CASE STUDY

Thermal Design for Smartwatches: Ensuring Performance and Safety

Developed and curated by the Ansys Academic Development Team

Sahil Sura

education@ansys.com

Ansys Software Used

This resource uses Ansys Discovery™, a 3D product simulation software, and Ansys Mechanical™, a structural finite element analysis software.

Summary

Explore the application of Ansys Discovery and Ansys Mechanical tools in addressing thermal management challenges in the design of a smartwatch for triathletes. This case study delves into thermal simulation techniques to analyze heat transfer mechanisms—conduction, convection, and radiation—under real-world conditions, including standing, running, swimming, and cycling. Ansys Discovery software facilitates rapid early-stage design exploration, allowing engineers to quickly identify promising design directions and potential thermal risks. Subsequently, Ansys Mechanical software is employed to validate these designs with advanced techniques, ensuring detailed examination of transient thermal behaviors and high-fidelity results. Key design features, such as an aluminum disc for heat dissipation and optimized insulator thickness, are validated to maintain skin-contact temperatures below the safety threshold of 49°C. This comprehensive approach underscores the robust simulation capabilities of Ansys tools, offering valuable insights to optimize wearable device designs for reliability, safety, and performance in extreme operational scenarios.

Table of Contents

1. Abstract

This case study explores the application of Ansys Discovery and Ansys Mechanical tools in addressing thermal management challenges in the design of a smartwatch for triathletes. Equipped technologies like GPS, biosensors, SIM cards, Bluetooth, and LED displays, which when in peak function, lead to thermal congestion in the electronics enclosure, the smartwatch requires efficient thermal management to ensure user safety and comfort during endurance events such as running, swimming, and cycling. Thermal simulations were conducted to analyze heat transfer mechanisms—conduction, convection, and radiation—under real-world conditions, including standing, running, swimming, and cycling. Ansys Discovery tools facilitated rapid early-stage design exploration, while Ansys Mechanical software validated transient thermal behaviors in detail. Key design features, such as an aluminum disc for heat dissipation and optimized insulator thickness, were validated to maintain skin-contact temperatures below the safety threshold of 49°C. The study underscores the robust simulation capabilities of Ansys tools, offering engineers valuable insights to optimize wearable device designs for reliability, safety, and performance in extreme operational scenarios.

2. Introduction to Heat Transfer in Smartwatches

Thermodynamics describes the direction and magnitude of energy transfer between equilibrium states, but in engineering applications, the rate and mechanism of transfer are equally important [1]. However, in practical engineering applications, the nature and timing of energy interactions play a crucial role along with the quantity of energy transferred reaching the equilibrium states. The study of heat transfer complements thermodynamics by exploring the mechanisms, rate, and spatial distribution of thermal energy for analyzing and designing practical engineering systems. Heat transfer occurs through three primary mechanisms: conduction, convection, and radiation, each playing a vital role in managing thermal energy in various applications [2]. In essence, incorporating heat transfer concepts into a design ensures that heat is efficiently managed, thereby safeguarding the appliances and users involved. Hence devising a heat path to manage thermal congestion is the aim of the following case study when designing an efficient high-performance smartwatch. The reader is referred to the Ansys Education Resource on heat transfer [1], covering the underlying concepts in detail, while this case study will focus on their application in product design.

2.1 Safety thresholds and physiological considerations

Wearable devices like smartwatches have rapidly become an integral part of an athlete's daily life, offering convenience, continuous health monitoring and connectivity. Their constant, close contact with the body, however, introduces unique engineering challenges—chief among them, managing and limiting heat transfer to ensure user comfort and safety. Prolonged or excessive thermal exposure from device surfaces can compromise user well-being, underlining the necessity for rigorous thermal design. To address these concerns, it is essential to understand the physiological responses of human skin to heat. Sustained exposure to high temperatures may cause skin damage ranging from mild irritation, rashes to severe burns or scarring, depending on the intensity and duration of heat contact [3]. For instance, temperatures of 49 °C endured for ten minutes can inflict deep dermal injury and charring of the epidermal layer [4].

By integrating knowledge of skin safety thresholds into device engineering, designers can ensure that smartwatch surfaces remain safely below critical temperature limits—typically below 49 °C—under all expected conditions. With these safety benchmarks established, the following section explores the core engineering considerations and advanced methodologies for designing smartwatch systems that effectively manage thermal output, safeguarding both the user and the performance of the device.

2.2 Thermal Design Requirements for Wearable Devices

A comprehensive understanding of the anatomical configuration (see Figure 1) of the concerned smartwatch is essential for achieving effective thermal safety. The structure consists of three primary casing segments: "casing 2," incorporating belt lugs and fabricated from Aluminum 6061 to fulfil Mechanical strength requirements sandwiched between "casing 1" and "casing 3," both produced from rigid PVC to optimize mass distribution and support key design objectives (see Figure 1). The display assembly is comprised of the touchscreen interface, screen, and associated circuitry with the electronics enclosure positioned below subject to this assembly. To ensure the safety of the display assembly, a thermally actuated cutoff switch is provided which cuts off the energy supply to the display assembly as soon as the temperature of the assembly reaches 60 °C.

To promote uniform heat dissipation originating from the dominant thermal source which is the electronics enclosure (consisting of battery and other electronic module assemblies), an aluminum disc is secured atop the enclosure. This disc component facilitates even thermal spread and expedites heat transfer to the external casing, enabling subsequent dissipation into the ambient environment. For user protection, specifically at the skin-contact surface, an insulating layer is adhered to the disc, followed by a back glass that is integrated into the overall watch housing for enhanced visual appeal.

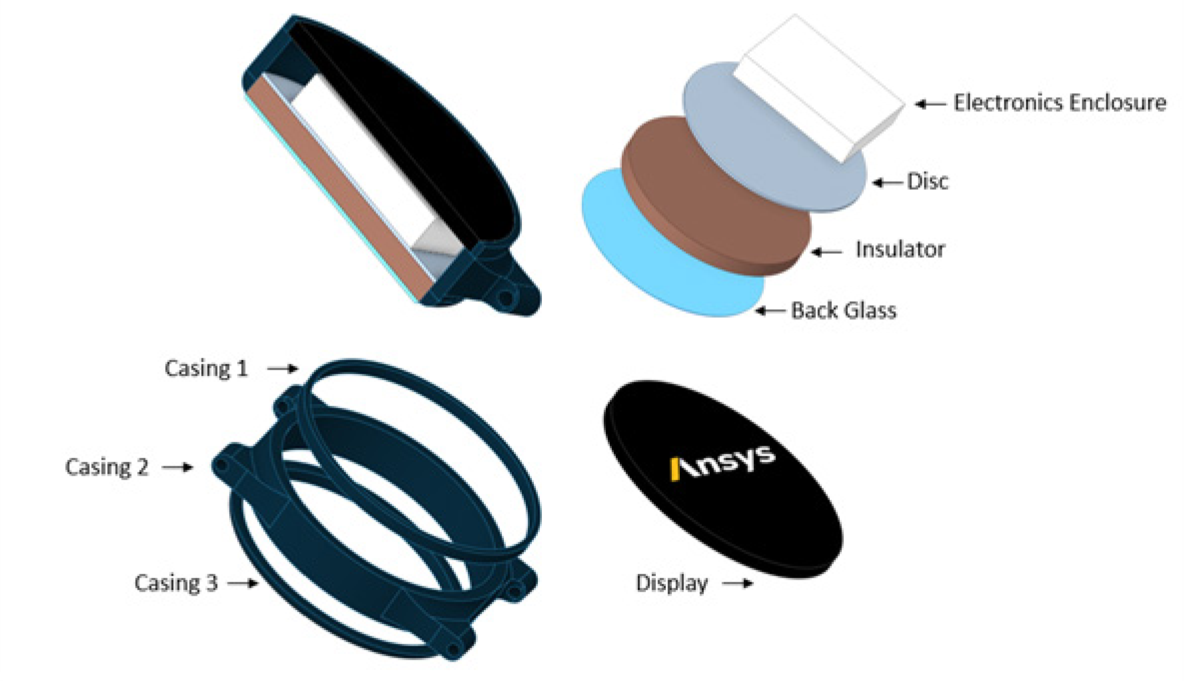


Figure 1: Cross section of the smartwatch with the respective components

Device thickness remains a critical parameter influencing consumer acceptance, with typical commercial smartwatches exhibiting a profile between 9 mm and 15 mm. In the present design, the thickness of the insulating layer plays a pivotal role in determining both the overall device dimensions and the resultant temperature at the skin interface. Furthermore, it affects the material usage and thus cost and CO_2 footprint which design engineers must consider. This should not be the focus here but can also be assessed within as the part is designed in Ansys Discovery software (see Figure 2) or with more detailed information within the materials databases of Ansys Granta tools.

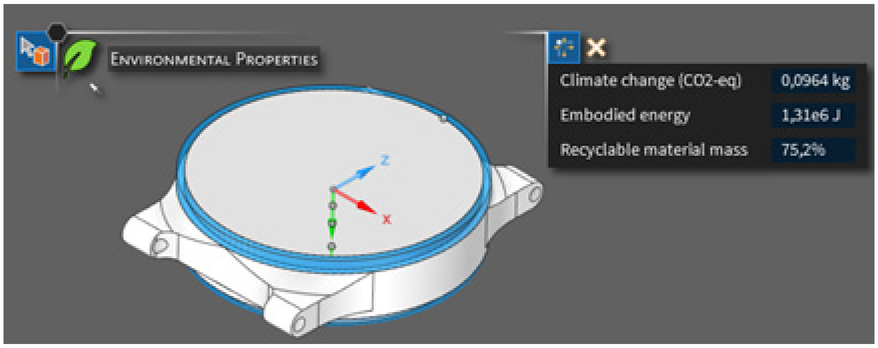


Figure 2: The measurement tool in Ansys Discovery software enables – among other things – a quick up-front assessment of the climate change equivalent and embodied energy of your product for given material choices.

3. Methodology & Results

3.1 Investigation Into Insulator Thickness

Accordingly, quantifying the optimal thickness of the insulator emerges as a fundamental phase of the design workflow. Ansys Discovery software was employed to evaluate this parameter within a three-dimensional computational model under varying physical conditions. Given the placement of the insulator between the aluminum disc and the back glass, maintaining precise contact interfaces was deemed crucial for efficient thermal management. A parameterized investigation was conducted to ascertain the ideal insulator thickness from the 3 options as per design requirement, utilizing the History-Tracking feature to monitor associated variables. The History Tracking feature supports transparent documentation of design modifications and simulation steps, providing traceability and facilitating comparison between different iterations (see Table 2, Figure 3 for additional information).

The resulting thermal performance of the back glass surface was evaluated with material properties as mentioned in Table 1, under the condition of the electronics enclosure operating at 60 °C, with outcomes summarized in Table 2.

Based on thermal analysis, an insulator thickness of 3 mm was determined to be optimal for maintaining the surface temperature within established safety parameters while preserving the preferred form factor of the watch. Subsequently, the finalized watch design incorporates an insulator thickness of 3 mm with overall thickness of 11.5 mm.

Table 1: Material properties for the concerned thermal simulation

Component Name	Material name	Density (kg/m3)	Specific heat (J/kgK)	Thermal Conductivity (W/mK)
Display	Display unit	2000	1100	10
Casing 1, Casign 3	Plastic	1392	1049	0.1753
Electronics Enclosure	Electronics enclosure	2500	479.4	50.4
Casing 2, Disc	Aluminum Alloy	2713	915.7	155.3
Insulator	Insulator	25	1900	0.01
Back Glass	Glass	2125	779.7	1.149

Table 2: Temperature at contact with respect to insulator and overall watch thickness

Insulator thickness (mm)	Thickness of Watch (mm)	Temperature at contact (°C)
1	9.5	41
2	10.5	34
3	11.5	31

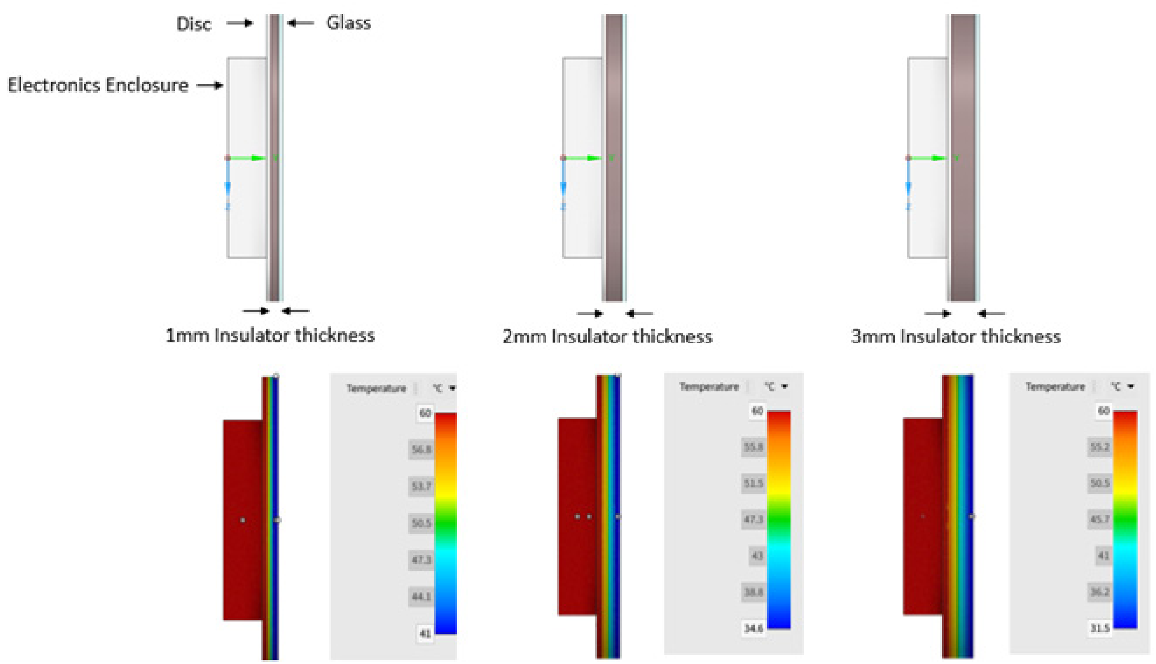


Figure 3: Temperature distribution for cases with varying insulator thickness

3.2 Scenario-Dependent Thermal Simulation

Four distinct thermal scenarios were simulated to replicate real-world conditions and verify the design's resilience. Each scenario applied specific boundary conditions to test the smartwatch's thermal response, ensuring energy dissipation is controlled for safety and optimal performance.

Table 3: Boundary conditions for cases for simulation

Standing	**Running**	**Swimming**	**Cycling**
At ambient temperatures without any drastic changes in environment	Usually conducted in areas with tree cover to ensure athletes won't exhaust due to sun causing accidents during the swim	In open waters and relatively cold temperatures as compared to on land	Conducted on well serviced wide roads due to which the area is exposed to direct sun
Ambient Temperature: 25°C	Ambient Temperature: 25°C	Ambient Temperature: 10°C	Ambient Temperature: 25°C
Daily usage scenario when non-athletic activities are performed	Body **temperature rises with time** due to exertion causing sweating	**Rapid temperature change** as environment changes	Prolonged **exposure to radiation** with higher wind speeds
Boundary Conditions:	**Boundary Conditions:**	**Boundary Conditions:**	**Boundary Conditions:**
Electronics enclosure operating at 60°C Free natural convection at 25°C, with convective heat transfer coefficient of 10 W/m²·K	Electronics enclosure operating at 60°C Temperature of the surface of contact makes equilibrium with the rising temperature of the skin Due to sweating, the convective heat transfer coefficient at the surface of contact changes for specific temperature as sweating and temperature vary linearly as per assumption Free natural convection at 25°C, with convective heat transfer coefficient of 10 W/m²·K	Electronics enclosure operating at 60°C Temperature of the surface of contact makes equilibrium with the decreasing temperature of the skin due to rapid change in surrounding temperature, the convective heat transfer coefficient at the surface of contact changes for specific temperature and vary linearly with temperature as per assumption The convective heat transfer coefficient of the surroundings changes	Electronics enclosure operating at 60°C Temperature of the surface of contact makes equilibrium with the rising temperature of the skin Forced convection at 25°C, with a varying convective heat transfer coefficient as shown in table in W/m²·K Radiative heat transfer effects also to be considered

Objectives:

- Temperature at the contact surface should be considerably less than 49°C
- Temperature of the display assembly should be less than 60°C
- Understand thermal effects of environmental changes on the device

3.2.1 Regular Daily Activity – Standing Case

The thermal modelling employed simulates the case of an athlete in daily conditions with the watch at its peak usage. The peak usage of the technologies in the smartwatch like GPS, SIM calling, Bluetooth, etc. causes the electronics enclosure to operate at a temperature of 60°C and considering regular environment with an ambient temperature of 25°C and the natural convection, convective heat transfer coefficient (h) is considered 10 W/m².k.

The simulation was conducted using Ansys Discovery software, applying solid thermal boundary conditions to evaluate the temperature distribution on the back glass surface and display assembly. The thermal interaction with other components was also considered to get a comprehensive idea of the design. Thus, the heat is not only passed through insulator but also to the aluminum casing through the disc.

As mentioned, thermal simulations are not only to get an idea of the temperatures (see Fig. 4.a, b) or the flux (see Fig. 4.c), etc. but underlying objective of any simulation is to ensure that the energy is not trapped and passed on to the sink in a controlled manner ensuring design safety.

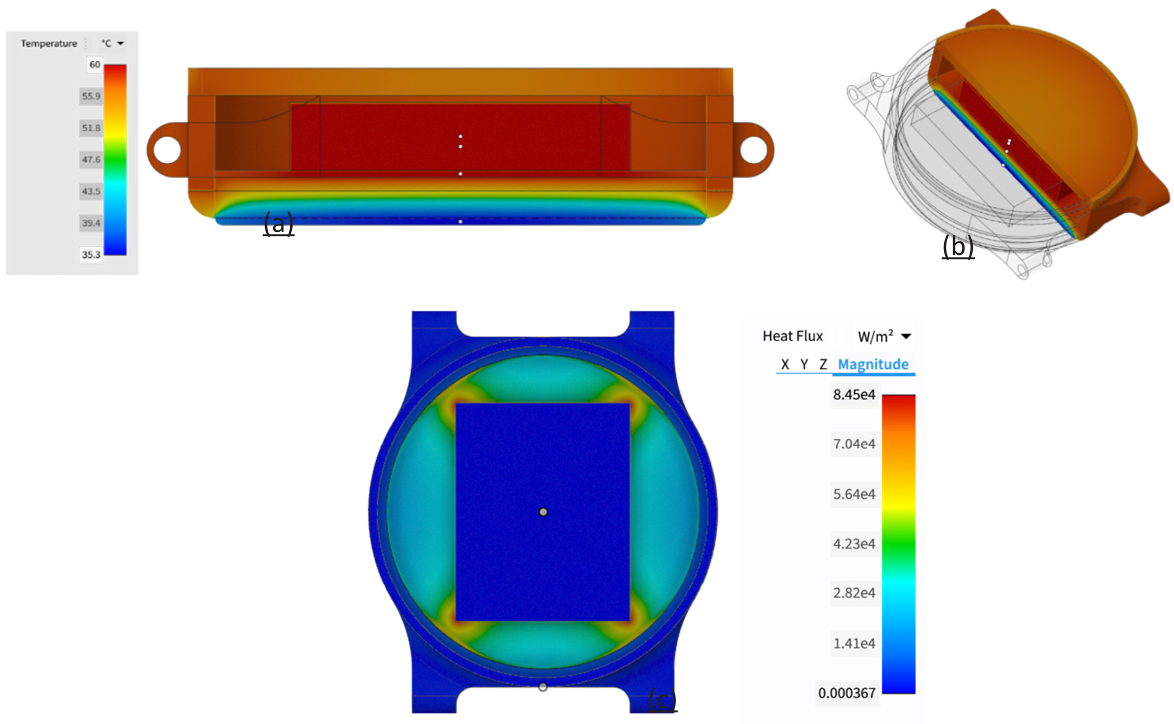


Figure 4: (a) Temperature distribution across the smartwatch, (b) Isometric cross section for the thermal distribution, (c) Heat Flux distribution for the watch with an insulation of 3mm thickness

The thermal evaluation conducted using Ansys Discovery software served as a strategic entry point for conceptual validation. Where design simulated under regular daily activities meets the criterion of the contact surface temperature 35.5°C , well below 49°C and the display assembly temperature as 52°C below 60°C for electronics enclosure operating at temperature of 60°C.It can be seen that the heat flux is in an expected range as a considerable heat transfer is being forced via the aluminium disc. This design decision ensures the heat is being transferred through the disc with a higher rate ensuring safety of other components.

3.2.2 *Integrated Simulation Workflow Utilizing Ansys Discovery and Ansys Mechanical Tools*

Ansys Discovery software serves as an effective platform for rapid conceptual exploration during early-stage design, but the complexity of the design requirements increases, Ansys Mechanical software is necessary. Considering simulating the watch simulated in running environment, the simulation parameters change with time. Ansys Discovery software is capable of showcasing results in a time dependent manner, however for setting up boundary conditions varying with respect to time, Ansys Mechanical software becomes essential for detailed finite element thermal analysis. Ansys Mechanical software supports a comprehensive range of advanced heat transfer simulations, including steady-state and transient analyses, nonlinear material behaviors, contact modeling, and coupled problems. The platform offers advanced mesh control and boundary condition management, which are critical for achieving high-fidelity results in complex engineering applications.

The setup of the analysis from Ansys Discovery software can be simply transferred to Ansys Mechanical software with a single click from transfer group under simulation tab (see Fig. 5). This ensures all the aspects of the simulation are correctly transferred to Ansys Mechanical software including materials, scoped boundary conditions, etc. Since a steady state thermal analysis was performed in Ansys Discovery software, the respective analysis system was set up in Ansys Workbench for Ansys Mechanical software. With a simple drag and drop of any requested analysis system on to the steady-state analysis system would ensure that the engineering data, geometry, and the model is transferred to the transient thermal analysis system. (Please note there is a difference in sharing and linking the systems, for current scenario the results of the steady-state thermal analysis would not serve as input to the transient thermal analysis thus the solution was not shared as input to the transient system.)

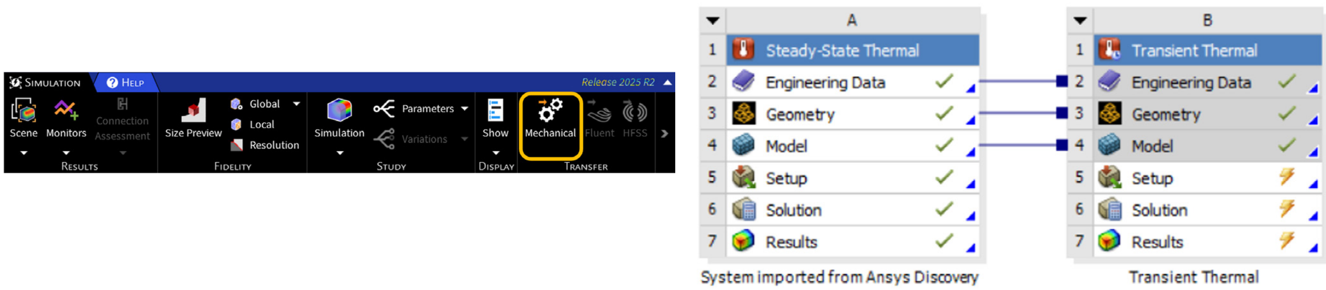
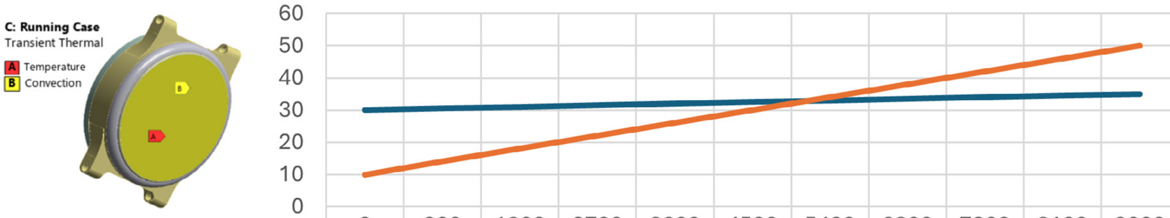


Figure 5: Transferring the Analysis Setup from Ansys Discovery software to Ansys Mechanical software

3.2.3 *Sporting Activity – Running Case*

For this case when the athletes are wearing this watch, at the contact surface the temperature along with the convection coefficient changes with time as sweating happens. The transfer of heat from the gaseous phase to liquid phase happens this is accounted by the change in the value of heat transfer coefficient. This time-dependent condition can be simulated using Ansys Mechanical with the Transient Thermal Analysis System to include the effects with respect to the time. For an athlete using a smartwatch at full capacity, increased exertion raises contact temperature and causes sweating, altering the convective heat transfer coefficient. To closely match near physical conditions and ease of defining boundary conditions, temperature and convective heat transfer coefficient are modeled as linearly varying over 9000 seconds. With the testing and physical references, usually it takes 9000 seconds i.e., 2.5 hours to raise the skin and body temperature by 5°C for an athlete in running conditions. Considering linear variation for convection coefficient accounting for sweating, its value changes from 10 W/m2.k to 50 W/m2.k, variation of both these boundary conditions and the scoping surfaces is shown in Figure 6. This is an added boundary condition to the thermal heating of the electronics enclosure. Thus, the objective would be to check if the temperature of the display assembly reaches over 60 °C considering its shutoff condition for the contact temperature from 30 °C to 35 °C. Hexahedral elements were used to ensure a resistance free thermal load transfer, and meshing was performed with a multizone method with an element size of 0.5mm, with an average element quality of 0.8. The contacts are assumed to be bonded contacts for maintaining the simplicity of the simulation.

The temperature attained from the simulation states 57.5 °C (See Figure 7.a) for the display assembly which is less than the acceptable cutoff temperature. Though the heat flux is on higher end, it is acceptable considering the conditions and the design. A particular pattern is observed for thermal distribution observed in the heat flux (see Figure 7.c), though intuitively a uniform distribution was expected. This generated pattern is due to the geometric design and its effects on the heat flux distribution. The rectangular shape of the electronics enclosure facilitates the areas near its corners as the shortest way for energy transfer, because of which there is higher heat flow through the area. These abrupt geometric transitions may cause a non-uniform heat transfer in many cases but in this case geometric transitions merits in rapid heat transfer serving the purpose. It can be stated that the objective of placing an aluminum disc maximizing the heat transfer to itself was achieved.



	0	900	1800	2700	3600	4500	5400	6300	7200	8100	9000
temperature (°C)	30	30.5	31	31.5	32	32.5	33	33.5	34	34.5	35
convective heat transfer coefficient (W/m2K)	10	14	18	22	26	30	34	38	42	46	50

Figure 6: Time (s) vs Temperature and Convective Heat Transfer Coefficient for the running case

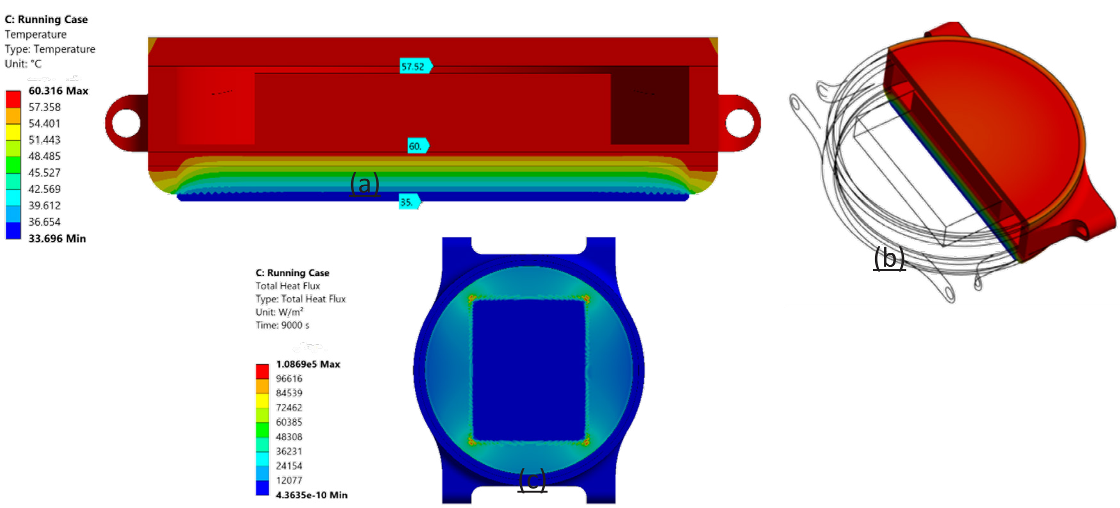


Figure 7: (a) Temperature distribution across the smartwatch, (b) Isometric cross section for the thermal distribution, (c) Heat Flux distribution for running case

Some anomalies exist on close observation. By including Thermal Error result objective issues like thermal overshoot can be identified (see Figure 8). Thermal overshoot is a numerical artifact where the predicted temperature exceeds realistic values based on the applied boundary conditions. This helps identify regions of high error and therefore show where the model would benefit from a more refined mesh or higher order elements (linear elements are more susceptible) to get a more accurate answer. Error result objects identified areas of potential attention. Since FEA is an approximate mathematical solution, convergence issues can arise for a given time step. Consequently, mesh refinement or a reduction in the timestep for a critical thermal change help address this issue. With the error being in ranges of 10-3, no major changes are expected.

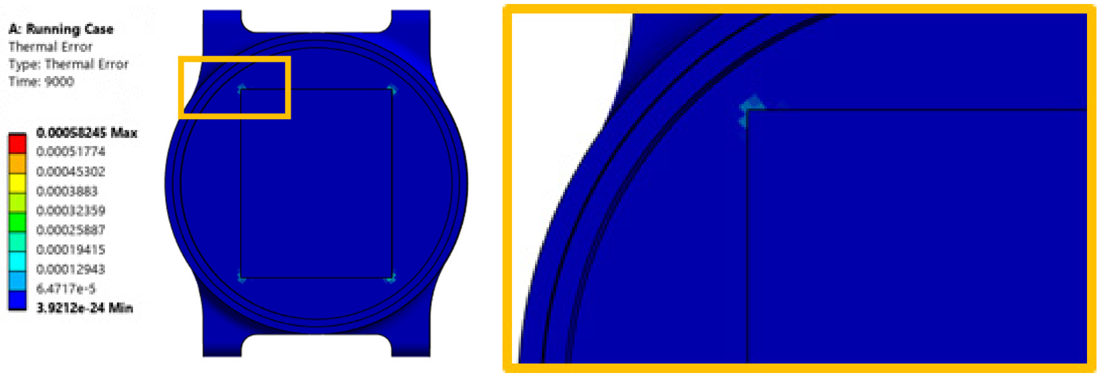
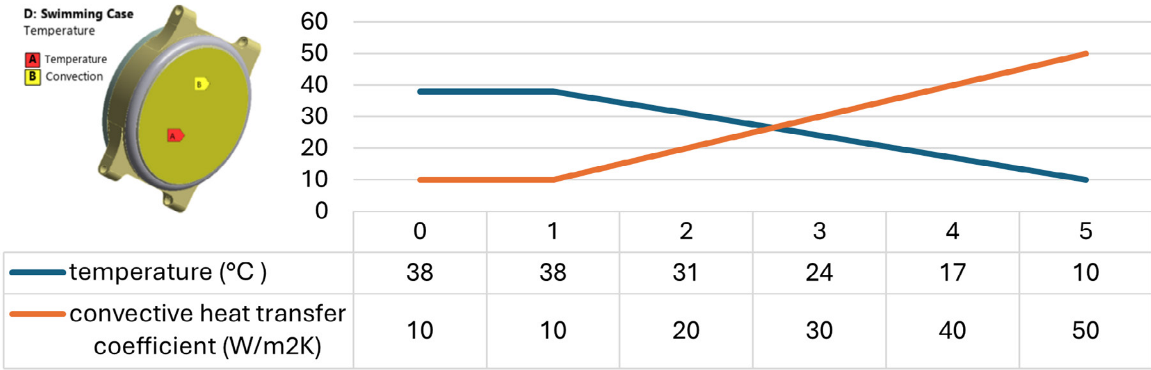


Figure 8: Maximum Thermal error observed on disc component for running case

3.2.4 *Sporting Activity – Swimming Case*

Whether participating in a triathlon or taking a cold plunge, smartwatches of athletes must be able to endure considerable temperature drops. This may cause an uneven temperature distribution and if the devised heat path is not functional for dissipation, there can be thermal damage to the components. Thus, simulating this with near physical boundary conditions is very critical.

After a heavy workout, the body temperature tends to increase to dissipate more heat. Starting off with 3°C higher from the end point of the running case where the temperature of the contact surface of skin and watch can rise from 35 °C to 38°C. In this scenario, it is assumed that the athlete jumps into a water of assumed temperature 10 °C. Figure 9 summarizes the thermal boundary conditions applied to the back glass. This variation of the parameters is considered linearly with time for 5 seconds and shown in the following graph. Observed temperature for the display panel is approximately 22 °C (see Figure 10.a, b) which is less than the cut-off range of 60°C. It can be observed that the heat flux (see Figure 10.c) is an order of magnitude higher than in the running case due to the short timespan in which the heat is being rapidly dissipated via the aluminum disc.



	0	1	2	3	4	5
temperature (°C)	38	38	31	24	17	10
convective heat transfer coefficient (W/m2K)	10	10	20	30	40	50

Figure 9: Time (s) vs Temperature and Convective Heat Transfer Coefficient for the swimming case for the back glass

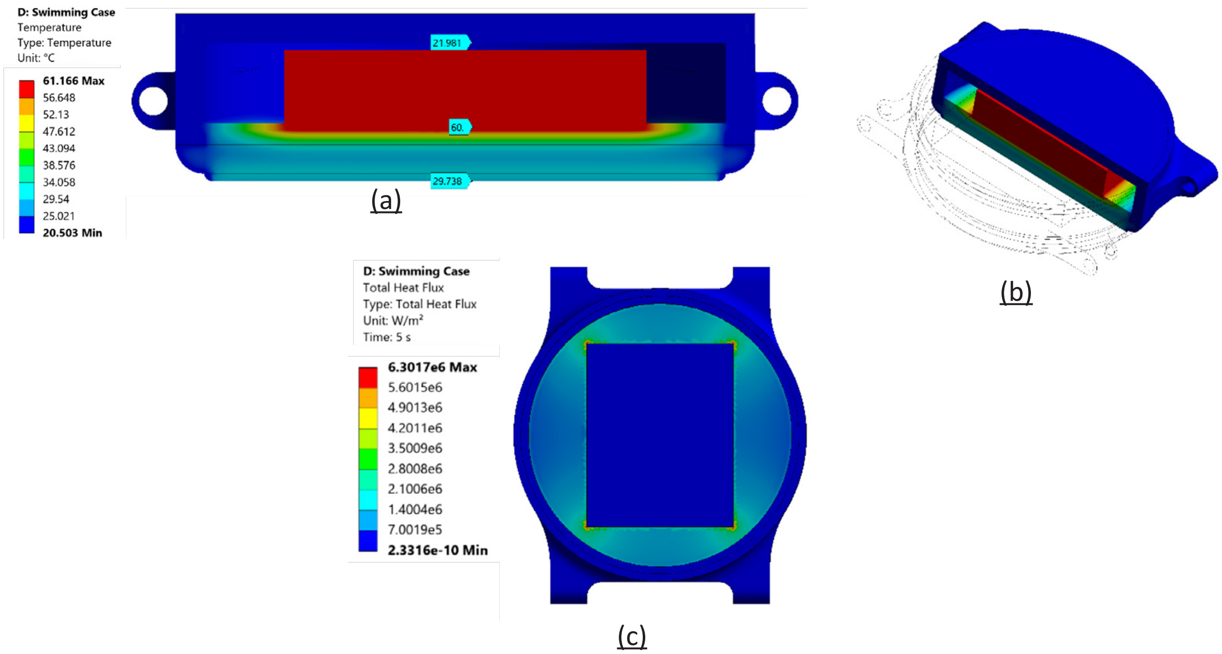


Figure 10: (a) Temperature distribution across the smartwatch, (b) Isometric cross section for the thermal distribution, (c) Heat Flux distribution for swimming case

The thermal error value was found to be 0.55, which is slightly greater than 0.5, the default value of oscillation limit (see Figure 11). It is a dimensionless quantity which measures how well transient thermal response is captured during each step and is a product of the response eigenvalue and the current time step size. Whenever the error is larger than this limit, it may indicate that the timesteps may be too large to obtain stability for the solution. In this case where there was a huge temperature drop in few seconds, the time steps needed to be very small, but this increases the computational expense.

While minor thermal deviations in noncritical areas slightly exceed the threshold, they are manageable with small adjustments and do not compromise solution reliability.

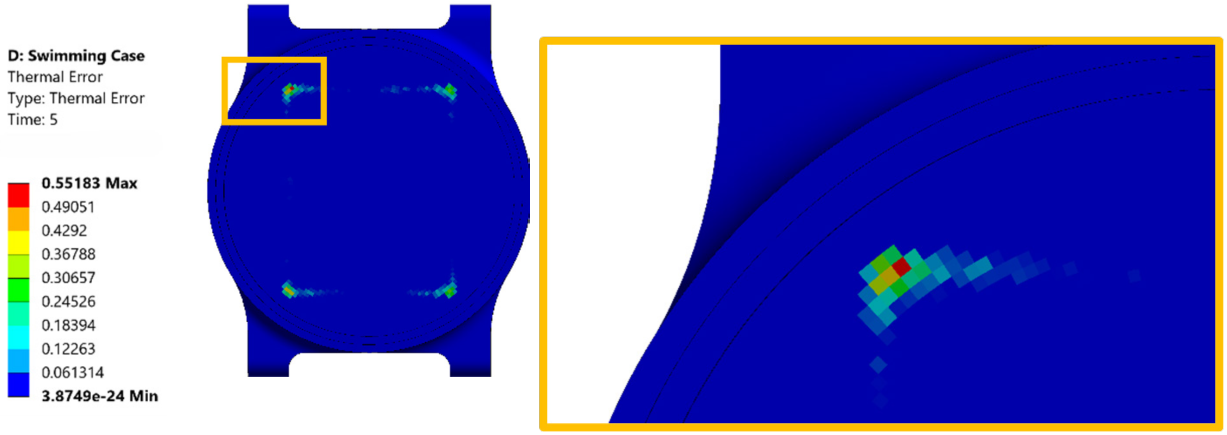


Figure 11: Maximum Thermal error observed on disc component for swimming case

3.2.5 Sporting Activity – Cycling Case

Considering cycling, events often happen on roads with rare forest cover or at altitudes, increasing exposure to solar radiation and winds. The variation of the convection coefficient with respect to time is nonlinear and random in nature as shown in Figure 12 for the ambient temperature of 25 °C. This randomness is due to the varying cyclist speed and the wind speed. Assuming that this pattern (see Figure 12) is representative for a ride the simulation considers only two minutes.
The objective of this simulation case is to apply convective and radiative boundary conditions to evaluate the thermal response at the contact surface and for the display assembly.

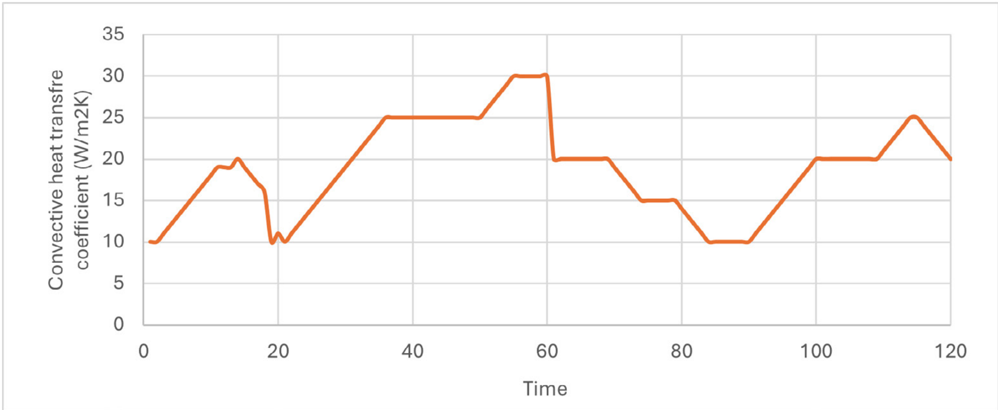


Figure 12: Time (s) vs Convective Heat Transfer Coefficient for the cycling case

Adding this data manually would be a tedious task in Ansys Mechanical software, thus with the help of the “Tables” feature it is just a task of importing a .csv file. Select the variables, set the delimiter accordingly and by simply clicking “Import” a table would be formulated. Rename the table and set the value for the convective coefficient in the convection boundary condition using the tabular data (see Figure 13). For simplicity the ambient temperature is set as constant at 25 °C. Considering the simulation time of 2 min, the effects of sweating and body temperature rise can be ignored for simplicity.

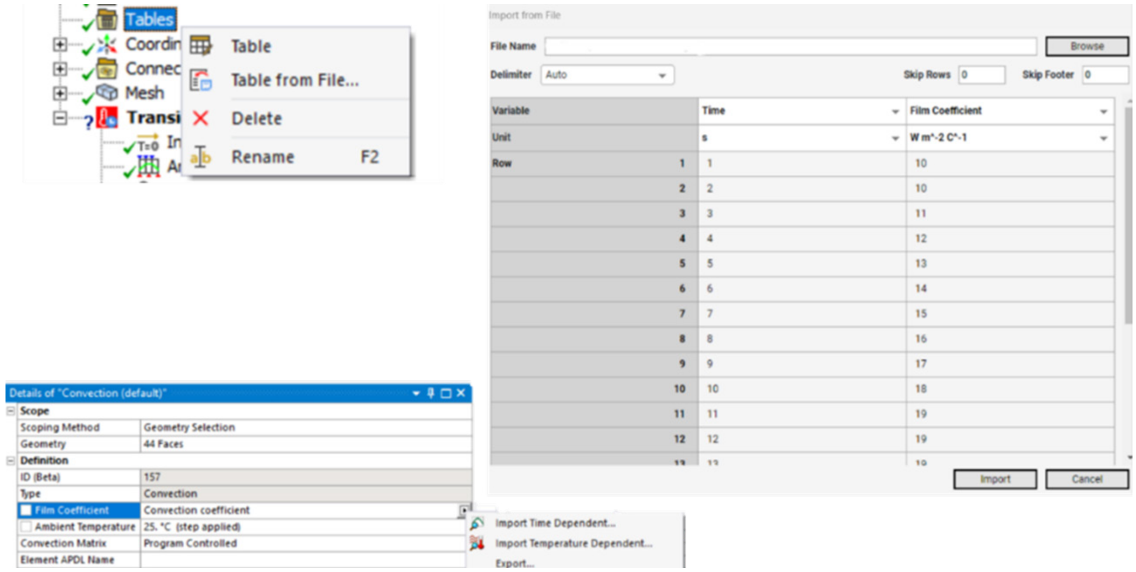


Figure 13: Figure applying boundary conditions using table

To simulate solar irradiation equivalent to 1 sun—a standard value of 1000 W/m² representing typical sunlight intensity at Earth's surface—a radiative surface measuring 100 mm × 100 mm × 1 mm was placed 500 mm above the display unit within an open enclosure (see Figure 14). Surface-to-surface radiation was defined, and radiation was applied to exposed surfaces. With a surface emissivity of 0.8, the internal heat generation was calculated as follows to achieve the desired irradiance:

To simulate 1 sun irradiance (defined as 1000 W/m²) on a radiative surface of 0.1 m × 0.1 m × 0.001 m:

Surface area (A) = $0.1 \times 0.1 = 0.01\ m^2$

Volume (V) = $0.1 \times 0.1 \times 0.001 = 0.00001\ m^3$

Emissivity (ε) = 0.8 (assumed equal to absorptivity α)

Total absorbed power (Q) = $G \times A \times \alpha = 1000 \times 0.01 \times 0.8 = 8$ W

Internal heat generation (q''') = $Q / V = 8 / 0.00001 = 800{,}000\ W/m^3$

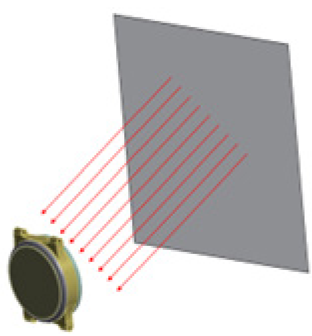


Figure 14: Test setup for radiative heat transfer

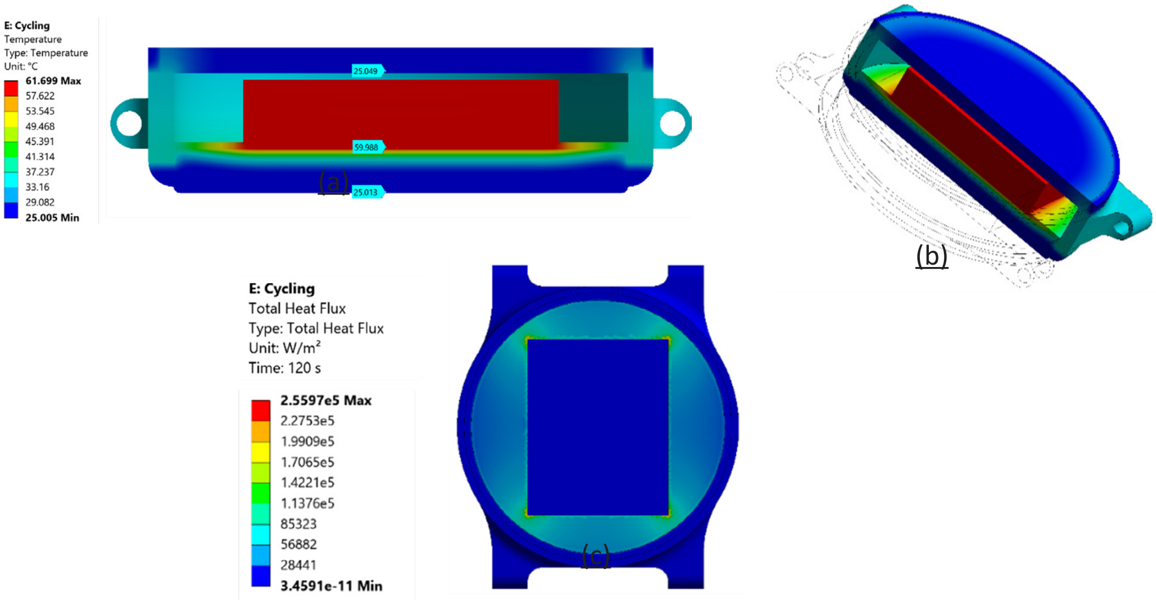


Figure 15: (a) Temperature distribution across the smartwatch, (b) Isometric cross section for the thermal distribution, (c) Heat Flux distribution for cycling case

With the hexahedral mesh having 0.5mm element size, the temperature at the contact surface of watch and skin is 25.013°C (see Figure 15.a, b), while the display assembly temperature is 25.049°C (see Figure 15.c).

Observed from the results the heat transfer via the aluminum disc dominates ensuring the design intent of the same is met. Thus, the radiation effects seem to be negligible for this case. When comparing to the standing, running and swimming cases, the non-uniform convective distribution ensures the temperatures obtained are much less for the back glass and display. While there exist thermal deviations observed as thermal error (see Figure 16) on the aluminum disc, considering their maximum magnitude in ranges of 10-4 they seem to be non-critical.

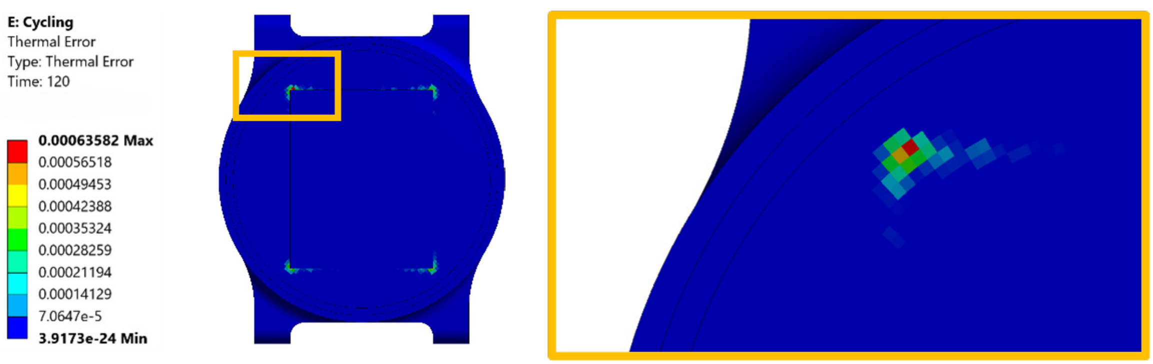


Figure 16: Maximum Thermal error observed on disc component for cycling case

4. Conclusion

In closing, this case study highlights the effectiveness of a multi-tiered simulation approach in wearable device thermal analysis. By first using Ansys Discovery software for rapid conceptual exploration, engineers quickly identified promising design directions and flagged potential thermal risks—such as varying insulator thickness. This was followed by detailed validation in Ansys Mechanical software, enabling thorough examination of heat transfer phenomena. Critical parameters like heat flux, temperature profiles, and convective coefficients were accurately captured, as shown in Table 4, and verified using thermal error outputs. Although, the thermal error slightly exceeded typical thresholds, deviations were confined to noncritical areas and manageable through minor design refinements. This emphasizes the importance of selecting appropriate time steps to balance solution stability and computational efficiency, especially when modeling rapid thermal events with steep temperature gradients. Importantly, the simulation confirmed that the aluminum disc effectively channels heat away from sensitive regions, supporting the original design intent and enhancing reliability. With daily and sporting activities with a smartwatch, environmental changes did affect component temperatures but did not compromise safety or performance. This workflow demonstrates best practices in engineering analysis—blending speed and precision to accelerate development. It ensures wearable devices meet innovation goals while satisfying validation, regulatory, and user requirements. These insights not only shorten development cycles but also equip teams to deliver real-world-ready wearable devices confidently.

Table 4: Simulation results summary

Activity	Maximum Temperature (°C)		Heat Flux (W/m^2) (Maximum)	Thermal Error (Maximum)
	Back Glass	Display		
Standing	35.5	52	8.45 e4	n/a
Running	35	57.5	1.08 e5	5.8 e-4
Swimming	38	22	6.30 e6	0.55
Cycling	25	25	2.55 e6	6.3 e-4

5. References

[1] F. P. Incropera and D. P. DeWitt, Fundamentals of Heat and Mass Transfer, 6th ed. New York: Wiley, 2007.
[2] Lecture Unit: Introduction to Heat Transfer with Ansys Discovery – Ansys
[3] The Johns Hopkins University, The Johns Hopkins Hospital, and Johns Hopkins Health System: Burns and Wounds; https://www.hopkinsmedicine.org/health/conditions-and-diseases/burns; accessed September 2025.
[4] MyCPR NOW: How Hot Does Water Have To Be To Burn You?; https://cprcertificationnow.com/blogs/mycpr-now-blog/how-hot-does-water-have-to-be-to-burn-you; accessed September 2025.
[5] ANSYS, Inc. ; Ansys Discovery Documentation: https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v252/en/discovery/UDA/user_manual/modeling/design/topics/c_HistoryTracking.html?q=history%20tracking; Version 2025R2.
[6] https://ansyshelp.ansys.com/

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited. The full Academic Terms & Conditions can be found using this link.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com