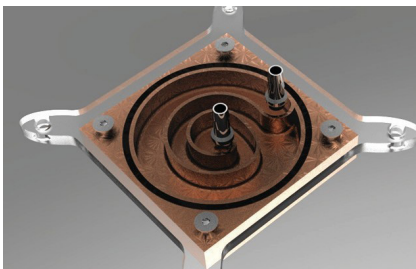


Ansys AIM Makes It Easy to Optimize CPU Cooler Design

Central processing unit (CPU) coolers are frequently used in custom-built or customized computers used by gamers and other computer performance enthusiasts. Accurate simulation of CPU coolers requires the capturing of fluid flow and the prediction of heat transfer at the boundaries between fluids and solid components. Unlike most other simulation tools targeted at design engineers, Ansys AIM includes inflation layers and conjugate heat transfer (CHT) — both of which are explained in more detail in the solution section below — in its multiphysics toolset, enabling design engineers to accurately predict heat transfer across fluid–solid interfaces without ever leaving the immersive AIM user interface that guides them through the simulation process.



Realistic rendering of CPU cooler model

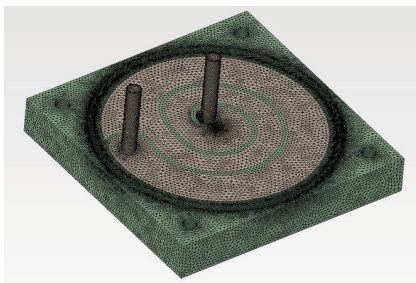
/ Introduction/Challenge

The CPU at the heart of every computer generates heat that must be removed from the chassis of the computer to prevent the CPU from overheating and eventually destroying itself. The faster computers are run the more heat they generate; the amount of heat that can be removed from the chassis is often the limiting factor on computing speed. The CPUs of mass market computers are typically cooled either by convection cooling – transferring heat from a power device by natural air flow – or forced air cooling, which uses a fan to blow air across a power device. Companies or enthusiasts who build high-performance computers need to find a way to remove the additional heat generated by their CPUs and graphics processors. Several companies have filled this void by developing more efficient forced air cooling systems and liquid cooling systems.

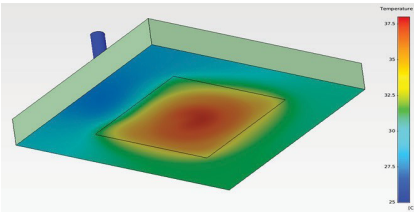
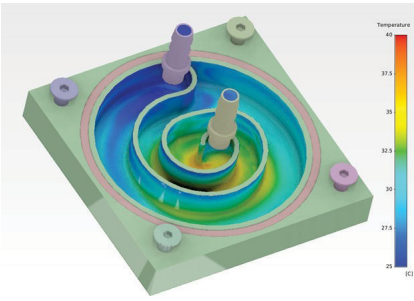
In designing liquid cooling systems, engineers need to track the flow of heat through the liquid coolant from the CPU to ambient temperature. Furthermore, in order to predict the temperature of the CPU itself, engineers also need to predict the temperature of metal components such as the base of the CPU. High-end computational fluid dynamics (CFD) packages use conjugate heat transfer (CHT) to predict the heat of metal components as part of a CFD simulation. But these software packages are usually too complicated to be run by design engineers, and instead are operated by analysts who are generally not available to the smaller companies that build CPU coolers. The alternative is to run a CFD analysis to generate temperatures at the fluid–solid interface and export these temperatures for use as boundary conditions to a thermal–mechanical simulation tool. This is a time-consuming process and requires purchasing and learning two different simulation software packages.

/ Solution

AIM incorporates the complete simulation toolset needed to design CPU coolers within a single immersive user environment that supports structural, electrical, fluid, thermal and electromagnetic physics. In this application, an engineer used Ansys SpaceClaim to produce the geometry for a water-based CPU cooler with a copper base and flow channel, plexiglass cover, aluminum flow connections and a rubber sealing gasket. The thermal contact areas correspond to the available cooling area defined in the Intel i7 processor Thermal Specification. The engineer used SpaceClaim's volume extraction function to rapidly generate the flow domain, and both the solid and flow geometries were imported into AIM. The engineer used the version of Keyshot 3D rendering and animation software that is included with SpaceClaim to produce a realistic rendering of the CPU cooler.



CPU cooler mesh including flow domain and solid components



Operating temperatures plotted on top and bottom of a CPU

The engineer meshed the geometry using pre-defined curvature and proximity settings and body size control to limit the maximum element size to 1 mm, which conserves computing resources over the majority of the mesh. He then increased the mesh density in the most critical section of the mesh by defining inflation layers that increase the cell count in the flow domain along the copper base to accurately represent the boundary layer flow. Inflation layers accurately capture the large changes in velocity, pressure and temperature in the direction normal to the wall that play a major role in heat transfer from a solid to an adjacent fluid. Templates can be used in AIM to guide the user through the process of establishing inflation layers as well as other meshing parameters.

Thermal performance is dominated by the flow domain plus thermal transfer through the copper base, so only those regions were included in the flow calculation. CHT was used to analyze the heat transfer between the solid and fluid domains, coupling conduction in the solid bodies with the convective and radiative heat transfer in the surrounding fluids. AIM includes fast and accurate temperature and fluid force mapping from a CHT solution to thermal-stress results as illustrated by the structural deformations induced from the temperature field in the CPU water cooler.

The flow conditions specified a mass flow of 0.015 kg/s at an inlet temperature of 25 C. The heat flow from the CPU was set at 140 W, the maximum heat dissipation of an Intel i7 processor, across the thermal contact area, while all other boundaries of the CPU were insulated.

The simulation predicted a maximum temperature at the CPU contact location of 37.1 C, well within the acceptable operating temperature range of the CPU. Likewise, the flow pressure drop of 1900 Pa was within the capabilities of the pump head used in this application. The simulation results could have been mapped onto the rendering of the CPU cooler to make them easier to understand, especially for managers and others who were not involved in the design process.

/ Results

As the first comprehensive integrated simulation environment directed at engineers rather than analysts, AIM reduces the number of tools, time and level of analysis expertise required to design a cutting-edge CPU cooler. This application demonstrates how AIM enables design engineers who are not simulation specialists to analyze the complete physics of a liquid CPU cooler to optimize its design. AIM's immersive user environment guided the engineer through the multiphysics workflows needed to get the design right the first time. The simulation results provided key insights into the design, such as the temperature of the CPU and the pressure drop of the cooling system. Although it was not necessary in this example, engineers can easily go one step further and define parametric values for geometrical dimensions and boundary conditions, and optimize the design based on these values.

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.

© 2021 ANSYS, Inc. All Rights Reserved.