ANSYS® Icepak®

12.1

12.1 RELEASE

ANSYS Icepak Features

Model Building Features

- Object-based modeling with predefined objects
  - Cabinets, blocks, fans, blowers, printed circuit boards, racks, vents, openings, plates, walls, ducts, sources, resistances, compact and detailed heat sinks
- Comprehensive object shapes
  - Blocks, cylinders, ellipsoids, elliptical cylinders, concentric cylinders, prisms and ducts of arbitrary cross-section
- Rectangular or circular fans with hubs, guards and power specifications
  - Specify operating and nominal RPM to update fan curve
- Polygonal and circular shapes for fans, vents, resistances, plates, openings and network objects
- Object libraries
  - Fans, heat sinks and thermoelectric coolers
- Library functions to store or retrieve groups of objects
- Graphical model management
- Material property database
- Undo and redo functions
- Error and tolerance checking
- Flexible and customizable units
- Graphical alignment tools
- Parametric geometry and boundary conditions
  - User-defined trials
  - Design optimization module
- Geometry import from MCAD/ECAD:
  - IGES, STEP
  - IDF wizard to import EDA PCB layout
  - MCM/BRD import for trace and via import
  - MCM import for BGA, Flip Chip and MCM packages
  - TCB neutral file ECAD import
  - Gerber file for trace and via import
  - Import and export geometry information to spread sheets
- Model summaries in HTML format
- User-defined macros
- Power map import from IC design tools

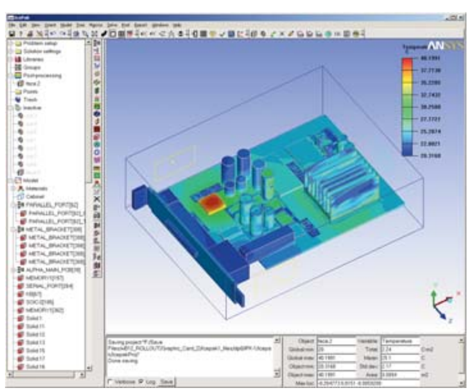
Powerful Fluid Dynamics Software for Thermal Management of Electronic Systems

For more than a decade, leading companies around the world have relied on ANSYS® Icepak® technology to provide robust and powerful fluid dynamics software for electronics thermal management. The electronics cooling capabilities of ANSYS Icepak software have set the standard for speed, accuracy and ease of use. By predicting fluid flow and heat transfer at the component, board or system level, the software improves design performance, reduces the need for physical prototypes and shortens time to market in the highly competitive electronics industry.

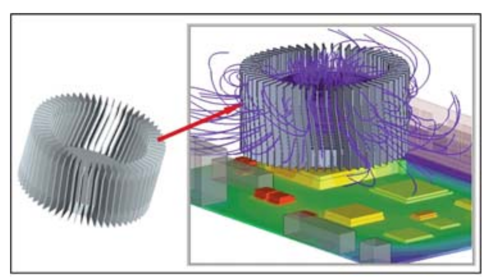
Rapid Model Development for Packages, Boards and Systems

To enable the rapid simulation electronics cooling applications, ANSYS Icepak software contains a streamlined user interface that speaks the language of electronics design engineers. Models are created by simply dragging and dropping icons of familiar predefined elements — cabinets, fans, circuit boards, vents, heat sources, heat sinks, etc. — to create models of complex electronic assemblies. These "smart objects" capture geometric information, material properties and boundary conditions, all of which can be fully parametric for performing sensitivity studies and design optimization.

To further accelerate model development, the software imports both electronic CAD (ECAD) and mechanical CAD (MCAD) data from a variety of sources. Geometry imported from ECAD and MCAD can be combined with smart objects to quickly and efficiently create models of electronic assemblies. The software also includes extensive libraries for standard materials, packages and electronic components such as fans — including fan geometry and nonlinear fan operating curves.



ANSYS Icepak software predicts the temperature profile in a computer graphics card



Simulation results of a fan-cooled processor heat sink attached to a printed circuit board

ANSYS Icepak Features

Automatic Mesh Generation

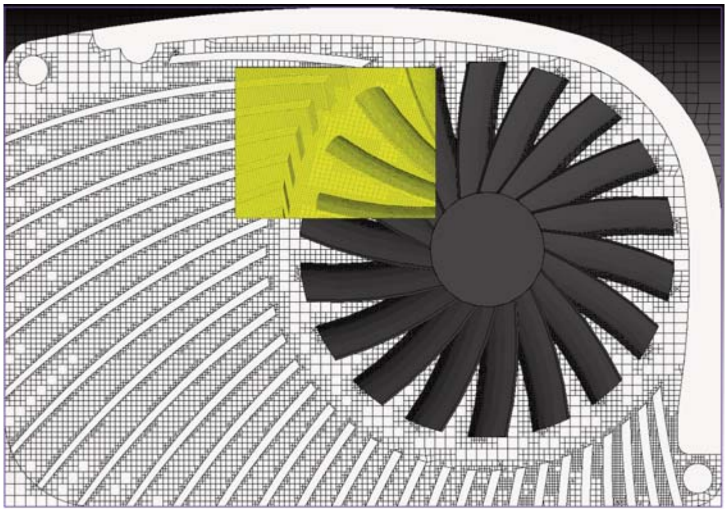
- Hexahedra, tetrahedra, pentahedra, prisms and mixed element mesh types
- Fully automatic, unstructured mesh for true geometry representation
- Automatic structured Cartesian meshing
- Automatic non-conformal for unstructured and structured meshing
  - Simplifies meshing, reduces cell count and increases solution speed
- Embedded non-conformal meshing
- Automatic hex-dominant mesher (HDM)
- Automatic multilevel meshing for increased geometric shape fidelity
- Automatic non-conformal regions without the use of separately meshed assemblies
- Automatic tetrahedral mesher
- Automatic mixed non-conformal meshing
- Coarse mesh generation option for first-pass analysis
- Full user control of meshing parameters and mesh deployment
- Mesh quality evaluation tools

Boundary Conditions

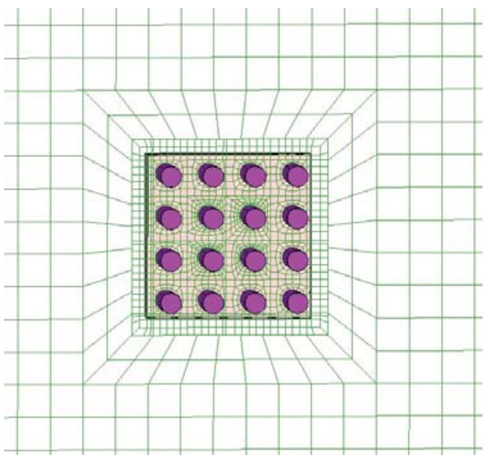
- Wall/surface boundary conditions for heat flux, temperature, convective heat transfer coefficient, radiation, symmetry and periodic conditions
- Openings and vents
  - Inlet/exit velocity, mass flow rate, exit static pressure, inlet total pressure, inlet temperature and turbulence parameters
- Grilles and resistances with an automatic loss coefficient based on free area ratio
- Fans with options for mass flow rate or fan performance curve
- Rotational speed for cylindrical/circular objects
- Recirculating boundary conditions for external heat exchangers
- Planar heat exchanger
- Network objects for modeling heat exchangers
- Time-dependent and enhanced temperature-dependent sources
- Time-varying ambient temperature
- Profiles of velocity, temperature, heat flux/heat transfer coefficients on openings and walls
- Automatic enhanced correlation-based heat transfer coefficient boundary conditions
- Time-dependent pressure
- Time-dependent electric current
- Electric current and voltage boundary conditions

Flexible Automatic Meshing Technology

ANSYS Icepak software automatically generates highly accurate, conformal meshes that represent the true shape of electronic components rather than rough, stair-step approximations. Meshing algorithms generate both multi-block and unstructured, hex-dominant meshes, which distribute the mesh appropriately to resolve the fluid boundary layer. While the meshing process is fully automated, users can customize the meshing parameters to refine the mesh and optimize the trade-off between computational cost and solution accuracy. The meshing flexibility of ANSYS Icepak software results in the fastest solution times possible without compromising accuracy.



Multilevel hex-dominant mesh of a heat sink and fan assembly imported from CAD geometry



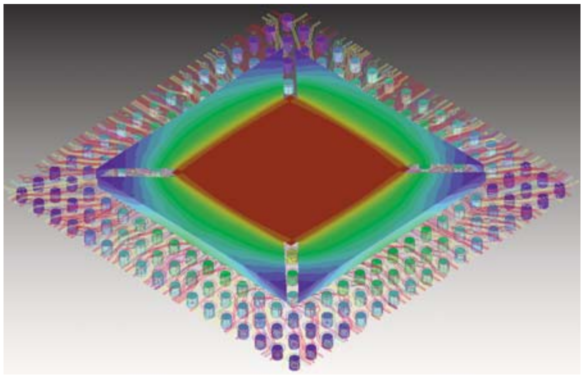
Mesh around pin fin heat sink follows the geometry of the part without any approximation

Robust and Rapid Numerical Solutions

ANSYS Icepak software uses state-of-the-art technology available in the ANSYS® FLUENT® CFD solver for the thermal and fluid-flow calculations. The CFD solver 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 multigrid scheme to accelerate solution convergence for complex conjugate heat transfer problems. The ANSYS Icepak solver provides complete mesh flexibility and allows the user to solve even the most complex electronic assemblies using unstructured meshes, providing robust and extremely fast solution times.

Advanced Physical Models

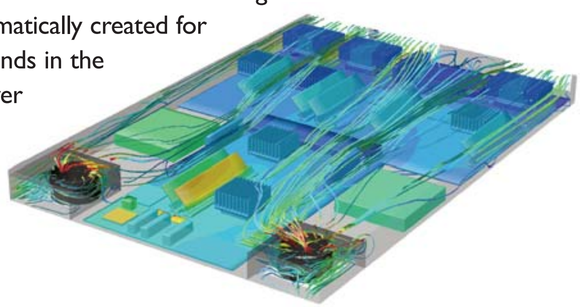
ANSYS Icepak software contains a broad array of advanced physical models, which ensure users can solve fluid flow and heat transfer physics with the confidence that simulation predictions will be accurate and dependable. The software includes several popular k-epsilon turbulence models along with advanced thermal modeling features such as contact resistance modeling, periodic boundaries, anisotropic conductivity and nonlinear fan curves.



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

Results Visualization and Reporting

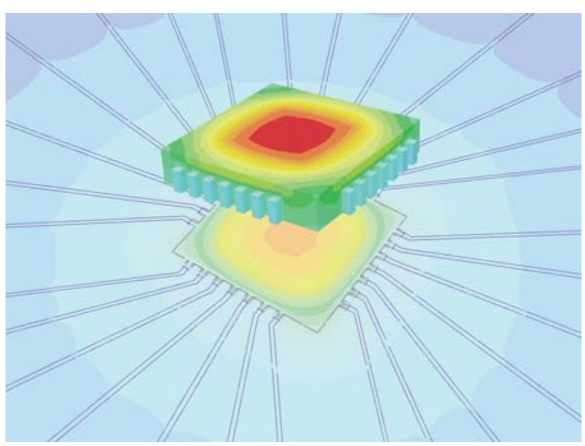
Within ANSYS Icepak software, a full suite of qualitative and quantitative post-processing tools are available to generate meaningful graphics, animations and reports to easily convey simulation results to engineers and non-engineers alike. Visualization of velocity vectors, temperature contours, fluid particle traces, isosurface displays, cut planes and XY plots of results data are all available for interpreting the results of an electronics cooling simulation. Customized reports, including images, can be automatically created for distributing results data, identifying trends in the simulation and reporting fan and blower operating points.



Fluid streamlines and temperature contours on a network server

Interfaces to Electrical and Mechanical Simulation

ANSYS Icepak technology provides interfaces to SIwave™ and ANSYS® Mechanical™ software to provide a full suite of tools to address the electrical, thermal and structural simulation requirements of the electronics design engineer. SIwave software extracts frequency-dependent electronic circuit models of signal and power distribution networks from device layout databases for modeling integrated circuit packages and printed circuit boards. ANSYS Mechanical software provides a comprehensive set of capabilities for modeling both nonlinear structural mechanics and heat transfer for static and dynamic systems.



Temperature contours on a micro lead frame plastic quad (MLPQ) package in a natural convection chamber, with traces modeled for increased accuracy

ANSYS Icepak Features

Comprehensive Thermal-Flow Modeling

- Steady-state or transient analysis
- Laminar or turbulent flows
  - Ability to specify a laminar region in a turbulent model
- Forced, natural and mixed convection
- Conduction in solids
- Conjugate heat transfer
- Surface-to-surface radiation
  - Automatic view-factor calculation
- Discrete-ordinates radiation
- Volumetric resistances and sources for velocity and energy
- Joule heating in traces and conductors

Advanced Physical Models

- Mixing-length zero-equation turbulence model
- Two-equation k-ε turbulence model
- RNG k-ε turbulence model
- Realizable k-ε turbulence model
- Enhanced models for all two-equation turbulence models
- Spalart-Allmaras turbulence model
- Anisotropic thermal conductivity for solids
- Temperature-dependent material properties
- Contact resistance modeling
- Non-isotropic volumetric flow resistance modeling
- Internal heat generation in volumetric flow resistances
- Nonlinear fan curves
- Radiation view factor computation
  - Hemicube method
  - Adaptive method
- Lumped-parameter models for fans, resistances and vents
- Robust shell conduction model for inplane and lateral conduction
- Resistance network model for IC packages
- CFD solution coupled with RC network
- Network implementation to model external heat exchangers
- Ideal gas law

Solver Attributes

- ANSYS FLUENT technology
  - Robust and fast convergence
- Choice of first-order upwind or a higher-order scheme
- Automatic under relaxation
- Advanced stabilization methods
- Variable time stepping for transients
- Parallel solver available
  - Job scheduling available
- Batch queuing

ANSYS Icepak Features

- Grid-to-grid interpolation for restart
- Graphical convergence monitoring
- Option to decouple the flow and energy solution for forced convection solution

Results Visualization and Reporting
- Interactive, object-based visualization
- Display of velocity vectors, contours and particle traces
- Cut planes with dynamic movement
- Isosurface displays
- Display of orthotropic thermal conductivity
- Display of electric potential
- Point probes with XY plotting
- Contours of velocity, temperature, pressure, heat flux, heat transfer coefficient, flow rate, turbulence parameters, vorticity, etc.
- Animation of particle and dye traces
- Animation of vectors and contours
- Report generation
- Time history display
- Point objects
  - Convergence monitoring, post-processing, reports
- Fan and blower operating points
- Export data to Autotherm
- Export data to NASTRAN®, PATRAN® and I-DEAS®

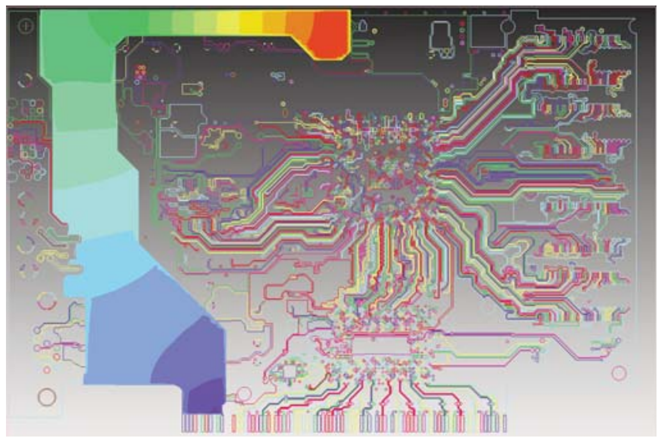
Online Help and Documentation
- Context-sensitive help
- Tool tips
- Tutorials
- Validation examples

Supported Hardware
- HP-UX 11i 64
- HP-UX 11.0 64
- SUN® Solaris™ 10 64
- Linux® Redhat® (3,4) Suse™ (9,10) 32
- Linux Redhat (3,4,5) Suse (9,10) 64
- Windows® XP, 2003 Server, Windows Vista® 32
- Windows XP, 2003 Server, Vista 64

Add-on Modules
- ANSYS Icepar
- ANSYS® Icepro™
- ANSYS Iceopt
- ANSYS Icegrb

Based on a SIwave simulation, the DC power distribution profile of printed circuit board layers can be imported into ANSYS Icepak software for a thermal analysis of the board. The coupling between SIwave and ANSYS Icepak software enables users to predict both internal temperatures and accurate component junction temperatures for printed circuit boards and packages.

Following an ANSYS Icepak simulation, the temperature data from a thermal-flow analysis can be imported into ANSYS Mechanical software to calculate the thermal stresses of the components. The interaction between ANSYS Icepak and ANSYS Mechanical software allows users to evaluate both the temperatures and resulting thermal stresses of the device.



Electric potential contours on a printed circuit board (PCB) trace

The ANSYS Advantage

With the unequalled depth and unparalleled breadth of engineering simulation solutions from ANSYS, companies are transforming their leading edge design concepts into innovative products and processes that work. Today, 97 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 multiphysics solutions to solve their complex engineering challenges. The engineered scalability of our solutions 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 almost 40 years as the industry leader — for the best in engineering simulation.

www.ansys.com

GSA Contract Holder

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

Image Credits: Some images courtesy FluidDA nv, Forschungszentrum Joülich GmbH, Heat Transfer Research, Inc., Riello SPA and iStockphoto.com

MKT0000461 11-09