International Journal of Science and Engineering Applications
Volume 15-Issue 09, 88 – 91, 2026, ISSN:- 2319 - 7560
DOI: 10.7753/IJSEA1509.1016

Thermal Simulation Methods for Air-Cooled Heat Sinks: A Review

Xiangyu Du
College of Automotive Engineering
Zibo Polytechnic University,
China

Abstract: With the continuous increase in power density of power electronic devices, thermal simulation analysis of air-cooled heat sinks has become an indispensable technical tool in product design and development. This paper systematically reviews the typical workflow and key technical procedures of thermal simulation methods for air-cooled heat sinks, including geometric modeling and model simplification, material thermophysical property assignment, computational domain mesh generation strategies, boundary condition and solver configuration, as well as result post-processing and identification of thermal weak links. On this basis, the main application scenarios of thermal simulation in air-cooled heat sink design are summarized, encompassing thermal performance evaluation, parameter sensitivity analysis, and heat source layout optimization. The analysis indicates that the accuracy of fluid-solid coupled modeling and mesh quality are the core factors affecting simulation reliability, while downstream hot air accumulation in unidirectional parallel-flow air cooling structures is a typical issue constraining the temperature uniformity of multi-chip modules. This paper also discusses the complementary relationship between simulation and experimental validation, and proposes directions for future methodological improvements, aiming to provide references for thermal simulation analysis and design optimization of air-cooled heat sinks.

Keywords: air-cooled heat sink; thermal simulation; computational fluid dynamics; fluid-solid coupling; simulation workflow

1.INTRODUCTION

With the continuous increase in power density of power electronic devices, heat dissipation has become a key factor restricting device reliability and service life [1]. Air cooling, with its advantages of simple structure, low cost, and convenient maintenance, has been widely applied in thermal management of medium and low-power electronic devices [2]. A properly designed air cooling system can effectively control the temperature rise of heating components, ensuring long-term stable operation of the equipment within permissible temperature ranges.

Thermal simulation technology is an indispensable analytical tool in the design process of air-cooled heat sinks. Compared with traditional empirical formula estimations, Computational Fluid Dynamics (CFD) methods can provide detailed distribution information of flow and temperature fields, helping designers intuitively identify thermal weak links and perform targeted optimizations [3]. With numerical simulation, multiple design schemes can be evaluated and screened before physical prototyping, thereby reducing late-stage design changes and shortening the development cycle [4].

ANSYS FLUENT, as a general-purpose computational fluid dynamics software, possesses a rich set of physical models and numerical algorithms, and is widely applied in the field of thermal analysis for electronic devices. This paper systematically reviews the full-process key technologies of thermal simulation methods for air-cooled heat sinks, summarizes the typical application scenarios in heat sink design, and discusses the limitations and improvement directions of current methods, aiming to provide references for simulation analysis and design optimization of similar products.

2. THERMAL SIMULATION MODEL CONSTRUCTION METHODS

2.1 Geometric Modeling and Model Simplification

The geometric model for thermal simulation of air-cooled heat sinks typically consists of core components including heating elements (chips), thermal conductive substrates, heat pipes, cooling fins, and the external air fluid domain. Since actual structures contain numerous fine features (such as bolts, chamfers, and identification markings), performing simulation calculations directly on the original geometry would lead to a sharp increase in mesh count while having negligible impact on the primary heat transfer paths. Therefore, modeling should follow the principle of "preserving primary heat transfer paths while simplifying non-critical details," removing local features that have minimal influence on temperature and flow field distributions, and applying reasonable equivalent treatments to complex assemblies while ensuring the integrity of thermal pathways.

Model simplification should focus on the following aspects: the thermal conduction links between heat sources and cooling structures must be completely preserved; the inlet and outlet positions and flow spaces of the fluid domain should be accurately represented; the effective heat transfer area of cooling fins must not be reduced; and for anisotropic thermally conductive components such as heat pipes, equivalent thermal conductivity coefficients should be used to characterize their directional high thermal conductivity characteristics.

2.2 Material Thermophysical Property Assignment

Thermal simulation requires assigning accurate thermophysical properties to each component. Air-cooled thermal systems typically involve heterogeneous materials including air fluid, semiconductor chip materials, aluminum alloy cooling structures, copper-based heat pipes, and thermal conductive substrates. The key parameters for each material include density, specific heat capacity, and thermal conductivity, with additional dynamic viscosity required for fluids.

The thermophysical properties of different materials vary significantly: chip materials have relatively high thermal conductivity, facilitating rapid diffusion of heat from hot spots; aluminum alloys combine good thermal conductivity with lightweight advantages, making them a common material for cooling fins; heat pipes, due to their internal phase-change heat transfer mechanism, can achieve axial equivalent thermal conductivity tens to hundreds of times that of pure copper, serving as a critical structure for efficient heat transport; air has a thermal conductivity far lower than solid materials, constituting the primary source of convective thermal resistance, and its temperature-dependent property variations should be considered in higher-accuracy simulations.

2.3 Computational Domain Mesh Generation Strategies

Mesh generation is the critical step of discretizing the continuous geometric model into finite control volumes, directly determining the accuracy and efficiency of simulation calculations. Air-cooled heat sink simulations typically involve coupling between solid and fluid domains, which differ significantly in geometric scale and physical characteristics: the fluid domain is large in size but requires capturing near-wall boundary layers, while local fine structures in the solid domain (such as fins and heat pipes) require detailed resolution.

Mesh generation should follow the following basic principles: local refinement should be applied in regions with large temperature and velocity gradients (such as chip surfaces, fin gaps, and heat pipe bends) to ensure accurate capture of key physical quantities; appropriate boundary layer meshes should be set near fluid domain walls to resolve convective heat transfer; overall mesh quality must meet quality indicator requirements such as skewness and aspect ratio to avoid distorted elements that may cause numerical divergence or computational errors.

2.4 Boundary Conditions and Solver Configuration

Proper configuration of boundary conditions is an important guarantee for the reliability of simulation results. Typical boundary conditions for air-cooled heat sink simulations include: inlet boundary——specifying the flow rate or velocity of cooling air and inlet air temperature; outlet boundary——typically set as pressure outlet with zero gauge pressure; wall boundary——solid exterior surfaces set as convective heat transfer boundaries or adiabatic boundaries; internal interfaces——solid-solid contact surfaces set as coupled thermal conduction boundaries, and solid-fluid interfaces set as fluid-solid coupled heat transfer boundaries.

Solver configuration includes turbulence model selection, pressure-velocity coupling algorithm, discretization schemes, and convergence criteria. Airflow inside air-cooled heat sinks is typically in a turbulent state, requiring appropriate turbulence models; the solution of energy and momentum equations requires selection of suitable discretization schemes based on model characteristics to ensure numerical stability and computational accuracy.

3. SIMULATION RESULT ANALYSIS METHODS

3.1 Flow Field Analysis

Flow field analysis is an important step in evaluating the flow characteristics of air-cooled heat sinks. By extracting pressure distribution contours and velocity vector plots of the fluid domain, the flow trajectories and velocity variation patterns of airflow within the air channel can be clearly identified. Pressure gradually decreases along the flow direction, and the total pressure drop can directly serve as a basis for fan selection and air channel design. Velocity distribution reveals whether airflow uniformly sweeps across the surfaces of each cooling fin; regions with excessively low velocity typically correspond to thermal weak areas in heat transfer.

The core value of flow field analysis lies in identifying "flow dead zones" and "hot air accumulation" phenomena. In unidirectional parallel-flow air cooling structures, airflow passes through the cooling module along a fixed direction; the air preheated by upstream heat sources continuously rises in temperature during flow, and when it reaches the downstream, the heat transfer temperature difference decreases, resulting in reduced heat transfer between downstream fins and the air, forming a "hot air accumulation" effect. This is a typical flow field issue constraining the overall temperature uniformity of the system.

3.2 Temperature Field Analysis

Temperature field analysis is the most direct output of thermal simulation. By extracting temperature distribution contours of various solid components, key information such as chip peak temperature, substrate temperature gradient, and fin temperature stratification can be obtained. The overall temperature field typically exhibits a gradient distribution pattern of gradual increase along the airflow direction, which is an inherent characteristic of parallel-flow air cooling structures.

Temperature field analysis should focus on three aspects: first, whether the peak temperature of heating elements falls within safe operating limits, which directly determines equipment reliability and service life; second, the temperature differences among similar heating elements——excessive temperature differences indicate uneven heat load distribution, which may cause premature aging of some components; third, regions with large temperature gradients, which typically correspond to areas with high thermal resistance or poor cooling conditions, representing potential directions for structural optimization.

3.3 Identification of Thermal Weak Links

By synthesizing the analysis results of flow field and temperature field, thermal weak links in the cooling structure can be systematically identified. Typical weak links include: downstream leeward side regions——where heat transfer efficiency decreases due to hot air accumulation effects, resulting in significant temperature rise of components; regions with excessively low airflow velocity over fin surfaces——where convective heat transfer coefficients are low and heat cannot be effectively transferred to the air; cross-sectional abrupt changes or contact interfaces in the thermal conduction path——where heat flux concentrates, forming localized hot spots.

The identification of weak links provides direction for structural optimization. Conventional improvement measures include: adding flow-disturbing structures to disrupt the thermal boundary layer and enhance local convective heat transfer coefficients; optimizing fin spacing and height to balance heat transfer area and flow resistance; adjusting air channel layouts to improve airflow uniformity; and reallocating power density among heat sources to balance heat load distribution.

4. TYPICAL APPLICATION SCENARIOS OF THERMAL SIMULATION

4.1 Thermal Performance Evaluation and Scheme Screening

During the product conceptual design stage, thermal simulation can be used for rapid comparative evaluation of different cooling schemes. By establishing simulation models of multiple alternative schemes and comparing key indicators such as chip temperature, fin efficiency, and flow resistance under identical boundary conditions, designers can screen out feasible structural schemes early in the design phase, reducing the frequency of late-stage design changes.

4.2 Parameter Sensitivity Analysis and Optimization

Thermal simulation can systematically evaluate the influence degree of various design parameters on thermal performance, known as parameter sensitivity analysis. Typical design parameters include fin height, fin spacing, substrate thickness, number and arrangement of heat pipes, and airflow rate. By sequentially varying parameter values and observing trends in performance indicators, key parameters most sensitive to cooling effectiveness can be identified, enabling priority focus on these parameters for refined adjustment in optimization design.

4.3 Heat Source Layout Optimization

In multi-chip modules, the power density and layout positions of different heating elements directly affect the overall temperature uniformity of the system. Thermal simulation can help designers evaluate temperature distribution differences under various heat source layout schemes, thereby achieving balanced heat load distribution, reducing peak temperatures, and narrowing temperature differences among chips by rationally allocating the heat generation power of each chip or adjusting chip placement positions without altering the cooling structure.

5. LIMITATIONS AND IMPROVEMENT DIRECTIONS OF SIMULATION METHODS

5.1 Factors Affecting Simulation Accuracy

The reliability of thermal simulation is influenced by multiple factors: the accuracy of material thermophysical properties——actual material thermal conductivity varies with temperature, and room-temperature value assignments may introduce deviations; the representativeness of boundary conditions——actual operating conditions such as airflow fluctuations, ambient temperature variations, and assembly tolerances are difficult to fully reproduce in simulations; the degree of model simplification——excessive simplification may omit critical heat transfer pathways or local flow characteristics; and mesh resolution——overly coarse meshes cannot resolve near-wall temperature and velocity gradients, while overly fine meshes result in excessive computational costs.

5.2 Complementarity of Simulation and Experimental Validation

Experimental testing is an important means of validating the reliability of simulation models. Actual temperature and flow field data obtained through experimental methods such as thermocouple temperature measurement, infrared thermography, and particle image velocimetry can be compared and calibrated against simulation results. Experimental validation not only confirms the accuracy of the simulation model but also provides a basis for model calibration, such as adjusting equivalent heat transfer coefficients and modifying boundary conditions. Meanwhile, simulation can guide the arrangement of experimental measurement points, improving experimental efficiency.

5.3 Directions for Future Methodological Improvements

Future research can improve thermal simulation methods in the following directions: establishing more accurate temperature-dependent material property models to enhance simulation accuracy under high-temperature conditions; developing multi-scale coupling methods to simultaneously account for system-level flow and local fine-scale heat transfer; introducing uncertainty quantification methods to evaluate the influence intervals of input parameter fluctuations on simulation results; and combining optimization algorithms with surrogate model techniques to achieve efficient automatic optimization of cooling structures.

6.ACKNOWLEDGMENTS

The authors sincerely thank the School of Automotive Engineering of Zibo Polytechnic University for its strong support for this research.

7.REFERENCES

[1] Nair V, Baby A, M.B. A, et al. A comprehensive review of air-cooled heat sinks for thermal management of electronic devices[J]. International Communications in Heat and Mass Transfer, 2024, 159(PB): 108055.

[2] Zhang H C, Li Z, Zhang K H, et al. Research on heat dissipation characteristics of high-power air-cooled power amplifier module cooling system[J]. Power Electronics Technology, 2022, 56(6): 26-30. (in Chinese)

[3] Ding J, Zhang P. Thermal simulation and experiment of IGBT air-cooled heat sink for motor controller[J]. Journal of Power Supply, 2015, 13(2): 38-44. (in Chinese)

[4] Chen J J, Zhou L, Qiu Y H. Optimization design and evaluation of air cooling for IGBT heat sink[J]. Industrial Technology Innovation, 2020, 07(6): 45-49+66. (in Chinese)