http://dx.doi.org/10.5755/j01.eee.19.3.3695 ELEKTRONIKA IR ELEKTROTECHNIKA, ISSN 1392-1215, VOL. 19, NO. 3, 2013

Analysis of Heat Sink Modelling Performance

P. Tarvydas[1], A. Noreika[1], Z. Staliulionis[1]
[1]Department of Electronics Engineering, Kaunas University of Technology,
Studentu St. 50, LT-51368 Kaunas, Lithuania, phone: +370 37 300524
alius.noreika@ktu.lt

***Abstract*—With increasing complexity of electronic components and their systems, their cooling solutions become an important issue, and heat sinks are the most widely applied solution in industry. Theoretical, experimental and numerical methods can be used to determine a heat sinks thermal performance. Finite element method is probably the most suitable numerical technique to simulate thermal fields of heat sinks. The advantages of finite element modelling and analysis of heat sinks are discussed. Thermal modelling of heat sink mounted inside air channel was accomplished using finite element analysis software COMSOL. Algorithm for heat sink model building was offered. Finite element mesh generation and problem solution performance was evaluated using created algorithm. Performance evaluation results are presented.**

***Index Terms*—Finite element method, heat sink, automated modelling.**

I. INTRODUCTION

The primary purpose of heat sink is to reduce the temperature of the electronic component attached to it. The excess heat is transferred into another medium, typically air. In this way heat-related failures of electronic components are prevented. There are several design aspects which are important in order to ensure the efficient heat emission from heat sink. Some of them are related to operating conditions, such as artificial air circulation, choice of thermal interface materials, or a proper attachment to the heat-emitting component. However, most of the factors which determine the efficiency of heat sink are related to its construction, e.g. larger active surface area, type of material it is manufactured of, overall shape or shape of separate elements of heat sink.

In modern industry the efficiency of a heat sink is typically determined using numerical modelling techniques; in relatively rare cases theoretical calculations are used. In computer-aided modelling and simulation, the finite element method (FEM) is probably the most often used numerical technique [1]–[9]. Currently this method is accepted as one of the most advanced engineering techniques due to its flexibility and solid mathematical and algorithmical basis. Computer software based on FEM is an irreplaceable part of heat sink design and is used in different stages of development, e.g. modelling, performance optimisation or analysis of their characteristics.

Mean time between failures of any electronic component or device is directly related to its operating temperature. Increased temperature shortens normal lifetime. Its exact effect can be determined mathematically; one possible method is by using Arrhenius equation, which relates how increased temperature accelerates the age of a product compared to its normal operating temperature [2]

$$A_f = Ae^{\left(\frac{E_a}{k}\left\{\frac{1}{T_u}-\frac{1}{T_t}\right\}\right)}, \tag{1}$$

where A_f – acceleration factor, A – a proportional multiplier, which can be a function of temperature, i.e. $A=A(t)$, E_a – activation energy in electron-volts (eV) is related to physical mechanism of electrical failure [3], k – Boltzmann's constant ($k=8.617\cdot10^{-5}$ eV/Tk), T_k – temperature, K, T_u – reference temperature, K, T_t – test temperature.

In practice such dependency means, that lowering the temperature by 10ºC potentially doubles the reliability of the product; and vice versa: increasing temperature by 10ºC decreases the lifetime by 50%. This is a solid motivation for development of better cooling solutions for electronic components.

II. THEORETICAL BASIS FOR MODELLING OF HEAT TRANSFER

Conduction, radiation and convection are the three fundamental mechanisms which define the distribution of thermal energy. As stated by first law of thermodynamics, such thermal processes take place when there is a difference in temperature, in our case between particular electronic component and ambient air. These processes follow the principle of conservation of energy and internal energy of modelled object is a logical quantity to solve for. However, in practice it is expressed in terms of temperature, since this parameter is more easy and convenient to measure.

Navier-Stokes equations are used to characterize the single-phase, heat-conducting and non-turbulent (or time-averaged turbulent) fluid (gas or liquid) flow interfaces. The most general form of these equations is [4]:

$$\frac{\partial\rho}{\partial t}+\nabla\cdot(\rho\mathbf{u})=0, \tag{2}$$

$$\rho\frac{\partial\mathbf{u}}{\partial t}+\rho(\mathbf{u}\cdot\nabla)\mathbf{u}=\nabla\cdot[-p\mathbf{I}+\tau]+\mathbf{F}, \tag{3}$$

$$\rho C_p\left(\frac{\partial T}{\partial t}+(\mathbf{u}\cdot\nabla)T\right)=-(\nabla\cdot\mathbf{q})+\tau:\mathbf{S}-$$

Manuscript received March 19, 2012; accepted May 22, 2012.

$$-\frac{T}{\rho}\frac{\partial \rho}{\partial T}\bigg|_{p}\left(\frac{\partial p}{\partial t}+(\mathbf{u}\cdot\nabla)p\right)+Q\,, \qquad (4)$$

where ρ – the density (kg/m^3); $\mathbf{u}$ – the velocity vector (m/s); p – pressure (Pa); τ – the viscous stress tensor (Pa); $\mathbf{F}$ – the volume force vector (N/m^3); Cp – the specific heat capacity at constant pressure (J/(kg·K)); T – the absolute temperature (K); $\mathbf{q}$ – the heat flux by conduction (W/m^2); Q – contains heat sources other than viscous heating (W/m^3); $\mathrm{S} = 0.5(\nabla \mathrm{u} + (\nabla \mathrm{u})^T)$ – the strain-rate tensor (1/s); “:” – denotes a contraction between tensors defined by $\mathbf{a}:\mathbf{b} = \sum_{n}\sum_{m} a_{nm} b_{nm}$.

When Navier-Stokes law is expressed in terms of temperature the resulting heat equation for fluids is (4).

III. MODEL BUILDING

Finite element method based software package COMSOL Multiphysics and its Conjugate Heat Transfer interface was selected to perform the modelling. This module provides the necessary tools to analyse heat transfer from heat sink to coupled single-phase laminar flow of air [4].

The sequence of heat sink energy transfer modelling using finite element method is analogous to modelling of other electronic devices and their systems, for example, when modelling distributions of electromagnetic fields [5]. In general case, it is possible to distinguish the following main stages of modelling: 1) formulation of modelling problem; 2) creation of complete geometrical model; 3) assignment of material properties and boundary conditions to specific areas of model; 4) selection of finite element shape and finite element mesh density; 5) solution of equation system, which characterizes thermal field; 6) analysis of obtained results. Generalized modelling algorithm of heat sink cooled by air flow is given in Fig. 1.

Depending on application area heat sinks can be manufactured in large, moderate and miniature (for surface-mount devices) configurations. However it is always preferable to produce a heat sink of less size while maintaining the same level of thermal efficiency. Pin-based construction of active surface ensures relatively low thermal resistance. Round pins provide a large surface area, and when they are placed symmetrically in respect of heat sink base, their efficiency does not depend very much on the direction of the incoming air. Such pin configurations are especially practical for use in embedded systems. Heat sinks with low pin density are usually employed for environments where speed of air ranges from 0 to 2 m/s; higher pin density is used when speed of airflow varies from 2 to 4 m/s and when cooling fan has to be placed directly on top of the pins. Copper and aluminium are the primary materials used to produce these components.

Round pin heat sink construction type (Fig. 2) and aluminium material were selected to perform modelling of moderate density heat sink for fan-cooled applications [8], [9]. The heat sink is placed inside an air channel of rectangular shape (Fig. 3), which imitates a part of an air intake pathway used to cool electronic components. The dimensions of this air channel construction model and heat sink placement coordinates x, y, z can be changed arbitrarily during geometric model creation stage. Such configuration can also form a part of larger cooling system. Air flow is horizontal to heat sink plane and enters from one side of air channel and exits at the opposite side. The type of heat sink construction or separate parameters (number of pins, their shape and size) can be also manipulated according to specific needs.

Geometrical model of heat sink can be created using COMSOL Multiphysics tools or, alternatively, it can be also imported from other CAD software by using different supported exchange file formats, e.g. widely known IGES, SolidWorks, Parasolid and similar file types.

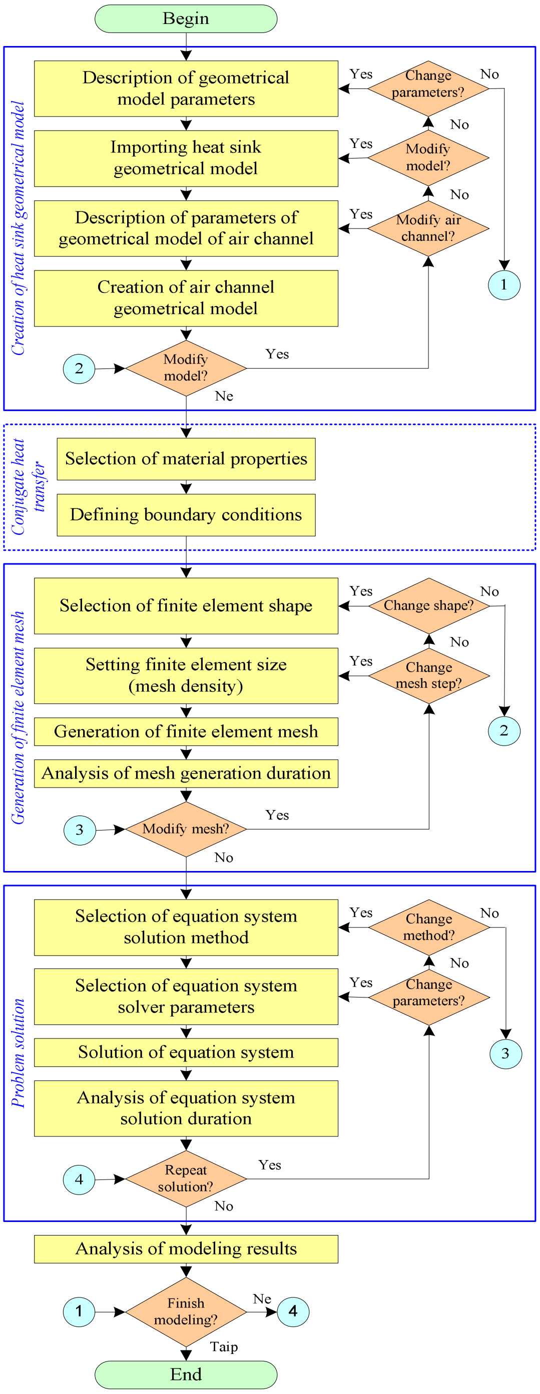
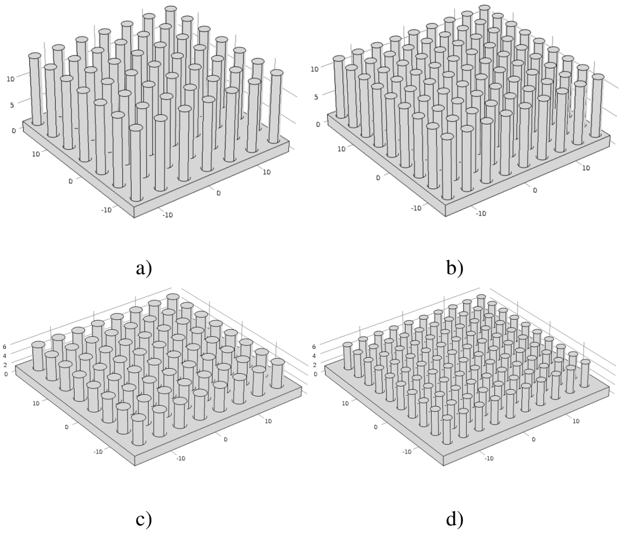


Fig. 1. Heat sink modeling algorithm.

Selected theoretical heat sink geometrical model consists of two parts (Fig. 2, bottom right): heat sink base and round pin fins. Heat sink base width – 38 mm, depth – 38 mm, height – 2 mm. The number of heat sink pins 100 fins. Round pin radius – 0.875 mm, height – 5 mm, pin displacement along *x* axis – 3.5 mm, pin displacement along *y* axis – 3.5 mm. Heat sink material: aluminium. The base surface of the heat sink receives a 1.5W heat flux from electronic component attached to it. External surfaces (i.e. boundaries) of the model are thermally insulated.


a) b)

c) d)

Fig. 2. Examples of heat sink geometrical models.

Rectangular channel was selected to provide air flow (Fig. 3). The following air channel dimensions are selected: width – 100 mm, depth – 60 mm, height – 15 mm. The problem solving task is to determine a thermal balance between heat sink heated by thermal power source placed underneath and flowing air inside of air channel with initial room temperature of 20□C. The thermal field is continuous across the interface of different materials (i.e. heat sink aluminium and air). No other heat sources are defined and it is considered that the walls of air channel do not conduct heat.

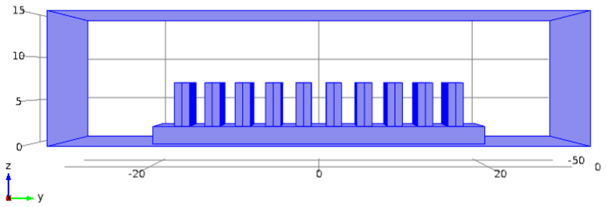


Fig. 3. Heat sink placed inside of air channel (frontal view).

The flow field is computed by determining one momentum balance for each space coordinate and a mass balance. The entering air flow velocity 10 cm/s corresponds to laminar flow and an assumption is made that at the outlet a constant pressure is present with no stresses perpendicularly to the outlet. At all solid surfaces, the velocity is set to zero in all three spatial directions.

The library of heat sink geometrical models was created during experimentation, which can be extended with additional heat sink construction models. During creation of heat sink geometrical model a possibility to manipulate base and pin dimensions, pin shape and their number without changing the rest parameters of the model was implemented.

IV. FINITE ELEMENT MESH GENERATION AND MESH GENERATION PERFORMANCE ANALYSIS

In analysis of modelling performance the following PC configuration was used: motherboard – Asus P7P55D Deluxe, CPU – Intel Core i7-860, RAM – Corsair 2xTW3X4G1333C9, HDD – WD1001FALS, operating system – Microsoft Windows 7 Ultimate x64.

Unstructured tetrahedral mesh type was selected to generate finite element mesh in entire volume of the model. Unstructured tetrahedral mesh is versatile, since in can be used for any type of geometry with various aspects of topology and shape.

Software COMSOL Multiphysics provides two ways to generate finite element mesh: one way is to select required element size parameters with no restrictions; other way is to use element size predefined according to software specifications [6]. In the first case the following parameters can be used to modify size of finite elements: maximum element size, minimum element size, maximum element growth rate, resolution of curvature, resolution of narrow regions. In the second case, mesh is specified in approximate manner, by selecting extremely fine, extra fine, finer, fine, normal, coarse, coarser, extra coarse or extremely coarse mesh. The generated mesh consisting of 35437 finite elements is shown in Fig. 4.

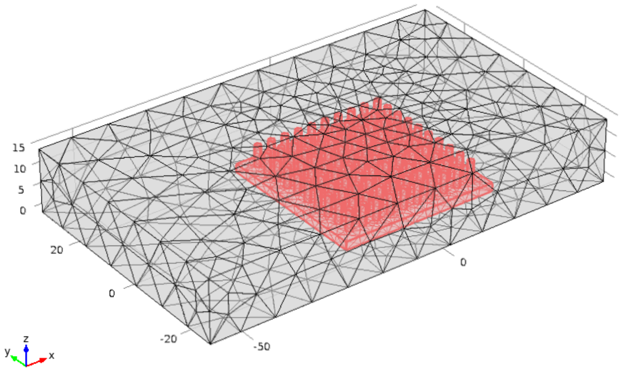


Fig. 4. Generated finite element mesh.

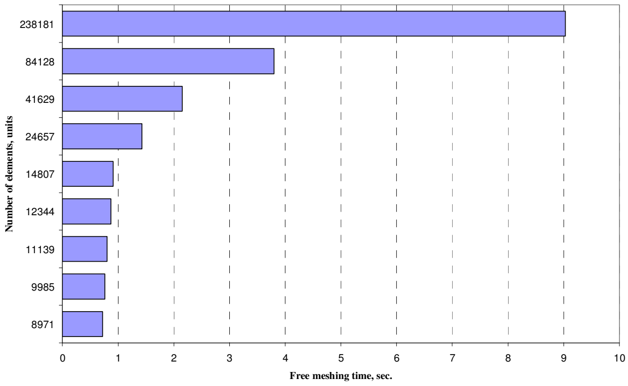


Fig. 5. Duration of free meshing.

After completing the free tetrahedral finite element mesh generation performance analysis procedure (Fig. 5), it can be stated, that mesh generation duration does not have significant impact on overall modelling process duration. Stages of geometrical model creation and problem solution require significantly more time.

Spatial modelling of devices or systems of complex shape using finite element method is relatively slow process. Therefore it is important to choose finite element mesh density in different areas of model optimally, because this factor influences the size of field-related equation system generated during solution of the problem. A selection of suitable equation system solver is another important factor; however we narrowed the scope of the work only to the analysis of the impact of the number of finite elements on the thermal field modelling performance.

V. SOLUTION OF EQUATION SYSTEM AND PERFORMANCE ANALYSIS

The following methods can be used to solve a heat transfer-related linear equation system: generalized minimum residual iterative method, flexible generalized minimum residual iterative method, biconjugate gradient stabilized iterative method, conjugate gradients iterative method [6]. Generalized minimum residual iterative method solver was selected to perform the modelling.

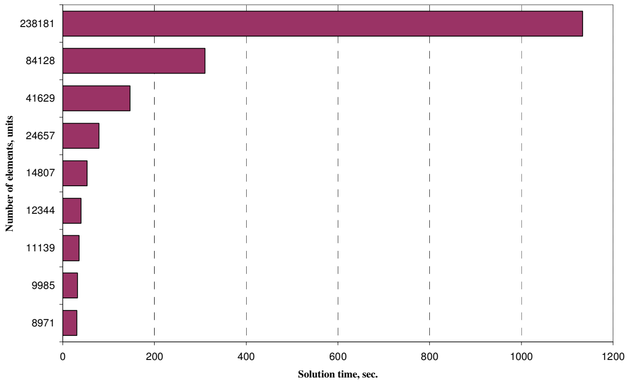


Fig. 6. Problem solution duration.

Computed results of performance analysis (Fig. 6) indicate the importance of selection of optimal finite element mesh density. This parameter essentially determined the solution duration. For example, after increasing number of mesh elements from 84128 to 238181, time needed to solve the problem increased 3.6 times.

VI. CONCLUSIONS

The created model of heat sink mounted inside an air channel can be used to measure the cooling capacity of heat sinks. Also, the developed modelling technique can be used to model heat sinks of different constructions.

After completing the free tetrahedral finite element mesh generation performance analysis it can be confirmed that time required to generate mesh (several seconds in our case) does not have any significant influence on overall duration of modelling process.

The solution time is related to the number of finite element mesh nodes by exponential law. Therefore in order to decrease the modelling duration it is very important to create an optimal geometrical model and to select and optimal density of finite element mesh. Solution time exceeds meshing duration by several tens of times and essentially determines the entire modelling duration. Heat sink modelling optimisation-related problems could be the topic for future research.

The created method of finite element mesh generation and equation system solution performance analysis can be applied to electronic devices or systems of any type.

REFERENCES

[1] S. S. Rao, “The finite element method in engineering”, Burlington, Elsevier Butterworth-Heinemann, 2011, p. 710.

[2] V. S. Speaks, “Reliability Engineering. Reliability and MTBF Overview”. [Online]. Available: http://www.vicorpower.com/ /documents/quality/Rel_MTBF.pdf

[3] M. Osterman. “We still have a headache with Arrhenius”, [Online]. Available: http://www.electronics-cooling.com/2001/02/we-still-have-a-headache-with-arrhenius/

[4] *COMSOL Heat Transfer Module User’s Guide,* COMSOL, 2010, p. 184.

[5] P. Tarvydas, V. Markevičius, A. Noreika, E. Pačėsaitė, “Modeling of electric field created by electronic optical system”, *Elektronika ir Elektrotechnika (Electronics and Electrical Engineering*), no. 1, pp. 20–23, 2005.

[6] *COMSOL Multiphysics Reference Guide,* COMSOL, 2010, p. 426.

[7] J. N. Reddy, D. K. Gartling, *The Finite Element Method in Heat Transfer and Fluid Dynamics,* 3rd ed., CRC Press, 2010, p. 524.

[8] O. C. Zienkiewicz, R. L. Taylor, P. Nithiarasu, *The Finite Element Method for Fluid Dynamics,* 6th ed., Elsevier Butterworth-Heinemann, 2005, p. 400.

[9] O. C. Zienkiewicz, R. L. Taylor, J. Z. Zhu, *The Finite Element Method: Its Basis and Fundamentals,* 6th ed., Elsevier Butterworth-Heinemann, 2005, p. 752.