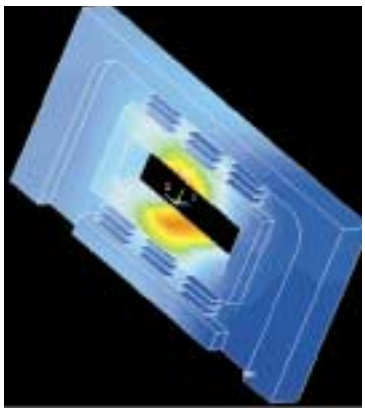
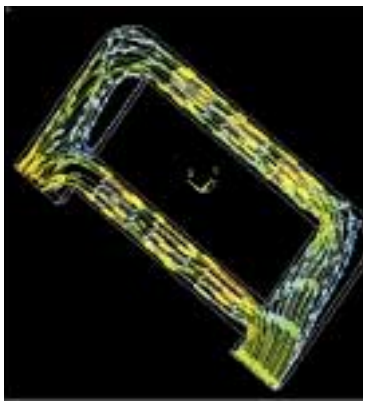
FLOTRAN Chills Out

Hot Aerospace Electronics

CFD tool determines thermal behavior in delicate equipment long before physical prototypes are built.





Water-cooling simulation of an electronic product. Shown are the temperature distribution within a heating panel (above) and the flow traces of the cooling water. Temperature error between test and analysis was less than 8 percent.

Mark Troscinski
Multiphysics Product Manager
ANSYS Inc.

Jiang Guangnan
Zhang Minyi
ANSYS/China

Modern electronic product designs for aerospace applications are driven to meet stringent weight and envelope restrictions. At the same time, the electric power requirements for these products have steadily risen, leading to increased waste heat generation rates. Typically, the equipment must function within wide variations in ambient temperature or withstand harsh environmental conditions. Often these systems are hermetically sealed to protect against contaminant ingress and other heat sources, and cannot be operated above 100 degrees Celsius. Based on U.S. Air Force statistics, more than 50 percent of electronic product failures are caused by thermal-related effects. Therefore, a sound thermal design is vital to ensure high product reliability.

For many years, empirical methods were exclusively relied upon for thermal designs, based on conservative, semirational formulae taken from thermal design guides. Physical prototypes had to be built and tested, and a design's success often depended on the experience level of the engineer. Time and cost constraints would often limit the number of candidate prototypes for testing, and the real optimum design was overlooked. If unforeseen problems surfaced after testing, the design had to be modified and retested, greatly prolonging the design cycle. With the serious competition of today's electronic product marketplace, designers truly feel pressure to "get it right the first time." A better approach is needed to enable rapid evaluation of design alternatives and dramatically reduce the overall design cycle time.

For many electronic devices and systems, the flow path for coolant is very complex, and stagnation and recirculation regions often characterize the resulting flow patterns. Quantifying local heat transfer effectiveness by traditional methods has truly become difficult to practically impossible. However, with today's advancements in computer hardware performance and capacity, computational fluid dynamics (CFD) simulation tools have helped revolutionize thermal design methods for electronics products. CFD can reveal comprehensive information on flow velocity, pressure and temperature in a range of designs well before the physical prototype stage. It allows thermal design engineers to solve practical flow problems within reasonable time and cost constraints.

Inside the CFD Box

The equations that mathematically describe general fluid motion govern conservation of mass (continuity), momentum (Newton's 2nd Law) and energy (1st Law of Thermodynamics). The Navier-Stokes (N-S) equations are a special form of the momentum equations for Newtonian viscous flows. These non-linear, second-order,

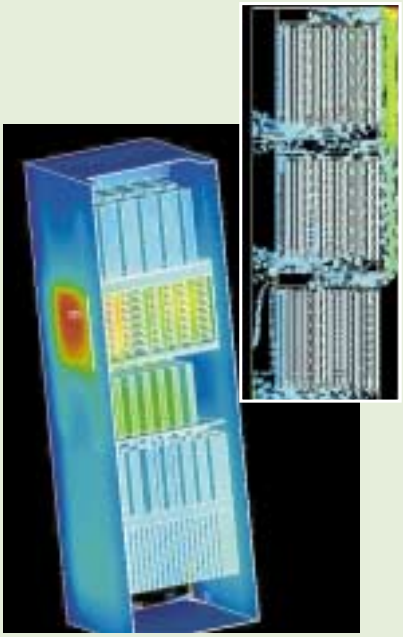
Electric power requirements for these products have steadily risen, leading to increased waste heat generation rates.

partial differential equations form the mathematical foundation of viscous flow theory and CFD. An analytically derived "exact" solution to a flow problem would yield a continuous description of the field variables (i.e., the velocity components, pressure and temperature) throughout the region of flow. However, the equations of fluid motion are highly non-linear, and closed-form solutions exist for only a handful of idealized cases. Instead, engineers must rely on computer-based flow simulation models that calculate approximate values of the field variables at discrete points in the flow. Solutions for complex flows of engineering importance are possible only through the numerical techniques of CFD.

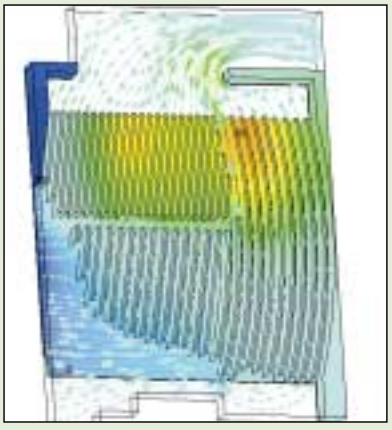
A CFD computer model represents a geometric region of interest called a flowfield, whose flow conditions are known or assumed at its boundaries and whose detailed internal flow characteristics are sought. A flowfield may be characterized as either an internal flow within solid walls or an external flow around an immersed body. CFD simulations may be steady-state or time-dependent. They may also include the effects of conduction and radiation heat transfer, turbulence, compressibility, buoyancy, multiple species transport, non-linear material properties and other more exotic effects. Depending on the existence of flow symmetry, CFD models may either be 2-D planar, 2-D axisymmetric or fully 3-D.

Finite element CFD modeling requires the flowfield to be discretized, i.e., spatially subdivided into a computational mesh consisting of individual elements interconnected at nodes. Common element shapes include quadrilaterals and triangles for 2-D problems; hexahedra or bricks, tetrahedral wedges and pyramids for 3-D applications. Elements contain interpolating shape functions, an assumed form of the field variable distributions within the element. Typically, polynomial functions of low degree are used because they are easy to integrate and differentiate. In general, the degree of the polynomial depends on the number of nodes assigned to each element and certain continuity requirements imposed at the nodes and along the element boundaries. Shape function equations yield a constant value throughout the element when the nodal values are numerically identical. They also ensure that the field variables are continuous across adjoining element boundaries.

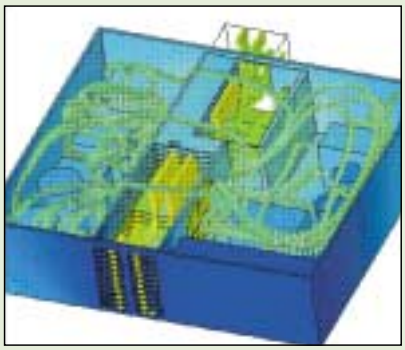
Solving partial differential equations on a computer requires that they be discretized, or converted to algebraic form. Numerical discretization techniques are used to approximate the partial derivative terms in the governing equations with pseudoequivalent algebraic expressions. In the CFD finite element method, the N-S system of partial differential equations is reduced to a finite system of algebraic equations and then solved using matrix solution techniques. Finite element discretization equations are formed in an element-by-element fashion, independent of the global mesh. Each element in the flowfield represents a separate subdomain over which the N-S equations are integrated. The element equations are termed matrix equations because they are collectively assembled in matrix form for simultaneous solution. For each element, these equations express the integrated form of the discretization equations in terms of nodal values of the field variables. The individual element matrix equations are then combined to form the system matrix equations



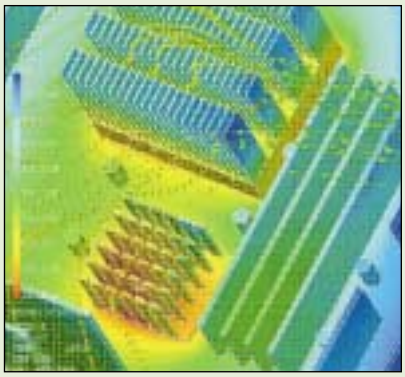
Conjugate heat transfer analysis of an airborne electronic product.



Temperature distribution of an electronic heat sink.



Forced-air cooling simulation of an airborne electronic equipment box.



Forced convection flow and conjugate heat transfer within detailed PCB components.

that express the behavior of the entire flowfield. Their assembly follows the topological configuration of the elements in the global system.

Iterative solution techniques are generally employed when solving CFD problems. An iterative method begins with an initial approximation to the true solution and attempts to develop a convergent sequence of approximations in a computational cycle. Convergence is reached if the change in the solution affected in two successive cycles is smaller than a specified tolerance. Segregated solution schemes solve each field variable matrix sequentially. Only one field variable is considered dependent in each equation, while all others are treated as known. A single solver iteration is a loop or sweep through the system of equations that pertain to a particular dependent field variable. The global iteration loop includes solver sweeps for all dependent variable equations. If a transient simulation is performed, a time-step loop exists around the global iteration loop. As global iterations elapse, the fluid elements in the flowfield's interior increasingly "feel" the effects of the boundary conditions. Solution convergence monitors indicate the normalized average change in each field variable between successive global iterations. The solution histories of the average, minimum and maximum values of each field variable are also useful tools for judging convergence.

Sound thermal design is vital to ensure high product reliability.

In broad terms, the finite element CFD model collectively solves the governing N-S equations within each element to determine the nodal values of the field variables. Once determined, the nodal values of the field variables and the element shape functions completely define the global behavior of the flowfield throughout the assemblage of elements. The ANSYS/FLOTRAN CFD software solves the governing Navier-Stokes equations for viscous fluid flow using the finite element numerical method. Conservation of mass, momentum and energy is solved within each fluid element, and for conjugate heat transfer problems, the energy equation is solved simultaneously within all solid regions as well.

Keeping It Cool

The cooling effectiveness of electronic products depends on the magnitude of the convective heat-transfer rate at fluid-solid interfaces. The term "convection" describes energy transfer between a surface and a fluid moving over the surface, when the surface and fluid are at different temperatures. And although diffusion mechanisms also contribute to this energy transfer, the dominant contribution typically comes from bulk fluid motion. As a matter of long-standing convention among thermal design engineers, convection rates (or heat fluxes) are often expressed in terms of effective film coefficients. The local film coefficient is expressed as the ratio of the local surface heat flux to the difference between the surface temperature at that location and some "bulk" fluid temperature. From a CFD modeling perspective, accurate predictions of local film coefficients are only achievable if the fluid velocity gradients near the surfaces are accurately captured, typically requiring strong mesh gradation near all wall boundaries in the model.

Since convection heat transfer is manifested by the boundary layers that develop along the surface, convection rates depend on a multitude of factors, including surface geometry, flow conditions and fluid properties. Specifically, a velocity boundary layer is characterized by the presence of velocity gradients and shear stresses, while temperature gradients and heat transfer characterize a thermal boundary layer. Surface friction and convection transfer rates are strongly dependent on whether boundary layers are laminar or turbulent. In the laminar boundary layer, fluid motion is highly ordered, and it is even possible to identify streamlines along which particles move. In contrast, fluid motion in the turbulent boundary layer is highly irregular and characterized by velocity fluctuations that enhance the transfer of momentum, energy and, hence, increase surface friction and resulting convection transfer

rates. In turbulent flow, the main flow stream is superimposed with small-scale rotational motions called eddies. Turbulent eddies have a wide range of sizes, or length scales, bounded from above by characteristic flowfield dimensions and from below by viscous diffusion mechanisms. Eddies continually agglomerate and disintegrate within one or two oscillation periods and, hence, have small time scales.

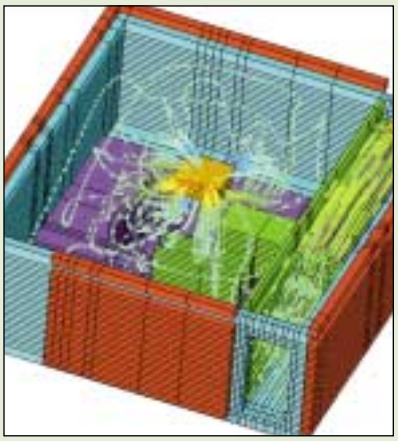
For most practical flow problems, present-day computers are not large and fast enough to solve the 3-D time-dependent N-S equations directly for all the important scales of turbulence. With today's commercial CFD software, turbulence itself is not directly computed, but rather its average effect on the mean flow is modeled using statistical techniques. This requires describing turbulent motion in terms of time-averaged quantities rather than instantaneous values. To describe a turbulent flow in mathematical terms, it is convenient to separate it into a mean flow and into a fluctuation, or eddying motion. Each field variable is now expressed as the sum of a mean component plus a fluctuating component. The fluctuation is the deviation from the mean value; its time-averaged value is zero. When the turbulent field variables are substituted into the governing equations, the equations are then time-averaged to yield the Reynolds-averaged equations for turbulent flow. These equations represent the time-averaged balances of mass, momentum and energy. An average scale of turbulence is substituted for the broad range of instantaneous scales encountered in actual turbulent flows.

Each component equation of the Reynolds-averaged N-S (RANS) equations contains mean velocity values plus three mean products, or correlations, of fluctuating velocities. These correlation terms, nine in total, represent the transfer of fluid momentum due to velocity fluctuations. They are known as the Reynolds stresses because they have the mathematical effect of stress. A RANS solution requires that relationships be provided for these correlations. The role of a turbulence model is to relate the Reynolds stresses to the mean-flow variables by using statistical methods and empirical data.

The k-ε turbulence model permits prediction of both near-wall and free-shear flow phenomena without adjustment to empirical constants or functions. Its premise is that turbulence behaves much like molecular diffusion, and momentum exchange arising from turbulence can be modeled through local addition of "extra" viscosity, termed the *turbulent eddy viscosity*. The eddy viscosity concept relates the Reynolds stress terms to the mean-flow strain rate in a fashion similar to molecular viscosity for laminar flows. Eddy viscosity is not a fluid property, but rather it is an entity that characterizes the statistical properties of turbulent fluid motion. In the k-ε turbulence model, a scalar eddy viscosity is obtained by solving additional transport equations for the kinetic energy of turbulence, k, and its dissipation rate, ε. Use of a scalar eddy viscosity assumes all turbulence is locally isotropic.

To properly resolve turbulent boundary layers with a numerical scheme requires an extremely fine near-wall mesh. The k-ε model utilizes wall functions to bridge the whole of the boundary layer to the fully turbulent freestream region. With wall functions, the Law of the Wall very near the wall and the Log-Law of the Wall further away are used to determine the wall velocity profile and shear stress. Wall functions reproduce the "logarithmic velocity profile" when uniform shear stress prevails, and energy generation and dissipation rates are in balance.

Based on U.S. Air Force statistics, more than 50 percent of electronic product failures are caused by thermal-related effects.



Fan model used in electronic system cooling of telecommunications cabin.