User Manual

MEMS Piezoelectric Vibrational Energy Harvesting Lab

Jorge Mario Monsalve Guaracao[1]

Welcome to the **MEMS Piezoelectric Vibrational Energy Harvesting Lab!** This nanoHUB tool will allow you to explore the performance of a system that is capable of scavenging potentially wasted energy from mechanical vibrations by converting them to electrical power. This is accomplished because of the *direct piezoelectric effect* which allows some special crystals to accumulate electrical charge when a stress is applied to them. In order to set-up the simulation, this tool will ask you to design and test your device within the following four stages on which different parameters are to be specified:

Designing the Structure and Choosing the Materials of the Cantilever

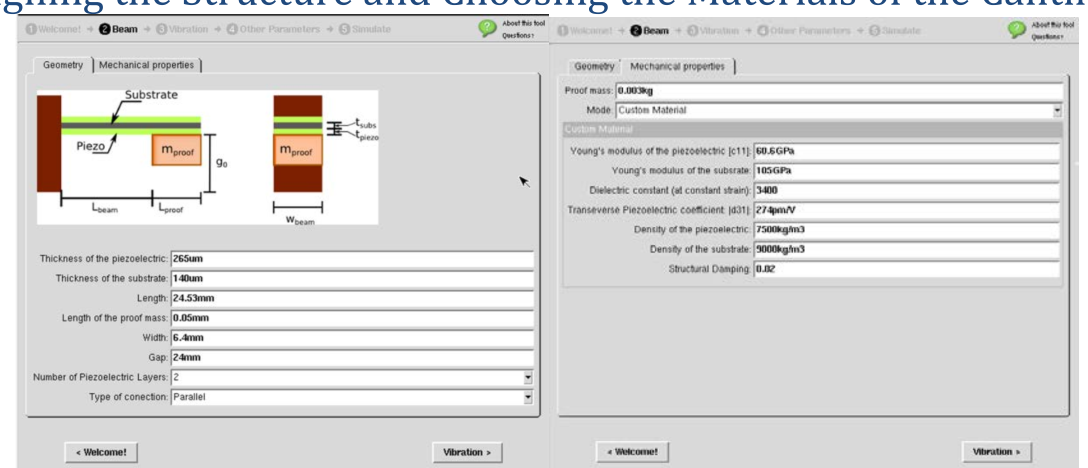

FIGURE 1 BEAM DESIGN TAB

The first step of the simulation is to design the geometry of the beam. As it can be seen, the harvesting device consists of a cantilevered beam with a mass attached on the end. This tool is currently capable to simulate two kinds of piezoelectric harvesters: *Unimorph* and *Bimorph*. The *Unimorph* beam consists of only one piezoelectric layer and a metallic substrate layer; whereas the *Bimorph* has two interconnected piezoelectric layers in a sandwich-like arrangement with the metallic substrate. Beams with more substrate or piezoelectric films (i.e. *Multimorph*) are not supported in this version.

---

[1] BS Mechanical & Electronics Engineering. Universidad de los Andes, COL. jm.monsalve886@uniandes.edu.co.
Undergraduate researcher at the nanoHUB SURF program, Purdue University.

In the “Geometry” tab all the dimensions (length, thickness, width, etc) are to be specified. If no substrate is to be simulated, it is possible to type ‘0’ for its thickness. In small devices the space that the proof mass occupies plays an important role; however, it is possible to also specify ‘0’ for its length if the “point-mass” approximation is to be used. The gap between the beam and the floor doesn’t play an important role at millimeter-sized devices; however, it could affect greatly the squeeze force for MEMS-scale cantilevers.

After specifying these dimensions, the materials for the beam and the value of the proof mass can be chosen in the “Mechanical Properties” tab. There are two modes to set the materials: “Material from Database” and “Custom material”. In the first mode you only have to pick some materials from within a list. A table with the values used for their properties is provided in the documentation. In the second mode it is possible to input manually all the relevant properties, which is useful when comparing against experimental results. The required properties include the modulus of elasticity and density for both materials as well as the transverse piezoelectric coefficient and dielectric constant for the piezoelectric films. The structural damping factor can be known from an experiment, or estimated with a reasonable value such as 0.02 for a 1% damping ratio.

Specifying the Input Vibrations

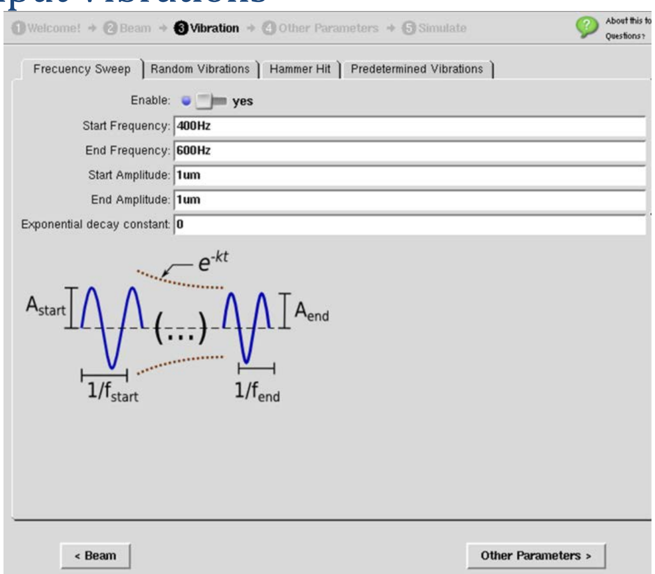

FIGURE 2 INPUT VIBRATION SET-UP

The next stage of the simulation allows testing the device under different excitation conditions. Here an input vibration signal will be generated according to the provided options. This tool allows generating sinusoidal, random and impulsive vibrations, and includes some predefined values for measured oscillations from different sources. Also, a recorded file can be uploaded. All of these can be tested at the same time since internally the program will add all the enabled options into a single input signal.

The first three tabs of the input vibration stage are meant to generate an ideal input signal, which is helpful for characterization purposes. The first option of this stage (Frequency sweep) calculates a sinusoidal signal with a linearly varying frequency and/or linearly varying amplitude. This is useful to let you know the influence of these variables in the output power. If desired, an exponential decay can also be simulated to study transient vibrations. It is important to note that this is the only tab that receives amplitude (in meters) as a parameter; the other ones will calculate directly the acceleration (in *g*). The second tab generates a random signal distributed with the Standard Normal function. Its “amplitude” will be the expected RMS acceleration. The third tab makes an impulsive train (series of “Hammer hits”) evenly distributed in the timespan. Here, the energy of each impulse (in *g*-ms) is asked which sets the peak acceleration depending on the internally calculated time step.

The other two tabs are meant to test more realistic oscillation conditions. The “Predetermined vibration” tab contains a list of common sources of vibrations. These were measured by S. Roundy (2003)[2] and basically consist of a monotonic sinusoidal wave at a specified frequency and amplitude representing the main measured harmonic plus a small amount of Gaussian noise. Finally, a measured vibration can be also uploaded as the input for the simulation. In this case, the program will ask for a comma-separated-value (CSV) file with the values of the acceleration in ***g*** located in a single **row**, separated from each other just by a comma. Also, it is necessary to indicate the sampling frequency of the measurement. Only in this case the simulation will last for the same period that the vibration was recorded and can’t be adjusted.

Selecting a Circuital Topology

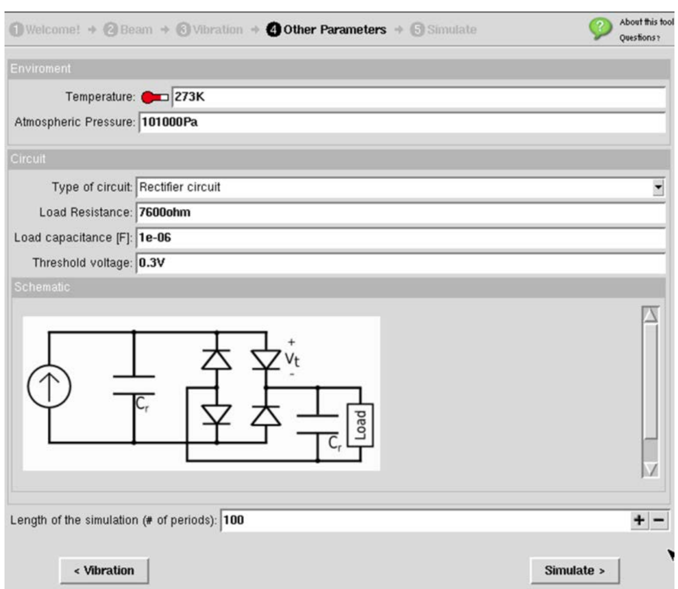

FIGURE 3 "OTHER PARAMETERS" TAB

[2] Roundy, S. (2003). Energy Scavenging for Wireless Sensor Nodes with a Focus on Vibration to Electricity Conversion. University of California, Berkeley.

The final stage is the selection of some other necessary parameters distributed in three groups: environmental conditions, circuit topology and length of the simulation. The environmental conditions alter the density and viscosity of the surrounding air. This affects the aerodynamic drag calculation, which becomes relevant at very small scales. In the circuit topology two options are supported: Resistive load and full-wave rectifier. In both cases a load resistance and capacitance are to be specified. In the resistive circuit the load capacitance would be connected in parallel to the load and is internally added to the capacitance of the piezoelectric; therefore, it can be set-up to zero if desired (this is clearly not desired if a rectifier is used). If a full-wave rectifier is chosen a threshold voltage for the diodes has to be supplied since within the program they are modeled as ideal switches. Finally, the length of the simulation is indicated. This will define the simulation period in terms of a reference frequency, which in most of the cases will be the natural frequency of the beam. Technically, the program makes an internal decision that depends also on the type of input vibrations: If it is a frequency sweep then it is calculated in terms of the lowest frequency; however, if the lowest frequency is too low it chooses the natural frequency as reference in order not to make the simulation very long.

Warning: convergence errors might happen if the RC constant is too low

Although the program has some reasonable bounds for the resistance and the capacitance of the circuit, a "Convergence error" might happen if the RC time constant is too low compared to the natural period of the mechanical structure. This makes the differential equation very unstable for a couple of its coefficients will be very high and the program will end the execution very early in the simulation. In other words, to avoid this error care must be taken no to simulate resistances that would be too close to the "short-circuit" condition. Also, this could happen if some capacitance is too low (or even zero!). One example of this situation is the case when the piezoelectric material has a very low permittivity which yields a small piezoelectric capacitance. This can be fixed by adding a load capacitance big enough to allow convergence. In case that an error of this kind raises consider to change the values of the resistance, the capacitor or even the material.

Interpreting the Simulation Results

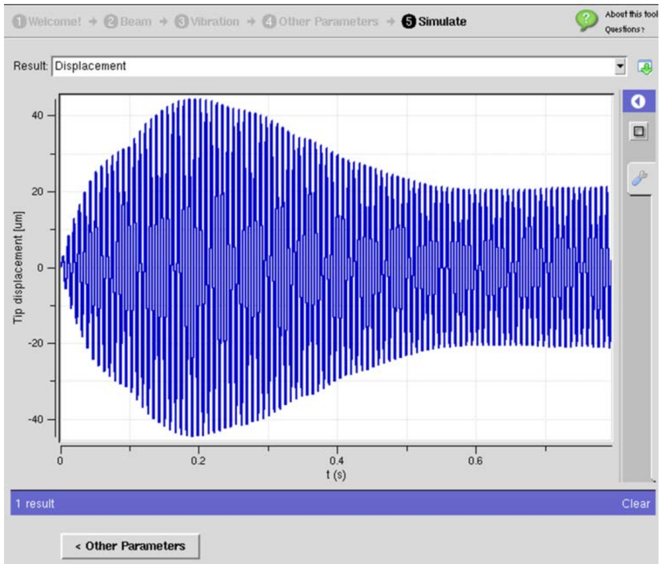

FIGURE 4 TIME DOMAIN DISPLAY OF THE TIP DISPLACEMENT

After the program finishes solving the differential equations, some statistics and a series of plots are presented in the “Result” menu. By default, the first result displayed is the summary of all the relevant calculations and statistics; that is, the resonance frequencies, the average and peak harvested power during the simulation, the volume of the device, among others described in Table 1. The second element in the list is a graph of the average power, which can be used to compare the effect of any parameter (e.g. load resistance, proof mass, etc) on the output power after different simulations.

TABLE 1 DESCRIPTION OF THE PERFORMANCE METRICS UN THE "RESULTS SUMMARY" FILE

Metric	Description	Units
Natural frequency	Resonance of the mechanical structure (a.k.a. “short circuit” resonance)	Hz
Open-Circuit resonance	Resonance of the voltage signal at high resistances	Hz
Average Harvested Power	Arithmetic mean of the generated power during the simulation	µW
Peak Power	Maximum electrical power during the simulation	µW
Estimated volume	$w \times L_{total} \times (2 \cdot z_{max} + t_{total}) + m_{proof}/\rho_{proof}$	cm^3
Power Density	P_{mean}/V_{total}	mW/cm^3
Input RMS acceleration	$(\Sigma a_i^2)^{1/2}$	*g* [RMS]
Power-to-acceleration ratio	P_{mean}/a_{RMS}^2	$\mu W/g^2$

Besides from these metrics, a series of plots can also be analyzed both in the time domain and the frequency domain. In the Results drop-down list you will find plots with the values of the output voltage, tip displacement and output power at any instant of the simulation. Also, a plot of the stress in the beam is shown to test both static and dynamic strength limits. It is worth to mention that this stress corresponds to the critical point at the base of the cantilever, both at the top and bottom layers (which could be made of different materials in a *unimorph*), and takes into account the static stress of the proof mass. For frequency analysis, the program displays the FFT[3] of the input and output signals as well as the Frequency Response Function. The last one has to be interpreted with care, since it will only yield accurate values depending on the kind of simulation. First of all, a Transfer Function makes sense only for linear devices thus it shouldn't be used for the rectifier circuit. Secondly, this function depends on the Fourier Transform of both the input and output signals; therefore, trustworthy results can only be achieved with an input with a high bandwidth such as an impulse, a long random vibration or a frequency sweep covering a very wide range.

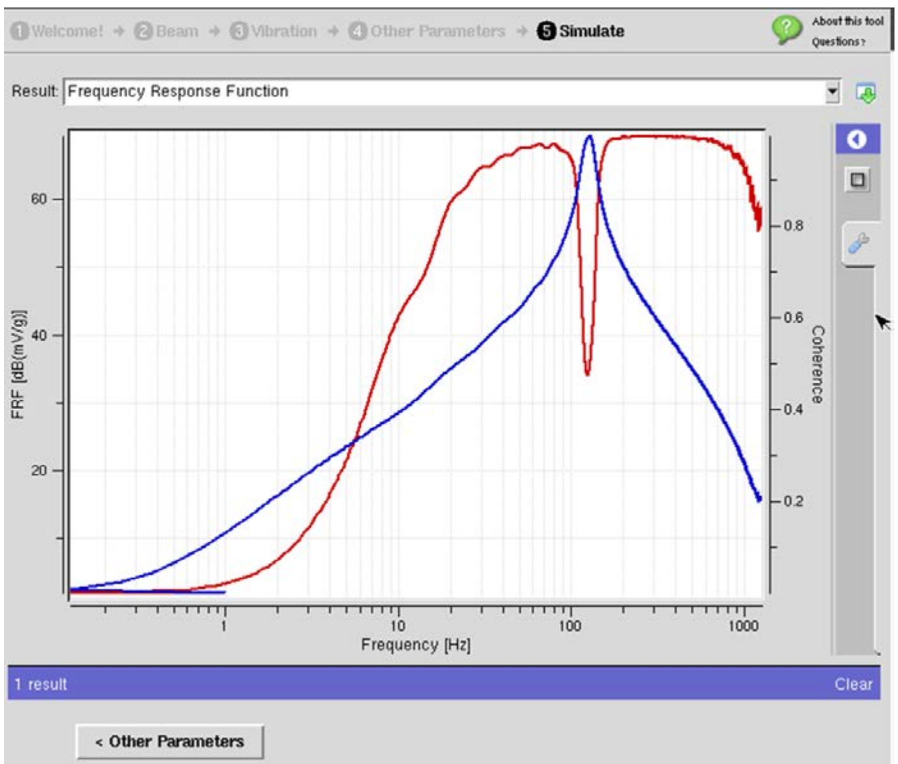

FIGURE 5 FREQUENCY RESPONSE FUNCTION RESULT

---

[3] Fast Fourier Transform. The program shows only the magnitude (or rather, twice the magnitude) of the Fourier Transform in the positive frequency spectrum. This allows knowing the frequency components in both signals.