Sensors and Actuators A 148 (2008) 168–175

Contents lists available at ScienceDirect

Sensors and Actuators A: Physical

journal homepage: www.elsevier.com/locate/sna

Thermal correction values for analysis of lineshape microstructure arrays

Kendall Teichert, Brian Jensen*

Department of Mechanical Engineering, Brigham Young University, Provo, UT 84602, USA

ARTICLE INFO

Article history:
Received 21 December 2007
Received in revised form 1 July 2008
Accepted 15 July 2008
Available online 5 August 2008

Keywords:
Thermal actuators
Finite difference method
Reduced-order models

ABSTRACT

Thermal modeling of microelectromechanical systems is a major area of research with many different approaches and complexities. In this paper a simplified, reduced order model for thermal analysis of lineshape microstructure arrays, such as those used in thermal actuators, is explained. Finite element modeling of these arrays was performed to determine thermal correction (K_t) values for the simplified thermal model. An equation is developed to describe a general MEMS design space of the independent parameters for implementation of the simplified model. Error in the equation fit is quantified and the fit is validated. Application of the model is also shown for single microstructures. Implementation of the simplified model using a finite difference model is presented and validated, with errors in predicted temperature rise of less than 5.4%.

1. Introduction

Various microelectromechanical systems (MEMS) depend on thermal phenomena for their operation. Much research has been done to capture these phenomena [1–5]. One such device is the chevron-shaped thermal actuator or thermomechanical in-plane microactuator (TIM) [6–10], as shown in Fig. 1. Understanding the thermal behavior of devices like these is crucial to proper modeling. In modeling these thermal attributes, the trade-off between simplicity and accuracy has resulted in several different approaches [6,7,11]. Work done by Lin and Chiao [12] developed a shape-correction factor that correlates the heat flux through the bottom of a lineshape microstructure to that of the heat flux out of the entire structure. This work has been used in several studies [7,13,14]. In Lin's work, a two-dimensional finite element model (FEM) of the cross-section of the structure was developed to determine this correction value for a single microstructure. This work expands this concept for application to multiple microstructures in air that are in an array configuration. This further development is necessary to account for heat interactions between structures in the TIM.

One particular motivation for this type of simplification of thermal modeling of lineshape microstructure arrays is shown in recent research by Harb et al. [15], in which they showed that increased efficiencies can be obtained in thermal actuators by lumping the actuation legs together so as to reduce heat loss to the surroundings. As mentioned by Harb et al. power reduction is significant for TIMs in order to make them viable for on-chip applications. Some previous modeling of the TIM with spread leg systems [7] used correction values based on Lin and Chiao's [12] one microstructure values. However, as these legs are brought closer to eliminate heat transfer to the surroundings, changes in these correction values occur. In order to properly analyze these actuators in this new "lumped" configuration, full three-dimensional modeling could be used as by Messenger [11], which requires immense computational power, or proper correction values could be developed to incorporate the interdependence between microstructures.

This paper presents an explanation of the significance of this expanded work. Finite element modeling was performed to analyze the static heat transfer of these device arrays. This model was used to explore the implications of this research over the desired design space. From this initial research, a final modeling approach was developed. Implementation of the simplification developed in this work will be demonstrated in a finite difference analysis of a TIM.

2. Modeling approach

2.1. Thermal correction value definition

The basic equation for heat transfer in a thin microstructure with temperature-dependent material properties is

$$\frac{\partial}{\partial x}\left(\kappa \frac{\partial T}{\partial x}\right) = q_{\text{gen}} + q_{\text{loss}} \qquad (1)$$

where x is the location along the beam, κ the thermal conductivity, T the temperature, q_{gen} the heat generation in the structure, and q_{loss} is the heat loss from the structure. However, the value

* Corresponding author.
E-mail address: bdjensen@byu.edu (B. Jensen).

0924-4247/$ – see front matter © 2008 Elsevier B.V. All rights reserved.
doi:10.1016/j.sna.2008.07.020

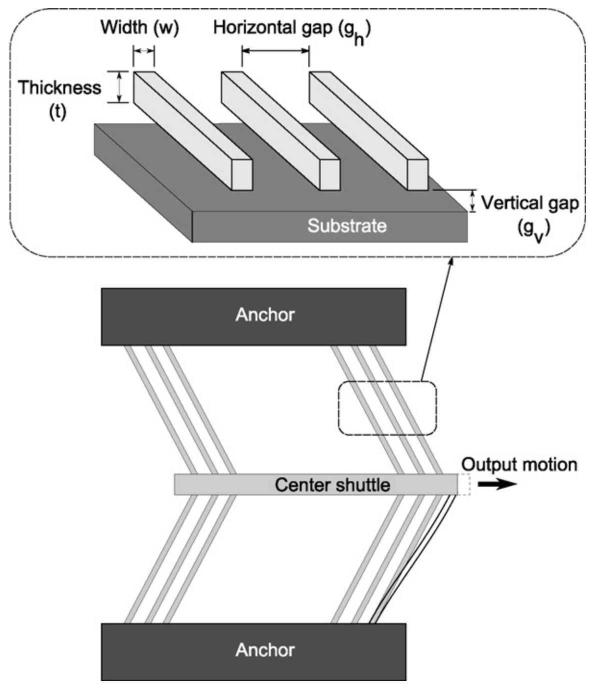

Fig. 1. Thermomechanical in-plane microactuator as an array of lineshape microstructures.

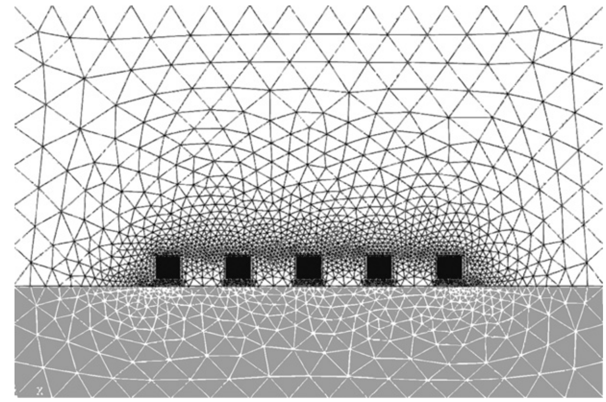

Fig. 2. Example meshing of 2D FEM.

of q_{loss} can be quite computationally expensive to determine in three-dimensional space. The purpose of the thermal correction value, K_t, is to allow accurate modeling to be done using a reduced order model. This approach is based on the work for a single microstructure presented in ref. [12], which is based on the assumption that the majority of heat loss is directly from the bottom of the microstructure to the substrate. Using this assumption, the heat loss would be calculated as

$$q_{loss\_reduced} = \left(\frac{A}{R_{th}}\right)(T_{sub} - T) \tag{2}$$

$$R_{th} = \frac{t_{air}}{\kappa_{air}} + \frac{t_{sub}}{\kappa_{sub}} \tag{3}$$

where $q_{loss\_reduced}$ is the reduced order model heat loss from the microstructure to the substrate, A the surface area of the bottom of the microstructure, t_{air} and t_{sub} are the thickness of the air and substrate, respectively, and κ_{air} and κ_{sub} are the thermal conductivities of the air and substrate, respectively. As mentioned, these equations assume only heat loss from the bottom of the structure to the substrate. Therefore, in order to compensate for the remainder of the heat flow a correction value, K_t, was defined as the actual heat transfer (q_{loss_actual}) that would occur out of the entire microstructure divided by the estimated value ($q_{loss\_reduced}$) that would be calculated by Eq. (2), or

$$K_t = \frac{q_{loss\_actual}}{q_{loss\_reduced}}. \tag{4}$$

In this way, the K_t value can be used to compensate for the loss of information due the reduction in order of a 1D model. As a result, Eq. (2) becomes

$$q_{loss\_actual} = (K_t)\left(\frac{A}{R_{th}}\right)(T_{sub} - T). \tag{5}$$

This approach is easily implemented into a basic finite difference model. The basic finite difference equation would then become

$$\kappa_i\left(\frac{T_{i+1} - 2T_i + T_{i-1}}{(\Delta x)^2}\right) + \left(\frac{\kappa_{i+1} - \kappa_{i-1}}{2\Delta x}\right)\left(\frac{T_{i+1} - T_{i-1}}{2\Delta x}\right) = -I^2\frac{R_e}{\Delta V} + K_t\left(\frac{T_i - T_s}{R_{th}\Delta x}\right) \tag{6}$$

with K_t as described previously; T the temperature of the node; T_s the substrate temperature; Δx the length of the beam associated with the node; R_{th} as described previously; I the electrical current through the node; R_e the electrical resistance associated with the node; ΔV the volume associated with the node; κ the thermal conductivity; and the subscripts $i-1$, i, and $i+1$ being the previous, current, and next nodes, respectively. The boundary conditions for Eq. (6) include a proscribed temperature at one end, and a symmetric or insulated boundary (corresponding to $\partial T/\partial x = 0$) on the other end. Further explanation of boundary conditions will be given in the K_t value implementation section. To implement this model, the dependence of K_t on the structure geometry must be known.

2.2. Finite element model

In order to characterize these K_t values, or more specifically the q_{loss_actual} values, a FEM was developed using a commercial finite element package (ANSYS). The model was based on a 2D cross-section of the structure. Fig. 1 shows the basic geometries of the model. A representative meshing of the FEM is shown in Fig. 2. This model consisted of three main components: air, substrate, and structure. The air was modeled with temperature-dependent material properties using infinite boundary condition elements on the outer edges (infin9 in ANSYS). The substrate was modeled as monocrystalline silicon, with infinite boundary condition elements on its sides only. The structures were modeled as polycrystalline silicon with temperature-dependent material properties. The material properties used throughout this work are the same as in Messenger [11], which included the temperature-dependent properties of the thermal conductivity of air and polycrystalline silicon, the resistivity of polycrystalline silicon, and the density of air. All outer edges of the model (air top and sides, and substrate sides and bottom) were constrained to the ambient temperature.

The initial modeling was done to explore effects due to changes in geometries, temperature, and number of structures in the array. The model was parameterized to be able to change microstructure thickness (t), width (w), vertical gap from top of the substrate to the bottom of the structure (g_v), horizontal gap spacing between structures (g_h), number of structures in the array, and the structure temperature. The bottom of the substrate was held at a constant