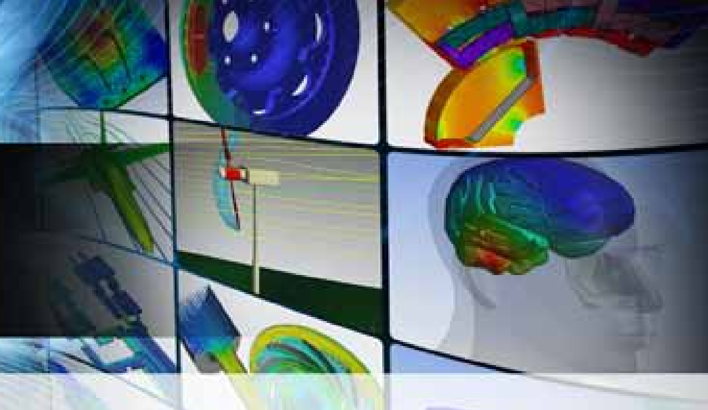
ANSYS® RELEASE 13.0



ANSYS Icepak

Powerful Fluid Dynamics Software for Thermal Management of Electronic Systems

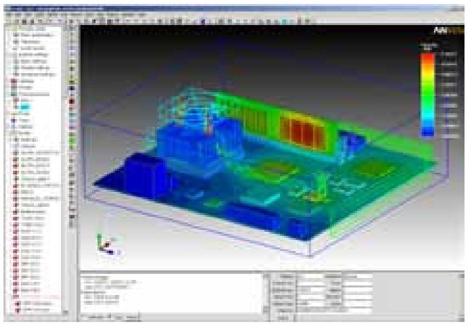
Leading companies around the world trust ANSYS® Icepak® software to provide robust and powerful computational fluid dynamics technology for electronics thermal management. In today's electronic devices, power requirements and smaller footprints require superior thermal designs. Overheating of electronic components degrades product performance and reliability, which results in costly redesigns. To ensure the adequate cooling of IC packages, printed circuit boards and complete electronic systems, engineers rely on ANSYS Icepak software to validate their thermal designs before building any hardware. ANSYS Icepak combines advanced solver technology with robust, automatic meshing to enable engineers to rapidly perform heat transfer and fluid flow simulation for a wide variety of electronic applications including computers, telecommunications equipment, semiconductor devices, aerospace, automotive and consumer electronics.

Rapid Thermal Simulation for Electronic Systems

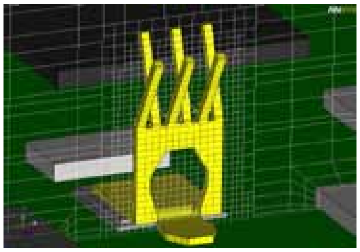
ANSYS Icepak contains a streamlined user interface that allows users to quickly create and simulate electronics cooling models of IC packages, printed circuit boards and complete electronic systems. Electronics cooling models are created by simply dragging and dropping icons of familiar predefined objects — cabinets, fans, packages, printed circuit boards, grilles, heat sinks, etc. — to create models of complete electronic systems. These smart objects capture geometric information, material properties, meshing parameters and boundary conditions — all of which can be parametric for performing sensitivity studies and optimizing designs. ANSYS Icepak contains extensive libraries of standard electronic components that can be used to further accelerate the development of thermal designs.

Accurate Thermal Analysis for Printed Circuit Boards

With ANSYS Icepak, users can import electronic CAD data from popular EDA layout tools for thermal simulation of printed circuit boards. Board dimensions, component layout information, and



Cut-plane velocity vectors and temperature contours for a fan-cooled rack-mounted computer



Multi-level hex-dominant mesh ("hanging-node") on a sheet metal heat sink represented directly as CAD geometry

Product Features

Model Building

- Predefined objects
  - Cabinets, fans, blowers, printed circuit boards, packages, grilles, openings, plates, walls, ducts, sources, resistances, blocks, compact and detailed heat sinks
- Object shapes
  - Blocks, cylinders, ellipsoids, elliptical cylinders, concentric cylinders, and prisms
- Direct CAD representation of components
- Rectangular or circular fans with hubs, guards and power specifications
- Thermal network modeling
  - Packages, heat exchangers, heat pipes and cold plates
- Object libraries
  - Fans, heat sinks, thermoelectric coolers, filters, packages and user defined libraries
- Comprehensive solid and fluid material property database
- Flexible and customizable units
- Parametric geometry and boundary conditions
  - User-defined trials
  - Design optimization with ANSYS Iceopt
- Import/export geometry to spread sheets
- Model summaries in HTML format
- User-defined macros for model creation

Electronic and Mechanical CAD Import

- IDF import for PCB layout
- MCM/BRD import for PCB traces and vias
- MCM import for packages
- Ansoft neutral file (ANF) import
- Gerber import with ANSYS Icegrb
- Cadence, Mentor Graphics, Synopsys and Zuken import with AnsoftLinks

Product Features

- IGES, STEP
- ANSYS® DesignModeler™ export to ANSYS Icepak objects
- Support for ANSYS Geometry Interfaces with ANSYS DesignModeler

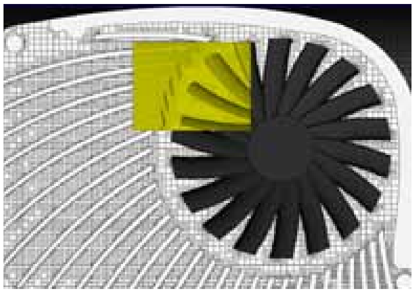
Automatic Mesh Generation

- Automatic hex-dominant meshing for true geometry representation
  - Automatic multi-level meshing
  - 2-D and 3-D cut cell techniques
  - Automatic non-conformal regions
  - Robust meshing of CAD geometry
- Unstructured hexahedral meshing
- Cartesian meshing
- Non-conformal regions
- Embedded non-conformal meshing
- Extruded meshes for packages and boards
- Coarse mesh option for first-pass analysis
- Mesh quality evaluation tools

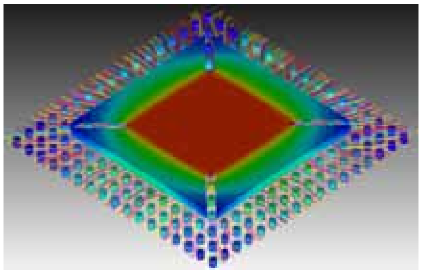
Boundary Conditions

- Temperature, heat flux, convective heat transfer coefficient, radiation, symmetry and periodic boundary conditions for walls and surfaces
- Inlet and outlet velocity, mass flow rate, outlet static pressure, inlet total pressure, inlet temperature and turbulence parameters for openings and vents
- Profiles of velocity, temperature, heat flux and heat transfer coefficients on openings and walls
- Grilles and resistances with automatic loss coefficient based on free area ratio
- Fans with options for mass flow rate or fan performance curve
- Rotational speed for cylindrical and circular objects
- Recirculating boundary conditions for external heat exchangers
- Time-dependent and enhanced temperature-dependent sources
- Time-varying ambient temperature
- Automatic correlation-based heat transfer coefficient boundary conditions
- Time-dependent pressure
- Electric current and voltage
- Power map import from IC package and PCB design tools
- Transient boundary condition import from spreadsheets

electronic trace and via information can all be imported into a thermal simulation. Using the trace and via information, a detailed thermal conductivity map of the board can be computed based on the copper content of the board layers. This allows the engineer to accurately represent the orthotropic thermal material properties of the board, which provides an increased fidelity in the prediction of the internal temperatures and component junction temperatures. Resistive heating in the individual traces carrying significant current can be modeled to further increase the accuracy of simulations.



Multi-level hex-dominant mesh (“hanging-node”) of a heat sink and fan assembly represented directly as a CAD geometry



Temperature contours on a 272-pin ball grid array (BGA) package on a substrate

Detailed and Compact Thermal Models for IC Packages

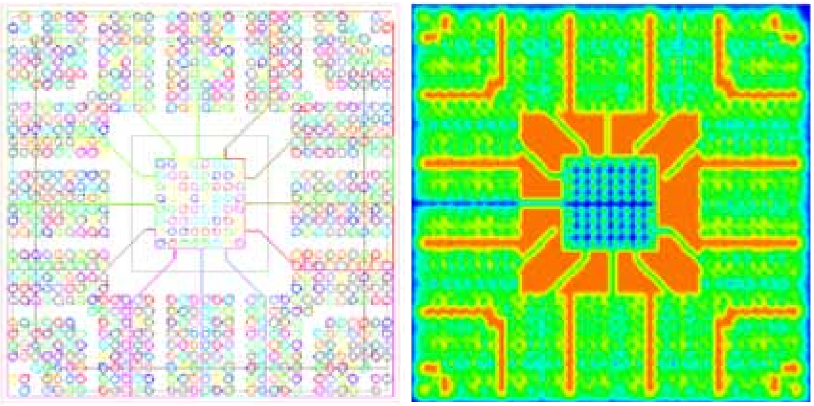
ANSYS Icepak includes options for both detailed and compact thermal modeling of IC packages. Based on electronic CAD data from EDA layout tools, users can import information such as substrate traces and vias, bond wires, solder bumps, die dimensions and solder balls into a detailed thermal model of an IC package. Using the substrate trace and via information, detailed thermal conductivity maps for the package substrate can be developed to accurately represent the orthotropic thermal material properties of the substrate. From a detailed package model, ANSYS Icepak contains an automated process for DELPHI package characterization (JEDEC compliant). The optimized DELPHI network model can easily be included in a system-level thermal simulation, which allows the engineer to accurately predict junction temperatures of IC components.

Flexible Automatic Meshing Technology

ANSYS Icepak contains advanced meshing algorithms to automatically generate high-quality grids that represent the true shape of electronic components. Options include hex-dominant, unstructured hexahedral and Cartesian meshing, which enable the engineer to automatically generate body-fitted meshes with minimal intervention. The user can localize the mesh density through nonconformal mesh interfaces, which allows the inclusion of a variety of component scales within the same electronics cooling model. While fully automated, ANSYS Icepak also contains many mesh controls that enable the customization of meshing parameters to refine the mesh and optimize trade-offs between computational cost and solution accuracy. The meshing flexibility of ANSYS Icepak offers the fastest solution times possible without compromising solution accuracy.

Robust and Rapid Numerical Solutions

ANSYS Icepak uses state-of-the-art technology available in the ANSYS® FLUENT® computational fluid dynamics (CFD) solver for the thermal and fluid-flow calculations. ANSYS Icepak solves fluid flow and includes all modes of heat transfer - conduction, convection and radiation - for both steady-state and transient thermal-flow simulations. The solver uses a multi-grid scheme to accelerate solution convergence for conjugate heat transfer problems. The ANSYS Icepak solver provides complete mesh flexibility, and allows users to solve even the most complex electronic assemblies using unstructured meshes, providing robust and extremely fast solution times.



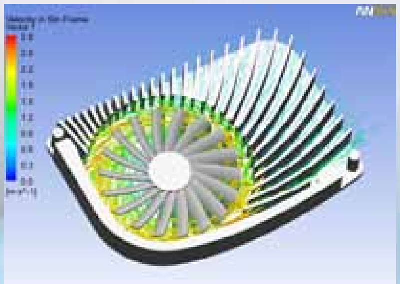
Detailed thermal conductivity map for an IC package substrate layer computed based on the electronic trace and via information

Results Visualization and Reporting

For post-processing, ANSYS Icepak contains a full suite of qualitative and quantitative tools to generate meaningful graphics, animations and reports. These can be used to easily convey simulation results to colleagues and customers. Visualization of velocity vectors, temperature contours, fluid particle traces, iso-surface displays, cut-planes and XY plots of results data are all available for use in interpreting the results of electronics cooling simulations. Customized reports, including images, can be automatically created for distributing results data, identifying trends in the simulation, and reporting fan and blower operating points. ANSYS Icepak includes ANSYS CFD-Post software for further post-processing of your results with advanced post-processing, graphics and animation tools.

Interfaces to Electrical and Mechanical Simulation

ANSYS Icepak provides interfaces to SIwave™ and ANSYS® Mechanical™ products, thus providing access to a full suite of tools to address electrical, thermal and structural simulation requirements. Based on an SIwave analysis, the DC power distribution profile can be imported into ANSYS Icepak to account for heating due to copper resistive losses. The coupling between SIwave and ANSYS Icepak enables users to predict both internal temperatures and accurate component junction temperatures for printed circuit boards and packages. Following



Velocity vectors for a heat sink-fan assembly, fan modeled using moving reference frame (MRF) fan model, image created using ANSYS CFD-Post

Product Features

Comprehensive Thermal Flow Modeling

- Steady-state or transient analysis
- Laminar or turbulent flows
  - Laminar regions in turbulent models
- Forced, natural and mixed convection
- Conduction in solids
- Conjugate heat transfer
- Radiation heat transfer
  - Surface-to-surface radiation
  - Discrete-ordinates radiation
  - Ray tracing radiation
  - Solar loading
- Volumetric resistances and sources for velocity and energy
- Joule heating in traces and conductors
- Thermal network modeling

Advanced Physical Models

- Zero-equation turbulence model
- Two-equation k-ε turbulence model
- RNG k-ε turbulence model
- Realizable k-ε turbulence model
- Spalart-Allmaras turbulence model
- Ideal gas law
- Anisotropic thermal conductivity for solids
- Temperature-dependent materials
- Contact resistance models
- Non-isotropic volumetric flow resistance
- Nonlinear fan curves
- Moving reference frame (MRF) fan
- Automatic radiation view factor computation
- Two-resistor, star and DELPHI network models for IC packages

Solver Attributes

- ANSYS FLUENT technology
  - Robust convergence for laminar and turbulent flows
- First-order upwind or higher-order scheme
- Automatic under relaxation
- Advanced stabilization methods
- Variable time stepping for transients
- Parallel solver available with ANSYS HPC
- Batch queuing
- Graphical convergence monitoring

Product Features

Results Visualization and Reporting

- Interactive, object-based visualization of results
- Contour and vector displays, cut planes, particle traces and iso-surfaces
- Point probes with XY plotting
- Animation of particle traces
- Animation of vectors, contours and cut planes
- Point objects
  - Solution convergence monitoring, post-processing, and reports
- Report generation
  - Solution overview
  - Trials report and plots
  - Power and temperature limits
  - Fan and blower operating points
- Time history displays
- Export results to ANSYS CFD-Post
- Export temperature data to ANSYS Mechanical

Online Help and Documentation

- Context-sensitive help
- Tutorials and validation examples

Supported Hardware*

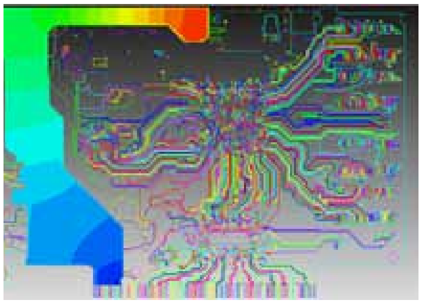
- HP-UX 11i v2 (B.11.23) 64-bit
- Linux Redhat (5), Suse Linux ES (10,11) 32-bit
- Linux Redhat (5), Suse (11) 64-bit
- Windows 7, XP, 2008 Server, Vista 32-bit
- Windows 7, XP, 2008 Server, 2008 Server (HPC), Vista 64-bit

Additional Modules

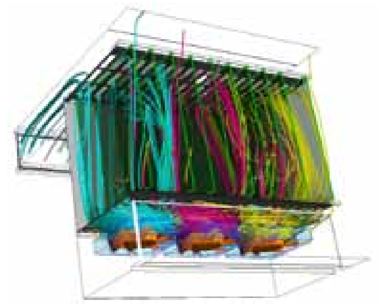
- ANSYS HPC
- ANSYS DesignModeler
- ANSYS Geometry Interfaces
- AnsoftLinks
- ANSYS Iceopt
- ANSYS Icegrb

* Refer to www.ansys.com for a current list of supported hardware and operating systems

an ANSYS Icepak simulation, a user can export the temperatures from a thermal flow simulation into ANSYS Mechanical using the ANSYS® Workbench™ platform. The coupling between ANSYS Icepak and ANSYS Mechanical enables the evaluation of temperatures and resulting thermal stresses of electronic components via an integrated set of software tools.



Electric potential contours on a printed circuit board trace, electronic trace information imported from electronic CAD



Velocity streamlines colored by fan for a card array in a VME format box cooled by three axial fans, image created using ANSYS CFD-Post

The ANSYS Advantage

With the unequalled depth and unparalleled breadth of ANSYS engineering simulation solutions, companies are transforming their leading-edge design concepts into innovative products and processes that work. Today, almost all of the top 100 industrial companies on the "FORTUNE Global 500" invest in engineering simulation as a key strategy to win in a globally competitive environment. They choose ANSYS as their simulation partner, deploying the world's most comprehensive multi-physics solutions to solve their complex engineering challenges. The engineered scalability of solutions from ANSYS delivers the flexibility customers need, within an architecture that is adaptable to the processes and design systems of their choice. No wonder the world's most successful companies turn to ANSYS — with a track record of 40 years as the industry leader — for the best in engineering simulation.

DASSAULT SYSTEMES Software Partner

CAA V5 BASED

ANSYS®

www.ansys.com

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

Toll Free U.S.A./Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com

GSA Contract Holder