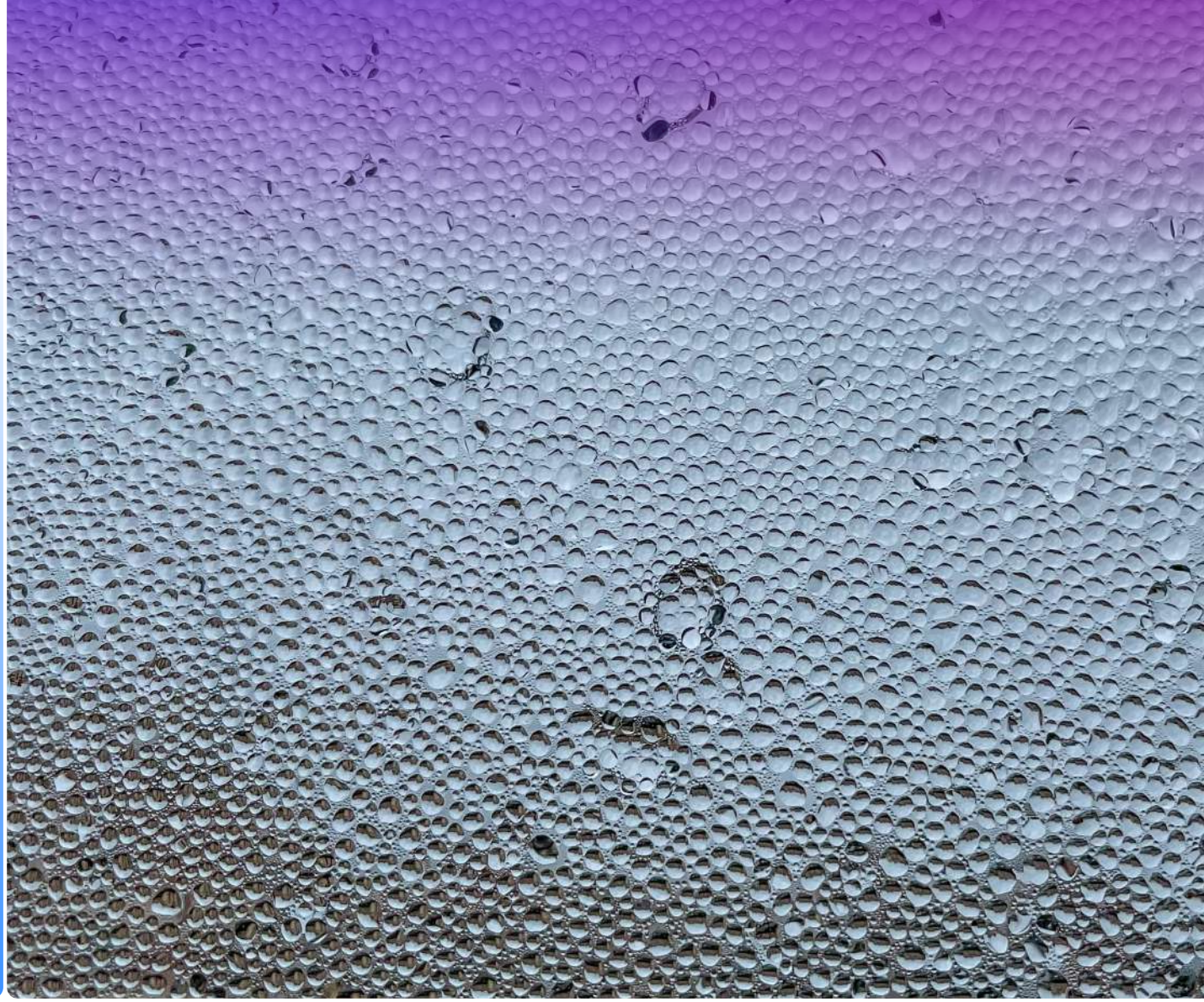
HCLTech | Supercharging Progress™

CFD-based prediction of wall condensation of moist air in an electronic enclosure using Eulerian Wall Film model



Introduction

Condensation is a phenomenon in which a fluid in the vapor phase turns into a liquid. It typically occurs when gasses or vapors are exposed to higher pressures or lower temperatures. This process occurs when a pure or multicomponent, saturated or superheated vapor encounters a surface or object with a temperature below its saturation temperature [1].

Humid or moist air contains water vapor, which condenses under certain conditions, such as encountering relatively cooler surfaces. This leads to the condensation of excess moisture or water vapor forming liquid water as the cooler air holds less moisture than warmer air. Water vapor condenses into liquid water when the air reaches a temperature where it can no longer hold moisture. This point is known as the dew point, which depends on the pressure and humidity of the air. Hence, water vapor condenses into liquid water when the temperature reduces below the dew point. For instance, when the surface of a bottle containing cold water [2] encounters humid air at a relatively cooler temperature than its dew point, the air close to the bottle cannot hold moisture, condensing the water vapor.

The process of humid air condensing on a cooler surface has a lot of implications in our daily lives. Formation of dew, fog, mist and other natural phenomena occurs due to condensation of water vapor. This causes dampness and blocks vision on window panes in a house, the windscreen of a car and other areas. Furthermore, it causes erosion of engineering equipment as the condensate formed on the walls is often cause for concern. It affects the intended performance of engineering systems, making it imperative to consider it during the design phase. Quantification of wall condensation helps designers to implement techniques to mitigate this risk. Several Computational Fluid Dynamics (CFD)-based techniques are available that can predict the condensate formation at the walls resulting from lower temperatures.

This whitepaper describes a CFD simulation that predicts wall film formation inside an electronic control box. The simulation is performed using the Eulerian Wall Film (EWF) model, which is available in the commercial CFD code ANSYS FLUENT.

Literature review

Marcus Lejon [3] investigated condensation models available in ANSYS CFX and ANSYS FLUENT, focusing primarily on ANSYS CFX. The study evaluated air and water vapor mixture condensation over a flat plate by leveraging various models in ANSYS CFX. Apart from this Morales-Fuentes et al. [4] investigated condensation on the surface of a heat exchanger implemented by utilizing a User Defined Function (UDF) in ANSYS FLUENT to account for wall condensation alongside using the available wall condensation model in ANSYS CFX. Barak et al.

[5] presented a prediction of moist air condensation on the walls of a heat exchanger, modeling the geometry of a square canal to address the condensation problem without using any UDFs. The wall condensation model in ANSYS CFX was the most suitable model.

Poredos et al. [6] introduced a semi-empirical correlation for predicting condensation from humid air inside vertical channels formed by flat plates predicting within 12% accuracy to test data. Additionally, Sun et al. [7] developed a new phase change model utilizing a UDF in ANSYS FLUENT to predict the evaporation and condensation process, showing good agreement between the analytical and numerical results. Liu et al. [8] devised a method employing the 3D transient CFD model, validated against experimental data, and implementing the CFD tool STREAM4.0 to solve the condensation problem, demonstrating good agreement between the experimental and predicted results. Narkhede et al. [9] presented a methodology for simulating humid air condensation over a vertical flat plate using the EWF multiphase model in ANSYS FLUENT software validated against experimental data, thus simulating condensation in the presence of air, which is a non-condensable gas, without the use of any UDFs.

In most simulation activities predicting condensation using ANSYS FLUENT, UDFs have been applied to capture the phenomenon accurately. The EWF model is an effective method to predict wall condensation of moist air without the need for UDFs. This article presents a simulation task performed using the EWF model in ANSYS FLUENT.

EWF model

The description and equations governing liquid film formation predicted by the EWF model are described in detail in the ANSYS FLUENT Theory Guide [11]. This has been summarized as below:

The EWF model predicts the creation of thin liquid films on the surface of walls. This model can be coupled with the species transport model to predict phase change between gas (vapor) and film material. The phase change rate is modeled using the diffusion balance model.

The rate of phase change is governed by:

$$\dot{m}_{phase} = \frac{(\rho D/\delta)}{\rho D/\delta + C_{phase}} C_{phase} (Y_{sat} - Y_i) \qquad \left[Kg/m^2\text{-}s\right]$$

ρ = density of the gas mixture

D = mass diffusivity of the vapor species

δ = cell-center-to-wall distance

C_{phase}= phase change constant

y_i = cell-center mass fraction of the vapor species

The saturation species mass fraction, y_{sat} , is computed as

$$Y_{sat} = \frac{p_{sat}(T)}{p} \frac{M_i}{\overline{M}}$$

P = absolute pressure of the gas mixture

M_i = Molecular weight of vapor species

= Molecular weight of liquid species

The saturation pressure Psat is a function of temperature (T), given by the below equation,

$$\log_{10} p_{sat} = -2.1794 + 0.02953(T-273.17) - 9.1837e^{-5}(T-273.17)^2 + 1.4454e^{-7}(T-273.17)^3$$

Condensation occurs when the vapor mass fraction exceeds the saturation mass fraction and condensation occurs. In contrast, the liquid film vaporizes when the vapor mass fraction is less than the saturation mass fraction.

There are a few limitations to this model. This model is available only in a FLUENT 3D solver. Multiphase models such as Volume of Fluid (VOF) may not interact effectively with the EWF model.

Methodology

This simulation is performed on an electronic control box. The control box functions as a closed enclosure containing a PCB and motor driver, which dissipates a defined amount of heat and is cooled naturally from outside the enclosure.

A computational grid of polyhedral cells and prism layers in the boundary layer has been generated. The geometry and mesh are shown in Figure 1. The display panel and switches have been removed from the geometry for simplicity.

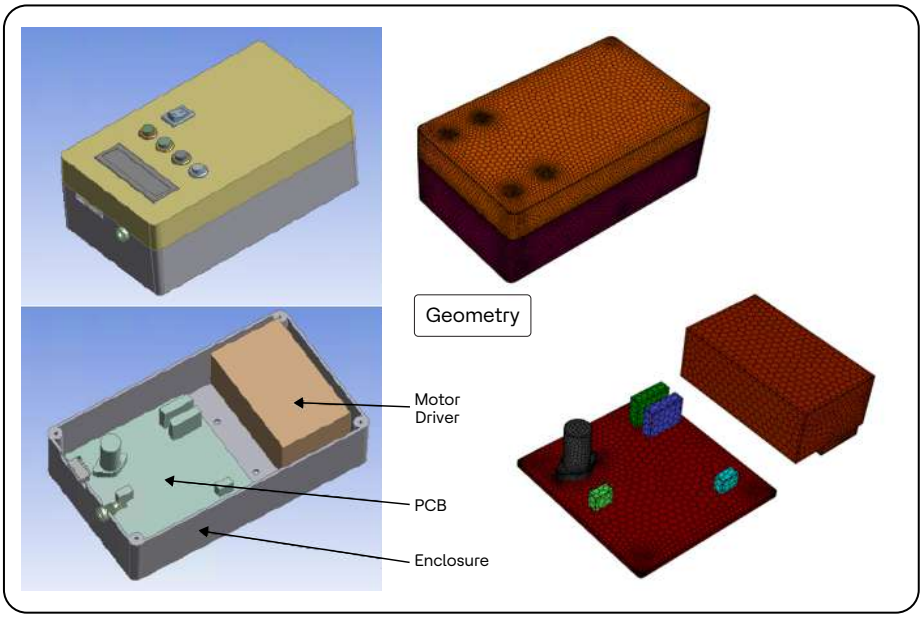


Figure 1- Geometry considered along with computational grid

During operation, the temperature inside the enclosure is well above the ambient temperature, ensuring no condensation occurs near the walls. Once the unit is switched off, the air inside the enclosure cools because of the ambient temperature and condensation film forms near the walls.

To simulate this phenomenon, a steady-state simulation with power dissipation was performed to forecast components and air temperatures within the enclosure, as depicted in Figure 2. The PCB and motor driver are assumed to be copper and aluminum enclosure for simulation. A convective heat transfer coefficient of 10 W/m^2K, along with a free stream temperature of 5^0C, was applied as a boundary condition on the enclosure's outer wall surface. The fluid is assumed to be a mixture of 2 species - dry air and water vapor.

This humid air is modeled through species transport equations in ANSYS FLUENT, with a water vapor mass-fraction of 0.01, implying that air mass-fraction is 0.99 and a relative humidity of around 50%. From the psychrometric chart, it can be noted that the dew point temperature of the air at RH is 50% and the temperature at 25^0C is approximately 14^0C [10]. Hence, water vapor in the air is expected to condense when the pipe wall temperature reduces to below 14^0C.

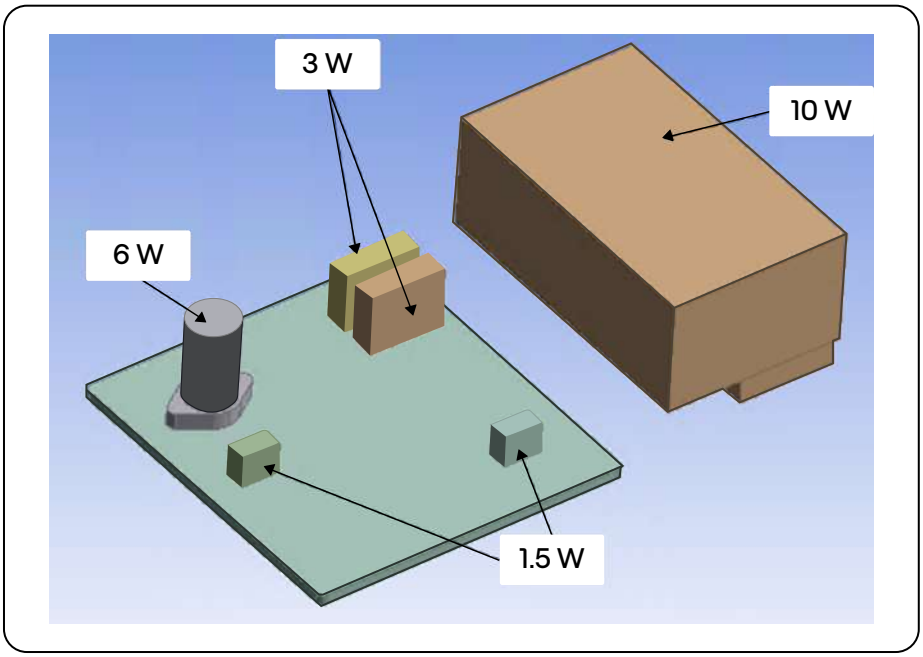


Figure 2- Power distribution inside an enclosure

The steady-state result served as the initial condition for a subsequent transient simulation initiated after switching off the heat dissipation values of components. As the ambient temperature outside the enclosure is cooler, the temperatures inside decrease below the dew point temperature. The EWF model was switched on and activated on the inner walls of the enclosure to enable the formation of a liquid water film attached to the wall. The maximum wall thickness was set to one mm. Since the EWF model can predict the amount of condensate formed, the multiphase was switched off.

The transient simulation was performed with a steady-state temperature distribution as an initial condition for one hour (3,600 seconds) with a timestep of 0.1 seconds.

Results

As mentioned in the previous section, a steady state simulation was performed initially with power dissipation values to determine the temperature distribution inside the enclosure. The fluid and component temperatures are shown in Figure 3.

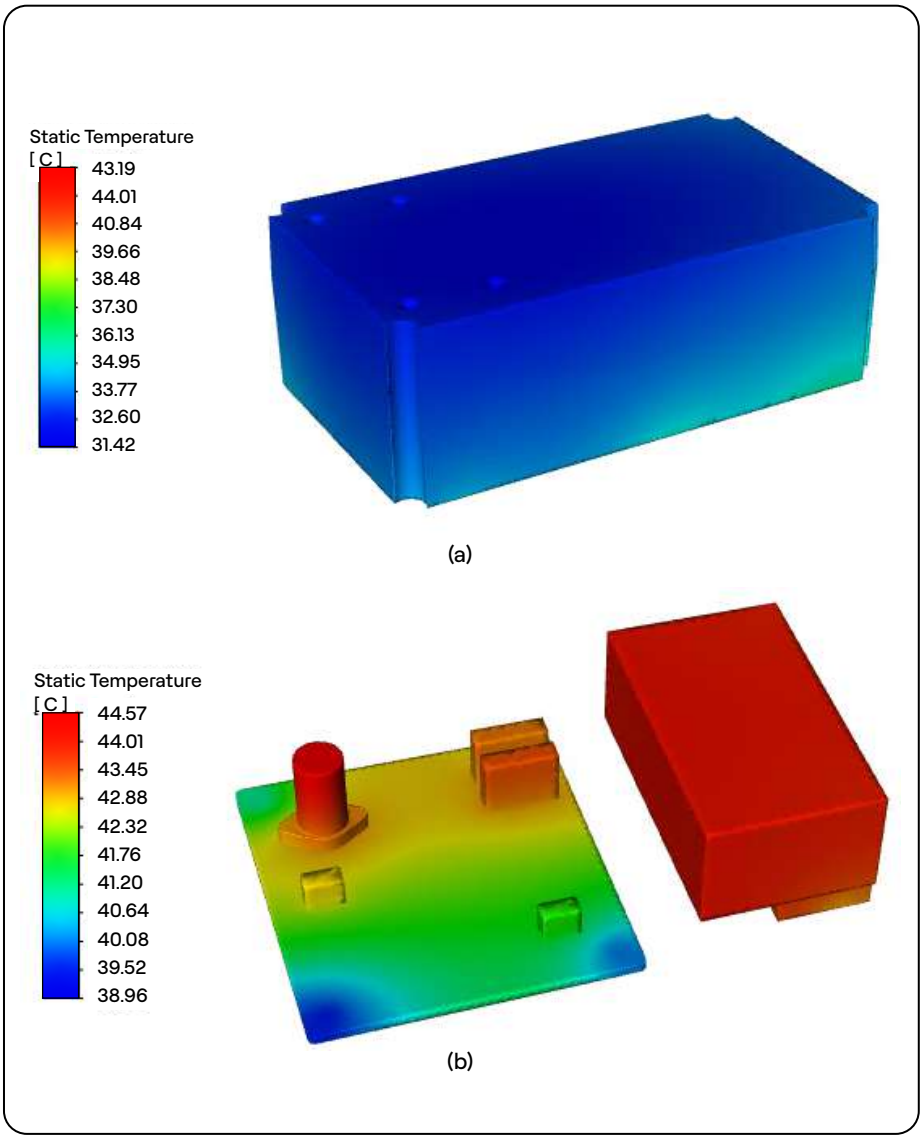


Figure 3- Steady state temperature distribution of (a) fluid volume inside enclosure and (b) heat dissipating components

As demonstrated in Figure 3, the minimum temperature of the fluid is 31^{0}C, which is well above the dew point of humid air. This implies no water film formation on the wall if the components dissipate heat.

A transient simulation was performed for 3,600s with the EWF model activated and power dissipations switched off. The growth of liquid water film with respect to time was captured. The result was condensation started immediately when the near-wall temperature reached a dew point temperature of around 14^{0}C. Figure 4 shows a plot representing the variation in wall film thickness. This value represents the maximum film thickness on the wall. Moreover, the corresponding minimum temperature value on the enclosure's inner wall has been plotted, indicating that when the minimum temperature reaches around 14^{0}C, the water film starts forming.

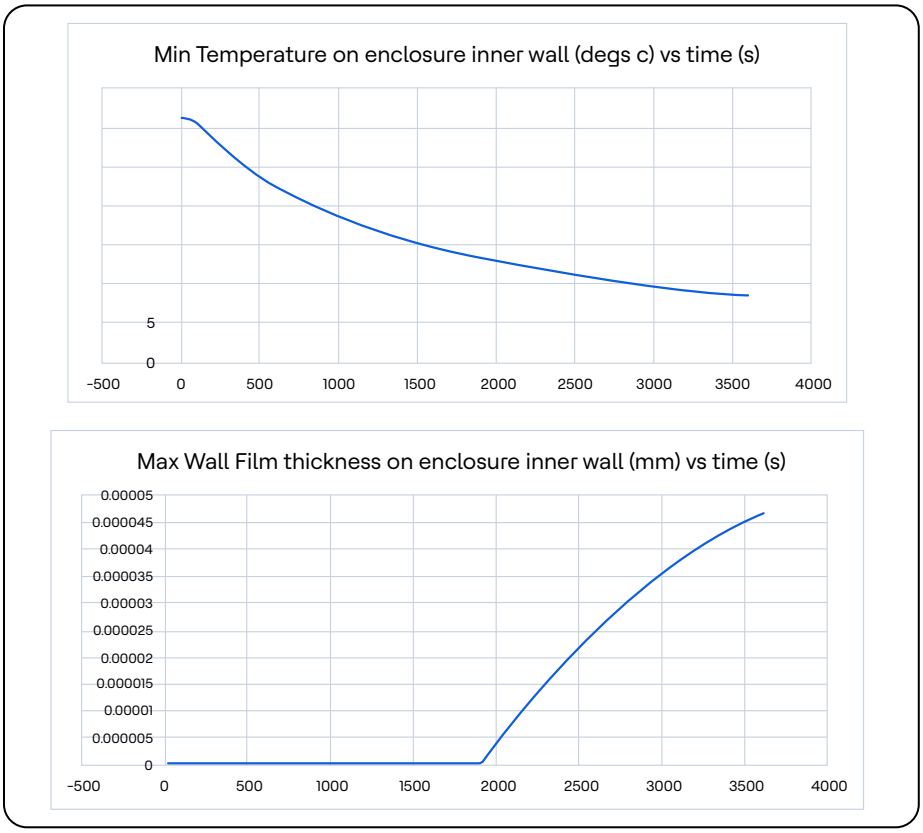


Figure 4- Plot representing (a) minimum temperature on enclosure inner wall and (b) liquid film thickness growth with respect to flow time

Figure 5 shows the variation of film thickness on the inner wall of the enclosure at the end of a one-hour duration. Film thickness is at its maximum at the location above the PCB, where temperature is at a minimum.

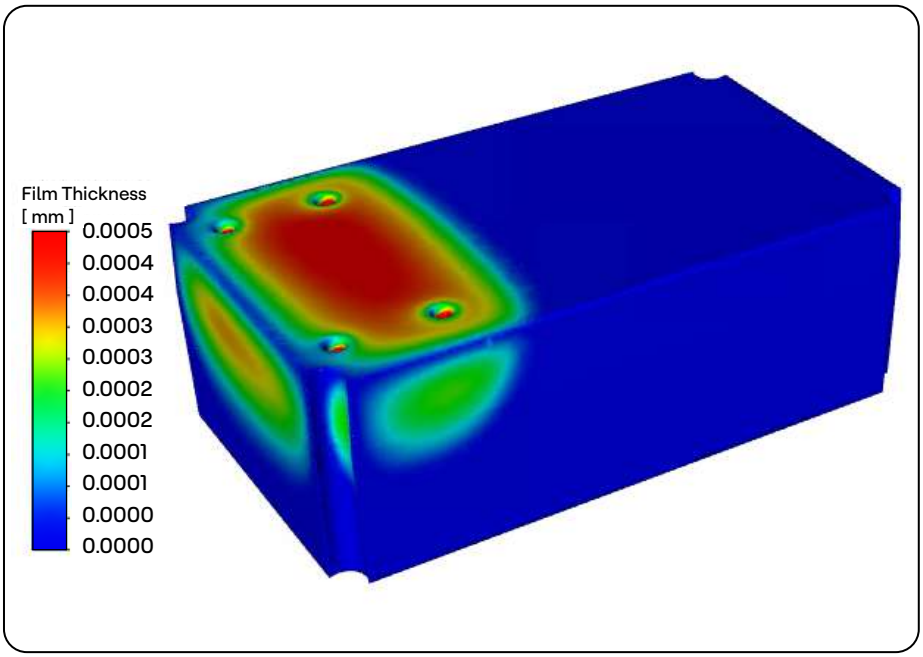


Figure 5 – Wall film thickness contour plot at the end of 1 hour

This result provided an idea of the location where condensate formation is possible in the given unit. The quantity of condensate would depend on accurate values of heat dissipation and temperatures achieved inside the unit. The overall conclusion is that the EWF model can predict thin films at walls as and when the temperature reduces to below the dew point.

Conclusion

A CFD model has been built to predict thin wall film formation on the inner wall of an electronic enclosure due to humid air condensation. It has been concluded that the existing EWF model in ANSYS FLUENT can predict the condensate formation on the walls without UDF. A comparison with experimental data will help to validate the model as a further step. This model can help designers identify problem areas while designing equipment — like heat exchangers, automobile headlamps, the windscreens of cars and others — where wall condensation is a serious concern.

References

[1] Amir Faghri, Yuwen Zhang, 8 - CONDENSATION, Transport Phenomena in Multiphase Systems, Academic Press, 2006, Pages 581-677, ISBN 9780123706102, https://doi.org/10.1016/B978-0-12-370610-2.50013-1

[2] https://www.mechstudies.com/what-condensation-definition-meaning-water-cycle-examples-reactions/?expand_article=1

[3] Lejon, Marcus. "Wall Condensation Modelling in Convective Flow." (2013)

[4] Morales-Fuentes, A., et al. "Wet-dry thermohydraulic performance comparison on fin-and-tube geometries." Applied Thermal Engineering 214 (2022): 118881

[5] Barák, Jan, Karel Fraňa and Jörg Stiller. "Condensation of Moist Air in Heat Exchanger Using CFD." International Journal of Mechanical and Mechatronics Engineering 8.1 (2014): 61-66

[6] Poredoš, Primož, et al. "Condensation of water vapor from humid air inside vertical channels formed by flat plates." Iscience 25.1 (2022)

[7] Sun, Dongliang, Jinliang Xu and Qicheng Chen. "Modeling of the evaporation and condensation phase-change problems with FLUENT." Numerical Heat Transfer, Part B: Fundamentals 66.4 (2014) 326-342)

[8] Liu, Jing, Hiroyoshi Aizawa and Hiroshi Yoshino. "CFD prediction of surface condensation on walls and its experimental validation." Building and Environment 39.8 (2004): 905-911

[9] Narkhede, Harshal, P. R. Dhamangaonkar and K. Parashar. "CFD Simulation for Condensation of Humid Air Over Vertical Plate with Eulerian Wall Film Approach." Recent Advances in Fluid Dynamics: Select Proceedings of ICAFFTS 2021. Singapore: Springer Nature Singapore, 2022. 425-439

[10] PSYCHROMETRIC CHART http://www.uigi.com/UIGI_SI.PDF

[11] ANSYS FLUENT Theory Guide https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v201/en/flu_th/flu_th.html

Author information

Vijay Kumar Turaga

Vijay has been working at HCLTech since 2010. He has almost 27 years of industry experience in the CFD domain. He has a master's in mechanical engineering from IIT Madras, Chennai, specializing in internal combustion engines. He has handled a wide variety of industrial problems involving conjugate heat transfer, air-fuel mixing and combustion, rotating machinery, multiphase and more. He is a part of the MCOE CFD and thermal team at HCLTech, working across all verticals such as aerospace, automotive, industrial, consumer products, hi-tech, semi, telecom and medical devices.