Simulation of Heat Generating In a Vibrating Structure Using COMSOL Multiphysics

Ali Kamil Jebur
University Of Technology
Ali_kamel42002@yahoo.com

Abstract

This paper dealt with heat generating in a beam structure model subjected to small vibrations to know the viscos elastic behavior under heat and vibration. The model first computed coupled thermal – structural interaction. The results obtained from this analysis of the model treated by the finite element method to calculate amount of heat generation in the material. A transient heat transfer analysis then simulated the slow rising temperature in the beam using these heat source terms.

The model has been constructed from two blocks, the first block from Aluminum while the second block made from β –Titanium. The model was constrained from one side, while the other side free, so vibrations that occur along the model. These vibrations led to heat generating, so yields that residual stresses through the model. The result obtained represented in curves which give good agreement with international published researches

Key words: Random vibration ,Curvilinear stiffener ,Stiffened plate ,viscoelastic Materials

الخلاصة

في هذا البحث تم التعامل مع نموذج يتضمن توليد حرارة في دعامة تم تعرضها الى اهتزازات بسيطة لمعرفة السلوك المرن اللزج للتركيب تحت تأثير مشترك للحرارة والاهتزازات .الموديل الرياضي الذي تم بنائه تناول حسابات مشتركة حرارية مع حسابات تصاميم واجهادات. والنتائج المستحصلة من الحسابات والتحليل للنموذج المأخوذ خلال هذا االبحث تضمن حسابات انتقال وتوليد الحرارة بضروف غير مستقرة وكيفية انتقالها خلال المعادن والمواد المستخدمة في هذا البحث حيث تم توضيف نمذجة رياضية باستخدام طريقة العناصر المحددة .ومن ثم تم نمذجة واحتساب الارتفاع في درجات الحرارة ببطئ .

النموذج المستخدم تم بنائه من جزئين مكونين من معدنين مختلفين الاول من الالمنيوم والثاني من بيتا تيتانيوم.

النموذج مثبت ومسنود من طرف اما الطرف لثاني فهو حر لغرض توليد الاهتزازات خلال النموذج والتي تؤدي الى توليد الحرارة نتيجة الاهتزازات ومن خلالها تم التوصل الى مقدار الاجهادات المتبقية في النموذج. اظهرت النتائج المستحصلة والممثلة في المخططات توافق جيد مع البحوث المنشورة عالميا.

الكلمات المفتاحية: اهتزاز عشوائي ، المبيس المنحنية ، لوحة اشتدت، والمواد اللزجة

Symbols	Meaning
ω	Angular frequency
k	Thermal conductivity matrix
ρC_p	Volumetric heat capacity
D	Elasticity matrix
ε	Strain vector
α_{vec}	Thermal expansion vector
T_0	*S*train reference temperature
t	time
T_0	Initial temperature across the beam
η	Loss factor
T_1	Linear temperature response in the frequency domain
Q_d	Internal work of the nonelastic (viscous) forces over the period
Q_d	Energy transfer between the mechanical and thermal domains

1. Introduction

When a structure is subjected to vibrations of high frequency, a significant amount of heat can be generated within the structure because of mechanical (viscoelastic) losses in the material. A second mechanism contributing to the slow temperature rise in a vibrating structure is called thermo elastic damping and represents the energy transfer between the thermal and mechanical domains [Duwel *et. al.,* 2006;Setareh, 2011].

There are many causes of high vibration in a generator field. The most common are mechanical imbalance, thermal sensitivity, misalignment and bearing degradation. Other causes are rubbing, bent overhangs, rotor stiffness asymmetry, out-of-round journals and other design deviations caused by abnormal in-service operation. Each of these causes has dominated frequency and characteristic response. The cause of vibration can be diagnosed by thoroughly analyzing the vibration data [Zhi Hua Nie *et. al.*,2010; Jiazhu and Frakn, 2011].

While the structural mechanics module works together with COMSOL Multiphysics, and can be integrated with other add-on modules to model many different applications, it does include a number of tailor-made multiphysics interfaces. For instance, the thermal stress interface is similar to the solid mechanics interface with the addition of a thermal linear elastic material model. It can be used in combination with various heat transfer interfaces to couple the temperature field to a structure's (material) expansion.A special joule heating and thermal expansion multiphysics interface combines thermal stress with joule heating and describes the conduction of electric current in a structure, the subsequent electric heating caused by the ohmic losses in the structure, and the thermal stresses induced by the temperature field [Jiazhu and Frank, 2011].

The model has been constructed from two blocks, the first block from Aluminum while the second block made from β –Titanium. The model was constrained from one side, while the other side free, so vibrations that occur along the model. These vibrations lead to heat generating, in this context; all of these considerations yield residual stresses during the model.

In this paper, a fully coupled thermoelastic response for a vibrating beam-like structure is computed by combining the stress-strain analysis with the linearized heat-transport equation. The analysis is performed in the frequency domain.

The temperature rise in the beam is then modeled as a transient heat-transfer problem,where the corresponding heat-transfer equation contains two source terms computed using the results of the frequency response analysis. These terms represent the heat generating due to mechanical losses in the material and nonlinear effects related to the thermoelastic damping.

2. Theoretical analysis

The beam consists of two layers of different materials. One end of the beam is fixed, and the other one is subjected to periodic loading that is represented in the frequency domain as $F_z \exp(j\,\omega\,t)$.

The starting point of the thermal analysis is the heat-transfer equation [Sisemore *et. al.,*1999]

$$T\frac{\partial s}{\partial t} - \nabla.(k\nabla T) = Q \qquad (1)$$

For a linear thermo-elastic solid, the entropy per unit volume is

$$S = \rho C_p \log\left(\frac{T}{T_0}\right) + S_{elast} \qquad (2)$$

where, in accordance with the Dulong-Petit law, the volumetric heat capacity is independent of the temperature. Furthermore,

$$S_{elast} = \alpha_{Vec} . D\left[\varepsilon - \alpha_{Vec}(T - T_0)\right] \qquad (3)$$

The initial state at $t = 0$ is stress-free, and the initial temperature is across the beam.

The temperature field is decomposed into small-amplitude periodic oscillations and slow temperature rise. The total (viscoelastic) stress that appears in the frequency response analysis is given by

$$S = D[(1 + j\eta)\varepsilon - \alpha_{Vec} T_1] \qquad (4)$$

The corresponding temperature oscillations in the physical domain are

$$\mathrm{Real}[T_1(x_i)\exp(j\omega t)] \qquad (5)$$

The elastic part of the entropy in the frequency domain is

$$S_{elast} = \alpha_{Vec}(\sigma - j\eta D_\varepsilon) \qquad (6)$$

Using the results of the frequency response analysis, model the slow temperature rise with the following equation:[Chandrashekara and Varadarajan, 1997]

$$\rho C_p \frac{\partial T}{\partial t} - \nabla . (k\nabla T) = Q_2 + Q_d \qquad (7)$$

Here T represents the temperature averaged over the time period $2\pi/\omega$, and the heat sources are

$$Q_2 = -\frac{1}{2} Real\left[T_1 Conj(j\omega S_{elast})\right] \qquad (8)$$

and

$$Qd = \frac{1}{2}\,\omega\pi \ \mathrm{Real}\ [\varepsilon . \mathrm{Conj}\ (D\varepsilon)] \qquad (9)$$

For S_{elast} and Q_d use the predefined variables Ent and Qdamp, respectively.

2.1. Finite Element Method

This method involves simple steps described as follow.

a. **Discretization of the domain**: Discretize the geometrically complex domain into set of finite elements called **elements**.
b. **Weak formulation of the differential equation over elements**: Multiply the equation by a weight function and integrate the equation over the domain.
c. **Local Approximation of Solution**: On each element let us attempt to compute the length.
d. **Assemble the Element Equations**: Collect the element equations to get a representation of the whole system. Assemble the element equations to obtain the global system of equations.
e. **Imposition of boundary conditions**.
f. **Solution of the algebraic system of equations**: Obtain the solution of standard matrix equation by direct or indirect (iterative) method.
g. **Post processing**: This final operation displays the solution to system equations in tabular graphical or pictorial form. Other meaningful quantities may be derived from the solution and also displayed.

The finite element solution converges to the true solution as the number of elements is increased. FEM is easy to use and it is also easy to approximate the differential terms of higher order. This method demands a good engineering judgment. The choice of type of element and other basis functions can be crucial [Wiechert ,1889].

3. Modeling of a beam like structure

From the model navigator window select 3D to build up the model in the software of COMSOL multiphysics, then select thermal – structural interaction of application modes and couplings from COMSOL multiphysics/structural mechanics module and select static and transient analysis of solid stress-strain with thermal expansion. This combines solid, stress strain with heat transfer by conduction, including thermal expansion in the structural loads using the temperature field. The following procedures are steps which have been used to build up the model:

3.1 Construction the model

From the command bar select Draw/ Block edit window should be enter all dimensions of the model as mentioned in the table 1. Select command Draw/ create composite object to assembly the two blocks of the model as one block.

Table. 1: Dimensions of the model

Name	Style	Length (*X, Y, Z*)	Axis base point (*X, Y, Z*)
BLK1	Solid	0.01, 0.001, 0.001	0, 0, 0
BLK2	Solid	0.01, 0.001, 0.001	0, 0, 0.001

3.2 Subdomain settings

In the subdoamin edit window from command physics, the main properties of the model should be enter. All properties of the model mentioned in the table 2 have been chosen by select subdomain 1 for Aluminum and 2 for Titanium beta-215 from the library material were existed in the COMSOL software.

Table 2: Subdomain settings of solid, stress-strain model

Subdomain selection	Description	Value	Unit
1	Modulus of elasticity	70	MPa
2		105	
1	Poisson ratio	0.33	Unit less
2		0.33	
1	Thermal expansion coefficient	$23 * 10^{-6}$	1/k
2		$7.06 * 10^{-6}$	
1	Density	2700	Kg/m^3
2		4940	

3.3 Boundary settings

From the command bar select Physics/ Boundary settings, the initial conditions of the model can be entering. Table 3 involves all expression of the initial condition. Select Multiphysics from the tool bar and switch from solid, stress-strain in to heat transfer by conduction then inter subdomain and boundary settings of the model as mentioned in the tables 4 and 5 respectively.

Table 3: Boundary settings of solid, stress-strain model

Boundaries	Initial Condition	Expression
2, 3, 5-9	Constraint	Free
1, 4	Constraint	Fixed
10, 11	Constraint	Free

Table 4: Subdomain settings of heat transfer model

Subdomain selection	Description	Value	Unit
1	Thermal conductivity	160	W/(m.K)
2		7.5	
1	Heat capacity at constant pressure	900	J/(kg.K)
2		710	

Table 5: Boundary settings of heat transfer model

Boundaries	Initial Condition	Expression
6, 10, 11	Thermal insulation	-
1, 4	Temperature	For transient analysis, $T_0 = T_e$ For stationary analysis, $T_0 = 0$
2, 3, 5, 7-9	Heat flux	For transient analysis, $T_{inf} = T_e$ For stationary analysis, $T_{inf} = 0$
		For transient and stationary analysis h = he

Where:

Value	Description
$h_e = 5\ W/(m^2.\ K)$	Heat transfer coefficient
$T_e = 300$ K	External temperature
$T_0 = 300$ K	Initial temperature

4. Meshing

Select initialize mesh from the command Mesh in the toolbar.

5. Solving the model

Select solve problem from the command bar select solve.

6. Plotting

From the tool bar, select Postprocessing/ Plot Parameters/ General, to display the solution of the model. To display total displacement, von mises stresses, temperature gradient and heat flux occur in the model, select subdomain then select plot type from predefined quantities. Curves of velocity and total displacement across z-axis drawn by select Postprocessing/ Cross Section Plot Parameters/ Line /Extrusion plot. For the cross section plot, should be entered the data mentioned in the table 5.

Table 5: Data of Cross section plot

Name	Value
X_0	0
Y_0	0
Z_0	0
X_1	0.01
Y_1	0
Z_1	0.001

4. Results and discussion

The velocity of vibration model along *Z*- axis shown in the figure 1, however, during this curve the value of velocity starts from 2.25 m/s at *Z* equal to 0.001. From this point, the velocity sharply falls down in to approximately 0.4 m/s, then drastically decrement from the top of the model until reached in to zero at the base of the model.

Vonmises stress developed during the model, focused at fixed side of the model as denoted with red color with high value 1.394 Gpa as shown in the contour of vonmises stress in the figure 2. Vonmises stress decrease and reached into minimum values at the free end of the model as denoted with blue color during contour of vonmises stress.

Stresses generated along *Z*-axis of the model clearly shown in the figure 3, as comparison this curve with that of vonmises stress, extremely same behavior between these curves. Where high stresses concentrated at the fixed side and decrease at the free end of the model. During this curve, stresses alternatively moves at the positive and negative sides along *Z*-axis. The values of the stress start from high value $1.1*10^8$ Pa at the fixed side then decreased in to $-0.8*10^8$ Pa at *Z* equal to 10^{-4}. Stress repeated move into positive side from this point and continuous alternatives moves along *Z*-axis until approached in to the free end of the model and reached to zero.

Figure 4, shows the total displacement that occur in the model and from this contour high displacement exist at the free end of the model. This behavior obviously shown during curve of total displacement in the figure 5. In this context, during this curve, the maximum total displacement found at the free end $8*10^{-4}$ then from this point the values of total displacement decreased until reached in to zero at the fixed end of the model.

The source of heat initiates from the second block of the model while approached to diminish at the first block of the model. This attitude shown in the figure 6 may be related to the type of materials that have been used to build up the model. Figure 7, shows the maximum value of heat source exist at the top block with $2.9*10^8$ W/m^3 at the fixed end of the model, and from this point the value of heat source decrement in to $0.5 *10^8$ W/m^3 at Z equal to 0.1. Then the value again back in to $2 *10^8$ W/m^3 and decreased as approached in to the base of the model.

Contour of temperature distribution in the model shown in the figure. 8, where maximum temperature 436.253 K occur at the region closed to fixed end, while the other regions shown graded values of the temperature until reach into 300 K of external temperature. Heat flux that generated along the model as a result to the vibrations of the structure, the behavior of heat flux differs in the transient and stationary cases. Where, in the transient case, the heat flux focused on the second block of the model. The maximum values reached into $1.162*10^6$, while for stationary case, the maximum value is equal to $4.554*10^6$, in this context of this case, this value concentrated at first block of the model and all of these data mentioned in the figures 9 and 10 respectively.

5. Conclusion

The free end of the model from one side enhances the variation process which starts with velocity 2.25 m/s at *Z* equal to 0.001 at the top of the model, fall down in to 0.4 m/s, and reached in to zero at the base of the model. Same behavior of stresses generated along *Z*-axis and vonmises stress. Maximum stresses and minimum displacement concentrated at the fixed side, while high displacement $8*10^{-4}$ exists at the free end of the model.

Maximum temperature 436.253 K occurs at the region closed to fixed end, while the other regions the shows graded values of the temperature until reach into 300 K of external temperature. The source of heat initiates from the second block of the model while approached to diminish at the first block of the model. These attitudes related to the type of materials which have been used to build up the model. The behavior of heat flux differs in the transient and stationary cases. In the transient case, the heat flux focused on the second block of the model. While for stationary case, concentrated at first block of the model.

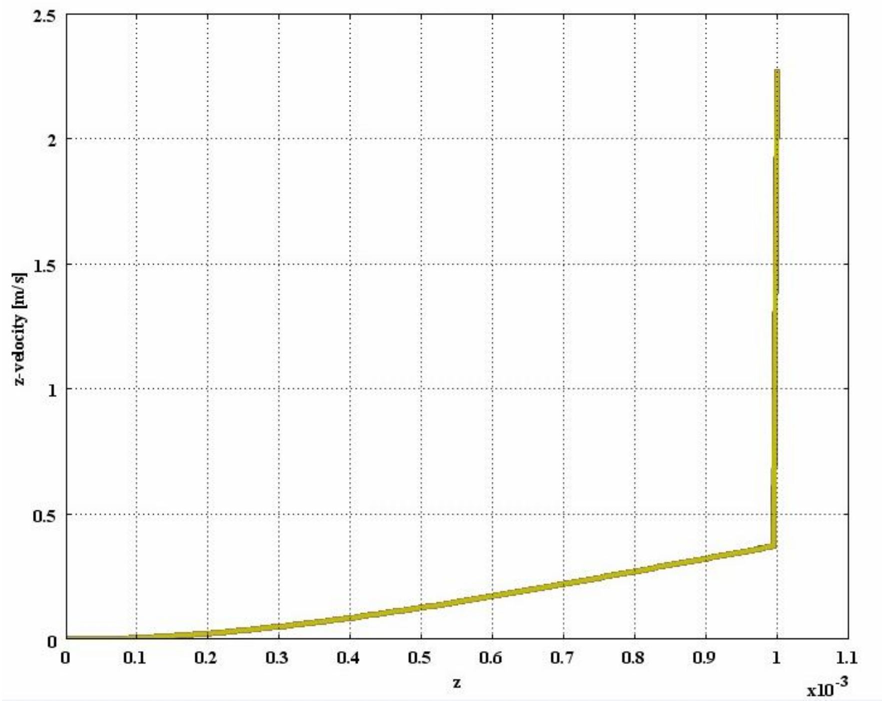

Figure. 1: Z- velocity of variation

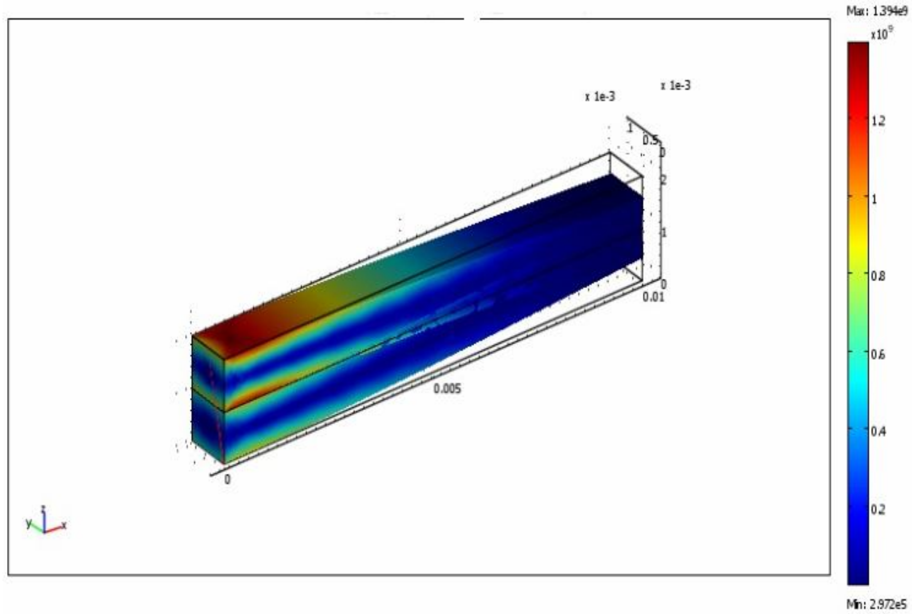

Figure. 2: Von Mises stress

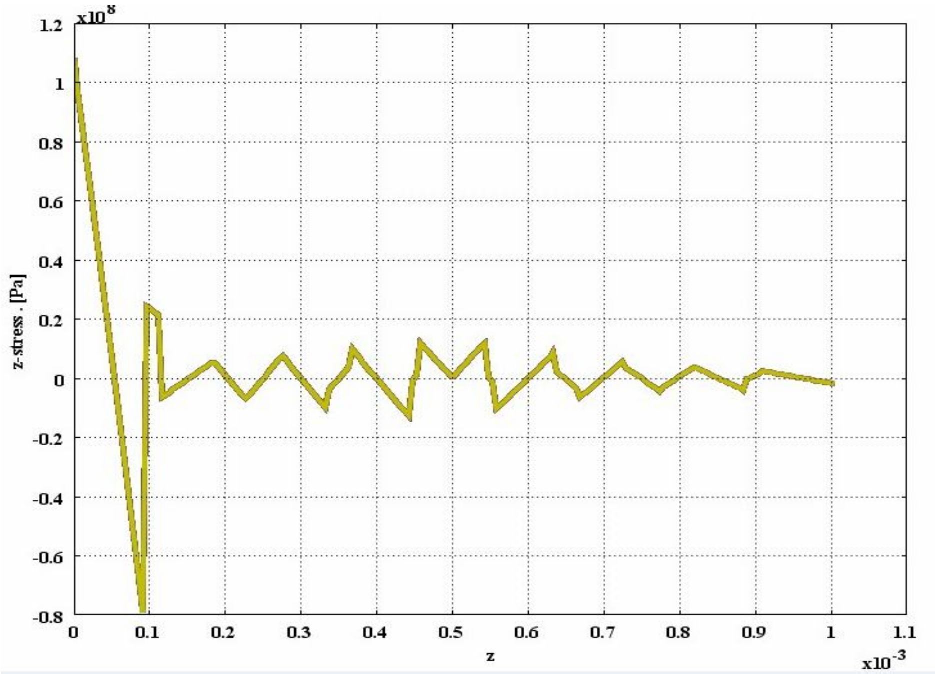

Figure. 3: Z- stress

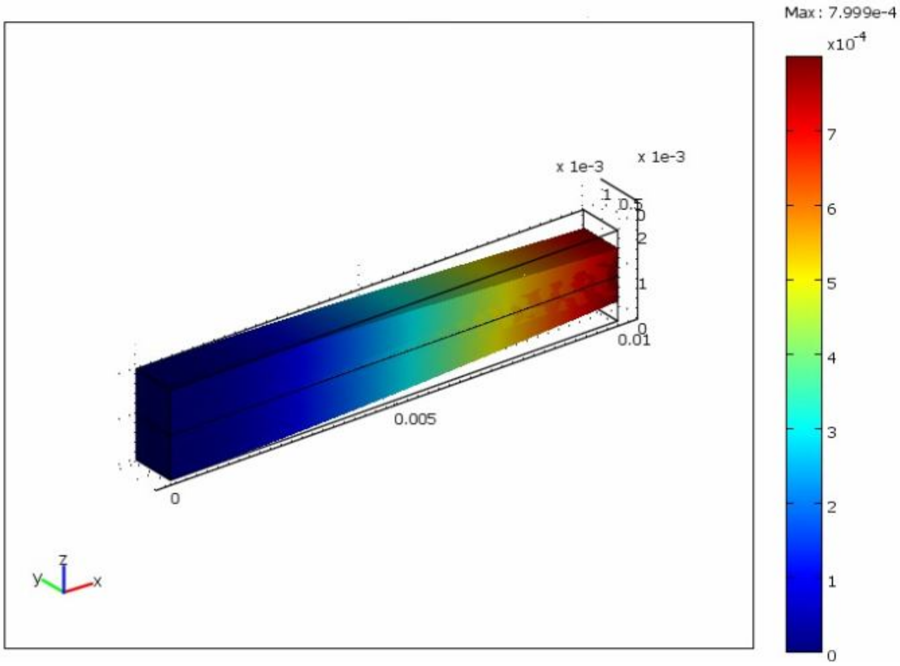

Figure. 4: Total displacement contour

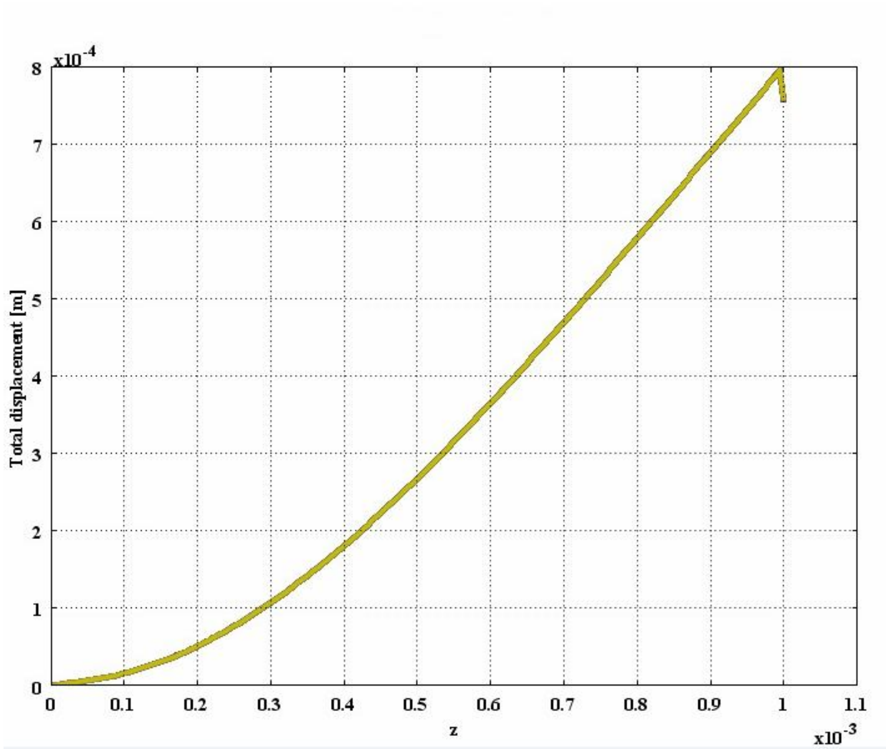

Figure. 5: Total displacement curve

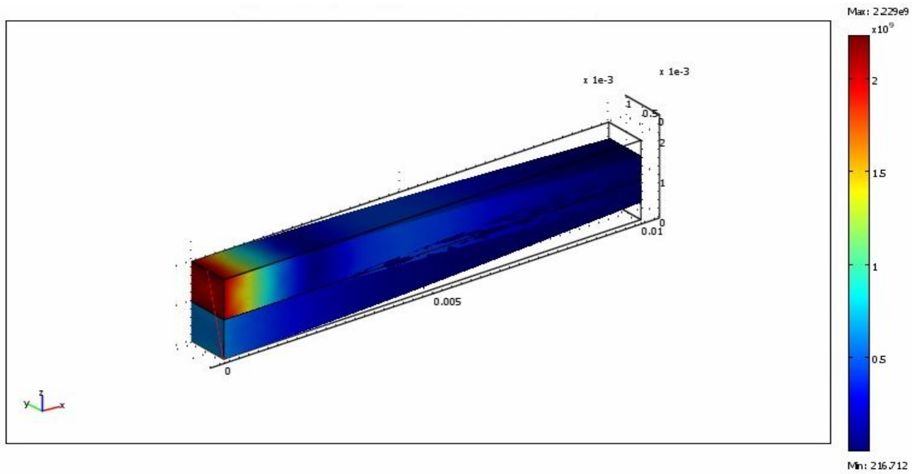

Figure. 6: Heat source

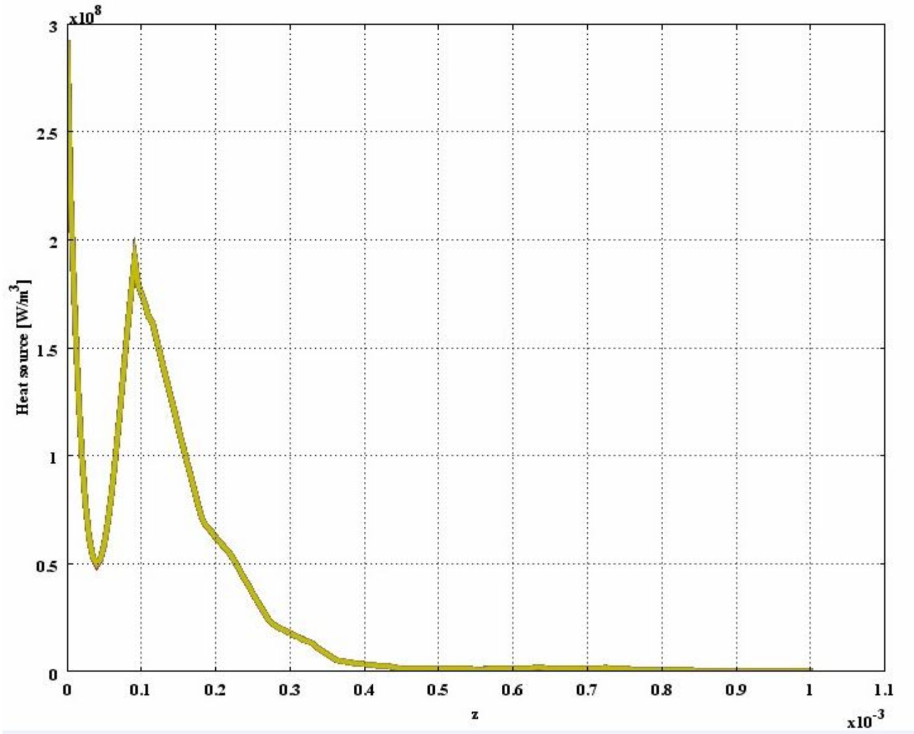

Figure. 7: Heat source curve

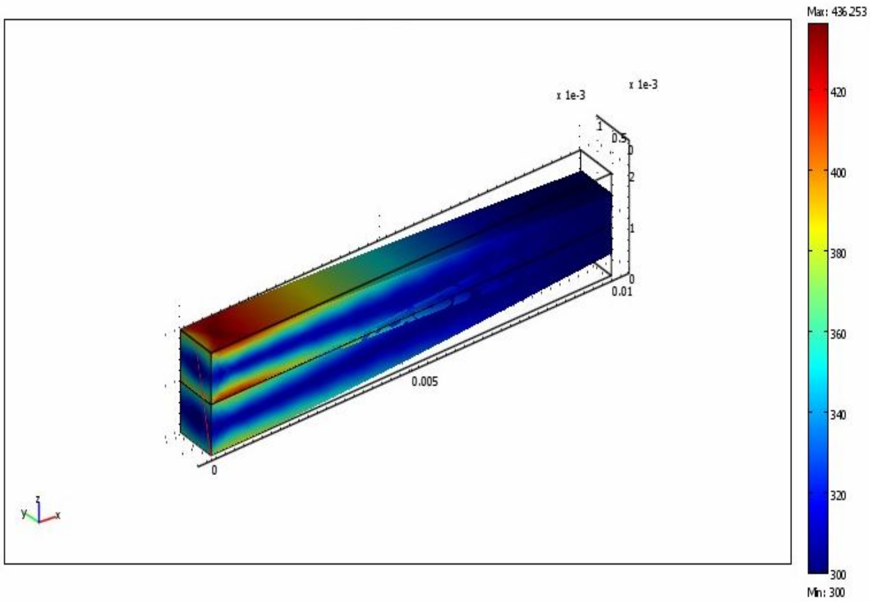

Figure. 8: Temperature distribution

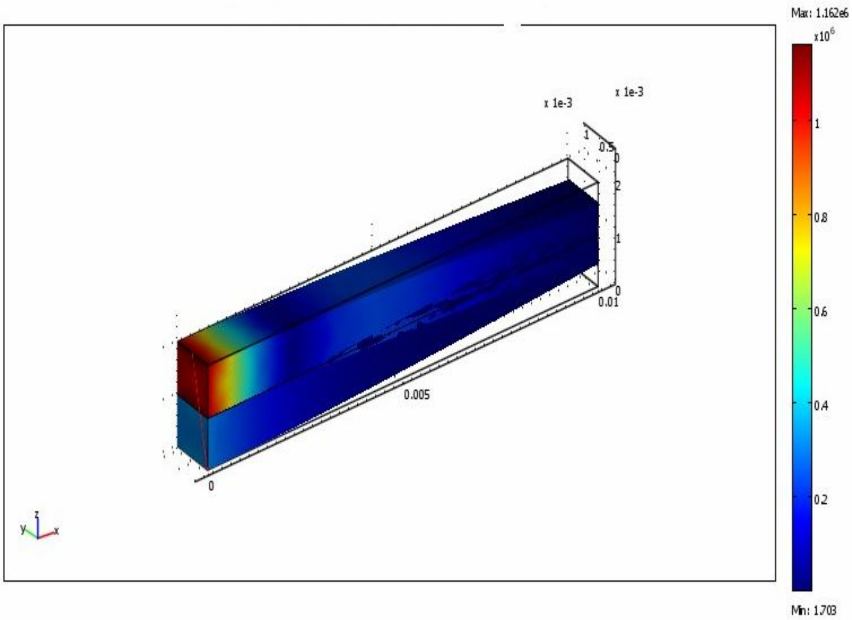

Figure. 9: Heat flux in transient case

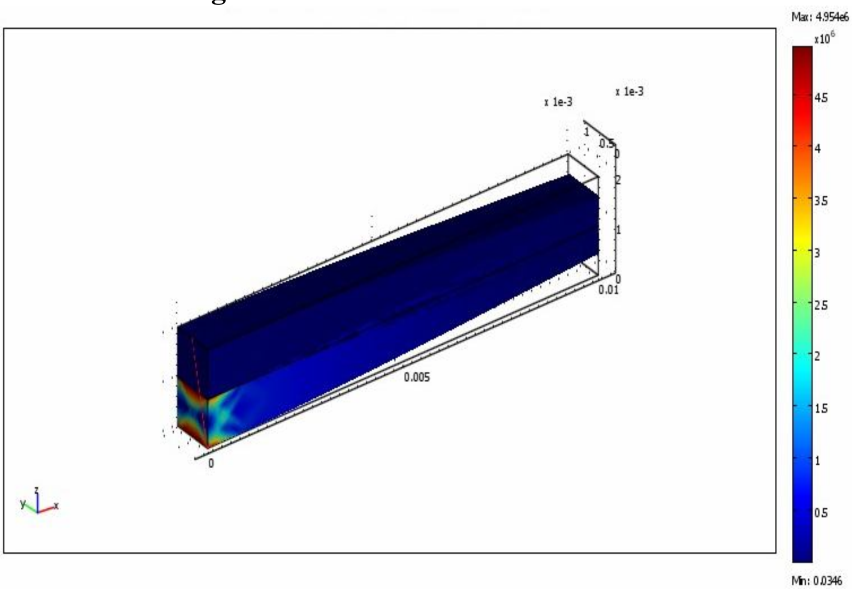

Figure. 10: Heat flux in stationary case

References

Duwel,A.,Candler, R.N.,Kenny, T.W. and Varghese, M. J. 2006. *Microelectromech. Sys.*, vol. 15, no. 6, pp. 1437–1445.

Setareh, M.,2011."Vibration Studies of a Cantilevered Structure Subjected to Human Activities Using a Remote Monitoring System."*J. Perform.Constr. Facil.*,25(2), 87–97.

Zhi Hua Nie *et. al.*, 2010, Advanced Materials Research, 163-167, 3935

Jiazhu HU and P.Frank PAI,2011. *Int.J. Bifurcation Chaos* **21**,3259.DOI: 10. 1142/ S0218127411030507

Jiazhu, HU and P. Frank PAI, 2011. *Bifurcation Chaos* **21**,3259. DOI: 10.1142/ S0218127411030507 Bifurcation Structure for Modulated Vibration of Strings Subjected to Harmonic Boundary Excitations

Sisemore, C.,Smaili, A. and Houghton, R., 1999."Passive Damping of Exible Mechanism System: Experimental and Finite Element Investigation," The 10th World Congress of the Theory of Machines and Mechanisms, Oulu, Vol. 5, pp. 2140-2145.

Chandrashekara, K. and Varadarajan, S.,2004 ."Adaptive Shape Control of Composite Beams with Piezoelectric Actuators," Intelligent Materials Systems and Structures 8, 1997, pp112-124e IEEE EMBS, San Francisco, 1-5 September, pp. 2758-2761.

Wiechert, E ,1889."Ueber elastische Nachwirkung", Dissertation, Königsberg University, Germany