



Case Study

Thermal Analysis of Heat Sinks with Ansys Discovery

Nick Stefani and David Mercier

Developed and curated by the Ansys Academic Development Team

education@ansys.com

Summary

This case study showcases how Ansys Discovery can be used to perform thermal analysis of a heat sink placed above two CPUs generating heat during intense computational tasks. It demonstrates how simulation can be used to test and improve the design of the heat sink and evaluate the effectiveness of the chosen cooling system.

Table of Contents

1. Background.....	3
2. Preprocessing	4
3. Results: From idle usage to intense computation	5
4.Ensuring Thermal Safety.....	5
4.1 Altering the material	6
4.2 Altering the Geometry.....	6
4.3 Altering the Heat Sink Operating Environment	7
5. Conjugate Heat Transfer (CHT) Simulation	7
6. Conclusions and Possible further Analyses	9
7. References	9

1. Background

Most desktops Central Processing Units (CPUs) must run under 70-80°C [1] to safely perform and avoid thermal degradation. However, during their usage CPUs can generate from a few watts to nearly 100W of heat [2], leading to dangerously high temperatures. Heat Sinks, shown in Figure 1, are components that sit on top of heat generating computer parts (e.g. CPUs) draw the heat away. The heat is conducted from the CPUs into the heat sink and then transferred to a surrounding fluid (gas or liquid) via convection.

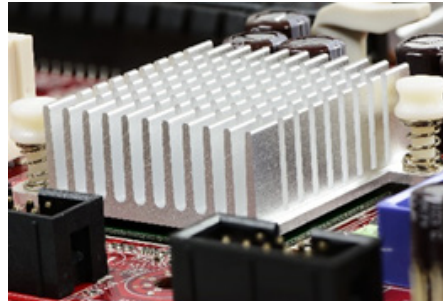


Figure 1. Heat Sink Picture

Table 1. Conduction and Convection Governing Laws

Conduction- Fourier's Law	Convection- Newton's Law of Cooling
$(1D) q''_x = -k \frac{dT}{dx}$ <p>q''=Heat flux q= Heat flow T= Temperature k=Thermal conductivity of the material</p>	$q'' = h(T_s - T_f)$ <p>q''= Heat flux, q= heat flow T_s= Temperature of the solid surface T_f= Temperature of the fluid surface h= Heat transfer or film coefficient</p>

The two prevalent heat transfer modes in the heat sink example are conduction and convection. These phenomena can be described through two different governing laws as shown in Table 1. In both conduction and convection, the transferred heat is proportional to the difference in temperature -a larger temperature difference leads to higher heat transfer- and a coefficient.

In conduction, this coefficient is thermal conductivity, which is a material property representing the ability of a specific material to conduct heat through its body. Materials with higher thermal conductivity will lead to larger heat fluxes. For example, common radiators in households are made of materials with high thermal conductivity such as metals and some technical ceramics, while households' wall cavities are usually made of foams or wood-like materials which have relatively small thermal conductivity and thus provide good insulation.

On the other hand, convection heat transfer is given by the energy exchange between a body and a surrounding fluid (liquid or gas) caused by molecular motion. In this case, transferred heat is proportional to the heat transfer (or film) coefficient. While thermal conductivity only depends on the material, the heat transfer coefficient (h) depends on many things, such as the fluid properties, its motion, and the contact geometry. Values for the heat transfer coefficient can be found in tables, which use assumptions based on typical use cases, but often this value needs to be calculated. To

learn more about the basics of heat transfer, check out our dedicated introductory lecture unit at this link: [Lecture Unit: Introduction to Heat Transfer with Ansys Discovery](#) and the two free self-learning tracks available on the Ansys Innovation Space: [Heat Transfer in Structures](#) (Ansys Mechanical) and [Heat Transfer in fluids](#) (Ansys Fluent). In this case study, for the given assembly shown in Figure 2, heat generation, conduction and convection will all be modeled in Ansys Discovery to investigate the temperature of the two CPUs under different working conditions and explore ways their maintain maximum temperature under 80°C.

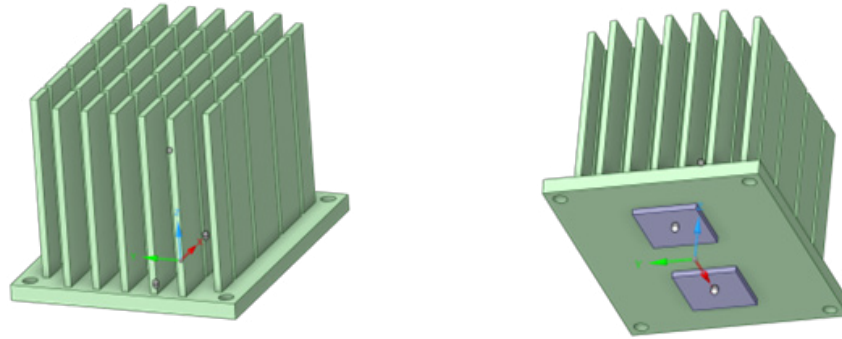


Figure 2. Heat Sink Geometry

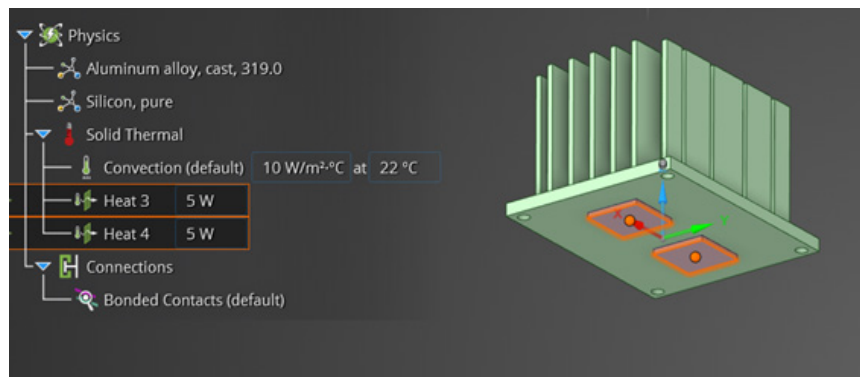


Figure 3. Preprocessing

2. Preprocessing

The first step in the preprocessing analysis is import the geometry shown in Figure 2 into Discovery. The CAD file of the assembly can be found in the downloaded case study folder under the name *Heat_Sink_Initial_Geometry.dsc*. Once the geometry has been imported, material assignment can be made with Silicon assigned to the two CPUs and *Aluminum alloy 6061* for the heat sink being a common material for these applications due to its good thermal conductivity.

With regards to heat generation, two boundary conditions of heat are applied to the bottom surface of the two CPUs with a value of 5 W, which is representative of the CPU performing in its idle state. However, this will be later changed to 50 W which is value characteristic of high computational efforts (e.g. when performing finite element analysis). When these heat boundary conditions are added, Ansys Discovery automatically adds convective heat transfer on the remaining surfaces with a heat transfer coefficient (h) value of 10 W/m²·°C and a surrounding fluid temperature of 22 °C resulting in the fully defined problem shown in Figure 3. The value of h used is representative of natural convection, which refers to the conditions in which the fluid motion is not generated by external sources (e.g. fan). Lastly

the convection boundary conditions, assumes that the heat is dissipated uniformly on all surfaces. This is however an approximation and its implication together with a more realist scenario will be investigated in the Conjugate heat transfer section of this report.

3. Results: From idle usage to intense computation

Once the model has been set up, the simulation can be run in Ansys Discovery. For this case, the explore stage was used which enables fast simulation results through a GPU solver, however the same analysis can be performed in the refine solver which make use of the flagship CPU based solver, also found in Ansys Mechanical.

Figure 4 shows the results of the simulation considering both the idle (left) and the intense computation scenario (right). The relative temperature distribution is the same across both geometries, but the absolute values are considerably different. In the idle state, the maximum temperature reached is 49.4 °C (located in the CPUs), which is safely below the critical temperature of 80 °C. The maximum temperature computed for the intense computation case reaches 299 °C, leading to possible damage in the components. In the next sections, various alternatives will be explored to bring the temperature below the critical level.

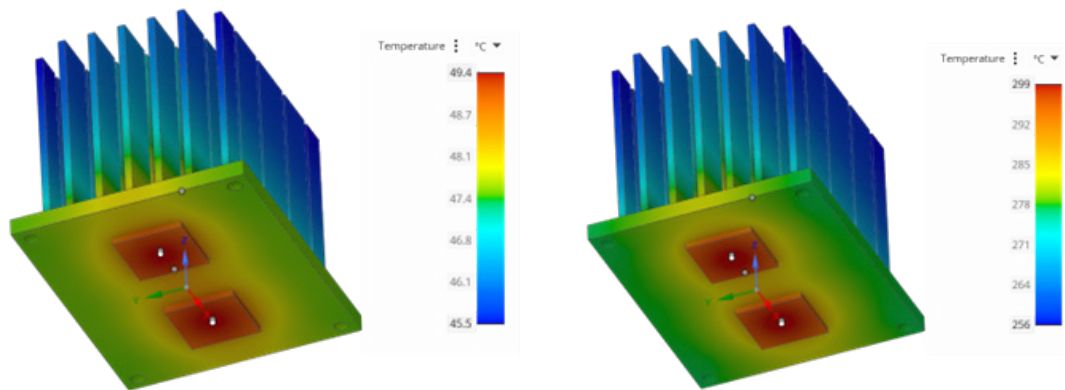


Figure 4. Temperature field results for two heat source scenarios: idle state 5 W/CPU (left), intense computation 50 W/CPU (right)

4. Ensuring Thermal Safety

In this case study, we will examine three different strategies for lowering the temperature of the CPUs: altering the material, altering the geometry, and altering the heat sink operating environment. Results for all simulation conditions can be found in Figure 5, where on the y-axis is displayed the maximum temperature and on the x-axis are present the design variations. While on the x-axis point 1 represent the idle state, and point 2 the intense computation scenario, points 3-5 are each respectively the three aforementioned alterations.

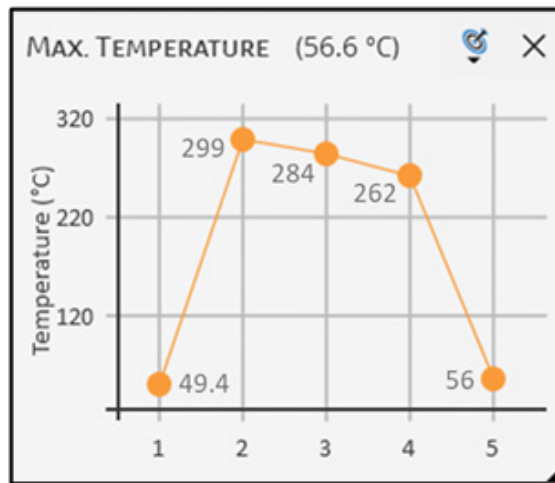


Figure 5. Exploring design scenarios: (1) 5 W/CPU, aluminum material, original geometry, natural convection (2) 50 W/CPU, aluminum material, original geometry, natural convection (3) 50 W/CPU, copper material, original geometry, natural convection (4) 50 W/CPU, copper material, increased convective surface, natural convection (5) 50 W/CPU, copper material, increased convective surface, forced convection.

4.1 Altering the material

To design a strategy to lower the temperature of the CPUs, and thus ensure thermal safety, it is necessary to consider how heat sinks work. These components capture the heat from the CPUs through conduction in the contact area, then the heat is transfer again through conduction in the body of the heat sink (e.g. the fins) and then dissipated by convection through air. Thus, the performance of the heat sink is related to its ability to conduct heat and then release to the environment. As discussed in the first section, the ability to conduct heat within a body is given by the thermal conductivity of a material, thus materials with higher thermal conductivity would in theory improve the ability of the heat sink to dissipate heat and thus lower the CPUs temperature. This is however a simplification, as other material properties such as thermal expansion coefficient, electrical insulation, electrical resistivity, and weight should be considered to find the optimal material for the design. To examine this in more detail, a separate case study (accessible at the following link [Heat Sink Case Study](#)) was carried out with [Ansys Granta Selector](#).

In this study for simplification, a common alternative to Aluminum with better thermal conductivity which is often used in these applications – Copper will be tested. As shown in Figure 5 (design points 2 and 3), changing the heat sink material to Cast High Conductivity Copper (394 W/m·°C) led to a maximum temperature decrease of 15 °C, which while going in the right direction, is not sufficient. It should be mentioned that copper (6.6 USD/kg) is more expensive than Aluminum (2.17 USD/kg) and thus the benefit of the lower temperature may not be justified by the increase in cost [3].

4.2 Altering the Geometry

Another possible solution to bring the temperature down is to increase of the amount of convective surface in the heat sink. To test this, the fins of the heat sink were increased in height by 5mm leading to a further temperature decrease of 22 °C and a maximum temperature of 262 °C (Figure 5 design point 4), which still far exceeds the threshold of 80 °C. One might suggest and the solution is to keep increasing the surface area of the heat sink until below 80°C temperatures are reached. While this in theory would further lower the temperature, in reality this is not feasible as electronic components are

often designed to minimized weight and volume, with each new generation of handheld electronics requiring lighter and thinner components. More creative geometries, which would increase the surface area to volume ratio could be explored but this is beyond the scope of this case study.

4.3 Altering the Heat Sink Operating Environment

The third parameter that can be explored to lower the maximum temperature experienced by the heat sink during use is the external operating environment. This can be investigated by reviewing the boundary conditions (BCs) established in the original simulation (Figure 3). Two solid thermal BCs are shown: the heat generation and the convection. The heat generation cannot be changed but the convection experienced by the heat sink could be altered. This BC is composed of two terms: the heat transfer coefficient and the surrounding fluid temperature. Lowering the temperature of the fluid could lower the temperature in the CPUs, however storing the electronic device in the fridge may not be a practical solution in terms of usability. The heat transfer coefficient that was chosen for this application was characteristic of natural air convection. Forced convection can be achieved by including a fan positioned within the electronic device and blowing air through the heat sink. This would raise the heat transfer coefficient from 10 to 100 W/m²·°C and lead to a maximum temperature of 56 °C (Figure 5, design point 5), ensuring thermal safety and showing why electronic devices such as computers & laptops contain fans for cooling purposes.

5. Conjugate Heat Transfer (CHT) Simulation

When introducing forced convection to the model, one of the assumptions made in the set-up of the boundary conditions needs to be re-evaluated. The convective boundary conditions assumes that the ambient air temperature is the same at all locations, giving uniform heat dissipation on all surfaces. This is not the case due to the air movement caused by the fan. As air moves across the heat sink, as shown in Figure 6, heat is absorbed and the overall air temperature increases. This lowers the temperature gradient between the heat sink and the air, thus reducing the air's ability to absorb further heat as it travels across the heat sink. Because of this, the two CPUs will have different temperatures.

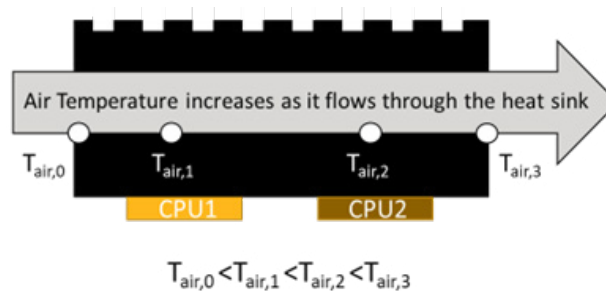


Figure 6. Air temperature flowing through the Heat Sink

In Ansys Discovery, conjugate heat transfer (CHT) simulation can be performed to simulate the air flow and the heat exchange on the fluid/solid interface. The first step is to create a box enclosure as shown in Figure 7a, which represents the fluid domain (material used is Air). Secondly, an inlet and outlet, as shown in Figure 7b, can be added respectively with a 1 m/s velocity and 0 pressure to provide directionality of the fluid flow. Before solving the model, it is necessary to increase the fidelity of our model (i.e. reduce the mesh size) to make sure that the fluid-solid interface can be captured. It is therefore recommended to set a mesh size of maximum 0.75 mm, which is half of the of the fins thickness (fidelity can be changed through the bottom central Fidelity Slider in the Discovery GUI, see [Quick Tips - Ansys Discovery Forum](#) for tutorial videos).

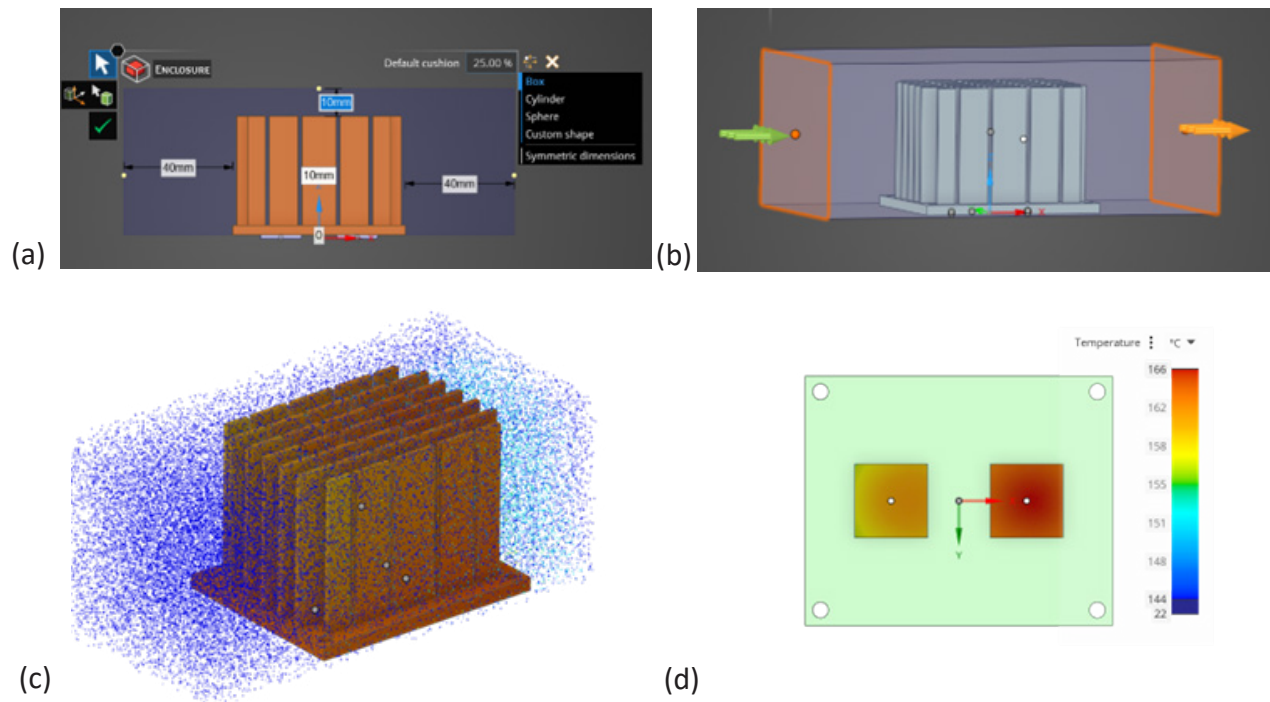


Figure 7. a) Creating fluid enclosure in Ansys Discovery, b) Location of inlet and outlet, c) Air particles flowing through the heat sink, d) temperature difference in CPUs

Once this is done, the simulation can be solved and the fluid flow observed through a variety of outputs visualization features, including the particle animations shown in Figure 7c. With CHT simulation, it is possible to capture the temperature differences in the two CPUs as displayed in Figure 7d, which represents a more realistic scenario compared to homogeneous convection of all surfaces.

The CHT results show that a 1 m/s air flow is not sufficient to lower the two CPUs temperatures below the acceptable max temperature of 80 °C, therefore higher airflow speeds are needed. This can be explored manually by changing the inlet speed boundary condition and running simulation multiple times; however, a more efficient solution is to use Ansys Discovery parametric analysis capabilities (available only in Explore mode). With this it is possible to automatically test several simulation variations back-to-back without requiring user input and plot the results in a graph (see [Quick Tips - Ansys Discovery Forum](#) for tutorial videos).

As per Figure 8, the results of this analysis show that a flow speed of at least 9 m/s is needed to bring the temperature of the CPUs below the critical threshold of 80 °C.

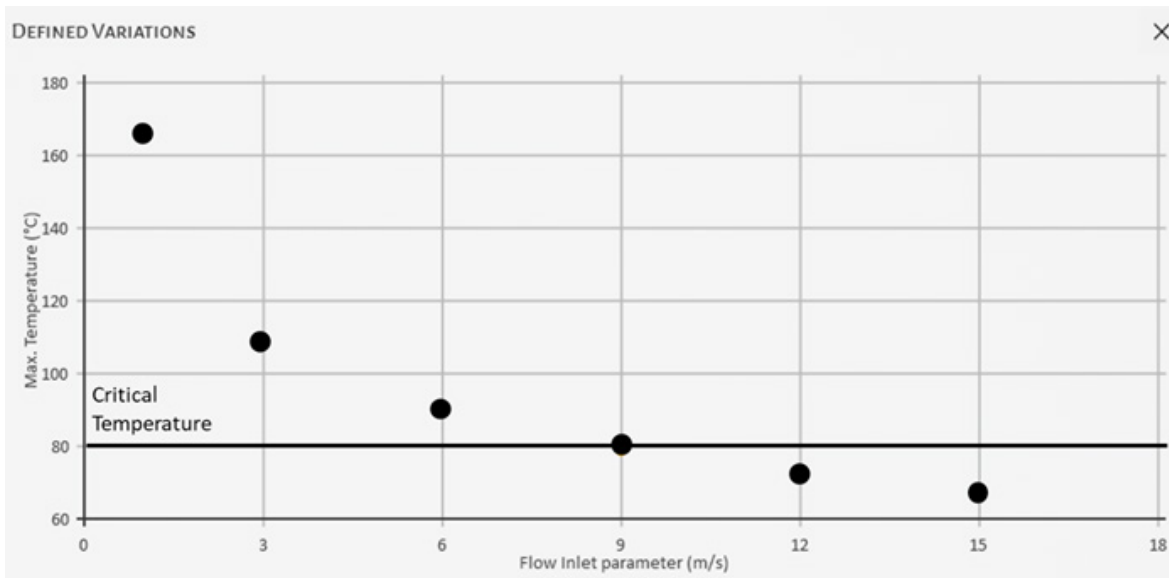


Figure 8. Result of the Parametric Analysis

6. Conclusions and Possible further Analyses

The use of heat sinks can guarantee thermal safety of CPUs in electronic devices. The design of these components can be performed through thermal simulation in Ansys Discovery and as shown in this case study, it was found that intense computation leads to unsustainable temperatures in the CPUs. The impact of the material and geometry on the performance of the heat sink was analyzed and while some improvements were observed, these were not sufficient to ensure thermal safety. Ultimately, it was determined that forced convection in the heat sink area is needed to ensure safe temperature margins. Moreover, it was discussed the need of conjugate heat transfer analysis to correctly capture the uneven temperature gradient in the assembly given by the directionality of the fluid flow. Lastly it was shown how simulation could be used to identify the requirements of the fluid flow such as speed (leading to fan selection) and choice of fluid. Additionally, it could be investigated how the positioning of the fan (i.e. inlet of the flow) affects the cooling of the heat sink or potentially if other type of fluids could provide better cooling (e.g. using liquids).

7. References

- [1] <https://doi.org/10.1145/1281700.1281716>
- [2] https://en.wikipedia.org/wiki/List_of_CPU_power_dissipation_figure
- [3] Data taken from Copper, cast (h.c. copper) and Aluminum 6061, T6 records in Ansys Granta Edupack Data 2022R2, Ansys Inc.

© 2023 ANSYS, Inc. All rights reserved.

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.

Document Information

This case study is part of a set of teaching resources to help introduce students to structures, fluids, or heat transfer (physics areas supported by Ansys Discovery).

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

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

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

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit **www.ansys.com** for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2023 ANSYS, Inc. All Rights Reserved.