Controlling temperature is critical to the efficiency, reliability, safety and durability of many products and processes. For example, running a chemical reactor at less than the appropriate temperature may reduce the product yield, while running at too high a temperature might cause a dangerous runaway reaction. Simulation can improve thermal management of products and processes by enabling engineers to understand the root cause of thermal problems so they can quickly correct them. Simulation also makes it practical to evaluate a wide range of alternative designs to optimize the design and ensure its safety under many different operating scenarios. Single physics solutions with relatively simple physical models are sufficient for some thermal management challenges. But many thermal management applications do not meet all of the assumptions on which the simpler models are based. Furthermore, many thermal management applications live at the intersection of multiple physics — for example, a case where heat from a fluid is transferred to a solid structure and later dissipated into the air — so they cannot be solved by single physics solutions.

This application brief will explain how accurate simulation of thermal management can enable your engineering team to quickly evaluate alternative product and process designs to increase efficiency, reliability, safety and durability in a competitive, time-critical environment. ANSYS provides more insightful solutions to a much wider range of thermal management problems. It does this by offering more powerful single physics solvers along with the ability to tie together multiple single-physics solvers to provide accurate answers to virtually any thermal problem. Thermal management is a critical computational fluid dynamics (CFD) app that you have to get right.

**ANSYS gives you the ability to delve into thermal performance to whatever depth you prefer**

Most products and processes have a thermal comfort zone — a range of temperatures in which they operate most efficiently. Operating outside this zone may cause problems such as failures of critical components or generation of thermal mechanical stresses that may lead to early failure. Investigating thermal effects in the laboratory is often not practical beyond a few sets of conditions, since it is time-consuming and costly to replicate complex thermal scenarios.
Simulating the thermal performance of a product early in the design phase can save large amounts of time and money by getting the design of the early prototypes right from a thermal management standpoint, thus reducing the need for additional prototypes that might otherwise be required to diagnose and correct thermal issues. Simple computational fluid dynamics (CFD) software can be used to analyze thermal issues such as determining how heat is transferred through a fluid. But many problems are more complex, such as those that involve multiple mechanisms of heat transfer, where heat is transferred through both solids and structures. Cases in which the fluids and structures involved in heat flow are closely coupled, so that thermal deflection of the structures affects the fluid flow, are also challenging. Engineers often need to understand how heat is transferred by a number of different mechanisms through a complicated interconnected system in order to understand how their product or process will perform under a given set of conditions.

ANSYS greatly simplifies the task of solving problems like these by providing best-in-class solvers for fluids, thermal properties, structures and electromagnetics, each of which provides the latest and most sophisticated physical models that make it possible to generate accurate solutions with the fewest possible assumptions. ANSYS also makes it easy to connect solvers together to perform fast, accurate, automated data exchange between meshes created for different physical domains. This makes it possible to follow the heat from wherever it is created to wherever it is transferred in order to generate more accurate solutions to a much wider range of problems than can be addressed with simpler single-physics solvers. These tools are seamlessly integrated within the ANSYS Workbench environment so engineers can simulate and diagnose thermal management problems ranging from the very simple to the most complex, with higher levels of fidelity, in less time than ever before.

Fast, accurate solutions for fluid-solid heat transfer problems with a single solver

Optimizing heat transfer between fluids and solids is critical in many types of industrial equipment such as pre-heaters, engine blocks, cylinder heads, turbine blades, chemical reactors and reformers. Conventional heat transfer analysis is often limited by dependence on wall correlations that are valid only for certain types of equipment and specific operating ranges; it does not provide the detailed understanding of local flow and heat transfer behavior required for designs to be truly optimized.

ANSYS users can define as many different physics in as many different domains as are needed to solve challenging engineering problems. For example, engineers can couple calculation of pure thermal conduction through solid materials with calculation of temperatures in working fluids using the
conjugate heat transfer (CHT) capability in ANSYS CFD software, eliminating the need for a separate structural model. A typical CHT simulation might involve three domains: a single-phase combusting air-fuel mixture outside a pipe, the pipe wall and multiphase flow of a liquid inside the pipe wall. CHT can model conduction through the wall. If the pipe is translucent, radiative heat transfer can also be modeled simultaneously among the three domains. If the pipe material is permeable, the diffusion of a contaminant or chemical could also be modeled.

The ability to use independent meshes in the solid and fluid domains saves time by making it possible to recycle existing models. Engineers can import an existing solid geometry mesh or create the geometry using the unstructured mesh generation capabilities in ANSYS SpaceClaim. Non-matching mesh technology is used to automatically connect fluid and solid interfaces so the fluid and solid meshes do not have to match node-to-node at the solid-fluid interface.

Siemens reduces wind tunnel testing time by 50 percent with thermal modeling

Regulations and customer demands put pressure on rail designers to deliver passenger coaches with comfortable climates. In the past, Siemens engineers spent about four months testing passenger coaches in a climate wind tunnel to validate the design of the heating, ventilation and air conditioning (HVAC) system. Now they use ANSYS CFD software to validate the HVAC design before building the first coach. CHT is used to model the boundary of the fluid interior to the exterior wall of the vehicle, and to predict surface temperatures of walls that may be touched by passengers, as well as channels that exchange heat with the inside of the car. The ability to accurately predict HVAC system performance with simulation enables Siemens engineers to get the design right the first time, making it possible to reduce the amount and cost of wind tunnel testing by 50 percent.
Generate more accurate thermal solutions by transferring thermal results to a mechanical solver

Solving thermal problems with CHT limits the engineer to the physical models that are contained within the CFD solver. More complex problems, such as those involving complex multimaterial structures or more accurate heat transfer coefficient (HTC) boundary conditions, can be solved by linking the CFD solver to a structural solver. ANSYS provides two different approaches: (1) transferring solid temperature fields from CFD CHT simulations to a structural thermal system, and (2) computing HTC values in CFD and passing them to a structural thermal system. Moving the thermal results calculated by CFD to a structural model makes it possible to put the full breadth of ANSYS Mechanical structural and thermal models to work. The temperatures or HTCs calculated by CFD, and also the surface loads if desired, calculated by the CFD solver are transferred across the physics interface through an ANSYS Workbench workflow, avoiding the need for complicated third-party integrations. The structural finite element analysis (FEA) code calculates the heat transfer and thermal fields in the structure as well as thermal–mechanical stresses. A characteristic of a one-way fluid structure interaction (FSI) solution is that the stresses and deformations calculated in the structural solver are not passed back to CFD to update the mesh and recalculate the flow.

Exhaust manifold heat transfer coefficient (HTC) values are computed in ANSYS CFD and passed to a thermal shell model in ANSYS Mechanical.

One-way coupled CFD and structural-thermal simulation solves cracking problem

An engine made by Ural Diesel-Motor Factory LLC that was designed several decades ago without the benefit of simulation experienced periodic cracking problems in the aluminum cylinder heads. The company’s engineers modeled the flow through the cooling jacket in ANSYS CFD; they also used ANSYS Workbench coupling to apply temperature fields determined by the CFD simulation in ANSYS Mechanical to calculate the thermal stresses associated with these temperatures. The calculated stresses were higher
than the yield strength of the aluminum material, explaining the cracking problem. Engineers changed the head material to cast iron, which has a considerably higher yield strength than aluminum. They performed the CFD simulation again and generated temperature fields for the modified head, then imported these temperature fields into ANSYS Mechanical. Engineers also used the ANSYS shape optimization module to reduce the mass of the head. Finally, they used ANSYS nCode DesignLife fatigue analysis software to calculate high-cycle fatigue safety factors. The new design delivers the long life and high quality that the company’s customers have come to expect.

Determine the impact of thermal deformation on flow with two-way coupled FSI

Applications such as leakage paths and thin film flows where thermal-stress induced structural deformation affects fluid flow present an even more difficult simulation challenge. These applications can be simulated with two-way coupled CFD and structural thermal simulation. With this approach, the fluid flow solution is applied to the structure and the deformation of the structure is in turn applied to the fluid flow at each time step of the simulation. ANSYS Mechanical uses coupled field elements to solve both thermal and structural degrees of freedom. The fluid mesh is automatically deformed or remeshed to account for deformation in the structural domain. Native Workbench system coupling enables engineers to easily set up coupled simulations using an intuitive project schematic. Once the participant solvers are fully set up and connected to system coupling, the remaining setup for the coupled CFD and structural thermal simulation — which includes analysis controls, data transfers and execution controls — takes place within the intuitive system coupling user interface.

Two-way coupled CFD and structural–thermal simulation of an exhaust manifold

Optimizing the design of an exhaust manifold is a challenging multiphysics design problem. Both structural and fluid heat transfer must be known to estimate cooling needs and predict the life of the components. Substantial improvements in the design of the exhaust manifold shown above were achieved by using two-way thermal FSI to accurately predict the temperature field of the structure. Two-way CFD and structural thermal simulation made it possible to set up a transient simulation that simultaneously solves the heat transfer coefficients (HTCs) and near wall temperatures in the fluid. The ANSYS CFD simulation modeled the CO2, CO, H2O and N2 mixture and calculated porous loss for the catalyst zone. The model used the Shear Stress Transport (SST) turbulence model. This information was then trans-

CFD coupled with structural simulation was used to calculate thermal stresses to diagnose a cracking problem in this aluminum cylinder head.

Simulation accurately predicted the exhaust manifold temperature by taking into account both structural and fluid heat transfer.
Thermal Management: A Critical CFD App You Have to Get Right

ferred to ANSYS Mechanical to solve for temperatures in the structure. A shell model was used and both convective and radiative heat transfer were modeled to determine heat loss to the surroundings. The wall temperatures calculated in ANSYS Mechanical were then transferred back to ANSYS CFD to update the fluid simulation. These calculations and data exchanges were repeated until a steady state was achieved. ANSYS streamlines the exhaust manifold design process by providing all the physical models and simulation tools needed to fully diagnose and optimize a heat exchanger within a single user interface.

Engineers achieve more accurate thermal results with ANSYS fluid and structural solutions

Thermal management is a concern for most products and processes developed today. Failure to properly manage thermal performance can lead to inefficient energy use, uncomfortable or even unsafe temperatures, suboptimal performance and lower than expected product life. Engineering simulation is essential to diagnosing and evaluating solutions for thermal problems early in the design process when they can be corrected at the least possible cost and time. To accomplish these goals, engineers need to be able to accurately predict thermal performance across a wide range of operating conditions. ANSYS is continually advancing its core CFD and structural solver capabilities to deliver the solution fidelity, robustness, usability, speed and optimization capabilities required to address the most difficult thermal design challenges. The complete ANSYS thermal simulation toolset is continually validated and revalidated to ensure that it solves the right equations while minimizing numerical, modeling and systematic errors.

ANSYS CFD software has been validated against more than 385 test cases using realistic geometry and a combination of physical models so you can be confident that your simulations will accurately reflect the real world. The test case shown above uses CHT to model natural or free convective heat transfer where heat is transferred between a solid surface and a fluid moving over it. Heat transfer in free convection situations depends on surface temperature, ambient temperature, air properties and the geometry of the heat transfer surface. This test case focuses on free convective and conjugate heat transfer characteristics of a vertical, isothermal plate contained within a naturally ventilated, rectangular cavity. The goal of the simulation was to predict the Nusselt number (Nu), the ratio of convective to conductive heat transfer across or normal to the boundary. The simulation results for a range of different cases correlated well with physical experiments.
High-Performance Computing (HPC) enables large, high-fidelity models to be solved

Thermal simulation requires substantial amounts of computing power because it frequently requires the use of complex physical models and multiple solvers that cross physical domains. The required level of computing power is further increased in applications where design optimization methodology is used to explore how various design alternatives will behave across a range of real-world operating conditions in the process of iterating to an optimized design. ANSYS High-Performance Computing (HPC) makes it practical to evaluate larger, higher fidelity models with more complex physics at so that you can gain insight at every stage of the product development process. ANSYS optimizes the entire HPC workflow by providing a unified environment – across solver components – for defining, submitting and monitoring parallel workloads.

ANSYS experts will transfer their knowledge and experience to you

Recent improvements in parallel scalability speed solution times for a 4-million-cell model

Recent improvements in parallel scalability speed solution times for a 4-million-cell model

ANSYS experts are well-equipped to help you overcome thermal management challenges

Experts from the ANSYS Customer Excellence (ACE) team are dedicated to helping you get the most from our advanced technology solutions. The ACE team includes over 450 Ph.D.s who have solved a vast array of engineering simulation problems so they are well-equipped to help you overcome whatever challenges you are facing. An ANSYS Mentoring Expert will work with you on a short-term basis to coach you on how to optimize simulations to ensure accurate results that match your product. The ANSYS Learning Hub provides an easy way to access a broad range of courses and other resources that cover most types of simulation targeted at improving your team’s ability to make more effective use of simulation.
Accurate simulation can help you improve the thermal performance of your products

From automobiles to high-speed semiconductors, thermal management concerns can often lead to wasted energy, reduced performance or premature device failure. This results in increased energy costs, warranty issues and additional design time that can reduce your product's competitive advantage. Simulation can help your engineering team look deeper into thermal design issues to see where heat is being generated and how it might be impacting surrounding materials and components. Simulation enables your engineers to evaluate more options more thoroughly than traditional prototype-based design and development methods. They can confidently determine if cooling or additional heating is needed and virtually explore ways to optimize designs to meet thermal management project goals all before building a physical prototype. ANSYS thermal simulation tools provide the wide array of physical models and integration between different physical domains that you need to get thermal management right.