## COMSOL

COMSOL is proprietary software which uses Finite Element Analysis to solve coupled systems of partial differential equations (PDEs). It provides a multi-domain physics workflow model for electrical, mechanical, fluid, chemical analysis. It is deployed in two parts: One with COMSOL and another with COMSOL server. The first type works on a single system as a standalone application, whereas the Server module allows deployment of simulations over a distributed systems. COMSOL originated in July 1986 as a work by Svante Littmark and Farhad Saeidi at the Royal Institute of Technology in Stockholm, Sweden to solve multi-physics problems. The module in COMSOL are categorized depending on its application areas, like Electrical, Mechanical, Fluid, Chemical, Multipurpose, and Interfacing.

COMSOL provides an IDE (Integrated Development Environment), which allows user to develop the simulation in steps:

1. Develop a 2D or 3D geometry. This will represent the shape on which the analysis will be performed.
2. Apply the boundary values and constraints. The constraints will represent the fixed values of Nodes which remains the same during each steps of solution.
3. Apply the Study, which uses the geometry and boundary values and perform the required differential equations to obtain a finite solution. The solution is only achieved when the iterative results converges that is, it is a minimization problem on the determinant of error matrix of solution.

The COMSOL is essential software for the following laboratory work, to design and analyze MEMS system.

Work Plane concept in COMSOL
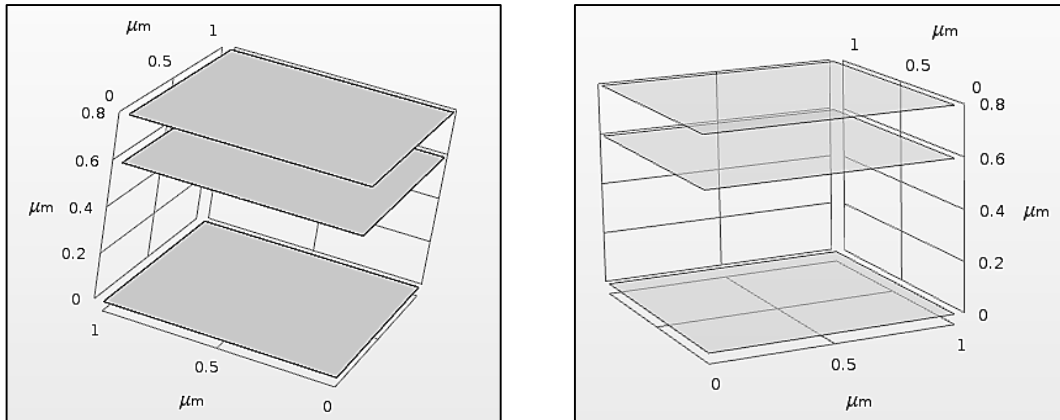
The MEMS design in this work are 3D geometry, but the drawing themselves are extrusion of 2D geometry. But 3D introduces Z-axis as additional co-ordinate. Hence, before 3D drawing can be done, a 2D drawing is first obtained. The 2D drawing have to be done on a 2D surface called a *work plane*. The work plane is a simple plane having a Z co-ordinate, on which the 2D geometry is constructed. Once the 2D drawing is completed over the work plane, it is extruded. The extrusion causes the 2D design to evolve into 3D. Extrusion requires specifying a depth parameter, using which the 2D geometry will be interpolated on the Z-axis that is, each (x,y) co-ordinate on the work plane is repeated along the z-axis from $z=z_0$ to $z=(z_0 + $ depth value). Therefore, a rectangle on a work plane will become a cuboid on a 3D geometry space, where the height parameter along the z-axis is the depth value specified during extrusion of the work plane. The z_0 parameter is initial z-value of the work plane in 3D space.

COMSOL allows declaration of multiple work plane with same or different z_0 parameter. This allows creating complex 3D structures from the 2D structure, which simplifies the geometry construction in COMSOL 5.3.

MEMS
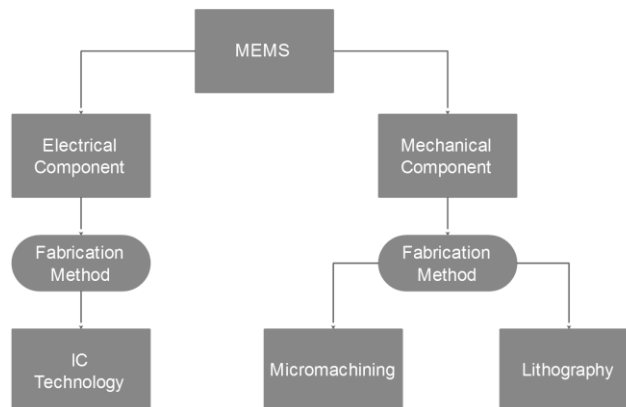
MEMS or Microelectromechanical systems is an integration of electrical and mechanical systems implemented using microsystem technology (MST). Similarly when MEMS is scaled down to nano-regime, they are named nanoelectromechanical systems (NEMS). The NEMS/MEMS finds application in chemical sensors, biosensors, aerospace, radio-frequency (RF) circuits, consumer electronics, etc. It is one of the most promising technologies of 21st century.

MEMS based devices varies in size which may range from few micrometers to millimeter range. These devices possesses the ability to sense, control and actuate on microscale. New material are used to design MEMS based on the requirements of the applications.

As MEMS is an integration between electrical and mechanical components working in co-ordination, they can must be fabricated using technologies of both domain. The electrical component is fabricated using the technologies required for fabricating IC technology such as Bipolar complementary metal-oxide-semiconductor (BiCMOS) processes. The mechanical components are fabricated using micromachining and lithography process. The lithography process called microstereolithography is used for polymer MEMS. Silicon micromachining is a dominant method for formation of mechanical components. Silicon wafers are more common for MEMS fabrication because they were already a substrate for silicon based IC industry.

MEMS devices therefore, possesses following advantages:

- Suitable for high-volume and low-cost production
- Reduced size, mass and power consumption
- High functionality
- Improved reliability
- Novel solutions and new applications

Analysis of MEMS structure using COMSOL

First we design the geometry in 3D. The designs are extrusions from 2D geometry drawn on 2D work plane. Then materials are assigned to each geometry sections, which describe their physical properties and also defines the physical constants required by the involved differential equations. Then using "Add Physics" option, the required physics options is selected. For this analysis, we select "Joule Heating and Thermal Expansion" under "Structural Mechanics" as the physics for study of the designed 3D structure. This option will add 4 subsections under Components task-

- Solid Mechanics
- Heat Transfer in Solids
- Electric Current
- Multiphysics

Solid Mechanics

The Solid mechanics defines the boundary conditions of the geometry as required to proceed with FEM analysis. The first of which are free and fixed constraint. The FEM requires the analyzer to specify free and fixed node for the geometry object. The fixed nodes are those whose value does not change in the solution iteration. These nodes are independent variables of the equation. Whereas the free nodes are those whose values are the solution obtained after each iteration. The free node values acts as the input for the next iteration and are always dependent parameters of the respective equations.

Then specify the initial values, which acts as values of the arbitrary constants which appears in the solution of differential equations in solid state mechanics. For a face selected, there can be two initial values: displacement in 3D space and corresponding directional velocity.

Finally, for each geometry specify the elastic nature. The elastic nature requires specifying Young's modulus and Poisson Ratio, which will help in solving for the deformation that the solid will undergo. This will be obtained from the material specified for the geometry under consideration.

Heat Transfer in Solids

These are parameters that will define the thermal behavior of the object and adds thermodynamics equations which can be used to obtain temperature at each node of the geometry, heat flux and radiation behavior. In these section, first the Density and Heat capacity are to be specified, which is again derived from the defined material. The initial values is the temperature specified for the geometries. Then we specify the temperature of the fixed faces and set other faces to "Thermal Insulation", which make these faces opaque to thermal radiation.

Electric Current

This section deals with behavior of the geometry when current or voltages appears at a node. For the geometry defined, the default values are not changed. However, a "Ground" and "Terminal Positive" options are added. The "Ground" defined for a face makes it connected to the 0V battery terminal. But "Terminal Positive" is the input terminal where current or voltage can be injected. Since we are using DC voltage source, we specify the "Terminal Positive" as a "Voltage" type node and connect a positive voltage source. This will cause current to flow into the geometry and thus a distribution is obtained.

Multiphysics

Since the study deals with multiple branches of physics which are developed independent of each other, we need to specify the coupling conditions required to couple equation from these different domains. These conditions are called boundary conditions, which if not specified, will make domain solution independent of each other. Thus these conditions are necessary for coupling solution in one physics domain to affect the solution obtained in another domain. COMSOL will automatically add the required boundary conditions necessary for the study concerned.

Study

Once the above tasks are complemented, "Add Study" is done which will add the required Multiphysics options, study options and consequent plots. For the analysis performed here, we add "Time-Dependent Model" as the study. This study will show the changes in the geometry with respect to time.

Mesh

Before FEM analysis can be performed, the geometry, which is a solid 3D structure must be broken down into meshes. The mesh process will divide the structure into small triangles which can be either fixed or free node, depending on the previously specified boundary conditions. This is then used by COMSOL solver to determine the degree of freedom, the number of solution equations. There are variety of Mesh methods that can be specified, but we have selected the default mesh method which is "Normal" sized "Tetrahedral", where no optimization is performed. It should be noted that, the Mesh method, the number of mesh directly influence the degree of freedom, number of equations, memory required and also the time taken to converge to the final solution.

Plot

The plot section contains the dataset which is used in the analysis of the solution of the geometry under study. There are variety of plot that can be specified, but only four of them are important for study:

- Stress (Solid) plot
- Temperature plot
- Isothermal Contours (the mesh faces which have same temperature values)
- Electric Potential (the electric potential at each mesh face)

Design-1

This structure is a simple structure with two geometry connected to two materials. Bottom solid is the silicon geometry above which a metal geometry is attached through which a current flows through. This current will cause the complete geometry to heat up. As there is no radiation, the heat will build up and increase temperature of the solid and cause deformation. Since there is no heat flux leakage, the temperature rise excessively.

Design-2

This structure is an improvement over Design-1. Here again there is no heat leakage, so the temperature build up excessively. But the deformation in each arm will cause a central block to mechanically displace caused by mechanical deformation caused by heating of the geometry caused by electrical current flowing through the metal arm. Therefore, mechanical movement is obtained using electrical potential, thus forming a MEMS device. But still the structure is not physically useful due to excessive temperature build up in the geometry.

Design-3

Design-3 improves the Design-2 geometry by adding a large boundary which assists in heat conduction and consequently avoid the excessive temperature buildup in the device. But still voltage application at "Positive Terminal" cause mechanical displacement. It can be seen that the temperature of the mechanical block and displacement arms are within tolerable range (300-400 K), thus making this MEMS device useful in physical situations.