then, analysis was carried out by defining appropriate boundary conditions and images and information of the ice melting on the windshield were obtained over certain periods of time. The results were compared and verified with the obtained test work.

2.   Materials and Methods

In this section, windshield defrosting analysis performed with the help of computational fluid dynamics is explained. The analysis results of the earth-moving machinery cabin windshield defrosting system developed by HİDROMEK have been shared. In the last part, the results were confirmed with the test data carried out by HİDROMEK.

2.1.   Creating Geometry

HİDROMEK earth-moving machinery mathematical model was used in the test data for defrosting analysis. Air is defined for the cabin area, glass is defined at the front of the cabin, and ice is defined in front of the glass. The cabin, glass and ice sheets are combined into a single model. The reason for this is to ensure that the node points intersect for the cabin, glass and ice while creating the mesh structure. While creating the mesh structure, simplifications were made in the model in order to limit the number of elements.
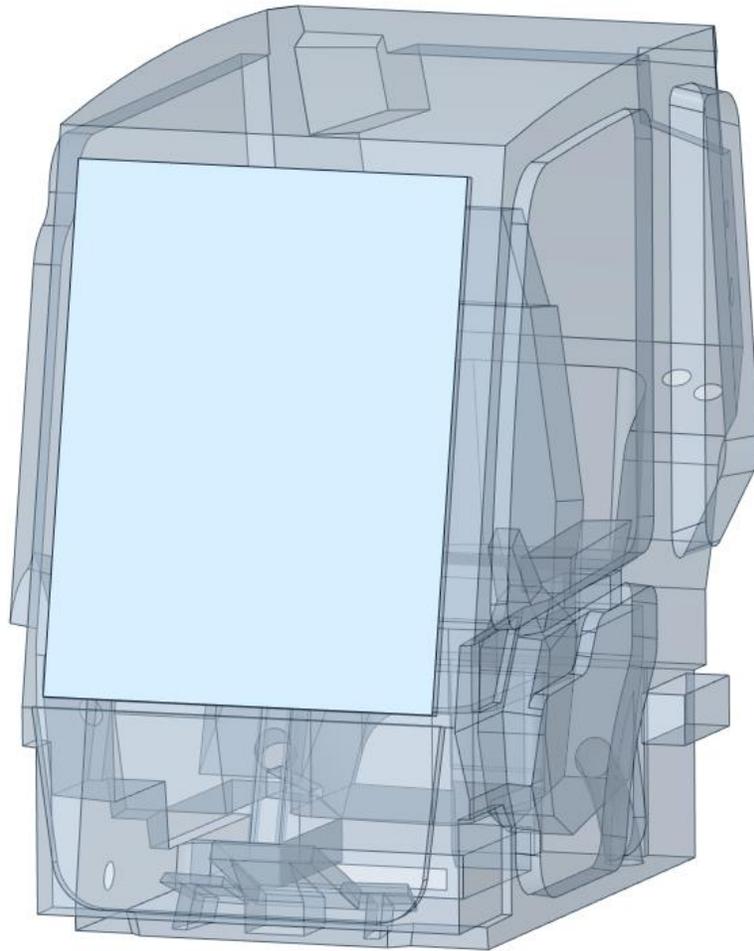
Figure 2: Cabin geometry

2.2. Creating the Mesh Structure

The next stage of computational fluid dynamics analysis is to create the mesh structure for the model. It can also be said as meshing.

Figure 3: Cabin mesh structure

Polyhedral elements were used in numerical studies. The geometry has become a single piece and the elements in the mesh structure have been matched. The model was then opened in the Mesh section and the mesh structure was created. The mesh structure consists of 2000000 elements.

2.3. Analysis Process

For computational fluid dynamics analyses, the mesh version of the model was opened in ANSYS Fluent and the necessary procedures for the analysis were initiated. The

analysis was solved as time dependent (Transient). Standard k-epsilon with two equations was chosen as the turbulence model. The Energy tab has been activated and the Solidification and Melting module used for defrosting analysis has been activated.

Material properties for air, glass and ice are defined in Fluent as given in the tables below.

Table 1: Material Properties of Air

AIR	
Density [kg/m^3]	1.225
Cp [J/kg-K]	1006.43
Thermal conductivity [W/m-K]	0.0242
Viscosity [kg/m-s]	1.7894e-05
Pure solvent melting heat [J/kg]	0
Solidification temperature [K]	0
Melting temperature [K]	0

Table 2: Material Properties of Ice

ICE	
Density [kg/m^3]	920
Cp [J/kg-K]	2040
Thermal conductivity [W/m-K]	1.88
Viscosity [kg/m-s]	0.00553
Pure solvent melting heat [J/kg]	334960
Solidification temperature [K]	271
Melting temperature [K]	273

Table 3: Material Properties of Glass

GLASS	
Density [kg/m^3]	2400
Cp [J/kg-K]	750
Thermal conductivity [W/m-K]	0.93

After the materials were defined and the material type of the relevant regions was introduced to the program, boundary conditions were entered. The main purpose here is for the hot air coming to the windshield area to hit the glass and provide heat transfer to the glass, and to transmit the heat to the ice on the outside of the glass and cause the ice

to liquefy. For boundary conditions, inlet vents are defined as "Velocity inlet" and outlet vents are defined as "Pressure outlet". In the Run Calculation section, the analysis was started by entering the number of time steps and the duration of these steps in seconds. The results and evaluations of the analyses will be shared in the following sections.

2.4. Test Process

A defrosting test was used to verify the work done on ANSYS Fluent. Test data were taken from HİDROMEK company. The defrosting test applied to the HİDROMEK earth-moving machinery is the verification test data of this study. The test lasted 60 minutes and the cabin temperature was set at -15°C throughout the test.

3. Result

In this section, the results of the computational fluid dynamics analysis performed for defrosting via ANSYS Fluent and the windshield defrosting test performed on the HİDROMEK earth-moving machinery are shared.

3.1. Defrosting Analysis Results

In the analysis solution, the solidification and melting modulus and the amount of ice melting depending on time were determined. The amount of defrosting on the windshield was displayed in the time-dependent energy solution of the analysis. The scale part takes values between 0 and 1. The color change from 0 to 1 goes from blue to red, with 0 indicating completely melted ice and 1 indicating completely solid ice. Starting from Figure 5, the display of the results depending on the time in the defrosting process is shown.
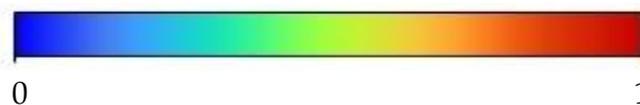

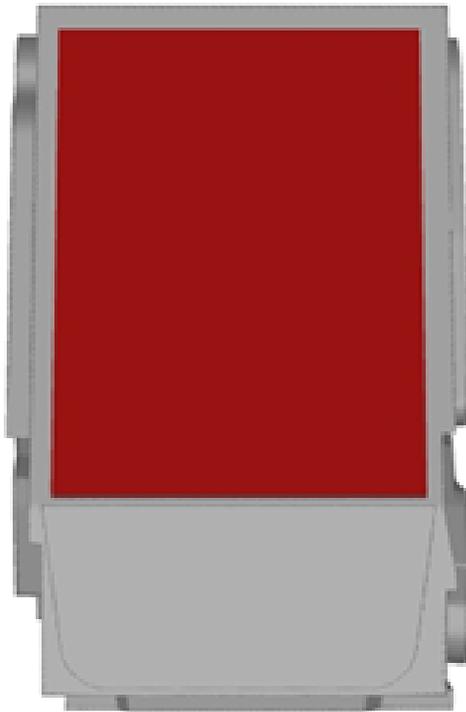0   1

Figure 4: Ice thickness scale
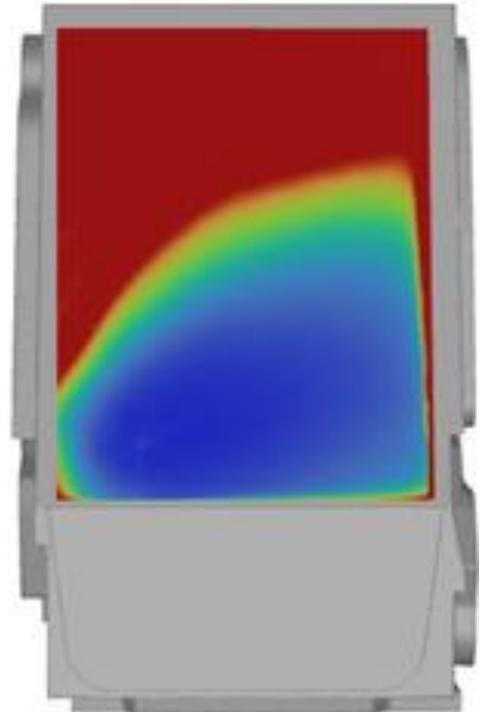
Figure 5: Analysis t=0. minute defrosting rate
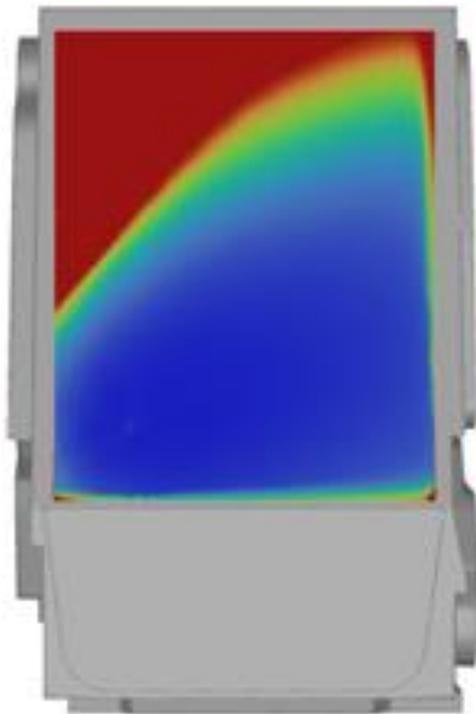

Figure 6: Analysis t=15. minute defrosting rate

Figure 7: Analysis t=30. minute defrosting rate

Figure 8: Analysis t=45. minute defrosting rate

Figure 9: Analysis t=60. minute defrosting rate

3.2. Viewing Test Results

Considering the results of the test study carried out in the climatic cabin; melting started in the regions where the speed of the air hitting the windshield is high, and after 40 minutes, it is observed that the ice is dissolved in approximately all regions. Figure 10 shows the defrosting images on the windshield of the HİDROMEK earth-moving machinery inside the climatic cabin.

Figure 10: Test t=0. minute defrosting rate

Figure 11: Test t=15. minute defrosting rate

Figure 12: Test t=30. minute defrosting rate

Figure 13: Test t=45. minute defrosting rate

Figure 14: Test t=60. minute defrosting rate

3.3.   Comparison of Results

In Table 4, the numerical analysis and test data are compared with the time-dependent defrosting images on the windshield. The dissolution behaviour of ice over time appears to be approximately similar. Minor differences between analysis and test, simplifications on the model due to solution constraints, variations in the mesh structure, natural temperature changes, etc. It was formed due to reasons.

Table 4: Comparison of Analysis and Test Results

Minute	Analysis	Test
t=0		
t=15		
t=30		
t=45		
t=60		

4. Discussion and Conclusion

In this study, computational fluid dynamics analysis and test work were examined. In the analysis performed via ANSYS Fluent, earth-moving machinery cabin CAD data was used and boundary conditions were adapted according to the situation in the test study. For the test study, HİDROMEK earth-moving machinery windshield defrosting test was carried out and the results were evaluated.

Defrosting times in the analysis and test data were compared and similar defrosting distributions were observed. Airflow over the windshield can be corrected and optimized by design iterations of the defroster nozzle using CFD simulation tools. Various designs can be evaluated using numerical methods in a short period of time, providing a lot of design flexibility.

The CFD results and temperature model for air flow are consistent with the test results, validating the use of numerical analyses. The study can serve as a model for using CFD simulations in air conditioning analysis instead of tests.

The defrosting performance was successfully predicted using CFD tools. Visualizing liquefaction rate CFD results helps identify areas to improve defrosting performance in the absence of prototype parts and costly weathering tests.

CFD simulation tools used for defrosting performance have the potential to significantly reduce the number of prototype parts and tests, reducing the timeline and associated costs. This benefits the overall product design and development process.

The defrosting analysis was validated with test data. In the light of this data, it has shown us that optimization of deicing speed can be achieved through virtual analysis in order to ensure driving safety, especially in regions where climatic conditions are difficult during winter months.

5.   Acknowledge

References

[1] Kumar V., Tare K., Sangeet K., 2010. Deployment of CFD for Optimization of the Air Flow Distribution Over the Windscreen and Prediction of Defrost Performance, SAE International 2010-01 – 1059.

[2] Sevilgen Gökhan, Kılıç Muhsin, 2012, 'Otomobillerde Camlar Üzerinde Oluşan Buğunun Hızlı Olarak Giderilmesinde Menfez Geometrisi, Hava Hız, Nem ve Sıcaklığın Etkisinin HAD Yöntemi İle Analizi', Uludağ Üniversitesi, BURSA.

[3] Nasr, K. J., and AbdoulNour, B. S., 2000. "Defrosting of Automotive Windshield: Progress and Challenges", Int. J. Of Vehicle Design, Vol 23, pp. 360-375, 2000.

[4] Farag, A. and Huang, L.-J. 2003. "CFD Analysis and Validation of Automotive Windshield De-Icing Simulation," SAE Technical Paper 2003-01-1079, 2003.

[5] Park, W.G., Park, M.S. and Jung, Y.R. and Jang, K.L. (2005), "Numerical Study of Defrosting Phenomena of Automotive Windshield Glass", Numerical Heat Transfer, Part A: Applications, 47:7, 725-739.

[6] K. Willenborg, J.F. Foss, R.S. AbdulNour, J.J. McGrath, B.S. AbdoulNour, 1997. A model defroster flow, in: Proceeding of the Eleventh Symposium on Turbulent Shear Flows, vol. 2, pp. 15.25–15.30.

[7] Y. Ikeda, N. Katoh, N. Ishii, T. Kuriyama, 1997. Numerical analysis of the airflow on windows from defroster nozzles, JSAE Paper 924076 (in Japanese).

[8] M. Sugano, T. Yamada, Y. Takesue, T. Yasuki, 1994. Numerical analysis of defroster cleaning pattern, JSAE Paper 9432912 (in Japanese).

[9] B.S. AbdoulNour, J.F. Foss, 1997, Computational and experimental predictions of automotive windshield flow, in: Proceedings of ASME Fluids Engineering Division Summer Meeting, ASME Paper No. FEDSM97-3022.

[10]   Kılıç, M., Sevilgen, G. 2012. The effects of using different type of inlet vents on the thermal characteristics of the automobile cabin and the human body during cooling period. The International Journal of Advanced Manufacturing Technology, 60(5-8), 799-809.

[11]   Osman T., Özcan A. (2004) Uçak Kanatlarının CFD Analizi, İstanbul Teknik Üniversitesi, Fen Bilimleri Enstitüsü, İstanbul.

[12]   ANSYS Fluent Theory Guide, ANSYS, Inc. USA, 2016.

[13]   Roy Subrata, Nasr Karim, Patel Paresh, (2002) 'An experimental and numerical study of heat transfer off an inclined surface subject to an impinging airflow' International Journal of Heat and Mass Transfer 45 1615–1629.

[14]   ISO International Standard, Earth moving machinery, Operator enclosure environment, Windscreen defrosting system test method, 2009.