Effect of Airflow Speed on Cooling an Aircraft Electronics Box

Avanthika Vuppala

Department of Mechanical and Aerospace Engineering, Rutgers University, Piscataway, New Jersey 08854

Abstract

This project studies forced convection cooling of a heated aircraft electronics box using Ansys Fluent. A two dimensional flow domain was created with cool air entering from the left, passing over a heated rectangular electronics component, and exiting through a pressure outlet. The purpose of the simulation was to understand how inlet airflow speed affects cooling performance by analyzing velocity vectors, temperature contours, heat transfer rate, maximum static temperature, average outlet temperature, and mesh convergence. It was hypothesized that increasing inlet airflow speed would increase the convective heat transfer rate because faster airflow removes warm air from the heated surface more effectively. It was also hypothesized that increasing inlet airflow speed would decrease the maximum temperature near the electronics box. The results supported both hypotheses. As inlet airflow speed increased from 0.5 m/s to 2.0 m/s, the heat transfer rate increased from 27.7779 W to 69.1093 W, while the maximum static temperature decreased from 348.7544 K to 347.0937 K. These results show that stronger forced convection improves cooling performance, which is important because aircraft electronics must remain within safe temperature limits for reliable operation.

Introduction

This project simulates forced convection cooling of a heated aircraft electronics box using Ansys Fluent. Air enters the flow domain from the left side, passes over a heated rectangular electronics component, and carries heat away through the outlet. The simulation compares different inlet airflow speeds to see how the cooling performance changes, and the main results include velocity vectors, temperature contours, heat transfer rate, maximum temperature, and mesh convergence. The main physical relationship being studied is the relationship between inlet airflow speed and heat removal from the heated electronics box. The physics involved include airflow, forced convection, temperature gradients, and thermal boundary layer behavior near the heated surface. When cooler air flows over the heated box, heat transfers from the solid surface into the moving air. If the airflow speed is increased, the heated air near the surface is expected to be carried away more quickly. The velocity field and temperature field are connected because the moving air transports thermal energy downstream, which makes this project a coupled thermal and fluid flow simulation. Heat transfer rate is used as the main value for comparing cooling performance, while maximum temperature is also important because it shows whether the electronics box stays cooler when inlet airflow speed is increased.

The problem being solved is how airflow speed affects the cooling of a heated aircraft electronics box. Aircraft electronics can generate heat during operation, and too much heat can reduce reliability or cause failure. In aerospace systems, electronics are often placed in compact spaces where heat can easily build up, so engineers need to understand how airflow can remove heat from components before final cooling designs are built. This project uses CFD to study the cooling behavior without needing a physical test setup. Fluent is used to predict airflow patterns, temperature distributions, heat transfer rate, and maximum temperature. The same geometry and heating condition are used for each case, while only the inlet airflow speed changes. This makes it easier to compare the effect of velocity on cooling performance. The main hypothesis is that inlet airflow speed and convective heat transfer rate have a direct physical dependence. As inlet airflow speed increases, the heat transfer rate from the heated electronics box to the air is expected to increase because faster airflow removes warm air from the surface and reduces the thermal boundary layer thickness. The second hypothesis is that inlet airflow speed and maximum box temperature have an inverse physical dependence. As inlet airflow speed increases, the maximum temperature is expected to decrease because stronger forced convection carries more thermal energy away from the heated surface. The temperature field is also expected to depend on inlet airflow speed, with higher airflow speed reducing the size of the hot region near the electronics box and moving the heated wake farther downstream.

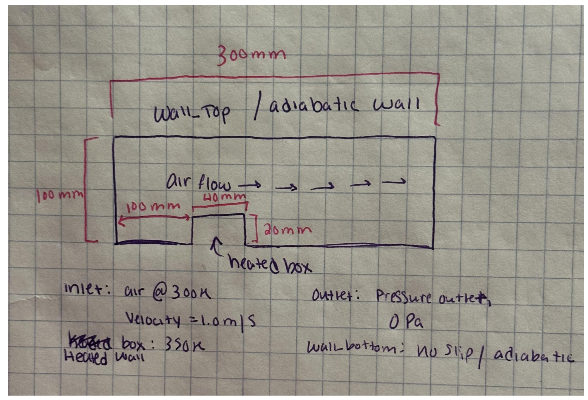

Figure 1: Hand-drawn sketch of forced convection cooling over a heated aircraft electronics box. Cool air enters from the left through a velocity inlet at 300 K, flows over the heated rectangular electronics box, and exits through the pressure outlet on the right. The electronics box surface is modeled as a heated wall at 350 K, while the top and bottom boundaries are treated as no-slip walls. The sketch shows the geometry, airflow direction, heat transfer from the box to the air, and the main boundary conditions used in the Ansys Fluent simulation.

This project simplifies an aircraft electronics cooling problem into a clear two dimensional Fluent model while still keeping the important coupled physics of airflow and heat transfer. Instead of modeling a full aircraft electronics rack, the project isolates inlet airflow speed as the main design variable. The project is also different because it includes a mesh convergence study before testing the airflow speed hypothesis. This makes the results stronger than a simple visual CFD comparison because the numerical setup is checked before drawing conclusions from the airflow speed data. Aircraft electronics support communication, navigation, control, sensing, and monitoring systems. These components generate heat during operation, and that heat must be removed so the electronics can continue working safely. If an electronics box is not cooled properly, heat can build up around the component. High temperatures can reduce performance, shorten component life, or lead to system failure. Cooling is especially important in aerospace systems because space, weight, and airflow paths are often limited. Engineers need cooling methods that are effective without adding unnecessary complexity or mass. Forced convection is useful because airflow can remove heat from hot surfaces and carry it downstream. Understanding how inlet airflow speed affects heat removal helps engineers decide whether stronger airflow meaningfully improves cooling performance. This project is motivated by the need to evaluate cooling performance early in the design process. CFD allows different airflow conditions to be compared before physical testing, which can save time and reduce trial and error design changes.

Aircraft electronics cooling has been studied using CFD because electronic equipment must remain within safe temperature limits during operation. Stancato, dos Santos, and Pustelnik studied electronic package cooling in an aircraft using CFD, which connects directly to this project because it includes airflow, electronics, and aircraft thermal management [1]. Their work supports the idea that simulation can be used to study cooling behavior before relying only on physical testing. Forced convection occurs when fluid motion removes heat from a heated surface and transports thermal energy downstream. In this project, air flows over a heated electronics box and carries thermal energy away from the surface. Ansys Fluent is useful for this type of problem because it can solve airflow and heat transfer behavior in the same simulation [2]. The Ansys forced convection tutorial supports the use of velocity fields, temperature contours, and heat transfer results to evaluate cooling performance [2]. These references support using inlet airflow speed as the main independent variable and heat transfer rate as the main dependent result. For this final paper, additional references are included to strengthen the background section, with at least five total references including scientific sources and an Ansys tutorial.

Methods/Design of Simulated Experiments

This project uses two main physics areas: airflow and heat transfer. The first physics area is airflow around the heated aircraft electronics box. Air enters the domain through a velocity inlet, moves around the heated rectangular component, and exits through a pressure outlet. The airflow pattern is shown using velocity vectors and velocity magnitude results. This part of the simulation helps show how air moves over the box and where cooling may be stronger or weaker. The second physics area is heat transfer from the electronics box to the surrounding air. The electronics box is treated as a heated surface, while the incoming air is cooler. Heat transfers from the box into the moving air through forced convection. Temperature contours are used to show where heat builds up and how the warm air moves downstream after passing over the box. Together, these two physics areas make the project a coupled fluid flow and heat transfer simulation.

The simulation was completed in Ansys Fluent using the finite volume method. The flow domain was divided into a mesh so the fluid flow and heat transfer equations could be solved over small control volumes. The energy equation was included because the project involves temperature changes and heat transfer. The same geometry was used for each case, while inlet airflow speed was changed to compare cooling performance. Several simplifications were used to keep the project manageable. The electronics box was modeled as a simple rectangle instead of a detailed circuit board. The simulation was two dimensional to reduce setup time and computation time. Radiation heat transfer was not included because the project focuses on forced convection, and the box was treated as rigid and stationary.

The geometry included a rectangular airflow domain with an inlet on the left side and an outlet on the right side. A rectangular electronics box was placed inside the flow domain, and air flowed over and around the box before leaving through the outlet. The geometry included the velocity inlet, pressure outlet, heated electronics box, no slip walls, adiabatic boundaries, and airflow direction. Convergence was tested using average outlet temperature as the dependent variable because this value is an engineering result directly connected to the cooling objective. Residual plots were still monitored to confirm that each simulation reached a stable numerical solution. However, the main convergence decision was based on whether the average outlet temperature approached a stable value as the mesh was refined. The main independent variable for mesh convergence was the number of elements. The coarse, medium, and fine meshes contained 708, 1352, and 3616 elements. The airflow speed study used the same selected mesh for each case, which made sure that changes in heat transfer rate and maximum temperature were caused by inlet speed instead of mesh differences.

Results and Discussion

Mesh convergence was tested before the airflow speed study by comparing average outlet temperature for different element counts. The coarse, medium, and fine meshes contained 708, 1352, and 3616 elements, with average outlet temperatures of 300.28365 K, 300.36691 K, and 300.31982 K. These outlet temperature values all stayed close to 300.3 K, which suggests that the solution was reasonably stable as the mesh was refined. Residuals were also monitored during each Fluent run to confirm that the numerical solution stabilized. The convergence plot uses average outlet temperature instead of iteration number because average outlet temperature is a physical result connected to the cooling problem. This preliminary convergence study supported using the selected mesh for the airflow speed study, so the same geometry and mesh were used for all airflow speed cases. This helped make sure that changes in the final results were caused by inlet airflow speed instead of changes in the numerical setup.

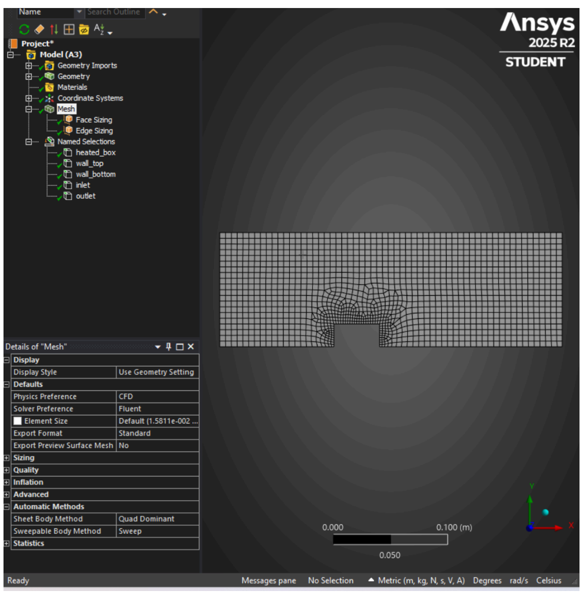

Figure 2: Meshed geometry of the airflow domain and heated aircraft electronics box. The mesh is refined near the heated box because this region is expected to have stronger velocity and temperature gradients. The medium mesh contains 1352 elements and is used as part of the mesh convergence study.

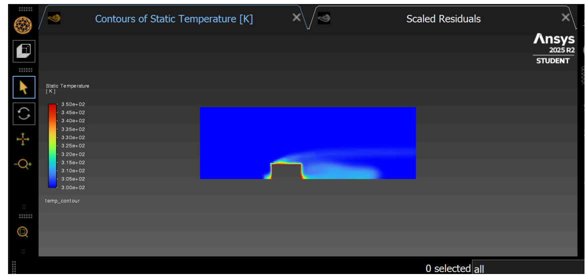

Figure 3: Temperature contour for forced convection cooling of the heated aircraft electronics box. The contour shows how heat moves from the heated component into the surrounding airflow and downstream region. This figure helps visualize where heat builds up and how the airflow carries thermal energy away from the box.

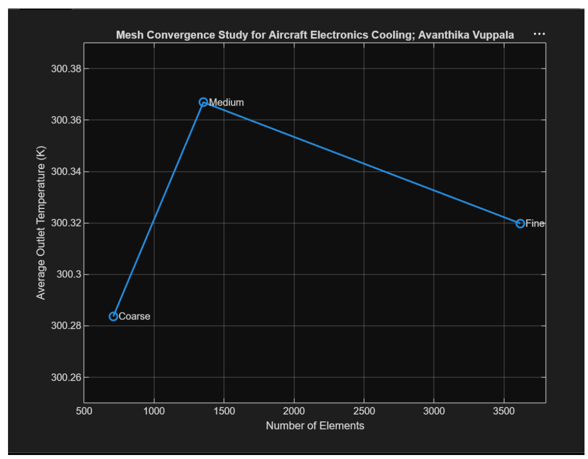

Figure 4: Mesh convergence plot showing average outlet temperature as a function of number of elements. The coarse mesh had 708 elements and an average outlet temperature of 300.28365 K, the medium mesh had 1352 elements and an average outlet temperature of 300.36691 K, and the fine mesh had 3616 elements and an average outlet temperature of 300.31982 K. Since all three values stayed close to 300.3 K, the preliminary solution appears reasonably stable as the mesh is refined.

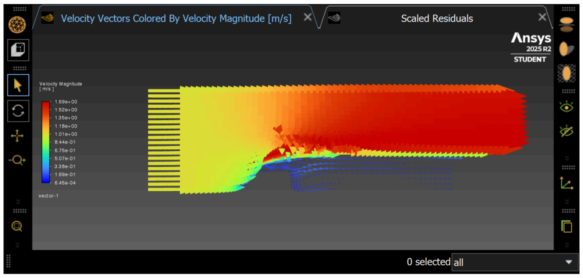

Figure 5: Velocity vectors colored by velocity magnitude for airflow over the heated aircraft electronics box. The vectors show air entering from the left, accelerating over the heated box, and forming a lower velocity region downstream of the box. This figure supports the hypothesis by showing how forced airflow carries heat away from the heated surface.

After the preliminary convergence study, the same geometry and mesh were used to test how inlet airflow speed affects cooling performance. The inlet speed was changed from 0.5 m/s to 2.0 m/s, while the inlet temperature, heated wall temperature, outlet pressure, and boundary conditions were kept constant. For each airflow speed, the heat transfer rate, maximum static temperature, and average outlet static temperature were recorded. These values directly test the hypothesis that increasing inlet airflow speed improves forced convection cooling. The airflow speed results are shown in Table 1. Overall, the data showed that heat transfer rate increased with inlet speed, while maximum temperature and average outlet temperature decreased.

Case	Inlet Speed	Heat Transfer Rate	Maximum Temperature	Average Outlet Temperature
1	0.5 m/s	27.7779 W	348.75436 K	300.49317 K
2	1.0 m/s	52.349417 W	348.22879 K	300.36948 K
3	1.5 m/s	60.549896 W	347.50009 K	300.28444 K
4	2.0 m/s	69.109317 W	347.09372 K	300.23383 K

Table 1: Airflow speed study for forced convection cooling of the heated aircraft electronics box. The inlet airflow speed was changed from 0.5 m/s to 2.0 m/s while the geometry, mesh, inlet temperature, heated wall temperature, and outlet pressure were kept constant. The heat transfer rate increased with inlet speed, while the maximum temperature and average outlet temperature decreased. These trends support the hypothesis that stronger forced convection improves heat removal from the heated surface.

The heat transfer rate was plotted against inlet airflow speed to test whether faster airflow removes more heat from the electronics box. The heat transfer rate increased from 27.7779 W at 0.5 m/s to 69.1093 W at 2.0 m/s. This increasing trend supports the main hypothesis because faster airflow removed warm air from the heated surface more effectively. It also shows that stronger forced convection increased the total heat removed from the electronics box. The maximum static temperature was also plotted against inlet airflow speed to test whether faster airflow lowers the peak temperature near the heated component. The maximum static temperature decreased from 348.7544 K at 0.5 m/s to 347.0937 K at 2.0 m/s. This supports the second hypothesis because stronger forced convection carried more thermal energy away from the heated surface and reduced the hot region near the electronics box.

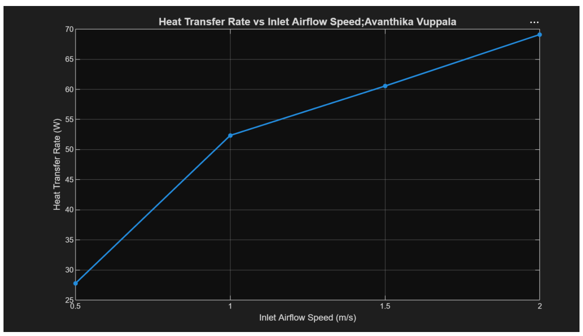

Figure 6: Heat transfer rate as a function of inlet airflow speed for the heated aircraft electronics box. The heat transfer rate increased from 27.7779 W at 0.5 m/s to 69.1093 W at 2.0 m/s. This supports the hypothesis that increasing inlet airflow speed increases convective heat removal from the heated surface.

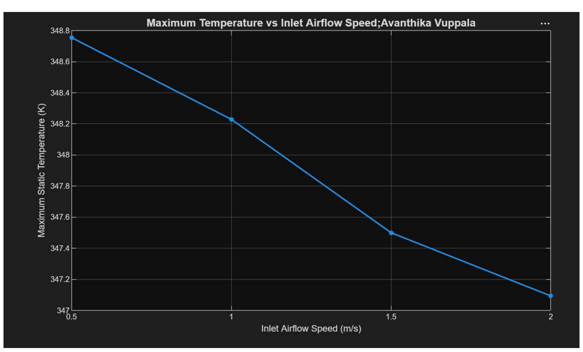

Figure 7: Maximum static temperature as a function of inlet airflow speed for the heated aircraft electronics box. The maximum static temperature decreased from 348.7544 K at 0.5 m/s to 347.0937 K at 2.0 m/s. This supports the hypothesis that stronger airflow reduces the peak temperature near the heated component.

The average outlet temperature was also plotted against inlet airflow speed. The average outlet temperature decreased from 300.4932 K at 0.5 m/s to 300.2338 K at 2.0 m/s. At first, this may seem like less cooling is happening, but it actually makes sense with the rest of the data. Faster airflow moves more air through the domain, so each unit of air has a smaller temperature rise even though the total heat transfer rate increases. In other words, more total heat is removed from the box, but that heat is spread across more moving air. This is why the heat transfer rate increased while the average outlet temperature decreased.

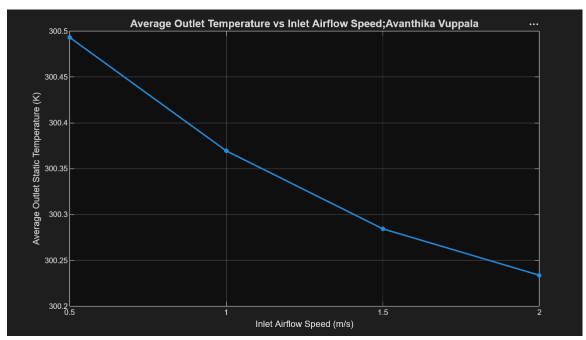

Figure 8: Average outlet static temperature as a function of inlet airflow speed. The average outlet temperature decreased from 300.4932 K at 0.5 m/s to 300.2338 K at 2.0 m/s. This happens because faster airflow moves more air through the domain, so each unit of air has a smaller temperature rise even

though the total heat transfer rate increases.

Overall, the airflow speed data supports both hypotheses. As inlet speed increased from 0.5 m/s to 2.0 m/s, the heat transfer rate increased from 27.7779 W to 69.1093 W, while the maximum static temperature decreased from 348.7544 K to 347.0937 K. These results line up with the original hypothesis because inlet airflow speed had a direct relationship with convective heat transfer rate and an inverse relationship with maximum box temperature. The temperature and velocity results also support the same physical explanation. The temperature contour shows where heat builds up and how warm air moves downstream after passing over the heated box. The velocity vectors show air entering from the left, moving around the box, and forming a lower velocity region downstream. Together, these results show that increasing inlet airflow speed improves the cooling performance of the heated electronics box by strengthening forced convection.

Conclusions

This project studied forced convection cooling of a heated aircraft electronics box using Ansys Fluent. The simulation used airflow over a heated rectangular component to study how inlet airflow speed affects cooling performance, and the main results included velocity vectors, temperature contours, heat transfer rate, maximum static temperature, average outlet temperature, and mesh convergence. The airflow speed study tested four inlet speeds from 0.5 m/s to 2.0 m/s. The heat transfer rate increased as inlet speed increased, while the maximum static temperature decreased, which supports the hypothesis that stronger forced convection improves cooling performance. These results are significant because aircraft electronics must remain within safe temperature limits during operation. If increasing airflow speed improves heat transfer, then airflow control can be an important part of aerospace electronics cooling design. The simulation also shows how CFD can be used to compare cooling conditions before physical testing, which helps connect the project results to real engineering design decisions.

The original hypothesis was that inlet airflow speed has a direct physical relationship with convective heat transfer rate. The data supports this hypothesis because the heat transfer rate increased from 27.7779 W at 0.5 m/s to 69.1093 W at 2.0 m/s. The second hypothesis was that inlet airflow speed has an inverse relationship with maximum box temperature. The data supports this hypothesis because the maximum static temperature decreased from 348.7544 K to 347.0937 K as inlet speed increased. Overall, this project is a simplified version of a real aerospace thermal management problem. Even though the geometry is simple, it still includes important coupled physics between airflow and heat transfer. The project is manageable because only one main variable, inlet airflow speed, is changed. Future work could include additional mesh sizes, domain size convergence, a more detailed electronics box, multiple heated components, or a three dimensional simulation.

References

[1] F. Stancato, L. C. dos Santos, and M. Pustelnik, "Electronic Package Cooling Analysis in an Aircraft Using CFD," SAE Technical Paper 2017-01-2158, 2017.

[2] Ansys, "Forced Convection in Ansys Fluent," Ansys Education Resources. Accessed: Jun. 16, 2026.

[3] A. Bar-Cohen, M. Iyengar, and A. D. Kraus, *Design and Optimization of Air-Cooled Heat Sinks for Sustainable Development*. Hoboken, NJ, USA: Wiley, 2012.

[4] Y. A. Cengel and A. J. Ghajar, *Heat and Mass Transfer: Fundamentals and Applications*. New York, NY, USA: McGraw-Hill Education.

[5] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. Harlow, England: Pearson.