TECHNICAL UNIVERSITY of LODZ
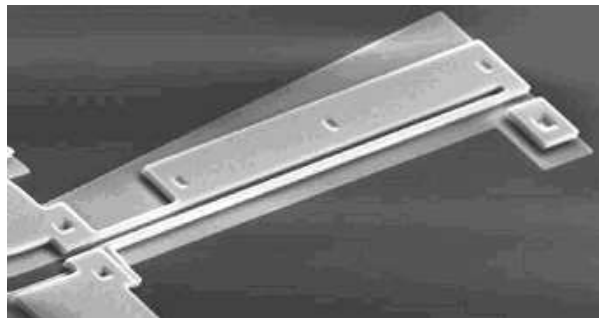
DEPARTMENT
of MICROELECTRONICS and COMPUTER SCIENCE

Tutorial for IFE Students of

RECENT TECHNOLOGICAL ADVANCES

Thermo-Electric Actuator

Przemysław Sękalski, M.Sc.
sekalski@dmcs.p.lodz.pl

Lodz, February 2003

1 Introduction

The standard form of the layout of the thermo-electric actuator is shown in Fig.1 , additionally the manufactured device is shown in Fig. 2. It is a 'U'-shaped structure that uses differential thermal expansion to achieve motion along the wafer surface. When a voltage is applied to the terminals, current flows through the device. However, because of the different widths, the current density is unequal in the two arms. This leads to a different rate of Joule heating in the two arms, and thus to different amounts of thermal expansion. The thin arm is often referred to as the hot arm, and the wide arm is often referred to as the cold arm.

Backward deflection of the device is possible by temporarily delivering too much power to the device. Then wider arm is exaggerating more than second one. Although too much power dissipated in the device can easily destroy it. At the right level the hot arm will plastically deform, decreasing its length. When the power is switched off, the hot arm should return to previous position.

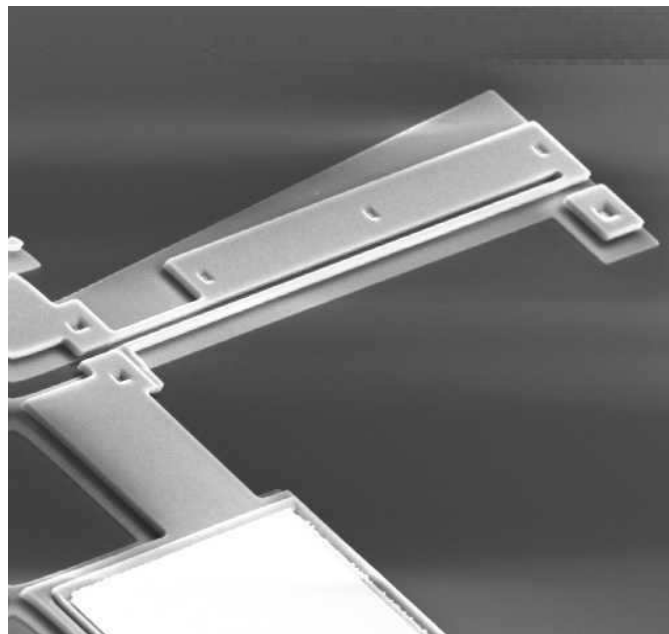**Fig. 1.** *Layout for standard electro-thermal actuator.*

Fig. 2. *SEM of an electro-thermal actuator. The cold arm of the thermal actuator has a width of 15 um and the hot arms have a width of 2 um.*

There is also the option of including a "hook" on the end of the design. This hook is a proven design for connecting the actuator to a rod (as in Fig. 3). It could be useful for constructing micromachines as micromotors because this solution increase the available force (Fig. 4).
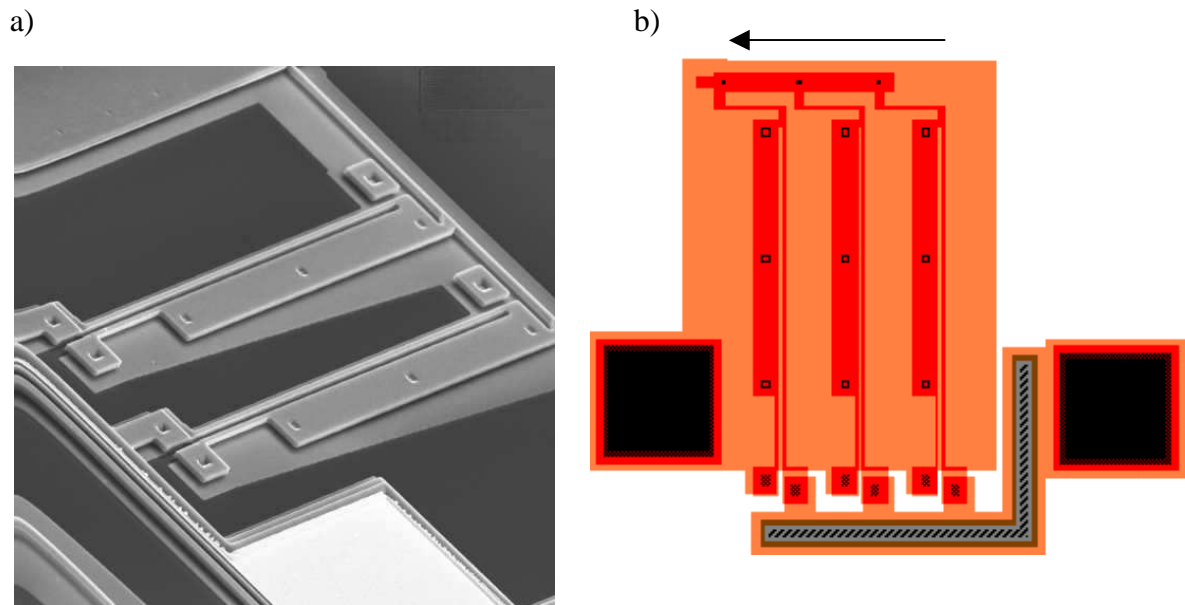
a)   b)

Fig. 3. *Coupled electro-thermal actuators SEM view a) and layout b). The cold arms of the thermal actuators have a width of 15 um and the hot arms have a width of 2 um.*
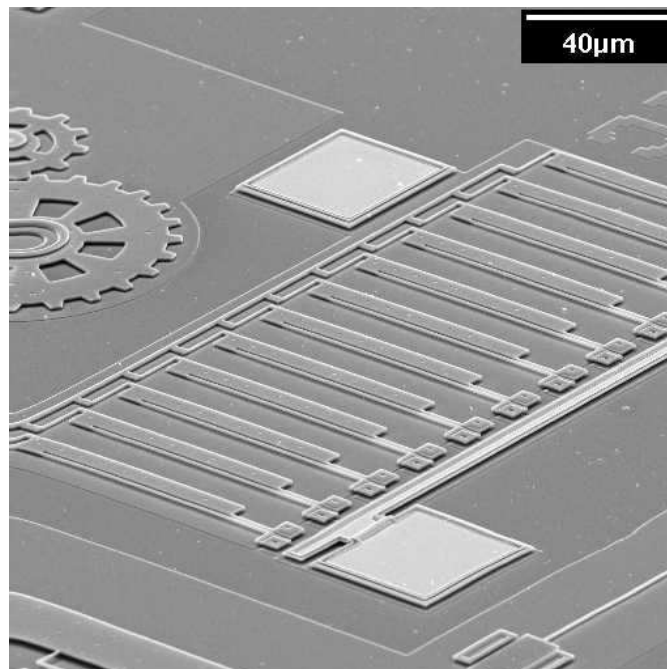

Fig. 4. *SEM of an array of electro-thermal actuators.*

2 Theory

2.1 Electrical

The resistance of the actuator can be determined by the following equation:

$$R = \left(\frac{L}{W_h t} + \frac{L - F}{W_c t} + \frac{F}{W_f t} \right) r$$

(1)

Where, R is the total resistance, L is the total length of the actuator, F is the length of the flexure, t is the actuators thickness, W_h is the width of the hot arm, W_c is the width of the cold arm, W_f is the width of the flexure, and r is the resistivity of the thermal actuator's material. For example for MUMPs technology the thickness t of the actuator will be either 2 µm or 3.5 µm, depending on whether the actuator is a POLY1 or double-thickness structure respectively. The exact value of the resistivity r should be determined from the process data, but for the MUMPS process is on the order of *2e-3* Ω*cm. A more accurate result can be obtained by replacing L with the total length less the width of the bar connecting the hot and cold arms (i.e. *Length -> Length - Cross*) for the W_h term.
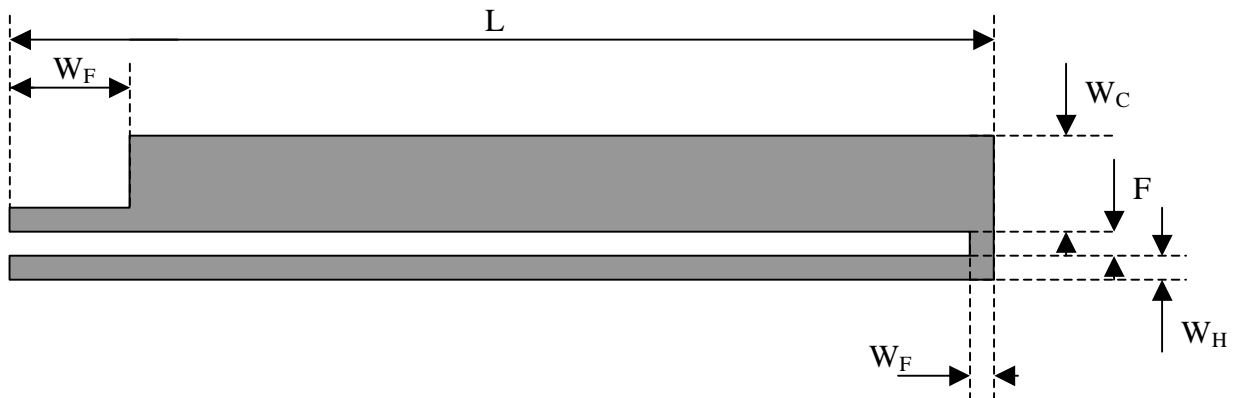

Fig. 5. *Sample of electro-thermal actuators.*

There is a major difficulty in apply the above equation. In thin beams (<10 µm wide), the resistivity could be significantly lower. This is due to excessive doping at the sidewalls of the beams. The wider the beam, the smaller the effect of side wall doping will be. Sidewall doping will affect 2-3 µm beams, but not 50 µm beams. Since the hot arm and the flexure will be very narrow, their resistivity will be significantly less than the quoted value. Using the above equation will grossly overestimate actuators' resistance.

2.2 Mechanical

A thermal actuator can be considered to be a cantilever. The cantilever has a no load deflection based on some input parameter (such as voltage or current). The following formula can be used:

$$d = d_0(a) + CF$$

(2)

Where, d is the deflection (length), d_0 is the no load deflection (length) and is a function of some input parameter a; C is the compliance (length per force), and F is the applied force. Typically, $d_0(0) = 0$. Values for d_0 have to be determined numerically. While a reasonable expression for the current density throughout the actuator could be found, the resistivity is a function of temperature. As the temperature will vary through out the device from room temperature to well over 1000^oK, this cannot be ignored. Also, the coefficient of thermal expansion is also a function of temperature.

Determining the value of d_0 requires an electro-thermal-mechanical simulation. See below.

It should be possible to determine the compliance C of an actuator with a closed form equation. However, the actuator cannot be considered as a simple cantilever, and so a more advanced approach is required. However, the compliance can also be determined using numerical methods. A mechanical simulation, easily and quickly performed using Ansys, can be used to determine the compliance.

Table 1: *Mechanical coefficients for several electro-thermal actuators (thickness = 2 μm).*

Length	Flexure	d_0(5 V)	Compliance	Avg. Err.	Max. Err.
200 µm	20 µm	-12.8 µm	0.7698	-	-
200 µm	30 µm	-10.5 µm	0.8261	35 nm	206 nm
200 µm	40 µm	-8.40 µm	0.8112	174 nm	324 nm
200 µm	50 µm	-6.14 µm	0.8578	24 nm	57 nm

The parameters in *Tab. 1* were determined by fitting the above linear equation to actuators simulated using Ansys. The applied force ranged from -5 µN to +15 µN. The quoted average and maximum errors are the difference between the fitted equation and the results from Ansys. Note that stick/slip motion will create much larger errors than those caused by fitting.

The no load deflection for a double-thickness actuator d_0 should be the same as the no load deflection for a single thickness actuator. However, the compliance C of a double-thickness actuator should decrease in proportion to the actuator increase in thickness:

$$C_{dt} = \frac{t_o}{t_{dt}} C_0$$

(3)

Where, C_0 and C_{dt} are the compliances of the single-thickness and double-thickness actuators, respectively. And where, t_0 and t_{dt} are the thickness of the single-thickness and double-thickness actuators, respectively.

Table 2. *Mechanical coefficients for several electro-thermal actuators (thickness = 3,5μm).*

Length	Flexure	d_0 (5 V)	Compliance	Avg. Err.	Max. Err.
200 µm	20 µm	-12.8 µm	0.4399	-	-
200 µm	30 µm	-10.5 µm	0.4690	-	-
200 µm	40 µm	-8.40 µm	0.4852	-	-
200 µm	50 µm	-6.14 µm	0.4909	32 nm	151 nm

2.3 Analytical calculation

It is also possible to obtain the result of the displacement of the end of electro-thermal actuator from basic analytic calculation. The brief description is presented below.

The surface heat flow φ [W/m^2] is given by the equation:

$$\varphi = \frac{\phi}{S} = -\lambda \mathrm{grad}T$$

(4)

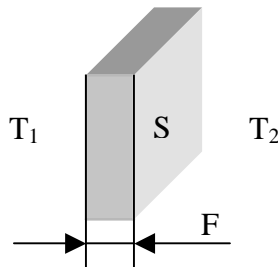where λ is a thermal conductivity, T is a temperature, ϕ is a heat, S is a surface.

Fig. 6. *Used notation.*

According to the Fig. 6 the heat can be described as:

$$\phi = \lambda S \frac{\Delta T}{F}$$

(5)

There is only a Joule heat in the structure therefore the difference of temperature can be found from equation:

$$\Delta T = \frac{U^2}{\Delta R} F \frac{1}{\lambda S}$$

(6)

The difference of resistance ΔR originates from the distinction of the hot an cold arm, therefore at last we can write:

$$\Delta T = \frac{U^2 W_C F}{\rho L (W_H - W_C) \lambda}$$

(7)

From the definition of the thermal coefficient of expansion (TCE) α we can write the expression for lengthen of the hot arm:

$$\frac{\Delta L}{L} = \alpha \Delta T$$

(8)

Otherwise, after some geometrical simplification ($\Delta L^2 << 2 * \Delta L * L$) the displacement of the actuator is given by:

$$\Delta x = L \sqrt{\frac{2 \Delta L}{L}}$$

(9)

Finally, the dispalcement of the end of the electro-thermo actuators Δx is given by:

$$\Delta x = L \sqrt{2 \alpha \frac{U^2 W_C F}{\rho L (W_H - W_C) \lambda}}$$

(10)

2.4 Creating a layout and building of 3D computational Mesh

2.4.1 Pre-Processing

The main aim of this point is creating a layout of 3D model of the electro-thermal actuator. The shape and necessary dimensions of the actuator are shown in fig. 7.
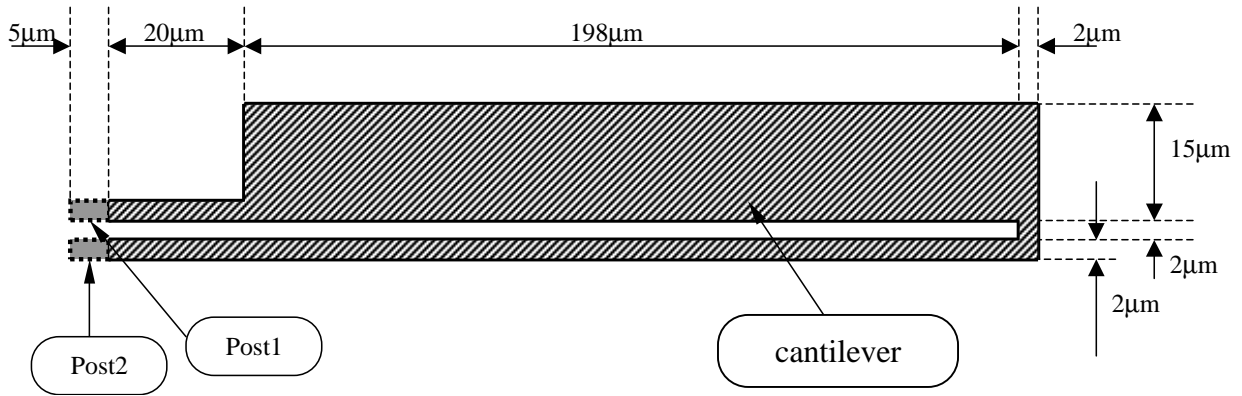

Fig. 7. *Layout of the simulating structure with dimensions.*

Using the knowledge you has acquired in previous tutorials you can try build this structure by your own, without aids given below and move directly to paragraph 2.4.2. The right top corner of *post2* should be in the coordinate origin. At point <220,2> add the pointer – small block 1um per 1um. It will be used to observe the actuator displacement.

2.4.1.1 Detailed description how create the layout.

2.4.1.1.1 Preparing

Start a new project in CFD – Micromesh.

Firstly set *Display Unit* as micrometers (from *View* in top toolbar select *Display Units*). Then click on *Change Boundary* button in toolbar, and set *Dimensions and Resolution* as it is given below:

- X : min = -10μm; max 240μm; resolution: 1000;
- Y : min = -5μm; max 20μm; resolution: 100;
- Z : min = 0μm; max 10μm; resolution: 50;

At the end, click on the *Change Grid* and set *Grid* to 1μm.

2.4.1.1.2 Creating layout

Click on *New Layer* button in the top toolbar to create a new layer. In the list of layers, click RB on the *NEW: Air (deposit)* layer, in the *Layer Properties* window, which appears, click *Edit Material* button. Click *Add* button in *Materials* window. In the *New Material* window type names of new layers to be added: **post1**, **post2**, **cantilever**. Close the *Material* window. Back in the *Layer Properties* window of the first created layer:

- in the *Material* pull-down list, change the name from **Air** to **post1**
- change *Thickness [μm]* to 4
- accept changes

Draw the first post of the actuator:

- Pos.X: –5; Pos.Y: 2; SizeX: 5; SizeY: 2

Add a new layer, in the *Layer Properties* window of the second created layer:

- in the *Material* pull-down list, change the name from **post1** to **post2**
- change *Thickness [μm]* to 4
- accept changes

Draw the second post of the actuator:

- Pos.X: –5; Pos.Y: -2; SizeX: 5; SizeY: 2

Add a new layer, in the *Layer Properties* window of the second created layer:

- in the *Material* pull-down list, change the name from **post1** to **cantilever**
- change *Thickness [um]* to 2
- accept changes

Draw the main part of the actuator using the polygon. Appropriate coordinates are given below:

X	Y
0	– 2
220	– 2
220	2
221	3
220	3
220	17
20	17
20	4
0	4
0	2
218	2
218	0
0	0

Add a new layer, in the *Layer Properties* window of the second created layer:

- in the *Operation* pull-down list, change the name to **mesh cells size**
- change *Mesh cells size* to 0.3
- accept changes

Draw areas where the mesh cells size will be smaller (it can be used for a thin beam) :

- Pos.X: 0; Pos.Y: -2; SizeX: 220; SizeY: 2
- Pos.X: 0; Pos.Y: 2; SizeX: **22**; SizeY: 2 (after meshing look carefully what causes parameter sizeX=22)

SAVE project as *cantilever*.

Click on the *Solid View* button in the top toolbar to visualize your work. Accept the default settings. You can verify whether you correctly have built the layout. You should achieve structure as presented in Fig. 8.
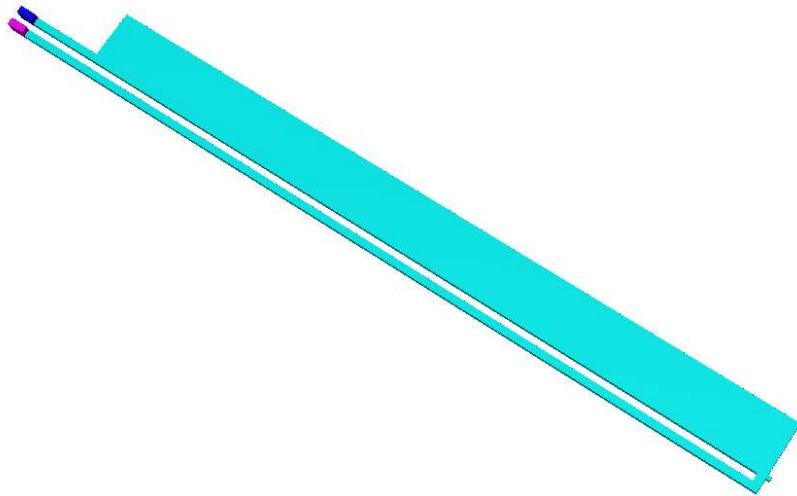

Fig. 8. *Model of the actuator.*

2.4.2 Build a 3D Computational Mesh

From the top toolbar select the *Prism-Hex Mesh* button to build a 3D computational mesh. The *Grid Parameters* window appears, set parameters as described below:

- Desired number of triangles per XY plane (including air): 1100
- Other leave as default
- Click on *Add Z Mesh* and set forward distribution from 0 to 2 um with six desired points.

Accept changes and start meshing.

At this moment you have a meshed fully usable 3D model of the thermo-electric actuator. Therefore you are ready to begin electro-thermo-mechanic simulations.

2.5 Setting up a simulation.

Using CFD-GUI set up parameters for simulations as given below.

2.5.1 PT (problem type)

According to description of action, we need to use four modules. *Electric* and *Heat transfer* for Joule heat and *Grid deformation* and *Stress* for deflection of the actuator.

2.5.2 MO (module options)

For increasing accuracy of results we should change parameters in module options in the stress panel as below:

Set Element Order to *Second* (for increasing accuracy)

Select *Coupling* to *Two way coupling* (for enable coupled thermo-mechanic calculation)

In *Electric* panel in *Electric Solver* options select *DC conduction*.

2.5.3 VC (volume conditions)

Select *post1* and *post2* and set them as an *Aluminum*. (you need to import proper values)

Select *cantilever* and set it as an *Silicon*

To enable deflection of the beam we need change volume conditions. Change setting *Module* to *Stress*, select *cantilever*, and apply stress. Select *solid type* to *elastic*. Verify if the reference temperature is 300K.

Change the *electrical conductivity* of the *cantilever* from 1e-4 to 1e-5Ωm.

2.5.4 BC (boundary conditions)

Apply the following BCs:

- select *air_cantilever*, then in *Stress* panel set *Sub_type* to *implicit pressure* (**this option appears only in old version of ACE**+) – it allows apply force to the cantilever. In newest version this step is done automatically.

- select *cantilever_post1* and *cantilever_post2*, then in *Stress* panel set *Sub_type* to *fixed displacement*. In *Electric* panel chose *potential* and set *voltage(real)* to value *+10V* and *–10V* (appropriately) - its cause fixing ends of cantilever and apply voltage to them.

- select *air_post1* and *air_post2*, then in *Heat* Panel set *sub_type* to *isothermal*, and set *temperature* to 300K – this steps sets endings of the cantilever as a heat sink.

2.5.5 IC (initial conditions)

Check up the *initial temperature* if it is *300K*.

2.5.6 SC (solver conditions)

Increase number of the maximum iterations to 20.

2.5.7 OUT (output data)

In *Graphic Panel*, set output variables you want to see in CFD-View. Because of specification of the problem interesting are:

Static temperature	*Principal strain*
Electrical conductivity	*Electric potential*
Displacement	*Conduction current density*
Principal stress	

2.5.8 RUN

Submit simulations under current name.

2.6 Displaying Results of Analysis.

Using CFD-View observe:
- distribution of temperature (T). Where is the hottest area? Why it is only in the center of the beam?

- internal stress (*vonMises*). Where are the biggest stress and strain? In which area the structure of actuator is the most fragile?

- x and y displacement (*xdisp* and *ydisp*), compare outcome with the result of an analytical calculation (see paragraph 2.4). The appropriate values of material constants can be found in VC module. Why the results are different? What kind of approximations are used during analytical analysis? Which result are closer to reality?

2.7 Additional simulations for eager students.

o Prepare transient simulation for voltage ±5V from 0 to 0,01s step 0,002s. Observe y displacement. Why at the first phase of movement it deflect more than at the end?

o Using Simulation Manager observe y displacement and temperature for different voltage from 0 to 50 V (step 10V).

o Make new mesh, increasing number of elements to 2200, and set simulation as the first one. What kind of difference you may observe? Is it necessary to make such a long simulation? Is it useful?

References

[1] John H. Comtois, and M. Victor. "Applications for surface-micromachined polysilicon thermal actuators and arrays," <u>Sensors and Actuators A</u>. vol 58, no 1, pp 19-25. (1997)

[2] John H. Comtois, M. Adrian Michalicek, and Carole Craig Barron. "Characterization of Electrthermal Actuators and Arrays Fabricated in a Four-Level, Planarized Surface-Micromachined Polycrystalline Silicon Process," (Invited Paper) <u>1997 International Conference on Solid-State Sensors and Actuators</u>. Chicago, IL, June 16-19, 1997. vol 2, pp 769-772. (1997)

[3] David A. Koester, Ramaswamy Mahadevan, and Buzz Hardy. <u>MUMPs Design Handbook Rev. 5.0</u>. Cronos.