**MAGNETO-ELECTRIC SIMULATION PROBLEM**

**Dear Sir**

My simulation problem is related to magneto-electric device. The brief introduction to the magnetostrictive and piezoelectric material is given below. Simulation 1 and Simulation 2 are the main problem.

**Magnetostrictive Material:**

Magnetostrictive materials are the class of material, when subjected to the external magnetic field, stress is generated in the magnetic material due to which dimensions of the magnetic materials changes. Ex: **Terfenol-D**, Co-Fe, Ni

**Piezoelectric Material:**

Piezoelectric material is the class of material, when subjected to the external force/ pressure/ stress the charge is accumulated on the surface of the material which give rise to a voltage signal. Ex: **Zno**, PZT

**Simulation 1:**

In first simulation I want to see the change in the length of the magnetostrictive thin film with application of external magnetic field.

L

Terfenol-D

z

SiO2

x

H

z

Silicon

H= External Magnetic field in +X direction

L= Length of the magnetic thin film

After the application of H (External magnetic field L will change) we want to simulate the change in L with the change in H.

In the figure I have shown the 2D thin film. Kindly consider a 3D thin film of magnetostrictive material.

**Simulation 2:**

If this rectangular magnetic thin film is placed below the piezoelectric thin film. The stress in the magnetostrictive thin film generated by the external magnetic field can be transferred to the piezoelectric thin film (ZnO). The stress then can be collected from the piezoelectric thin film in the form of the electrical signal. Hence in this way we can measure the magnetic field in terms of voltage signal across the piezoelectric layer. I want to simulate this behavior.

In second simulation I want to see the amount of stress generated in the magnetostrictive material and magneto-electric ME voltage coefficient I want to measure. The structure of this device is cantilever type structure.

L

z

ZnO

x

Terfenol-D

SiO2

Silicon

z

H

Kindly tell the appropriate physics to use and how to apply the boundary conditions for simulation 1 and simulation 2.

I have not found any tutorial example to simulate this device structure. I know it will include the magnetic field (mf) and solid mechanics and piezoelectric (pzd) module. But I am finding difficulty in applying the physics settings and boundary conditions.

I am attaching my device structure comsol file.

Waiting for a positive reply.

Sachin Dhariwal