3) Computer Aided Design of microsystems structures and elements

3.1. INTRODUCTION

MEMS and microsystems design differs from traditional engineering design is that in additional to the design for structural integrity and performance of the device or system, the designer's responsibility also include signal transduction, fabrication processes and manufacturing techniques, packaging, assembly and testing. There are three major interrelated tasks in microsystems design. The first one is the fabrication process flow design, the second one is the electromechanical and structural design, and the last is design verification which includes assembly, packaging and testing.

In figure 3.1 is presented an overview of the mechanical design of microsystems.

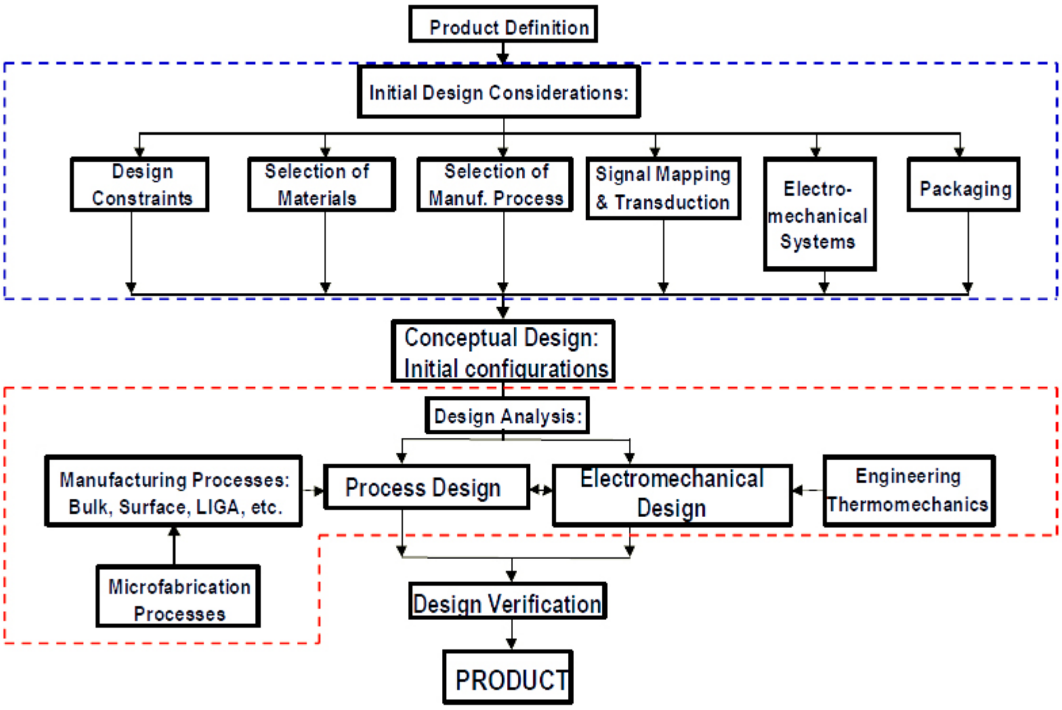


Fig. 3.1 An overview of mechanical design of microsystems

INITIAL DESIGN CONSIDERATIONS

a) Design constrains:

- Customer demands: applications, product specifications, operating environments
- Time to market
- Environmental conditions: temperature; humidity; chemical; optical.
- Size and weight limitations

- Life expectance
- Availability of fabrication facility
- Costs

b) Selection of materials: (for substrate, components and packaging materials)

- Substrate: Silicon, GaAs, Quartz and polymers
- Thermal/electric insulation: SiO2
- Doping materials: B, P and As
- Mask materials: SiO2, Si3N4, quartz
- Packaging materials: Adhesive, eutectic solder alloys, wire bond, encapsulation
- Photoresists for photolithography
- Thin films depositions

c) Selection of Manufacturing technique(s) and fabrication processes:

- Micromanufacturing techniques: Surface micromachining, Bulk manufacturing

d) Signal transduction: Types, Locations, Transduction methods, Interconnects

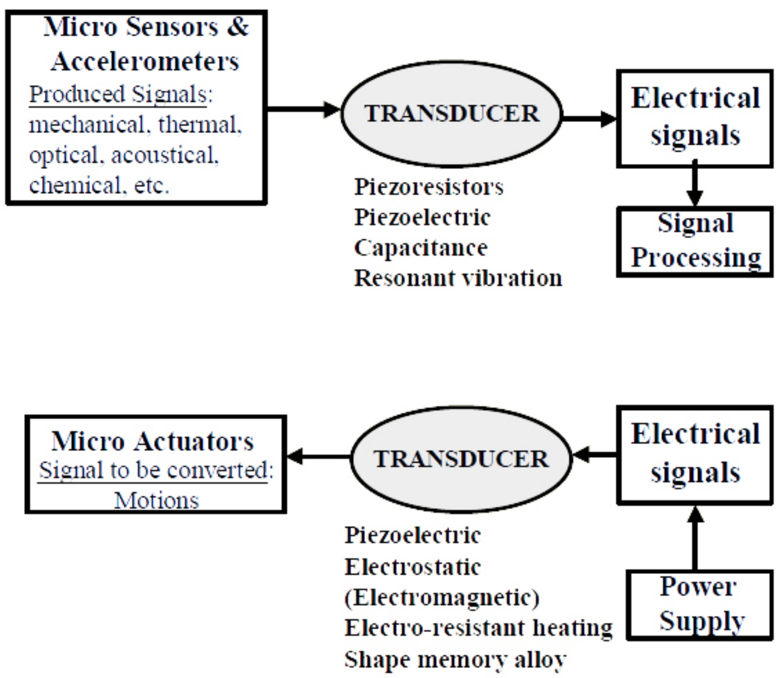


Fig. 3.2 Signal transduction Diagrams

e) Electromechanical systems:

- Power supply
- Interface of MEMS/microsystems and microelectronics

f) Packaging: Materials, Process design, Assembly strategy and methods, Testing

- Die passivation
- Media and System protection
- Electric interconnect
- Electrical interface

- Electromechanical isolation
- Signal conditioning and processing
- Mechanical joints (anodic bonding, TIG welding, adhesion, etc.)
- Processes for tunneling and thin film lifting
- Strategy and procedures for system assembly
- Product reliability and performance testing

The diversity and complexity of microsystems design used to take about 5 years to complete by the industry. Therefore, manually designed microsystems is no longer a viable option in practice. In 1990s the computer-aided design (CAD) was made commercially available to the industry. The design cycle has since drastically reduced to 3 to 6 months for new microsystems products using CAD as a tool.

In MEMS technology, CAD is defined as a tightly organized set of cooperating computer programs that enable the simulation of manufacturing processes, device operation and packaged Microsystems behavior in a continuous sequence, by a Microsystems engineer.

MEMS CAD software is used for matching system specifications, optimizing device performance, designing packages, validating fabrication process, shortening development cycle and reducing development cost.

In general, CAD for microsystems involves 3 major databases:

- Electromechanical design database
- Materials database
- Fabrication database

In figure 3.3 is presented the general structure of CAD for microsystems design.

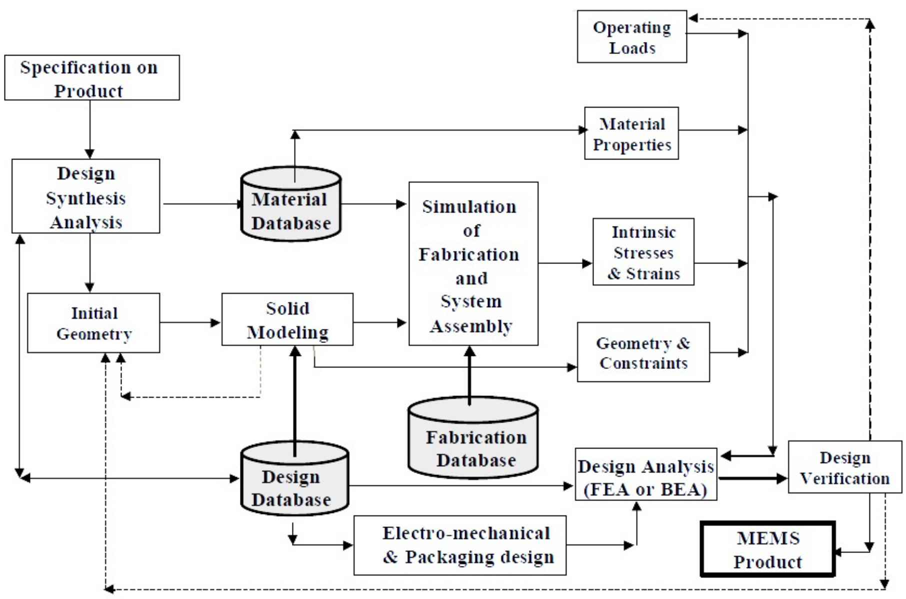


Fig. 3.3 General structure of CAD for microsystems design

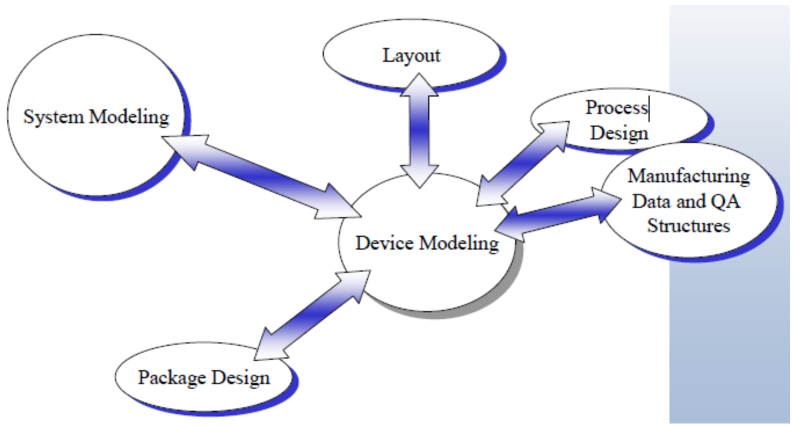


Fig. 3.4 MEMS CAD System flow

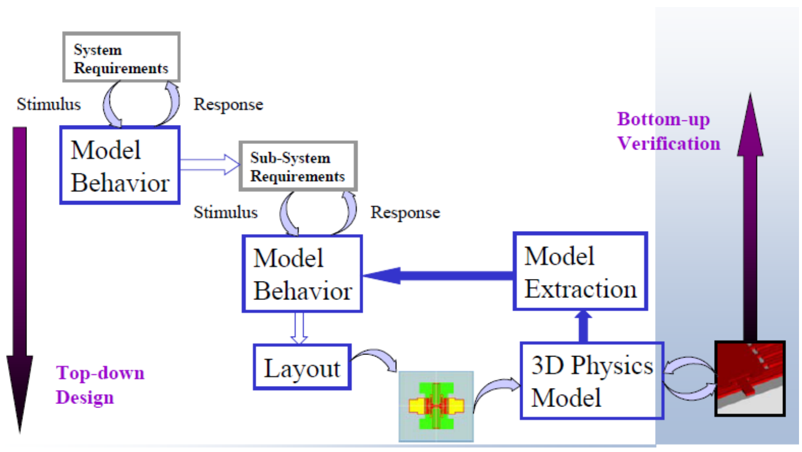


Fig. 3.5 MEMS CAD Design flow

3.2. SOFTWARE TOOLS FOR CAD AND DFM

<u>SELECTION OF A CAD PACKAGE:</u>

- User friendliness
- The adaptability of the package to various computer and peripherals
- Interface of this CAD package with other software, e.g. nonlinear thermomechanical analyses and the integration of electric circuit design
- Completeness of material database in the package
- The versatility of the built-in finite element or boundary element codes
- Pre- and post-processing of design analyses by the package
- Capability of producing masks from solid models
- Provision for design optimization
- Simulation and animation capability
- Cost in purchasing or licensing and maintenance

COMMERCIALLY AVAILABLE SOFTWARE:

- Coventorware from Coventor – http://www.memcad.com
- IntelliSuite from Intellisense Inc. (Corning) – http://www.intellisense.com
- MEMS ProCAETool from Tanner Inc. – http://www.tanner.com
- MEMScap from MEMScap Inc. – http://www.memscap.com
- SOLIDIS from ISE Inc. - http://www.ise.com
- COMSOL Multiphysics from COMSOL - https://www.comsol.com/

DEVICE DESIGN:

- Cadence/OrCAD 16.6-2015
- LEdit
- Spice
- MATLAB

PROCESS DESIGN:

- TSuprem (fabrication cross-section)
- IntelliSuite
- AnisE (bulk silicon etching)

ANALYSIS:

- MEMCAD
- IntelliSuite
- ANSYS
- CoventorWare
- COMSOL Multiphysics

a) CoventorWare

CoventorWare is a fully integrated MEMS design environment that offers the most productive path to manufacturable MEMS. Its unique capabilities let the user explore design alternatives efficiently and converge on a device that has the highest probability of success without using computationally intensive FEM tools. Then CoventorWare can be used to perform FEM/BEM analysis on critical areas to fine-tune the device.

CoventorWare provides electromechanical component libraries and field solvers. Therefore, design and simulations of many MEMS and microfluidics application can be realized with CoventorWare. For instance, accelerometers, switches, biochips, and mirror designs can be realized.

The software has three modules. These are Architect, Designer and Analyzer. Architect is a schematic-based system level modelling tool, in which different designs with different materials can be realized. Designer is a physical design tool that contains 2-D layout editor and the model generator. Analyzer is a physical and numerical analysis tool.

Analyzer has many solver types such as MEM Electro (EM solver), MEM Mech (mechanical solver) or Microfluidics solver.

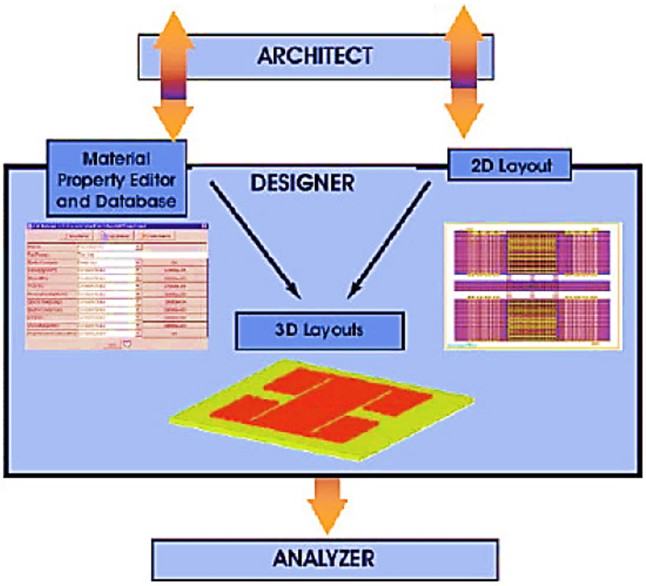


Fig. 3.6 CoventorWare tool flow

ARHITECT

Architect is a behavioral design and simulation environment using parameterized libraries. A realistic MEMS simulation requires that 3-D representations of multiphysics devices be created and boundary conditions applied such that the devices can be evaluated in multiple degrees of freedom. The complexity of the modeling and the computation of the equations used to represent the models demand a streamlined approach that enables the design procedure to be both fast and accurate.

Coventor's Architect is a new MEMS paradigm that uses a three-step approach for accurate high-level design and simulation:

- *Capture* a design in circuit form using libraries of parametric elements. The capture process uses placement and interconnect techniques familiar to digital designers.
- *Simulate* device behavior using SPICE-like simulation for mixed domain technologies. The result is an accurate and rapid solution available in tabular and graphical formats.
- Optimize design using advanced techniques such as sensitivity and Monte Carlo analyses. Hundreds of design iterations can be evaluated quickly to produce a device that minimizes the influences of manufacturing tolerances.

DESIGNER

Designer is an efficient front-end design tool for creating models of MEMS devices. It combines a fully functional 2-D mask layout program, process editor, materials properties database, and a preprocessor module for 3-D model generation and viewing. The models can be used for export to FEM/BEM simulation verification, while the 2-D files can be exported for mask set foundry processing.

Designer assists users in designing MEMS devices in the following ways:

- *2-D layouts* created, imported, generated, or exported Material properties database to store characteristics of the materials used in the fabrication process. Properties may be specified as scalars, polynomials, or in table format. The database is shared by other Coventor software.
- *Process editor* allowing real foundry steps to be emulated, including etching through multiple layers and partial backside etching. The process editor is used in 3-D model creation and in creation of layouts from high-level circuit schematics.
- *3-D models* automatically created from a 2-D description and process information and viewed in Preprocessor.

ANALYZER

Analyzer is Coventor's extensive suite of 3D field solvers designed specifically for MEMS applications. Analyzer gives the ability to analyze and simulate the behavior of MEMS devices that are subject to multiple physical effects. The sub modules are listed below:

- *MEMS Solvers*: Perform complex, fully coupled, multi-domain physics analysis with the comprehensive suite of MEMS-specific solvers. It also performs multi-domain interaction between true-coupled electrostatic, mechanical, PZE, and thermal analyses.
- *Query, View, and Compare Results*: Quickly examine results of analyses in selectable presentation formats. The powerful query capability enables the user to easily extract a wide variety of results over a wide range of parametric conditions.
- *Automatic Meshing*: MEMS structures will be meshed with precision using the automated mesh generator. Various mesh control techniques, such as boundary refinement and bias are available.
- *Microfluidic Solvers*: It Examines chemical transport and containment physics of lab-on-chip applications in DNA, protein, and chemical analyses. It also analyzes full 3D electrophoreses, electro-osmosis, and mixed electro-kinetics. Perform a multi-phase analysis to predict drop or bubble behavior for inkjet or dispensing applications.

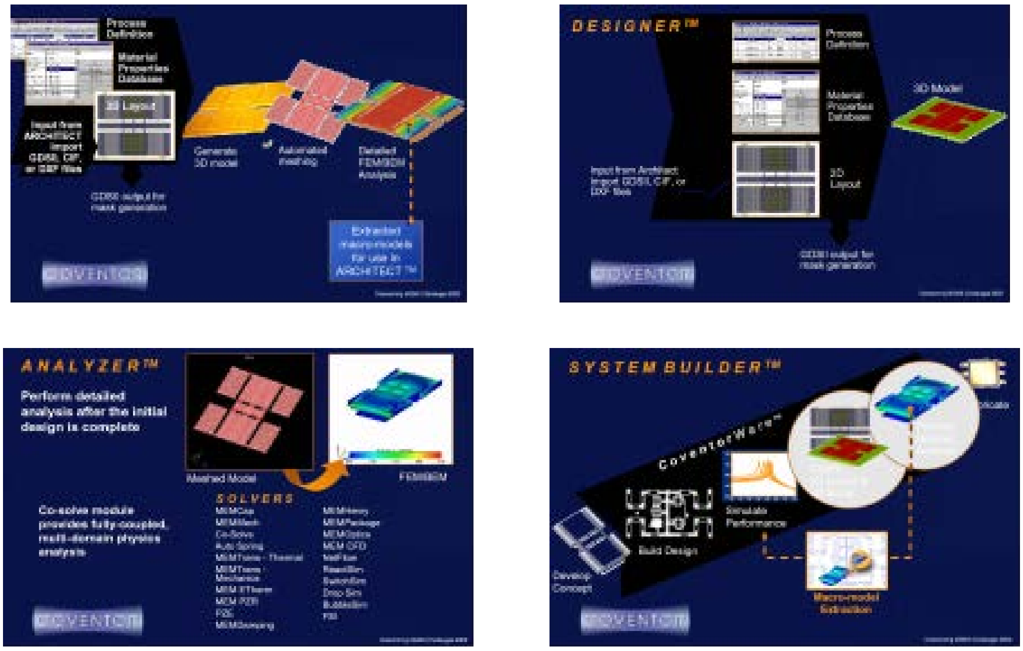


Fig. 3.7 CoventorWare software blocks

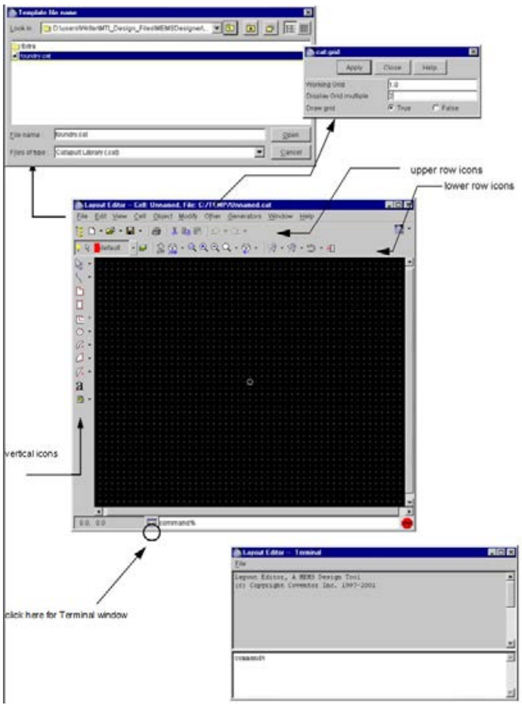


Fig. 3.8 Typical CAD Software: CoventorWare windows

TASK SEQUENCE ACCOMPLISHED BY A CAD TOOL

- Layout and process
- Topography simulation
- Boundaries, IC process results and material properties
- Mesh generation
- Device simulation
- System-Level simulation
- MEMS Control CAD

b) Intellisuite

Intellisuite is a registered trademark of IntelliSense Software Corporation. The MEMS device development using Intellisuite is given in the figure.

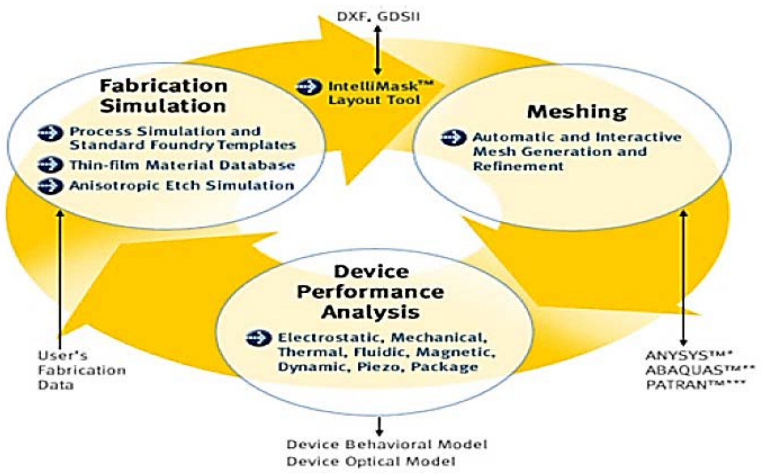


Fig. 3.9 MEMS device development using Intellisuite

The integrated seamless flow of Intellisuite is given in figure 3.10.

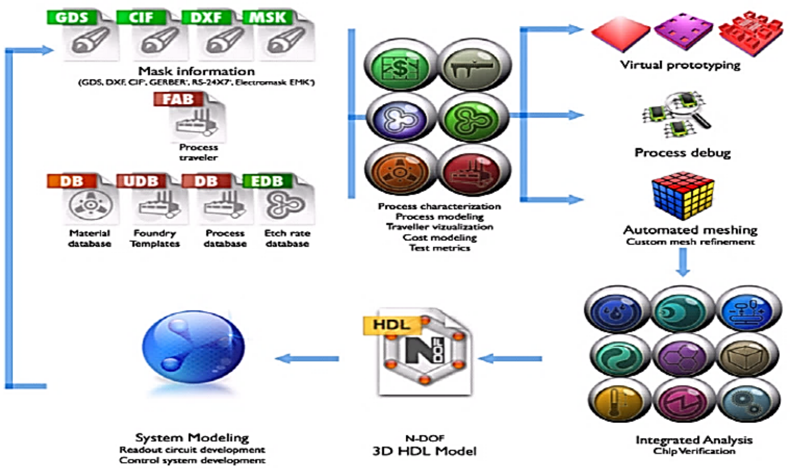


Fig. 3.10 Intellisuite tool flow

The software has several main modules and auxiliary modules. The main modules are Intelli Mask, Intelli Mask Pro, MEMaterial, IntelliFab, 3D Builder, AnisE, Thermo Electro Mechanical, Electro Magnetic, SYNPLE, RECIPE and Microfluidic. The auxiliary modules are 2D Viewer, 3D Geometry and EDA Linker.

There are two kinds of approach to model a structure: top down approach and bottom up approach.

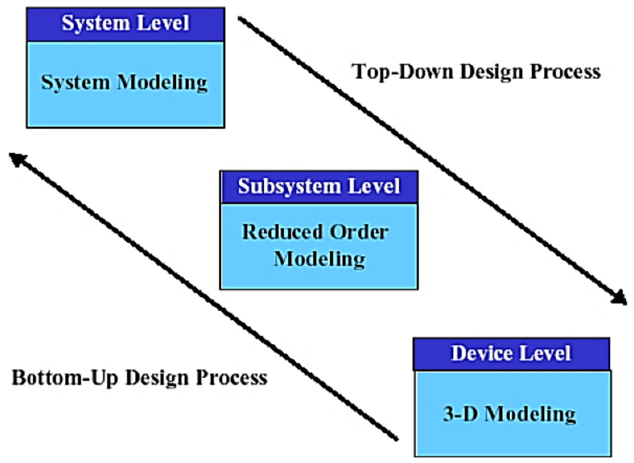


Fig. 3.11 Top-Down / Bottom-Up Design

TOP-DOWN APPROACH

In Top-Down approach, models are designed using the basic elements and then simulated using software module called SYNPLE. The basic elements used are rigid plate, linear beams, comb-drives, anchors, beam joint etc. The brain behind SYNPLE is a powerful new system level simulator designed from the ground up for the MEMS and Nano communities. SYNPLE ships with advanced algorithms to solve complex differential equations with ease. It allows users to perform several analyses that are essential for complex MEMS/System design and synthesis.

BOTTOM-UP APPROACH

In Bottom-Up approach models are developed using the following process:

- Mask layout creation using Intelli Mask:
  - Figure creation, Copying, Multi-copying
- Fabrication process simulation:
  - IntelliFab: Build and visualize the MEMS structure based on foundry.
  - MEMaterial: To deal with material properties based on process.
- Electro Mechanical analysis
  - Apply loads and boundary conditions
  - Mesh the structure - local refinement
  - Analyze the model for various input

c) COMSOL Multiphysics

COMSOL Multiphysics brings simulation and virtual prototyping to a far wider community of engineers. Key features are ready-made couplings between common physics, Model Tree, interactive meshing, merging components to build models, the ability to handle CAD assemblies, support for the multiphysics analysis of surface contact, and gains towards fully automatic solver selection. Users can also expand on the package's internal material database by enacting an online search of the Matweb database and directly importing material properties.

The MEMS Module is a collection of application modes and models for COMSOL Multiphysics with which one can model various MEMS devices and applications. It includes application modes for modeling of electrostatics, structural mechanics, piezoelectricity, film damping, and microfluidics. The MEMS Module Model Library shows how to use these application modes to model actuators, sensors, and microfluidic devices.

The MEMS Module seamlessly connects to COMSOL Multiphysics and the other add-on modules in the COMSOL Multiphysics product line. One can also view and modify the models in terms of the underlying PDEs. The software thus offers a unique transparency because the model equations are always visible, and you have complete freedom in the definition of phenomena that are not predefined in the module.

The table summarizes the most important MEMS couplings and some common devices using the MEMS Module.

	Electromechanical	*Fluid-structure Interaction*	*Microfluidics*
Phenomena/ coupling	Electro-structural Electro-thermal Thermo-mechanical Thermal-electric-structural Piezoelectric Piezoresistive Prestress modal analysis Stress stiffening	Moving boundary using ALE technique Squeezed-film damping	Pressure-driven flow Electroosmotic flow Electrophoresis Dielectrophoresis Electrothermal flow Mass transport
Devices	Cantilever beams Comb drives Resonators Micromirrors Thermomechanical actuators Inertial sensors	Mechanical pumps and valves	Lab-on-a-chip devices Microfluidic channels Micromixers Nonmechanical pumps and valves MEMS heat exchanges

The MEMS Module Model Library consists of a set of models that demonstrate the use of this module for modeling of various types of MEMS devices. The purpose of these models is to assist in learning, by example, how to create sophisticated multiphysics models for design and analysis of microelectromechanical systems.

In addition to serving as a reference, the model library is also gives a big head start while developing a model of a similar nature.

These models are divided into 4 different groups:

- Actuator models
- Sensor models
- Microfluidics models
- Piezoelectric models

d) <u>MEMS Pro</u>

The MEMS Pro package includes a schematic entry tool, an analog and mixed analog / digital circuit level behavioral simulator, a statistical analyzer, an optimizer, a waveform viewer, a full-custom mask level layout editing tool, an automatic layout generator, an automatic standard cell placement and routing tool, a design rule checking feature, an automatic netlist extraction tool (from layout or schematic), a comparison tool between netlists extracted from layout and schematics (Layout Versus Schematic), and libraries of MEMS examples.

MEMS specific key features also available in MEMS Pro encompass a 3D solid model generator using mask layout and process information, a 3D solid model viewer with cross-section capability, a process description editor, true curved drawing tools and automation of time consuming tasks using the MEMS Pro Easy MEMS features. Embedded features within ANSYS allow automatic mask layout generation from an ANSYS 3D Model and process description (ANSYS to Layout), as well as automatic MEMS behavioral Model generation in hardware description languages (Reduced Order Modeling).

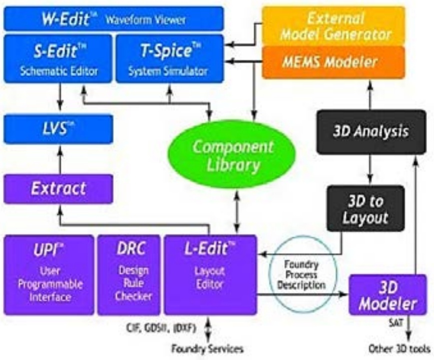


Fig. 3.12 MEMS Pro tool flow

SCHEMATIC CAPTURE (S-EDIT)

S-Edit is a fully hierarchical schematic capture program for MEMS and IC applications. The program also serves as a schematic entry front end to the T-Spice simulator, L-Edit/SPR automatic standard cell placement and router, and layout vs. schematic (LVS) netlist comparison programs. S-Edit and its associated libraries are technology independent; that is, the design may be built and tested before choosing a specific manufacturing technology and vendor. User-defined global symbols convey connection among nodes without wiring. S-Edit also supports global node naming so that a single symbol can represent several distinct nodes in the design. Using S-Edit, MEMS schematics can be designed to include signals in multiple energy domains. For example, the MEMS Library includes a set of examples of electro-mechanical schematic symbols and models.

SIMULATOR (T-SPICE PRO)

The T-Spice simulator provides full-chip analysis of analog, mixed analog/digital and MEMS designs using an extremely fast simulation engine that has been proven in designs of over 300,000 devices. For large circuits, the T-Spice simulator can be ten times faster than typical SPICE simulators. MEMS macromodels can be implemented in 3 different ways in T-Spice. In the simplest form, MEMS devices may be modeled using equivalent circuits of standard SPICE components. Another method is to create table models from experimental data or finite element or boundary element analysis of the MEMS devices. A third method is to use the external functional model interface. This last method allows quick and easy prototyping of custom MEMS macromodels using a C code interface.

The W-Edit graphical waveform viewer, embedded within T-Spice, displays analysis results, and automatically updates its display each time T-Spice simulates a circuit. Powerful optimization algorithms automatically determine device or process parameters that will optimize the performance of the design. Defining parameters to be varied, setting up optimization criteria, and choosing optimization algorithms is a cinch using the new Optimization Wizard. The Wizard prompts the user for the optimization criteria the program will need. Monte Carlo analysis generates “random” variations in parameter values by drawing them probabilistically from a defined distribution. This type of statistical analysis may be used to discover what effects process variation will have on system performance.

Layout Editor (L-Edit)

L-Edit is an interactive, graphical layout editor for MEMS and IC design. This full-custom editor is fast, easy-to-use, and fully hierarchical. Primitives include boxes, polygons, circles, lines, wires, labels, arcs, splines, ellipses and tori. Drawing modes include 90°, 45°, and all-angle layout. Shortcuts are also available for quickly laying out circles, tori, pie slices, and “curved polygons” with true curved edges. Designs created in L-Edit are foundry ready. The MEMS Pro Toolbar in Version 4.0 gives access to MEMS specific design features. It gathers the creation of splines and ellipses, the display of vertex information, and the use of Easy MEMS features like the polar array feature and the plate release feature. It also includes access to the process definition graphical interface, 3D modeler and viewer, MEMS specific DRC, and MEMSLib (the MEMS library).

3D Modeler

3D models can be created of the MEMS device layout geometry directly in LEdit using one of the many foundry fabrication process descriptions supported by MEMS Pro or by specifying the custom process. The 3D Solid Modeler permits views of surface and bulk micromaching steps including deposit, etch, and mechanical polishing. You can easily customize your view with features such as panning, zooming, cross-section modeling, and other viewing controls. 3D solid model geometry can be exported in a SAT, ANF or APDL format.

3.3. BEAM DESIGN

In MEMS design, the beam structure has many applications, including sensors, accelerometers, and RF switches. In the following part, will be described a simple example to understand how MEMS design is done.

In the following part the software tool that will be used is CoventorWare. It will be designed a solution for an electrostatically actuated fixed beam with conformal supports. The example is educative for editing process sequence, creating the 3-D model, and simulation with electrostatic and mechanical solvers. The main structure and analyses that will be done is shown below.

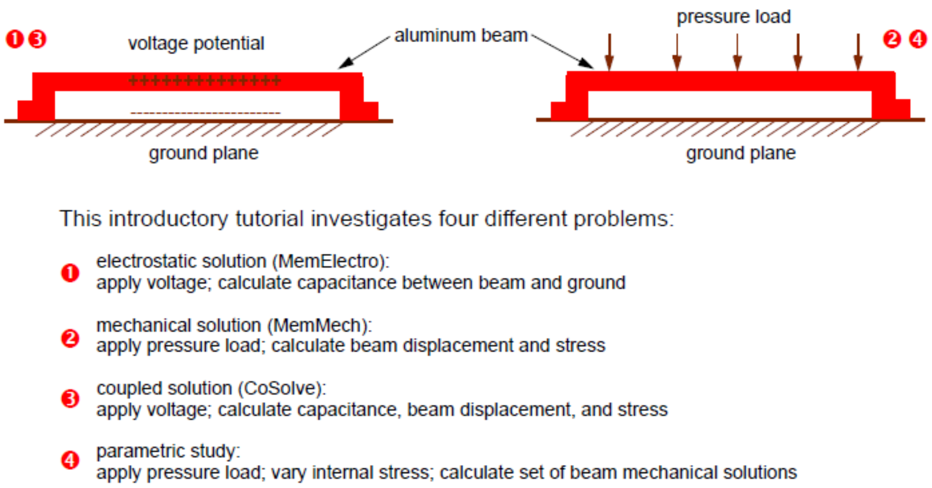


Fig. 3.13 Beam structure

To start the software, on your desktop, double click on the CoventorWare shortcut icon or from the Start menu select *Coventor* → *CoventorWare 2010*. The first time you run the software, you will have to set a work directory, a temp directory, and a shared directory (default location). These directories will not have to be set again. After we set the project directory our procedure will:

- Check and modify the Material Properties Database
- View and modify the process flow
- View and modify the 2-D mask design
- Build a 3-D solid model
- Mesh the model
- Name faces and conductors
- Simulate electrostatic and mechanical stimulus

MATERIAL PROPERTIES DATABASE SETUP

The Material Properties Database is the foundation of your design. This database stores properties for materials used for MEMS design. Materials or parameters can be added or modified at any time from this or from other dialogs in the software. The Function Manager provides access to the Material Properties Database in the top portion of the window, above the tabs. Material Properties Database access and setup is the first field in the Function Manager because the process is dependent on the materials in the Material Properties Database, and in turn, the rest of the design creation is dependent on the process file. The user designates the file path for the Material Properties Database file in the Users Settings dialog the first time CoventorWare is run. This setting can be changed from the *Tools* →*User Settings* menu.

In this example, the Material Properties Database is checked and validated before a process file is created because the materials used in the process file must exist and have the correct values. The process file takes the materials and their values from the Material Properties Database. In the next step, properties for both Silicon_100 and Aluminum Film will be checked. These materials are used to construct the beam.

The first step is to access the Material Properties Database. For this, in the Function Manager, click on the *Start MPD Editor* icon. In the Edit Materials dialog, click on the drop-down arrow beside the Materials field. Select the material named *ALUMINIUM(FILM)*. If the material exists, check the property values (see Fig 3.14) against those in Table 1. If the material is not on the list click on New Material and in the dialog that opens enter *ALUMINIUM(FILM)*. Set the properties from Table 1. Then, select *SILICON_100* from the Materials drop-down menu and verify its material properties (see Fig 3.15). When finished in the Edit Materials dialog, click on Close.

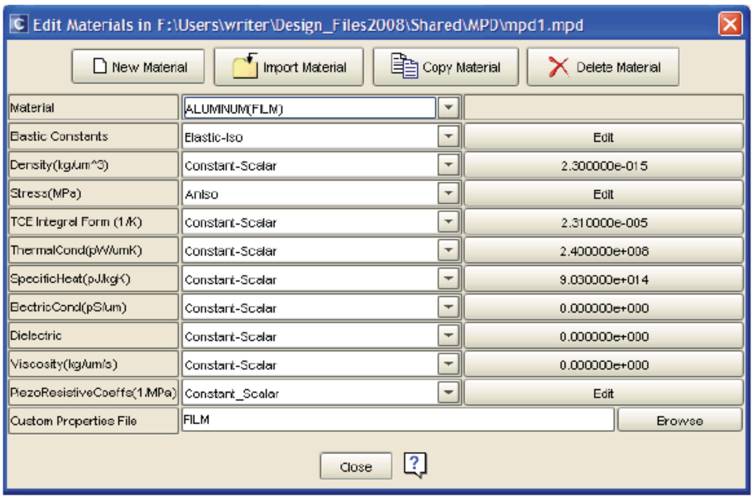


Fig. 3.14 Material Properties for *ALUMINIUM(FILM)*

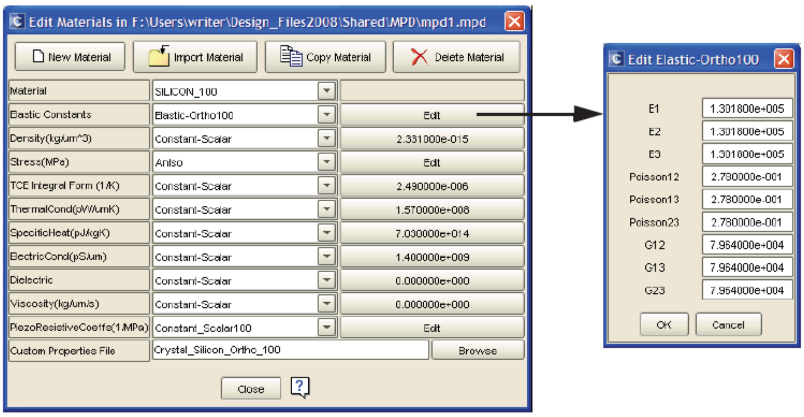


Fig. 3.15 Material Properties for *SILICON_100*

Property	Data Type	Aluminum(FILM)	Silicon_100	Units
Elastic Constants	Elastic-Iso, (Aluminum film) Elastic-Ortho100 (Silicon_100)	E: 7.70e+04 Poisson: 3.00e-01	E1, E2, E3: 1.3018e+005 Poisson12, 13, 23: 2.78e-01 G12, G13, G23: 7.964e+04	MPa
Density	Constant-Scalar	2.30e-15	2.331e-15	kg/μm^3
Stress	AnIso (S_x,S_y,S_z)	0,0,0	0,0,0	MPa
TCE	Constant-Scalar	2.31e-05	2.49e-06	1/K
Thermal Cond	Constant-Scalar	2.40e+08	1.57e+08	pW/μm · K
Specific Heat	Constant-Scalar	9.03e+14	7.03e+14	pJ/kg · K

Table 1 Material Properties for the example

PROCESS EDITOR

The process editor supplies the information to construct the 3-D model of the structure. The layers are created with deposition and etching steps. Adjustable process parameters are material thickness, sidewall angles, deposition type and mask polarity. The cross-section diagram of the structure is shown below.

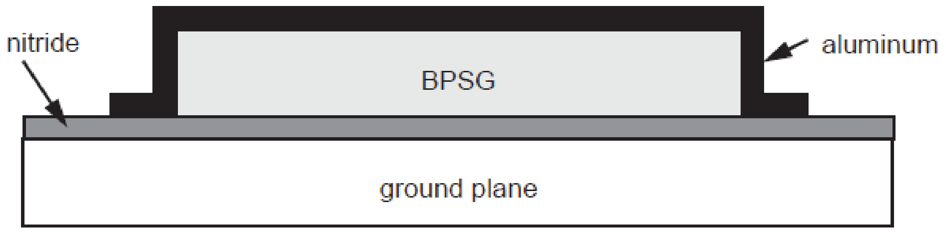


Fig. 3.16 Structure to be obtained

A ground plane is overlaid with a nitride layer for isolation of the beam. A sacrifice layer of BPSG (boron phospho-silicate glass) is deposited on the nitride layer. It is etched to define the areas where the beam will be anchored to the nitride layer. The BPSG thickness sets the separation between the nitride layer and the thin aluminum beam to be built on top. After the entire wafer is deposited with aluminum, a selective etch defines the actual beam dimensions. Note that the aluminum beam conforms to the deposited BPSG, with equal thickness on the top and on the sidewalls of the BPSG. The BPSG is called a sacrifice layer because it is etched away when the MEMS process is complete. When the BPSG is removed, the fixed beam is held to the nitride by its conformal supports in the region where the BPSG holes were created. The section of the beam in the middle is now free to deflect when an external force, such as a pressure or electrostatic load, is applied.

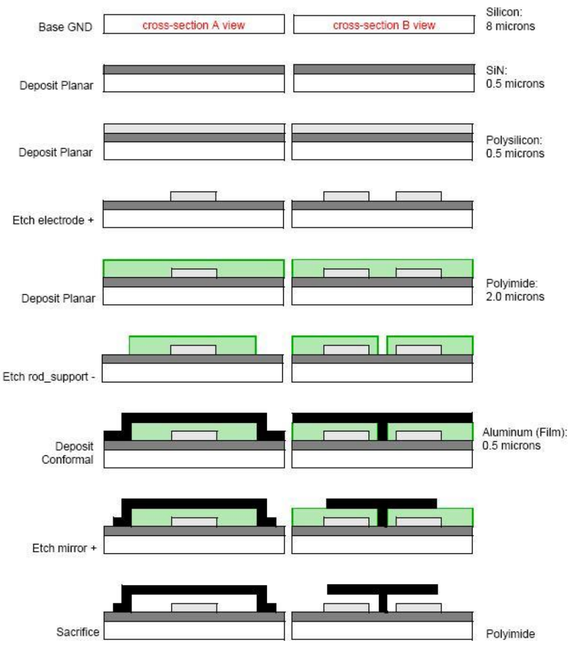


Fig. 3.17 Fabrication steps

The Process Editor is accessed from an icon in the top half of the Function Manager. The initial process flow does not show the deposit and etch steps required to form the aluminum beam. In the steps below, the process file will be modified so an appropriate model can be built.

In the Function Manager, click on the drop-down arrow to the right of *Process* filed. Select the *beam_init.proc* file located in the *BeamDesign\Devices* directory. Click on the *Process Editor* icon.

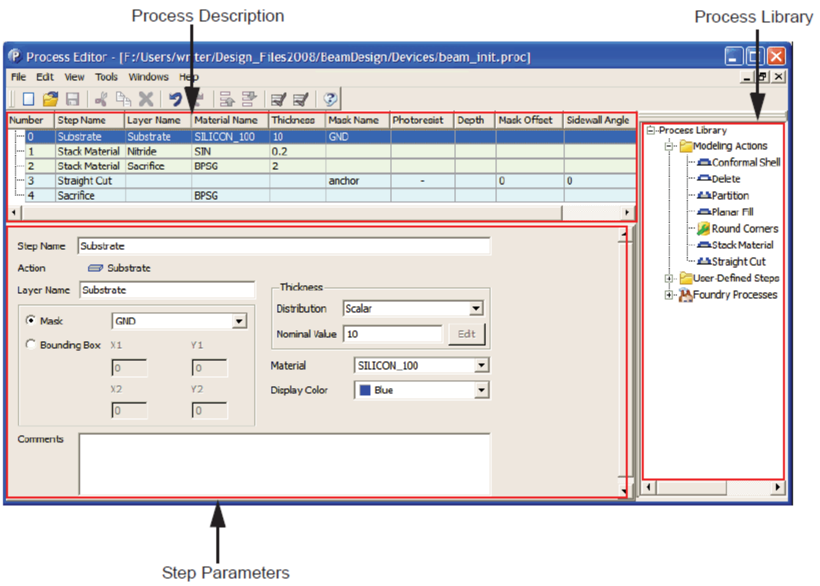


Fig. 3.18 Process Editor for the *beam_init.proc* file

The Process Description identifies all the steps used in creating the beam design. The Process Library provides the modeling step options. The Step Parameters pane allows the user to set parameters for a selected step in the Process Description. The Substrate in the Process Description pane corresponds to the ground plane, the next Stack Material corresponds to the Silicon Nitride layer, and the next Stack Material and Straight Cut steps correspond to the defined BPSG material on which the aluminum beam is placed. Finally, the BPSG delete step is shown at the end.

In the Process Description pane, highlight *Step 3 (Straight Cut)* by clicking on the fourth row. On the right-hand side of the Process Editor, in the Process Library pane, double-click on *Modeling Actions* to expand the folder. Right-click on *Conformal Shell* and select *Insert Below Current Step*. Right-click on *Straight C*ut and insert it below the current step. Highlight Step 4 (Conformal Shell) in the Process Description pane, and enter the following attributes into the Step Parameter fields:

- Layer Name: enter *beam* (or another descriptive name)
- Material: select *ALUMINUM(FILM)*
- Thickness Nominal Value: enter 0.5
- Display Color: select *red*

Select *Step 5 (Straight Cut)* and enter the following attributes:

- Select *Cut Last Layer Completely*
- Select *Front Side*
- Mask: enter *beam*
- Photoresist: select +

The beam process is now complete. The finished Process Description pane of the Process Editor should look as shown below.

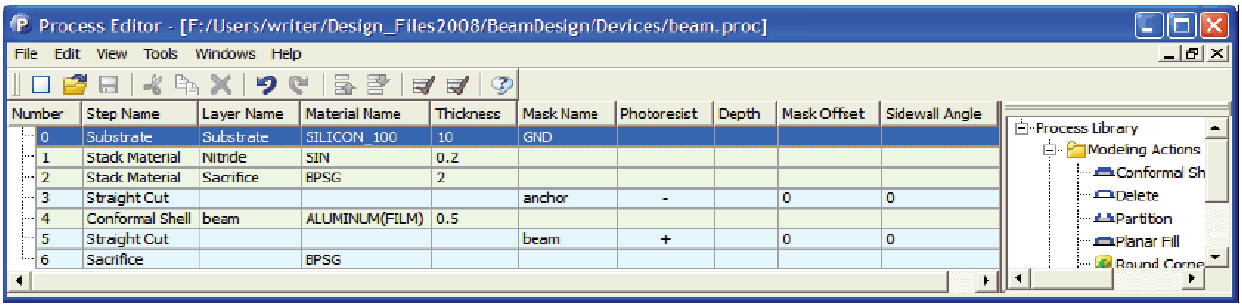


Fig. 3.19 Process Editor for the *beam_init.proc* file after edits

From the menu at the top of the Process Editor, select on *File→Save As*. Type the new file name *beam.proc* in the File name field. Click on Save. From the *File* menu, select *Exit*.

Layout Editor

- Layout process
  - Multi-layer mask sets
  - Cell hierarchy
  - Boolean operations
  - Curved shapes
- MEMS-specific features
  - Any-angle feature creation
  - Multi-copy by translation or rotation
- Links directly to process simulation and mesh generation
- Compatible with GDSII & DXF

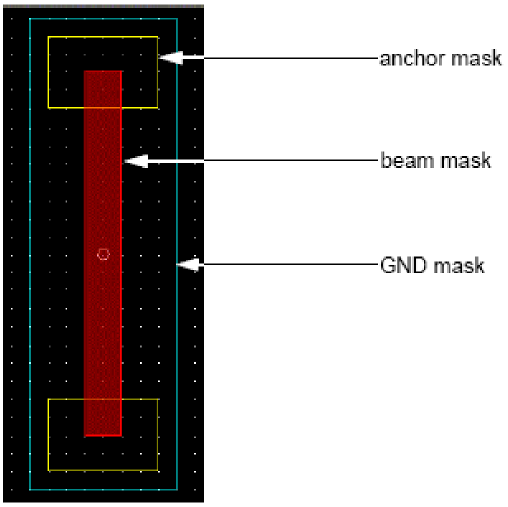


Fig. 3.20 2D layout of the beam

After defining material properties and the deposit and etch sequence, the next step in creating a MEMS design is creating the 2-D layout. The 2-D Layout Editor is used to create, import, view, and edit 2-D mask information. The Layout functions are accessed by selecting the Designer tab from the Function Manager. From this window, the user selects a cat file. The cat file is the storage location for the 2-D layout created or edited in the Layout Editor. This cat file can include any number of individual cell layouts. The 2-D layout information can be used to create foundry masks for building the MEMS device and is used as the source for rendering a 3-D model for meshing and solving. In the next step, the 2-D mask layout will be viewed in the Layout Editor.

From the Function Manager, click on the tab labeled *Designer*. Click on the Browse icon to the right of the *Layout* field. Select the *beam.cat* file in the *BeamDesign\Devices* directory. Click on *Open*. Click on the *Layout Editor* icon.

Building the Solid Model and Meshing

A 3-D model must be designed using the thickness and etch profile information from the process file and 2-D layout mask information. The Preprocessor tool can be used and the do necessary name settings and face selections. After that, a very important step is meshing the structure. Meshing is done to present the structure to the solver for finite element analysis. There are many types of meshing, and the user must select the one appropriate for his application. The most important areas must be meshed with a greater number of polygons to obtain accurate results, and the areas that do not present a very high interest can have a rougher mesh, for not loading the processor very much (thus obtaining a shorter simulation time). In our case, a Quadrilateral mesh type was selected for the ground and Manhattan Bricks was selected for the beam.

Building the Solid Model

To build the solid model of the beam, the first step is to click on the down arrow Model/Mesh field and select the option *Create a new model*. Then click on the *Build a New 3D Model* icon and in the Input dialog that opens enter *beam*. After the build is finished the Preprocessor will automatically open.

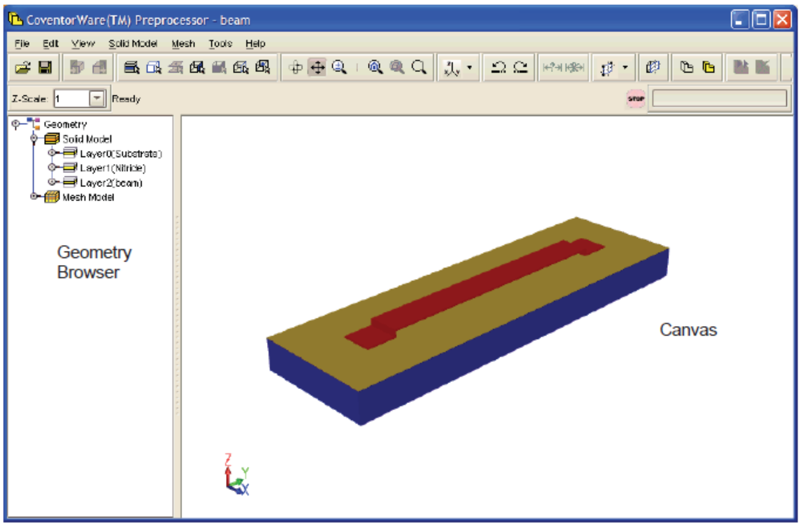


Fig. 3.21 Preprocessor Rendering of the Beam Model

Meshing

To assign the *Manhattan bricks* mesh type to the *Substrate* layer, from the Geometry Browser, click on the region containing the *Substrate* layer and then right click and select *Mesher Settings*. Select the *Manhattan bricks* mesh type. For setting the Element Size, enter the following:

- For X direction: 2
- For Y direction: 2
- For Z direction: 10

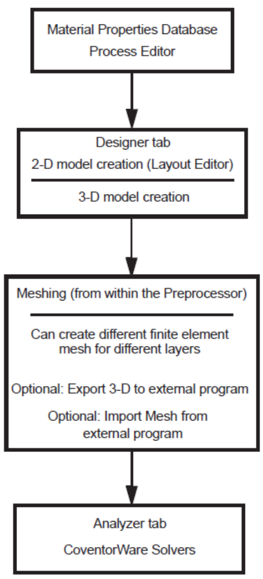


Fig. 3.22 Meshing in CoventorWare

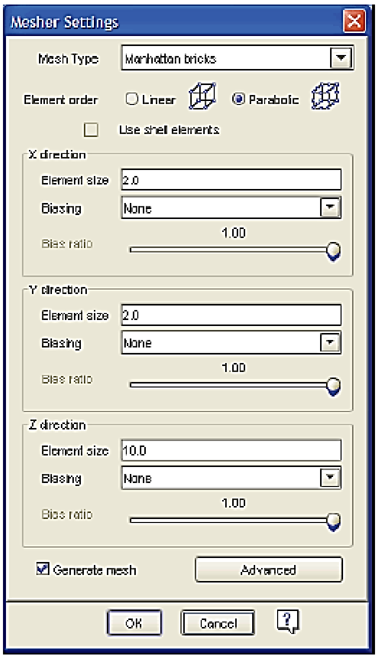


Fig. 3.23 Mesher Settings

To assign the *Manhattan bricks* mesh type to the *Beam layer*, click on the region containing the *beam* layer. Then apply the same mesher settings. After all the settings were done create the mesh by clicking on the *Generate Mesh* icon.

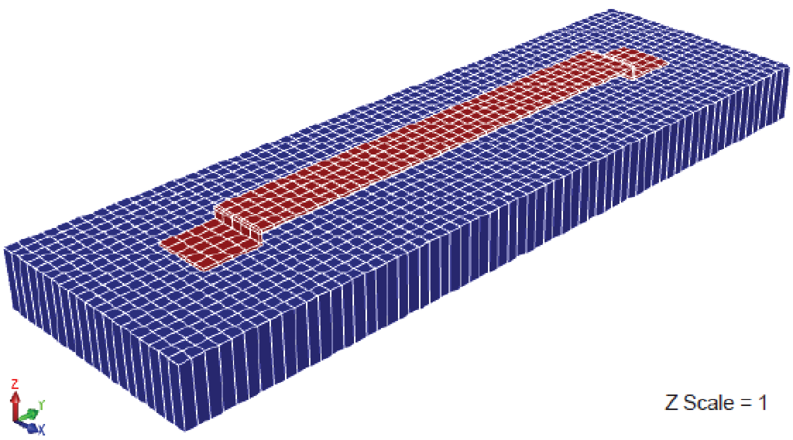


Fig. 3.24 Meshed Substrate and Beam

MEMELECTRO SIMULATIONS

First analysis is the uncoupled electrostatic simulation, MEMElectro. The MEMElectro solver produces an electrostatic solution by solving for the charge and capacitance interaction between the beam and the ground components. It uses the Boundary Element Method (BEM). MEM Electro computes the charge on each surface panel and presents a final solution with charge distribution calculated for all the panels in the model.

From the Function Manager, click on the *Analyzer* tab to set up the *Analyzer* tab to point to the *MemElectro* solver for the *beam* model. After the window opened click on the arrow to the right of the Solver field, and from the dropdown menu, select *MemElectro*. Then click on the arrow beside the Model/Mesh field and select the *beam* model. In the Analysis field, verify that *create a new analysis* is displayed. Click on the *Solver Setup* icon.

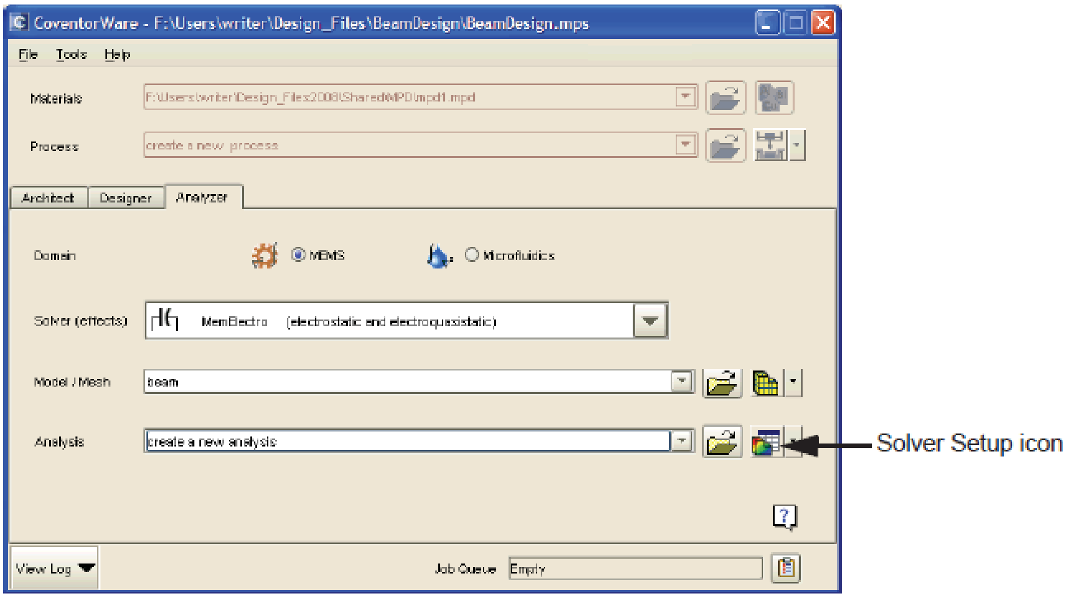


Fig. 3.25 MemElectro window

Conductor beam is adjusted to have 1 V potential. The resulting ground and self-capacitances of the structure can be seen below. Notice that beam-beam or ground-ground capacitances are the self-capacitances of the structure.

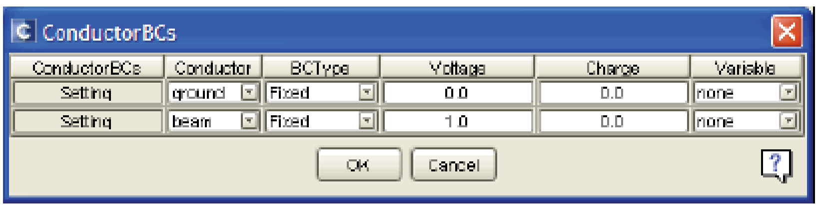
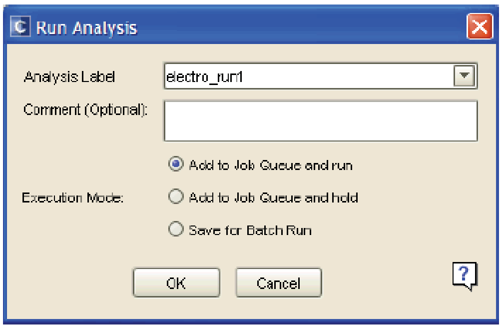


Fig. 3.26 ConductorBCs window

In the MemElectro BCs window click on ConductorBCs and modify the fields so the conductor beam has a 1 V stimulus applied. After the settings was applied click on Run to start the simulation. After the Run Analysis window opens enter *electro_run1* in the Analysis Label field and click OK.



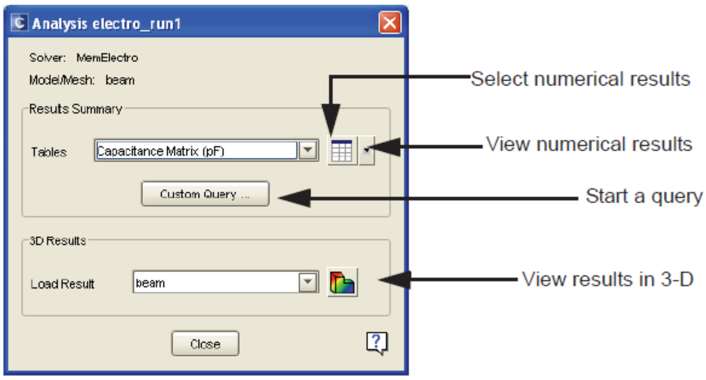
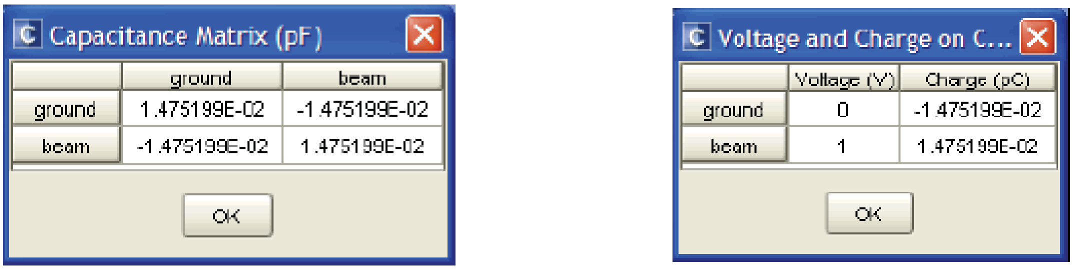


Fig. 3.27 Run Analysis window

Fig. 3.28 Analysis electro_run1 window

After the simulation is finished open the Analysis electro_run1 results by clicking on the *View Results* icon. To view the electrostatic results, click on the arrow beside the Tables field and select *Capacitance Matrix (pF)* from the drop-down menu and then click on the *View Table* icon.


Capacitance Matrix (pF)

	ground	beam
ground	1.475199E-02	-1.475199E-02
beam	-1.475199E-02	1.475199E-02

Voltage and Charge on C...

	Voltage (V)	Charge (pC)
ground	0	-1.475199E-02
beam	1	1.475199E-02

Fig. 3.29 Electrostatic results

To view the 3-D charge results click on the *View 3D Results* icon and then select *Coventor→Geometry Scaling*. In the Scaling panel change the Z Scale field to 5.

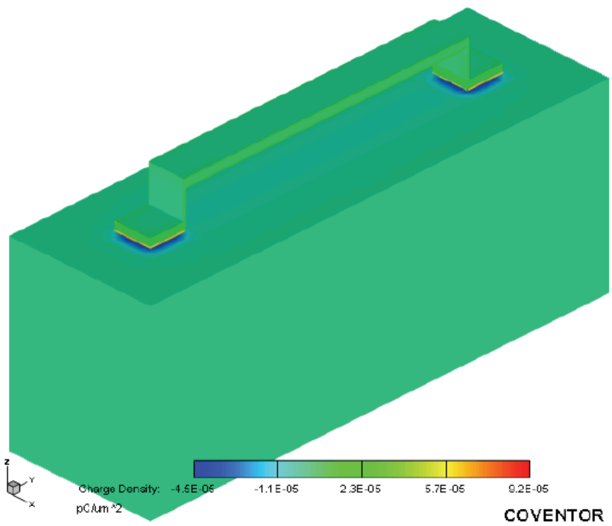
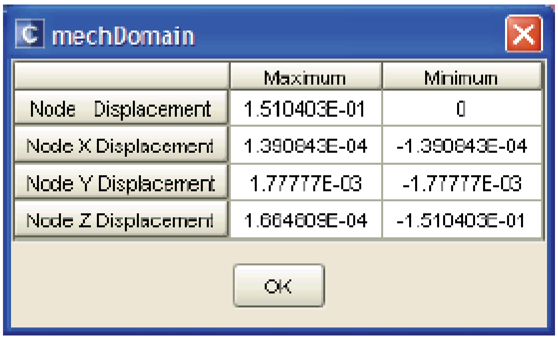


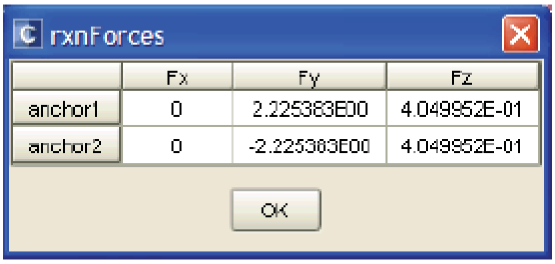
Fig. 3.30 The 3-D view shows the mapped charges

MemMech simulations

Mechanical analysis can be performed using MemMECH which solves for the mechanical stress and displacement at each node. From the surface boundary conditions window, pressure load of 0.001 MegaPascals is applied onto the top surface of the beam in –z direction. The results of displacements in each direction and the applied reaction forces on the anchors are shown below:



	Maximum	Minimum
Node Displacement	1.510403E-01	0
Node X Displacement	1.390843E-04	-1.390843E-04
Node Y Displacement	1.77777E-03	-1.77777E-03
Node Z Displacement	1.664609E-04	-1.510403E-01



	Fx	Fy	Fz
anchor1	0	2.225383E00	4.049952E-01
anchor2	0	-2.225383E00	4.049952E-01

Fig. 3.31 The results of displacements in each direction and the applied reaction forces

To set up the Analyzer tab to point to the MemMech solver for the beam model click on the arrow to the right of the *Solver (effects)* field, and from the drop-down menu, select *MemMech* and then select the *beam* model. After the selection was made click on the *Solver Setup* icon. After the verification of the default settings in the MemMech Settings dialog, click on the SurfaceBCs from the MemMech BCs window to open the SurfaceBCs dialog to fix the anchor faces. Modify Set 1 as described: for FixType select *fixAll*, for Patch1 select *anchor1*, for and1 select *or*, for Patch2 select *anchor2* and for the rest of the Set1 line does not change.

The next step is to apply a 0.001 MPa load to the top patch. To do this modify Set2 as described: for FixType select *LoadPatch*, for Patch1 select *top*, for LoadValue type select *Scalar* and for LoadValue enter *0.001*. Do not change any of the Boolean settings.

After all the settings were done start the MemMech simulation by clicking Run in the MemMech BCs window and save the analysis results as *mech_run1*. When the simulation is finished click on the *View Results* icon and select *mechDomain* from the Tables drop-down menu and then click on the *View Table* icon to review the mechanical results. Select *rxnForces* from the Tables drop-down menu and then click on the *View Table* icon to verify the rxnForces results.

To view the 3-D mechanical results click on the View 3D Results icon and from the Visualizer menu bar select *Plot→Contour/Multi-coloring*. In the dialog that opens, select *Mises Stress* from the drop-down menu then click on Close. Select *Coventor→Geometry Scaling* and set Scale Z to 5. Select *Deform Using Displacements*, and set the Exaggeration to *15*.

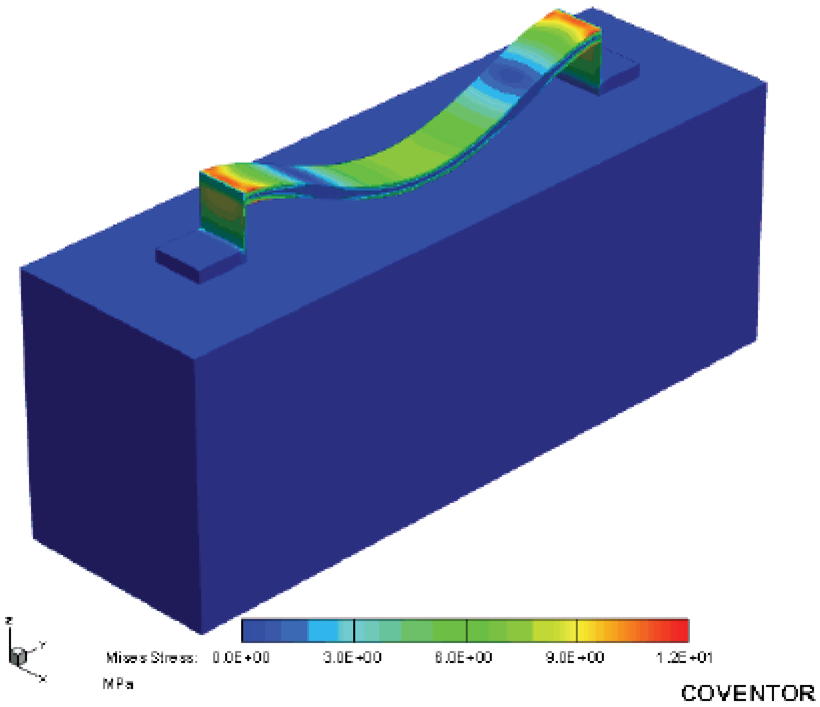


Fig. 3.32 3-D display shows surface stress along the beam surface which shows that maximum deflection is at the beam center and maximum surface stress is at the fixed anchors

<u>CoSolveEM simulations</u>

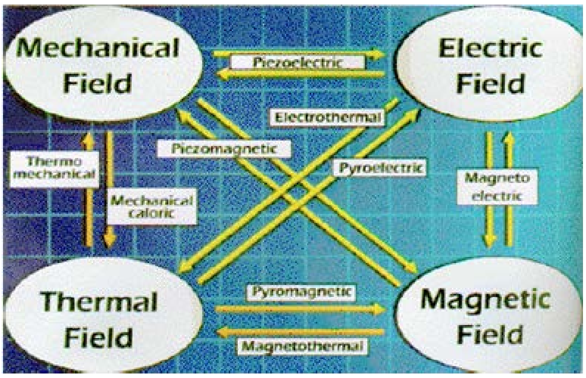


Fig. 3.33 Coupling effects

The CoSolveEM solver couples the electrostatic and mechanical solvers. The electrostatic results are input to the mechanical solver, and the results are used as feedbacks until convergence is achieved. Before starting with this analysis, the external pressure load is removed from the system. The voltage for the beam is set to 20 V.

The first step is to remove the pressure load applied in the MemMech simulation by opening the MemMech SurfaceBCs and changing the Set2 LoadValue to *0.0*. Then, from the *Analyzer* tab set the Solver field to *CoSolveEM*. From the Model/Mesh drop-down menu select the *beam* model and click on *Solver Setup*. In the CoSolveEM Settings dialog, set the Independent Variable to *Voltage* and then set the Analysis Option field to *Single Step* and the Iteration Method to *Relaxation*.

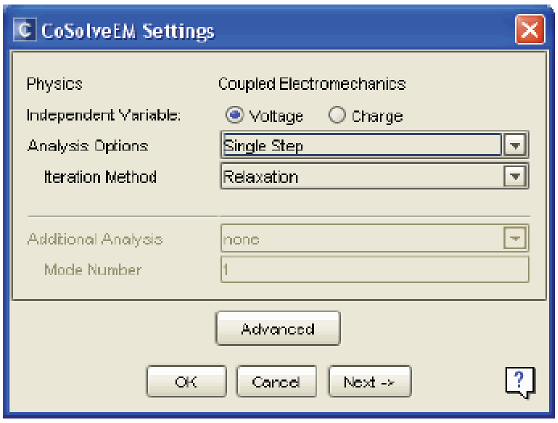


Fig. 3.34 CoSolveEM Settings

In the CoSolveEM BCs window click on the ConductorBCs and change the voltage for the beam to *20* V. Then click Run to start the simulation and save the analysis as *cs_run1*.

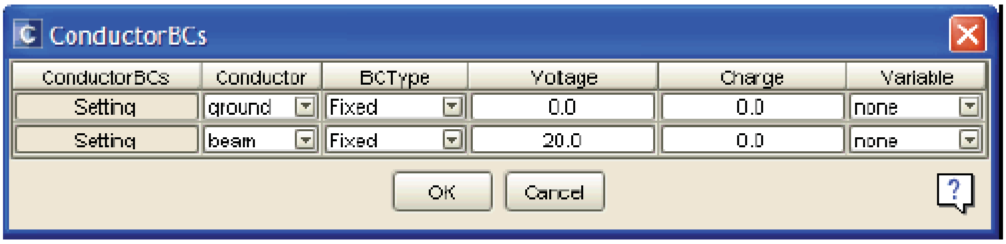


Fig. 3.35 ConductorBCs window

To view the numerical displacement results, click in the *View Results* icon and select *Displacement* from the Tables drop-down menu and click on the *View Table* icon.

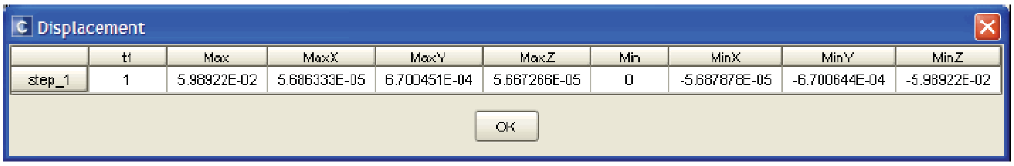


Fig. 3.36 Displacement results

To view the Capacitance results, select *Capacitance* and to view the Voltage results on Conductors select *Voltage*.

Capacitance

	t1	C_ground_ground	C_ground_beam	C_beam_beam
step_1	1	1.480666E-02	-1.480666E-02	1.480666E-02

OK

Fig. 3.37 Capacitance results

Voltage

	t1	V_ground	V_beam
step_1	1	0	20

OK

Fig. 3.38 Voltage results

TO view the Charge results on Conductors, select *Charge* and to view the Reaction Force results select *Reaction Force*.

Charge

	t1	Q_ground	Q_beam
step_1	1	-2.961333E-01	2.961333E-01

OK

Fig. 3.39 Charge results

Reaction Force

	t1	Fx_anchor1	Fy_anchor1	Fz_anchor1	Fx_anchor2	Fy_anchor2	Fz_anchor2
step_1	1	-2.439897E-04	1.172144E00	3.670744E00	4.663638E-04	-1.170707E00	3.678175E00

OK

Fig. 3.40 Reaction Force results

To view the 3-D charge results start the Visualizer by clicking on the *View 3D Results* icon. In the MemElectro Results frame select *Coventor→Geometry Scaling* and set the Z Scale to 5. Then select *Plot→Contour/Multi-coloring* and display the *Charge Density* results.

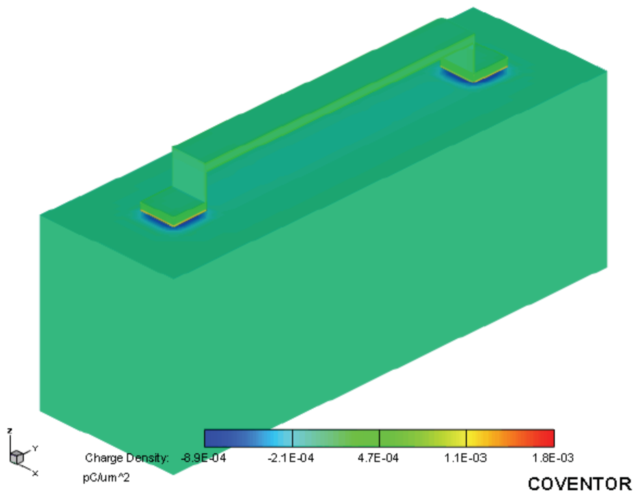


Fig. 3.41 CoSolve Surface Visualization of Results

To view the 3-D mechanical results select *Frame→Push Active Frame*. Then select *Plot→Contour/Multi-coloring* and *Mises Stress*. Set the Geometry Scaling dialog as: Scale Z to *5*, select *Deform Using Displacements* and Exaggeration to *50*.

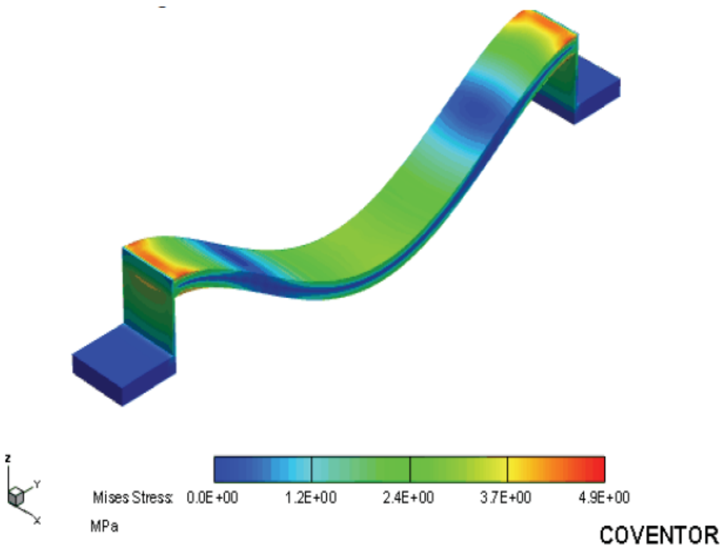


Fig. 3.42 CoSolve Mechanical Results

PARAMETRIC STUDY

Parametric study is used to see the effects changing stress step by step with a constant pressure load. The initial stress is changed while applying a constant pressure load of 0.001 MegaPascals. The step size was set to 5 steps.

From the *Analyzer* tab, select the beam model from the Model/Mesh field and click on the *Start Preprocessor* icon and the save the model as *beam_stress*. Then click on the *Part Selection* icon and then on the beam. Select *Edit→Properties* and click on the *MPD Editor* icon. After the MPD Editor window opens change the ALUMINIUM(FILM) Stress values to *10,10,0* for S_x,S_y and S_z. The next step is to select *MemMech* as the solver from the Analyzer tab and select *beam_stress* model. Select create a new analysis and click on the *Solver Setup* icon. In the Settings dialog set the Time Dependence back to *Steady-State* and in the MemMech BCs window click on SurfaceBCs. Set the parameters as shown below.

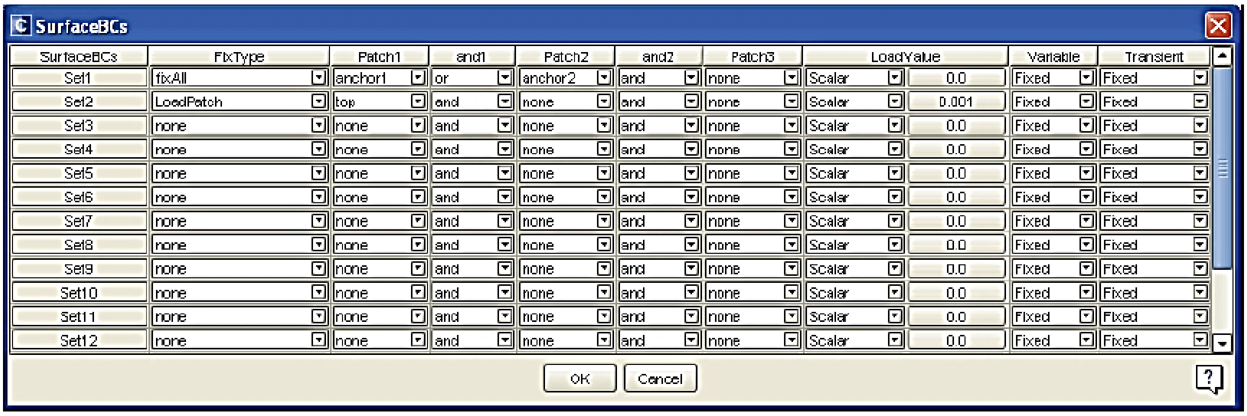


Fig. 3.43 SurfaceBCs window with the reapplied 0.001 pressure load to the top patch

To access the MemMech Parametric Study Function click on the Parametric Study in the MemMech BCs window.

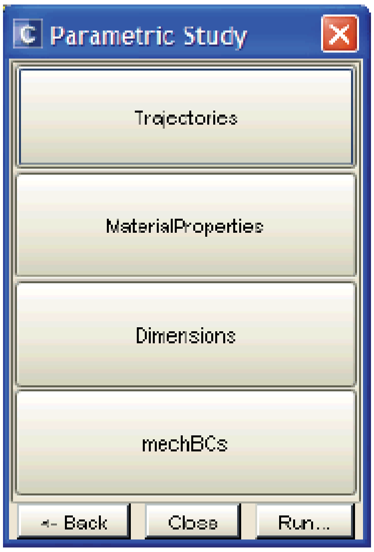


Fig. 3.44 Parametric Study window

Click on Trajectories and set the t1 TrajectoryType to *Delta*. Set the Delta trajectory of 5 steps by clicking on Edit. Set the values as shown below.

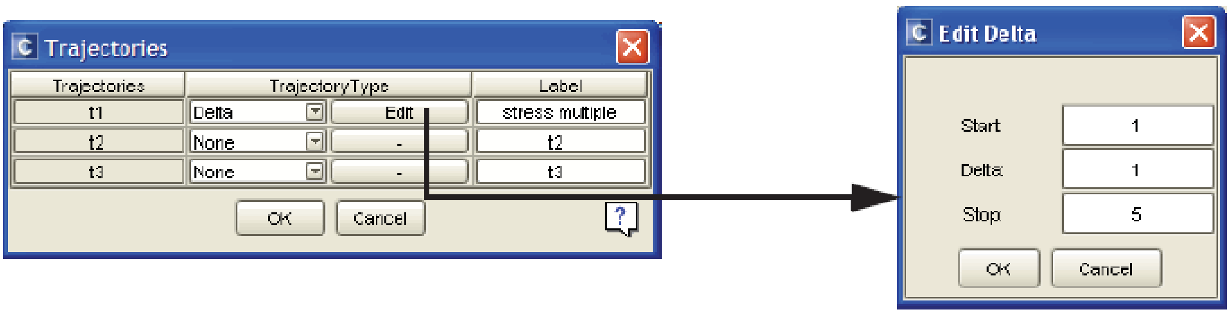


Fig. 3.45 Set up of the Delta trajectory of 5 steps

Click on MaterialProperties and set the MaterialProp1 Property to *Stress*. Set the Trajectory to *t1*, the Type to *Factor*, the ScaleFactor to *1.0* and the Material to *ALUMINUM(FILM)*. Then start the simulation by clicking Run in the Parametric Study window. In the Run Analysis window enter *mech_para*.

To view the numerical and graphical mechDomain results select mechDomain from the Tables drop-down menu for the numerical results (see figure 3.46) and for the graphical results (see figure 3.47) select mechDomain from the Graphs drop-down menu and then click on the View Graph icon.

mechDomain

	stress multiple	Node Displacement_Maximum	Node Displacement_Minimum	Node X Displacement_Maximum	Node X Displacement_Minimum
Step 1	1	1.593509E-01	0	6.067072E-04	-6.067072E-04
Step 2	2	1.668228E-01	0	1.093223E-03	-1.093223E-03
Step 3	3	1.737497E-01	0	1.581683E-03	-1.581683E-03
Step 4	4	1.802885E-01	0	2.071922E-03	-2.071922E-03
Step 5	5	1.865251E-01	0	2.608934E-03	-2.608934E-03

OK

mechDomain

Node Y Displacement_Maximum	Node Y Displacement_Minimum	Node Z Displacement_Maximum	Node Z Displacement_Minimum
3.853575E-03	-3.853575E-03	9.16316E-04	-1.593498E-01
6.953832E-03	-6.953832E-03	1.657806E-03	-1.668193E-01
1.008381E-02	-1.008381E-02	2.390588E-03	-1.737425E-01
1.32111E-02	-1.32111E-02	3.114649E-03	-1.802766E-01
1.633549E-02	-1.633549E-02	3.83073E-03	-1.865075E-01

OK

Fig. 3.46 Parametric Study mechDomain numerical results

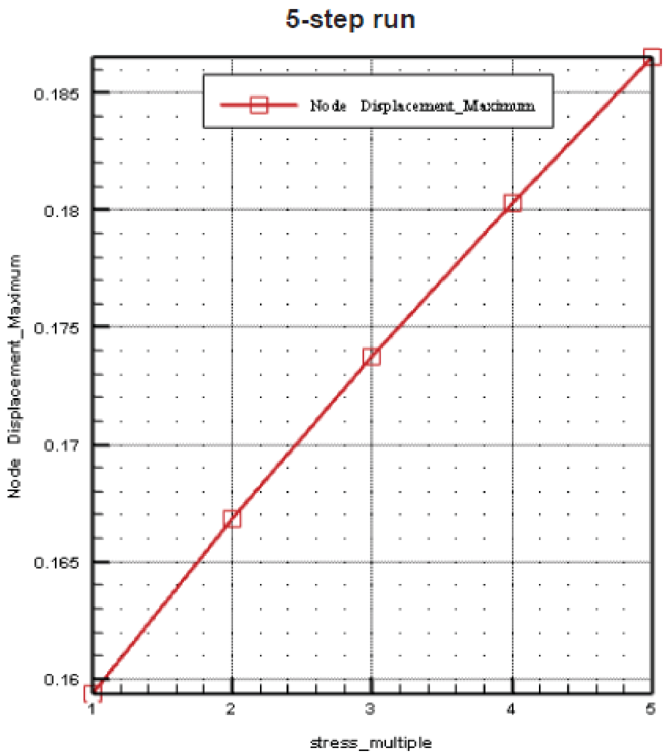


Fig. 3.47 Parametric Study mechDomain graphical results

To view the displacement results in the Visualizer, click on the arrow beside the Load Results view and select *All results* and then click on the *View 3D Results* icon. Select *Coventor→Part Visibility* and move the *ground* part to the Visible column and then select *Plot→Contour/Multi-coloring* and then select *Displacement Y* from the drop-down menu. Select *Coventor→Geometry Scaling* and set the Z Scale to *5*, check *Deform Using Displacements* and set Exaggeration to *10.* To view the results for each trajectory step select *Coventor→Result Selection*.

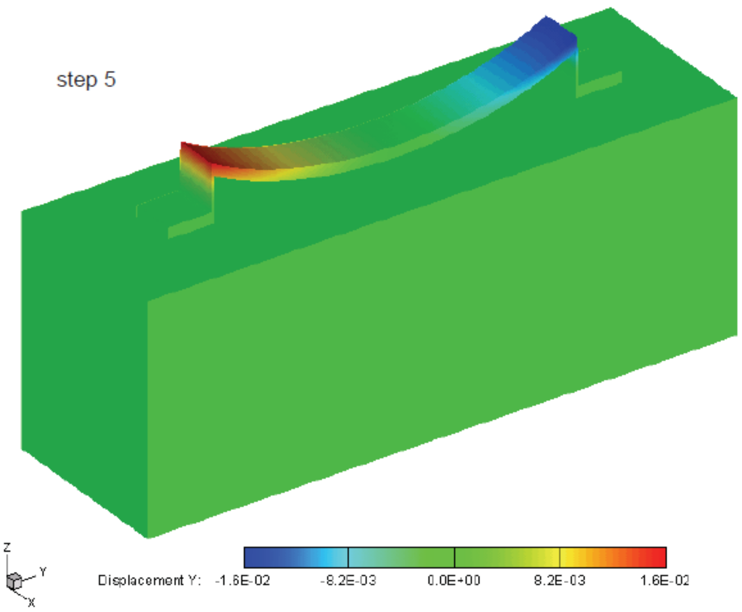
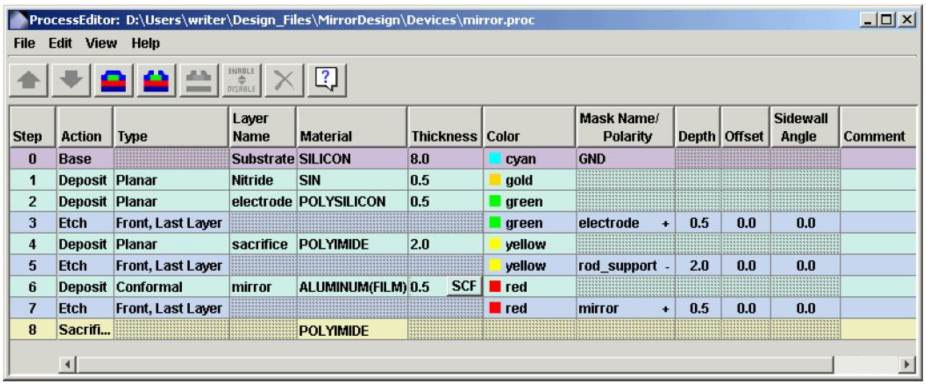


Fig. 3.48 Displacement in the y-direction

3.4. MIRROR DESIGN

3.4.1. <u>SIMPLE MIRROR DESIGN</u>

Let's see a more complex example: designing a mirror that rotates around an attached support. A thin multisided aluminum plate will rotate about an axis which is a support rod. The aluminum plate is rotated by applying voltages to the electrodes that are placed under the mirror ends. First the 3D structure is designed with the given deposition and etching steps.



Step	Action	Type	Layer Name	Material	Thickness	Color	Mask Name/ Polarity	Depth	Offset	Sidewall Angle	Comment
0	Base		Substrate	SILICON	8.0	cyan	GND				
1	Deposit	Planar	Nitride	SIN	0.5	gold					
2	Deposit	Planar	electrode	POLYSILICON	0.5	green					
3	Etch	Front, Last Layer				green	electrode +	0.5	0.0	0.0	
4	Deposit	Planar	sacrifice	POLYIMIDE	2.0	yellow					
5	Etch	Front, Last Layer				yellow	rod_support -	2.0	0.0	0.0	
6	Deposit	Conformal	mirror	ALUMINUM(FILM)	0.5 SCF	red					
7	Etch	Front, Last Layer				red	mirror +	0.5	0.0	0.0	
8	Sacrifi...			POLYIMIDE							

Fig. 3.49 Process Editor window

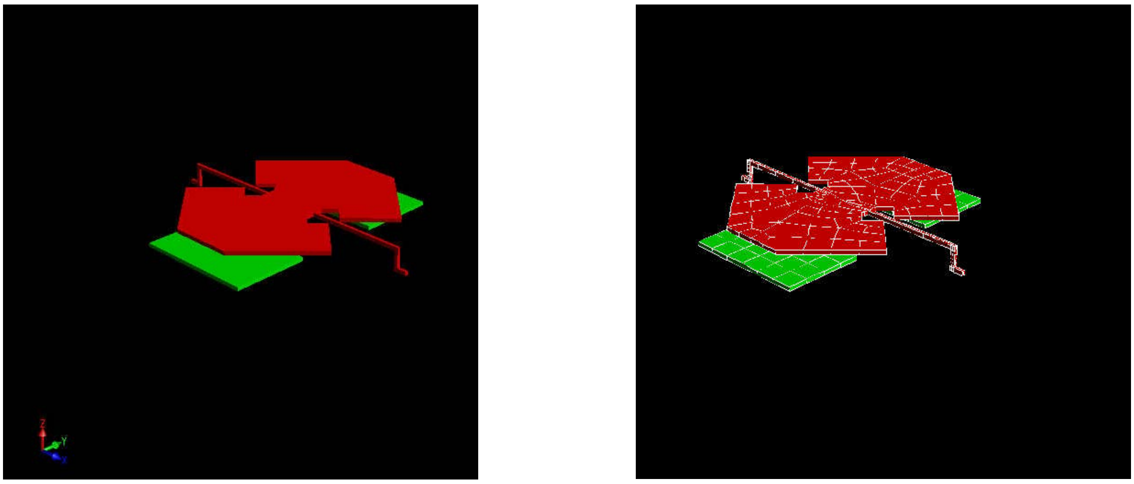


Fig. 3.50 The created 3D structure and the meshed structure

After that CoSolve single step simulations are done with voltages of 5 and 10V to electrode1 and 2 respectively. The results taken are shown below.

Capacitance (pF)

Capacitance (pF)	electrode1	electrode2	mirror
electrode1	1.145923E-03	-7.452819E-05	-1.071395E-03
electrode2	-7.452819E-05	1.174705E-03	-1.100176E-03
mirror	-1.071395E-03	-1.100176E-03	2.171571E-03

Close

Fig. 3.51 The Capacitance results

electroBCs

electroBCs	Voltage (V)	Charge (pC)
electrode1	5.000000E00	4.984319E-03
electrode2	1.000000E01	1.137442E-02
mirror	0.0	-1.635873E-02

Close

Fig. 3.52 The electroBCs results

mechDomain

mechDomain	Maximum	Minimum
Node Displacement	7.567401E-02	0.0
Node X Displacement	1.732652E-03	-1.730912E-03
Node Y Displacement	2.772496E-04	-4.828912E-03
Node Z Displacement	5.068669E-02	-7.567350E-02

Close

Fig. 3.53 The mechDomain results

rxnForces

rxnForces	Fx	Fy	Fz
end2	-7.421982E-02	1.111218E-03	9.985456E-03
end1	7.416848E-02	1.083967E-03	9.954309E-03

Close

Fig. 3.54 The rxnForces results

The probes are connected to the electrodes are the voltage values are recorded. One example is shown:

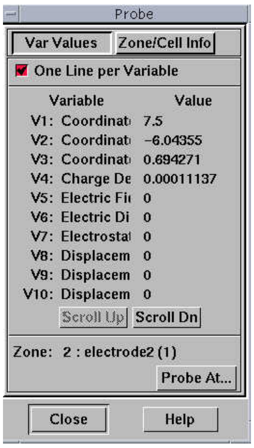


Fig. 3.55 Probe window

After that 3D stress results are investigated.

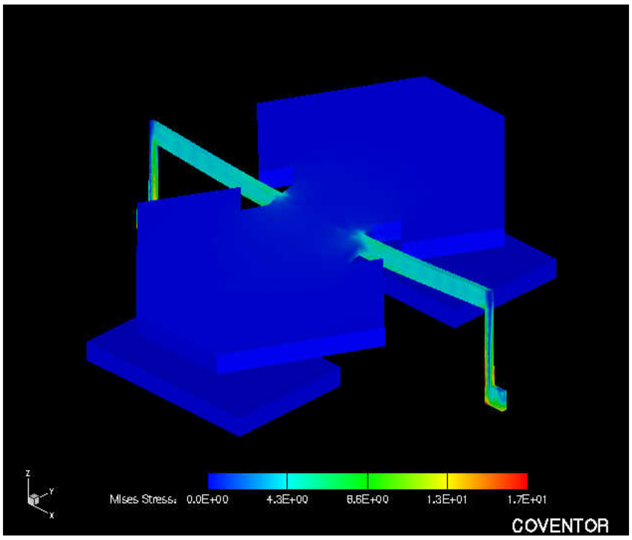


Fig. 3.56 The 3D stress results

After the single step simulation, multiple CoSolve analyses is done. Displacement values are given below.

Displacement	t1	t2	Max	MaxX	MaxY	MaxZ	Min	MinX	MinY
step_1	1.000000E01	0.0	9.518015E-02	1.723799E-03	6.049563E-03	7.320794E-02	0.0	-1.724804E-03	-2.218060E-
step_2	2.000000E01	0.0	3.877195E-01	2.045625E-03	2.426847E-02	3.286179E-01	0.0	-2.052585E-03	-2.334895E-
step_3	0.0	1.000000E01	7.857642E-02	1.714302E-03	1.075064E-04	5.858115E-02	0.0	-1.712931E-03	-5.235789E-
step_4	0.0	2.000000E01	3.833887E-01	2.048378E-03	2.333469E-05	3.249156E-01	0.0	-2.041549E-03	-2.414694E-

Close

Fig. 3.57 The displacements results

3.4.2. FOUR-LAYER OPTICAL MIRROR

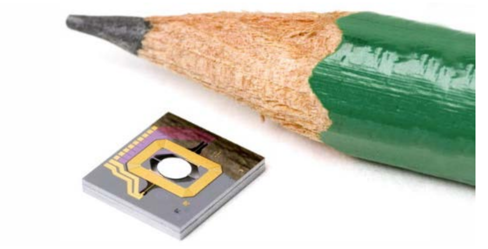


Fig. 3.58 Four-layer optical mirror

In the following part the software tool that will be used is CoventorWare. It will be designed a solution for four-layer optical mirror.

After the CoventorView started select the *Import Tutorial* icon and then select Designer. Enter the *designer.mps* and click Open. After select the *Start Layout Editor* and from the Layout Editor menu bar select *File→New→From Template…*.In the Template file name navigation window that opens select foundry.cat and then save the file as *micromirror.cat*. From the menu bar select *Other→Grid* and change the Working Grid to *1.0*.

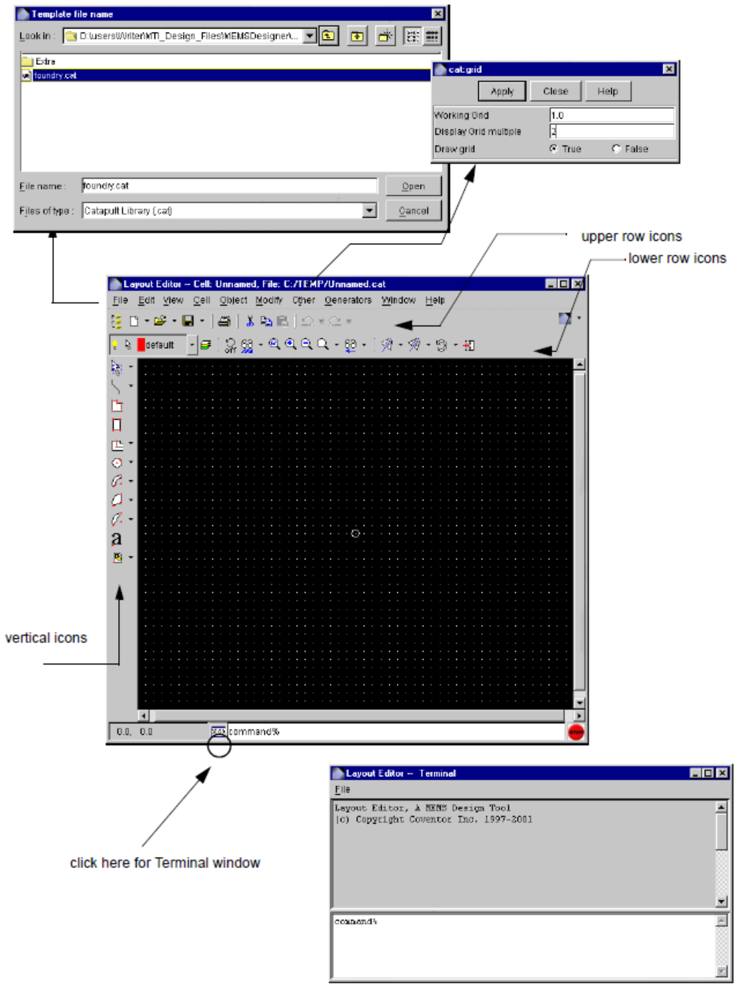


Fig. 3.59 Screen Illustrations

The next step is to draw a four-layer mirror using the values from the table below.

Layer	Color	Fill	Object	First coord	Value	Second coord	Value
poly2	blue	none	rectangle	lower left vertex	-50, 0	upper right vertex	50, 100
p12via	orange	none	circle	center	0, 50	point on radius	47, 50
metal	cyan	solid	circle	center	0, 50	point on radius	46, 50
poly1	green	solid	rectangle	lower left vertex	-54, -4	upper right vertex	54, 104

Fig. 3.60 Mirror Object values

On the top of the window, on the lower row icons click on the arrow to the right of the Current layer name and select *poly2*. Along the vertical icon column, select the *Rectangle* icon. While viewing the coordinate reading in the lower left corner of the canvas window, move the mouse to the left in an attempt to reach coordinate *-50, 0* and click. Move the mouse to coordinate *50, 100* and click. View your work by selecting the *View All* icon in the lower horizontal row.

Then select from the Current layer list *p12via*. Along the lower horizontal icon row, click on *Turn Repeat Mode On*. From the vertical icon column, select the *Circle* icon. In response to the Enter center coordinate prompt, move the mouse to coordinate *0, 50* and click (to define the center). In response to the Enter point on radius prompt, move the mouse to coordinate *47, 50* and click (to define the radius).

The next step is to select *metal* from the Current layer list. Place coordinate *0, 50*. Place coordinate *46, 50*. Right click to end the prompts for more circle coordinates.

The final step is to select *poly1* from the Current layer list. Select the *Rectangle* icon. Place coordinate *-54, -4* (zoom as necessary). Place coordinate *54, 104*. Right click to end the rectangle coordinate prompts.

Select *Save Cell As..* and in the window that opens enter *mirror*.

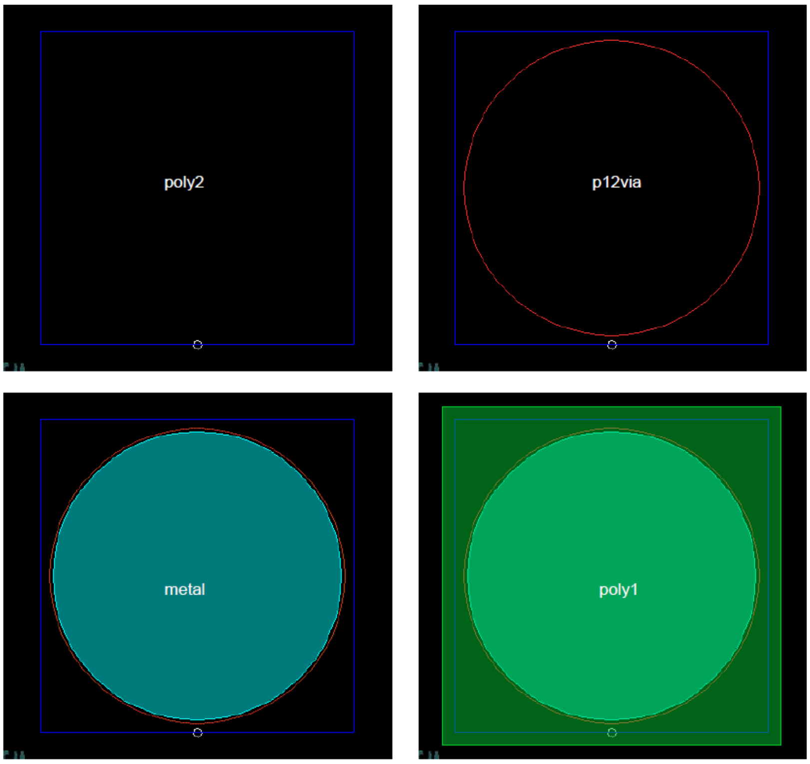


Fig. 3.61 Four sequenced layers

Along the top icon row, click on the arrow next to *New File* and select *New Cell*. From the vertical icon column, select the *Reference* icon. In the dialog window that opens, click on the Browse button next to Cell, and select *mirror*. In the Origin field, enter *0, 0*. Click on Apply and Close. Click on the View All icon in the lower horizontal row. From the menu bar, select *Cell→Copy ...*. In the dialog window that opens, next to the Source library field, click on *Browse...* and select the *partial_cells.cat* file. Next to the Source cell field, click on Browse... and select *popupmech*. In the Destination field, type *popupmech*. Click on Apply. Continue using the same copy procedure to copy the other four cells into your existing library. When finished, Close the dialog window.

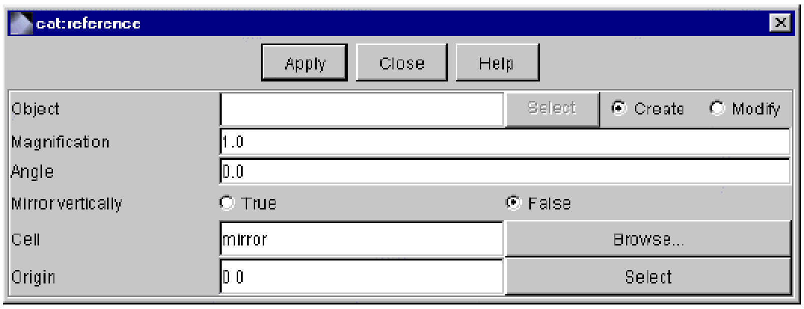


Fig. 3.62 Reference dialog window

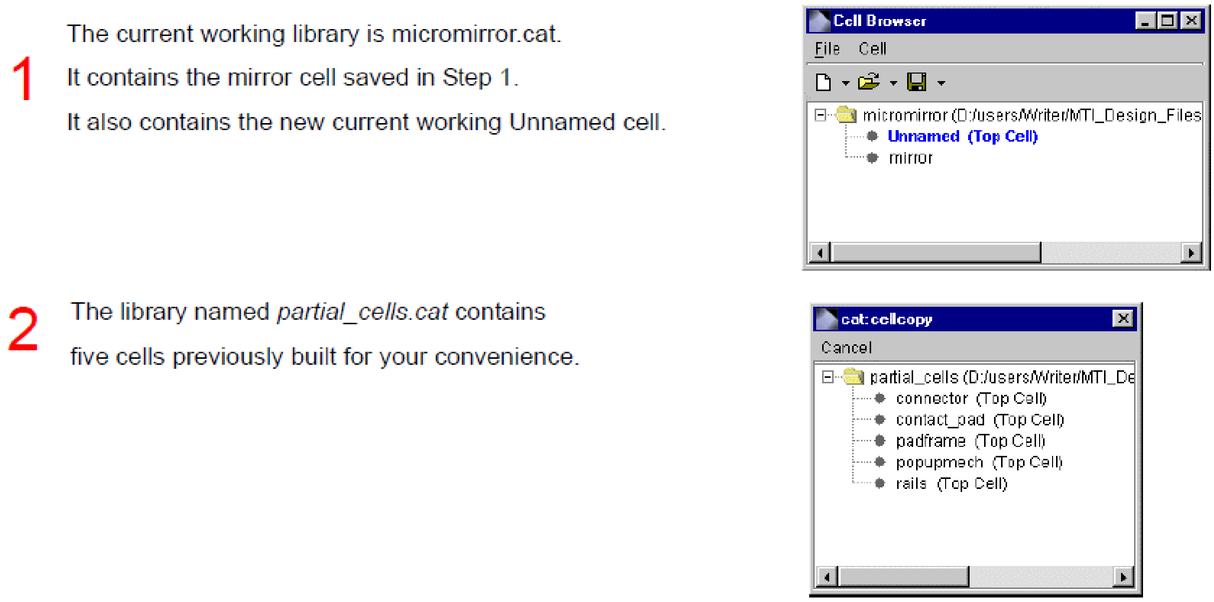


Fig. 3.63 Illustration of the concept of copied cells

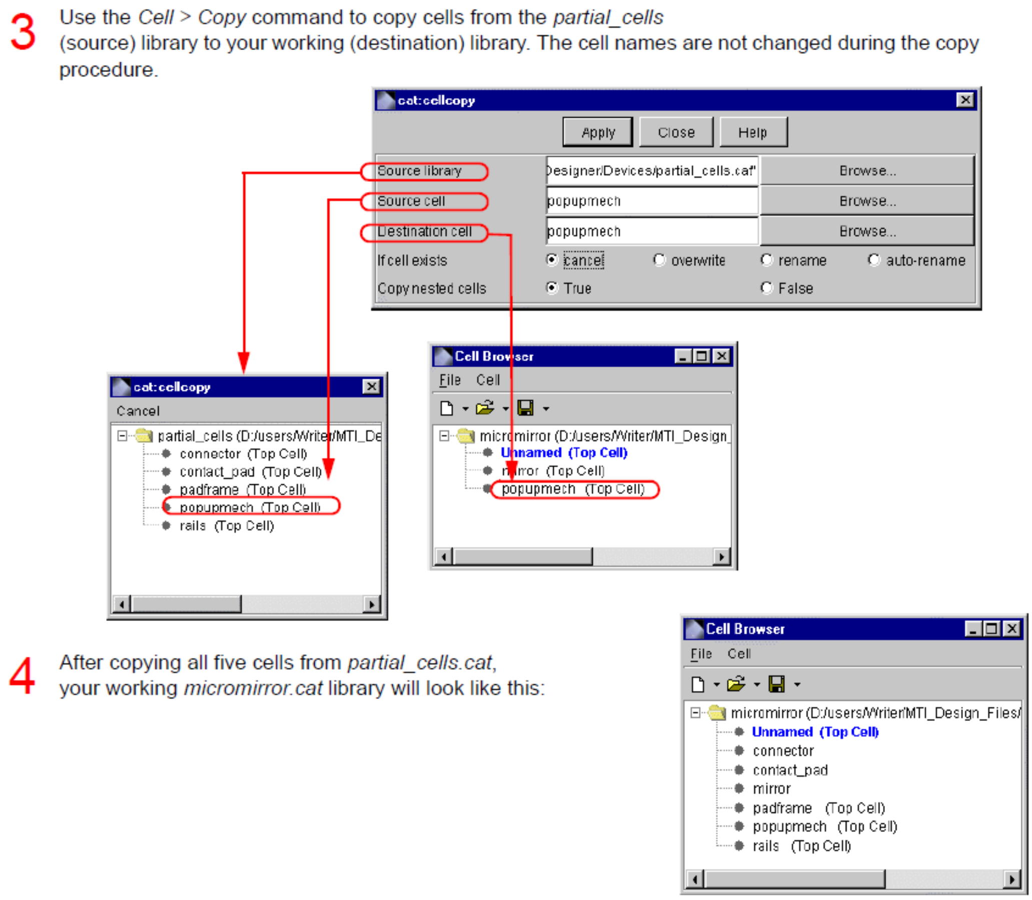


Fig. 3.64 Illustration of the concept of copied cells

To create the *popupmech* cell reference, from the vertical icon column, select the *Reference* icon. In the dialog window that opens, click on the *Browse* button next to Cell, and select *popupmech*. In the Origin field, enter *0, 0*. Click on Apply and Close. Click on *View All*.

To create a new *clip* layer first click on the *Layer browser* icon, next to the Current layer list and then, in the window that opens, from the menu bar, select *Layer→New…* In the dialog window, enter the Layer name *clip*. Click on the Color name arrow and select the color *red* from the menu. Click on Apply and Close.

To build a *clip* rectangle select the *clip* layer, the *Rectangle* icon and place coordinates *-59, 50* and *59, 105*. Right click to end the prompts for more rectangle coordinates.

Save the cell by using the arrow next to the diskette icon and then select *Save Cell As..*. In the window, enter *popup*; click on OK. Click on the diskette icon to save your *micromirror.cat* file.

To check the hierarchy of the reference cells from the menu bar, select *View→ Levels* and uncheck *Level 1*. When finished, from the menu bar, select *View→Levels→All*. In the Layout Editor Cell Browser window, click on the + next to *popup* and observe the reference cells.

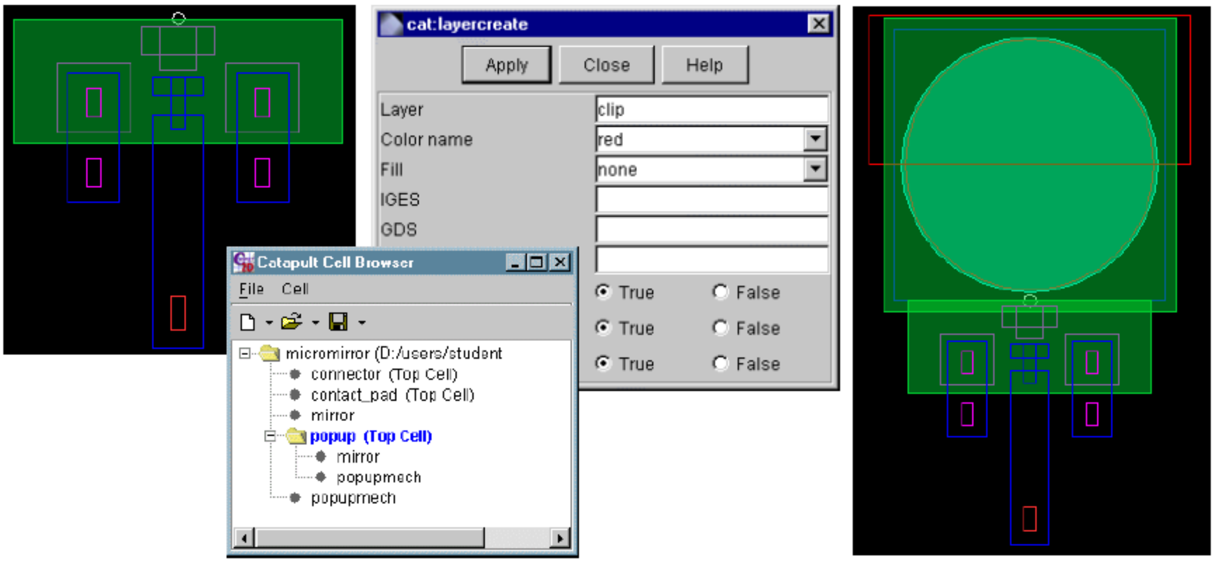


Fig. 3.65 Creating the clip layer

To set source catapult file and the Top Cell fields, on the *Layout* tab of the Function Manager, verify or set the Source Catapult File to *Designer\Devices\micromirror.cat*. Verify or set the Top Cell field to *popup*. To set the file path select the *Foundry* tab (see figure 3.66). To set the Process file path, in the window of the Process field, set the path to the

Designer\Devices directory and select the *foundry.proc* file. Navigate using the folder icon next to the field.

To view the Process Editor file select the Foundry tab and then click on the icon to the right of the Process File field (see figure 3.67) to open the process Editor. Close when finished.

To create the file name for the solid model, from the Function Manager, select the Solid Model tab. A default *sat* file will appear in the Solid Model field. Verify that the Solid Model File is named *micromirror.sat*.

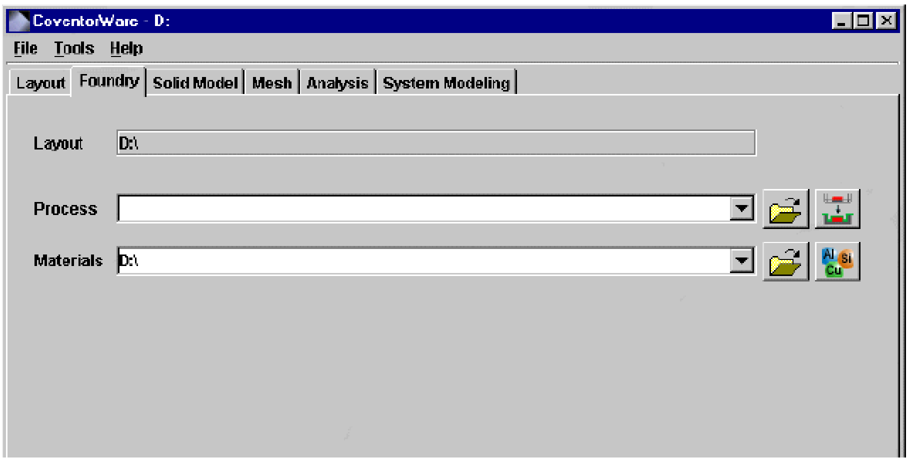
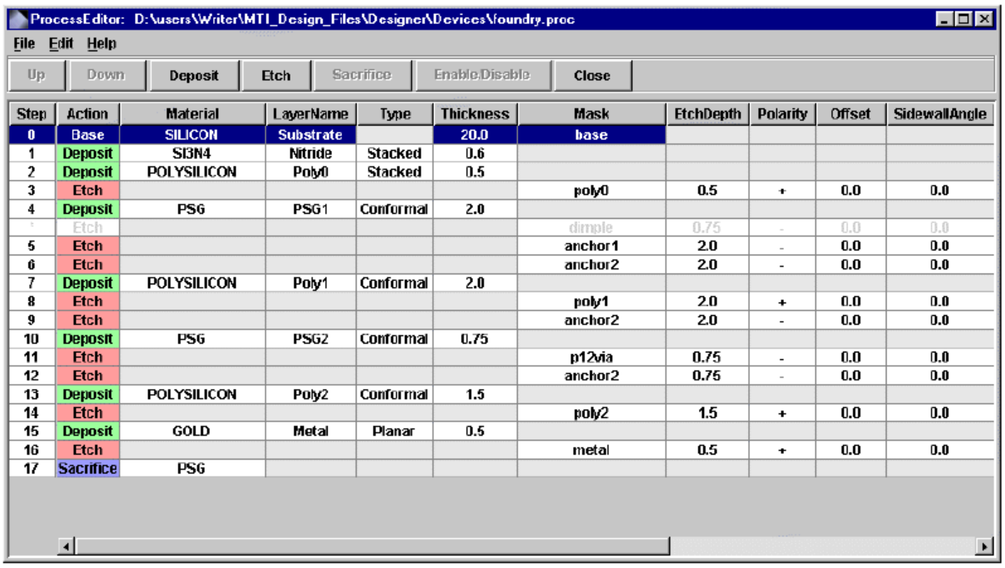


Fig. 3.66 Foundry tab



Step	Action	Material	LayerName	Type	Thickness	Mask	EtchDepth	Polarity	Offset	SidewallAngle
0	Base	SILICON	Substrate		20.0	base				
1	Deposit	SI3N4	Nitride	Stacked	0.6					
2	Deposit	POLYSILICON	Poly0	Stacked	0.5					
3	Etch					poly0	0.5	+	0.0	0.0
4	Deposit	PSG	PSG1	Conformal	2.0					
*	Etch					dimple	0.75	-	0.0	0.0
5	Etch					anchor1	2.0	-	0.0	0.0
6	Etch					anchor2	2.0	-	0.0	0.0
7	Deposit	POLYSILICON	Poly1	Conformal	2.0					
8	Etch					poly1	2.0	+	0.0	0.0
9	Etch					anchor2	2.0	-	0.0	0.0
10	Deposit	PSG	PSG2	Conformal	0.75					
11	Etch					p12via	0.75	-	0.0	0.0
12	Etch					anchor2	0.75	-	0.0	0.0
13	Deposit	POLYSILICON	Poly2	Conformal	1.5					
14	Etch					poly2	1.5	+	0.0	0.0
15	Deposit	GOLD	Metal	Planar	0.5					
16	Etch					metal	0.5	+	0.0	0.0
17	Sacrifice	PSG								

Fig. 3.67 Process Editor window

To build the 3-D model click on Build Model option. The Preprocessor opens automatically after a successful build. To view the 3-D click on the *View Model* option. At the top of the Preprocessor window, click on the *Isometric View* and *Normalize* icons to adjust the view.

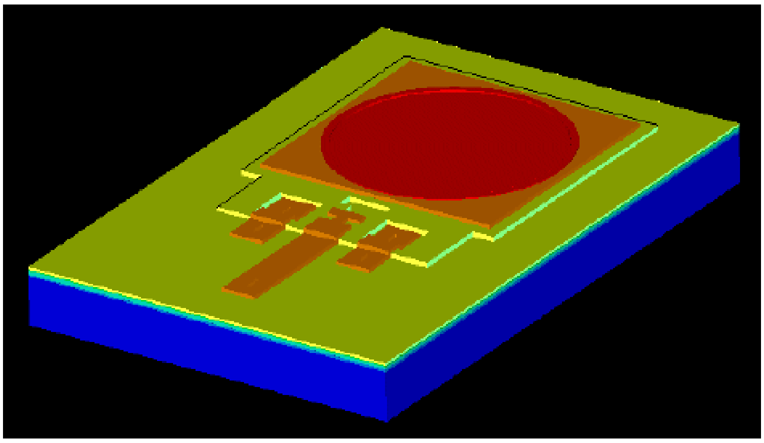


Fig. 3.68 Mirror Assembly

To repeat build and view with Clip layer first in the Function Manager, *Solid Model* Tab, check the box next to *Clip Device with Mask* and verify that the layer name field is set to *clip*. Verify that the Top Cell is still *popup*. Click on Build Model. Click on *Isometric View* and *Normalize*. In the Designer Setup window, uncheck the box next to *Clip layer*.

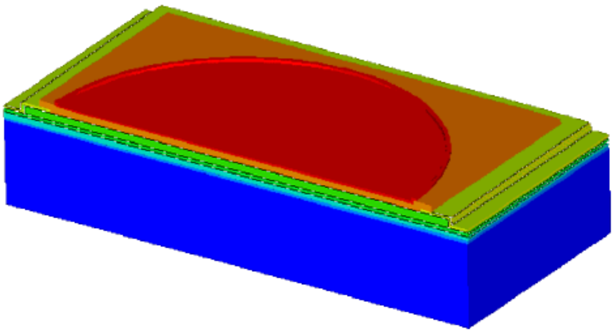


Fig. 3.69 Clip layer view

3.4.3. THERMALLY-ACTUATED POP-UP MIRROR

The thermally actuated mirror design has numerous applications, including optical switching and scanning. The mirror is controlled by an array of actuators connected to the mirror.

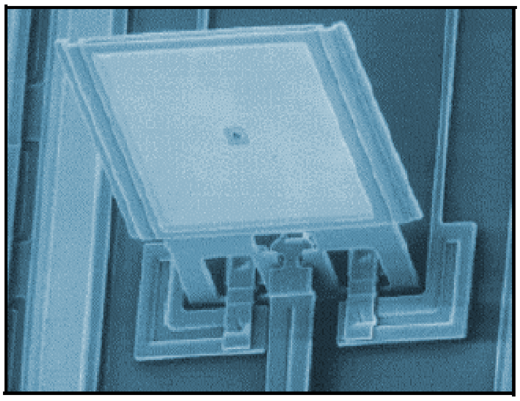


Fig. 3.70 SEM image of Pop-up mirror

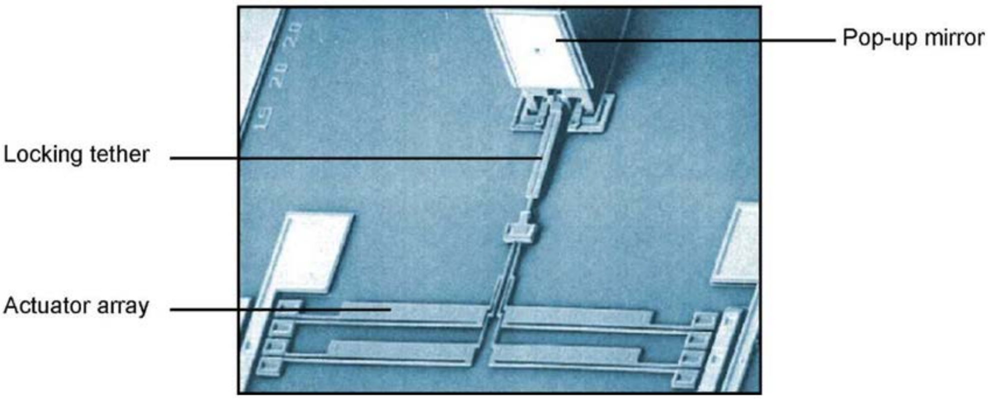


Fig. 3.71 SEM image of complete scanning micromirror assembly

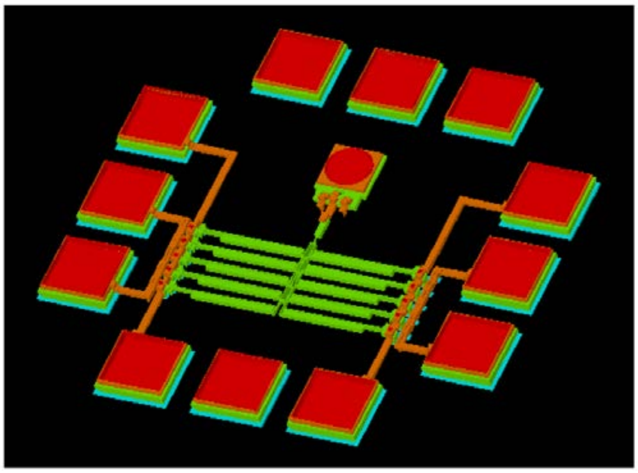


Fig. 3.72 Designer 3D model of micromirror assembly

The mirror that makes up part of the device is known as a pop-up mirror because it is flat when fabricated, but extends out from the wafer plane when in use. It uses a layer of gold as its optical reflector. In this design, the mirror is manually assembled to a nearly vertical position and locked in place by a rod connected to the actuator. The mirror rotates around pins that are held to the substrate by staple-like structures that function as hinges.

The thermal actuator, or heatuator, uses differential thermal expansion to provide high forces and substantial linear motion. To increase the available force, this design uses a parallel combination of ten thermal actuators. The heat is generated by passing current through the device. This actuator is designed to pull the mirror to avoid flexing the connecting rod. Each actuator is capable of over 20 µN of force and over 10 µm of deflection at an input electrical power of 25 mW (at 5 V and 5 mA). A 5 x 2 array used here can provide 200 µN of force while consuming 250 mW.

The conductors used to feed this device are thick enough to prevent significant parasitic voltage drops for 50 mA of current and to reduce the associated power loss. Because this is a significant amount of power, the fabricated chip must be bonded to a thermally efficient heat sink.

In the following part the software tool that will be used is CoventorWare. It will be designed a solution for thermally-actuated pop-up mirror.

After the CoventorView started select the *Import Tutorial* icon and then select Designer. Enter the *designer.mps* and click Open. After select the *Start Layout Editor* and from the Layout Editor menu bar select *File→New→From Template…*.In the Template file name navigation window that opens select foundry.cat and then save the file as *micromirror.cat*. From the menu bar select *Other→Grid* and change the Working Grid to *1.0*.

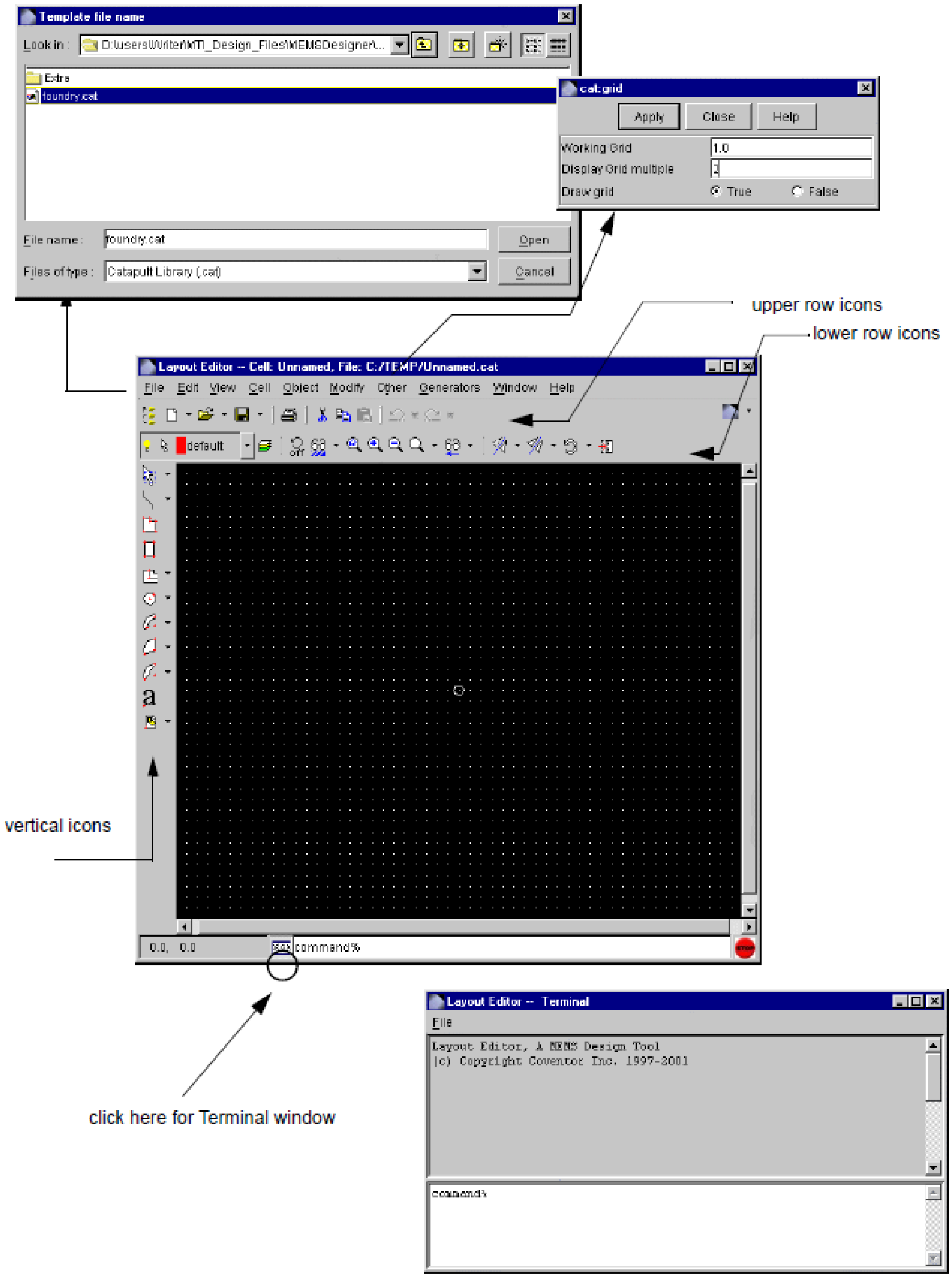


Fig. 3.73 Screen Illustrations

The next step is to draw a four-layer mirror using the values from the table below.

Layer	Color	Fill	Object	First coord	Value	Second coord	Value
poly2	blue	none	rectangle	lower left vertex	-50, 0	upper right vertex	50, 100
p12via	orange	none	circle	center	0, 50	point on radius	47, 50
metal	cyan	solid	circle	center	0, 50	point on radius	46, 50
poly1	green	solid	rectangle	lower left vertex	-54, -4	upper right vertex	54, 104

Fig. 3.74 Mirror Object values

On the top of the window, on the lower row icons click on the arrow to the right of the Current layer name and select *poly2*. Along the vertical icon column, select the *Rectangle* icon. While viewing the coordinate reading in the lower left corner of the canvas window, move the mouse to the left in an attempt to reach coordinate *-50, 0* and click. Move the mouse to coordinate *50, 100* and click. View your work by selecting the *View All* icon in the lower horizontal row.

Then select from the Current layer list *p12via*. Along the lower horizontal icon row, click on *Turn Repeat Mode On*. From the vertical icon column, select the *Circle* icon. In response to the Enter center coordinate prompt, move the mouse to coordinate *0, 50* and click (to define the center). In response to the Enter point on radius prompt, move the mouse to coordinate *47, 50* and click (to define the radius).

The next step is to select *metal* from the Current layer list. Place coordinate *0, 50*. Place coordinate *46, 50*. Right click to end the prompts for more circle coordinates.

The final step is to select *poly1* from the Current layer list. Select the *Rectangle* icon. Place coordinate *-54, -4* (zoom as necessary). Place coordinate *54, 104*. Right click to end the rectangle coordinate prompts.

Select *Save Cell As..* and in the window that opens enter *mirror*.

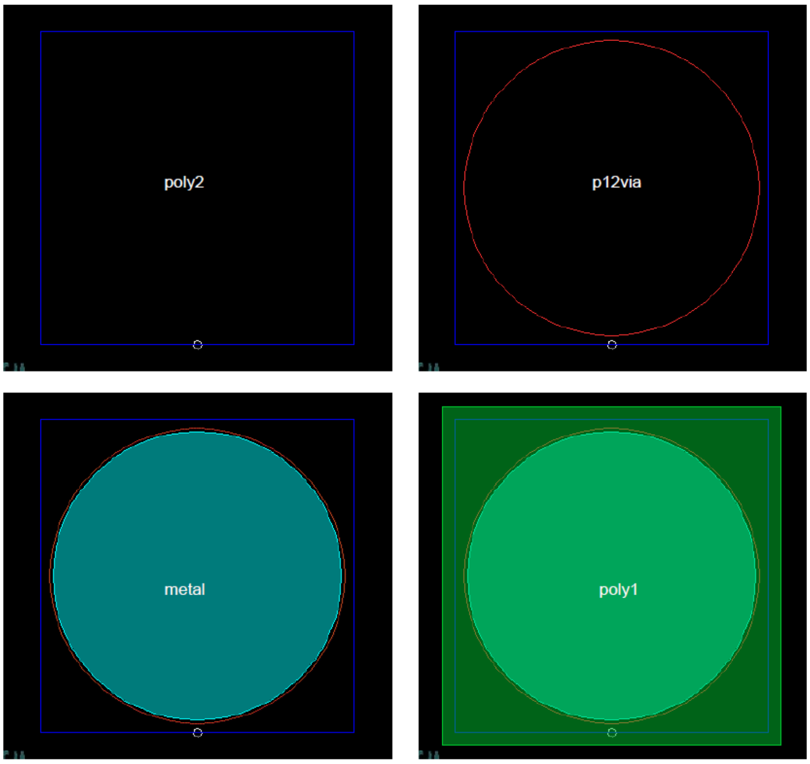


Fig. 3.75 Four sequenced layers

Along the top icon row, click on the arrow next to *New File* and select *New Cell*. From the vertical icon column, select the *Reference* icon. In the dialog window that opens, click on the Browse button next to Cell, and select *mirror*. In the Origin field, enter *0, 0*. Click on Apply and Close. Click on the View All icon in the lower horizontal row. From the menu bar, select *Cell→Copy ...*. In the dialog window that opens, next to the Source library field, click on *Browse...* and select the *partial_cells.cat* file. Next to the Source cell field, click on Browse... and select *popupmech*. In the Destination field, type *popupmech*. Click on Apply. Continue using the same copy procedure to copy the other four cells into your existing library. When finished, Close the dialog window.

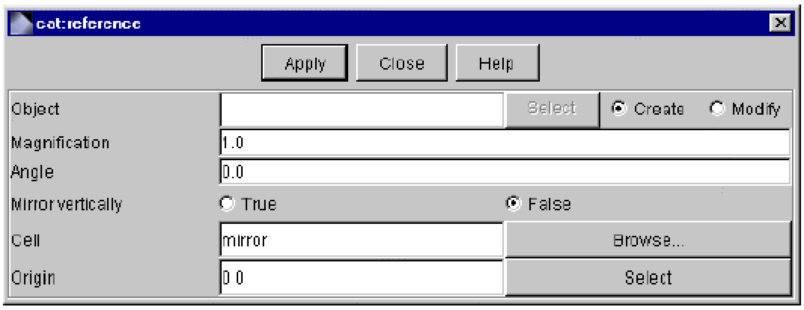


Fig. 3.76 Reference dialog window

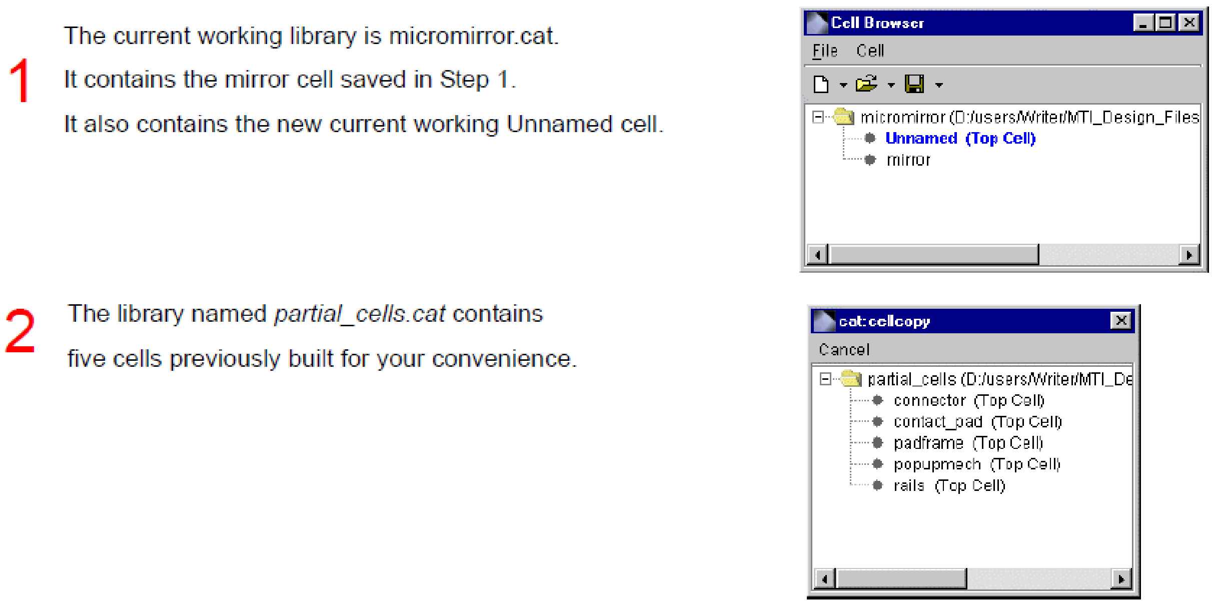


Fig. 3.77 Illustration of the concept of copied cells

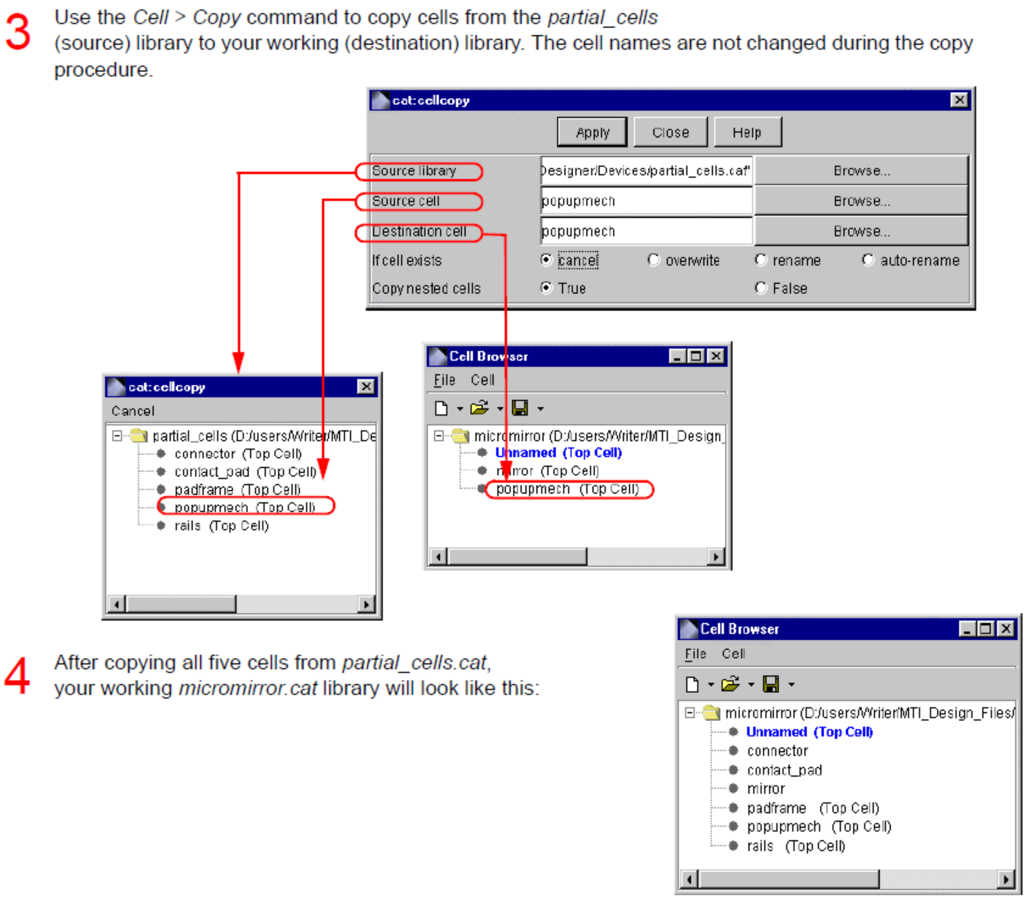


Fig. 3.78 Illustration of the concept of copied cells

To create the *popupmech* cell reference, from the vertical icon column, select the *Reference* icon. In the dialog window that opens, click on the *Browse* button next to Cell, and select *popupmech*. In the Origin field, enter *0, 0*. Click on Apply and Close. Click on *View All*.

To create a new *clip* layer first click on the *Layer browser* icon, next to the Current layer list and then, in the window that opens, from the menu bar, select *Layer→New…* In the dialog window, enter the Layer name *clip*. Click on the Color name arrow and select the color *red* from the menu. Click on Apply and Close.

To build a *clip* rectangle select the *clip* layer, the *Rectangle* icon and place coordinates *-59, 50* and *59, 105*. Right click to end the prompts for more rectangle coordinates.

Save the cell by using the arrow next to the diskette icon and then select *Save Cell As*... In the window, enter *popup*; click on OK. Click on the diskette icon to save your *micromirror.cat* file.

To check the hierarchy of the reference cells from the menu bar, select *View→ Levels* and uncheck *Level 1*. When finished, from the menu bar, select *View→Levels→All*. In the Layout Editor Cell Browser window, click on the + next to *popup* and observe the reference cells.

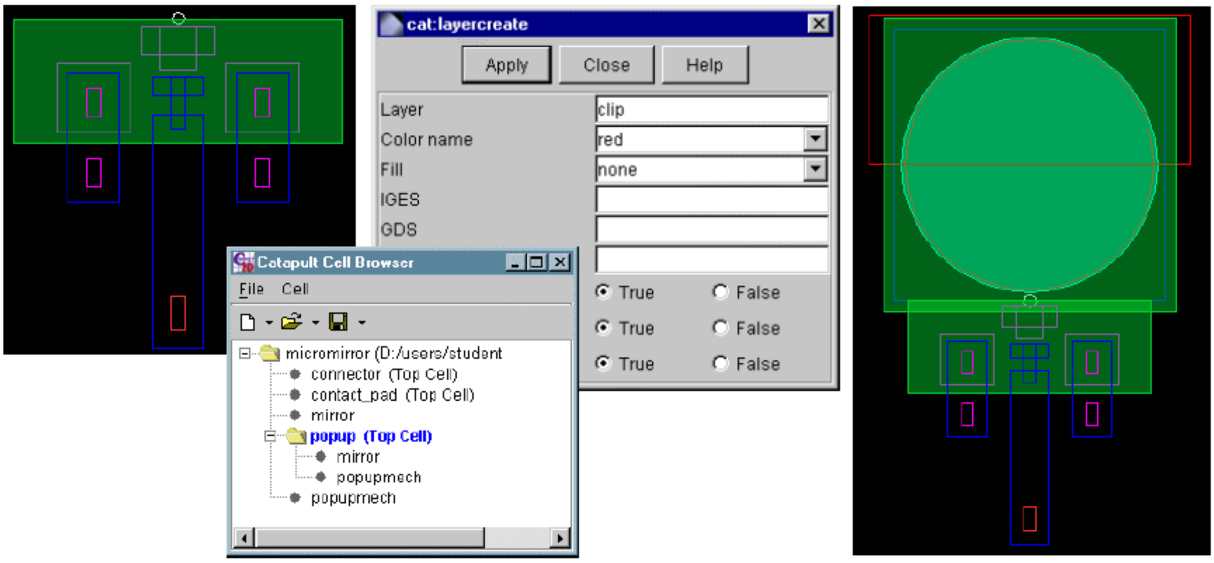


Fig. 3.79 Creating the clip layer

To set source catapult file and the Top Cell fields, on the *Layout* tab of the Function Manager, verify or set the Source Catapult File to *Designer\Devices\micromirror.cat*. Verify or set the Top Cell field to *popup*. To set the file path select the *Foundry* tab (see figure 3.80). To set the Process file path, in the window of the Process field, set the path to the *Designer\Devices* directory and select the *foundry.proc* file. Navigate using the folder icon next to the field.

To view the Process Editor file select the Foundry tab and then click on the icon to the right of the Process File field (see figure 3.81) to open the process Editor. Close when finished.

To create the file name for the solid model, from the Function Manager, select the Solid Model tab. A default *sat* file will appear in the Solid Model field. Verify that the Solid Model File is named *micromirror.sat*.

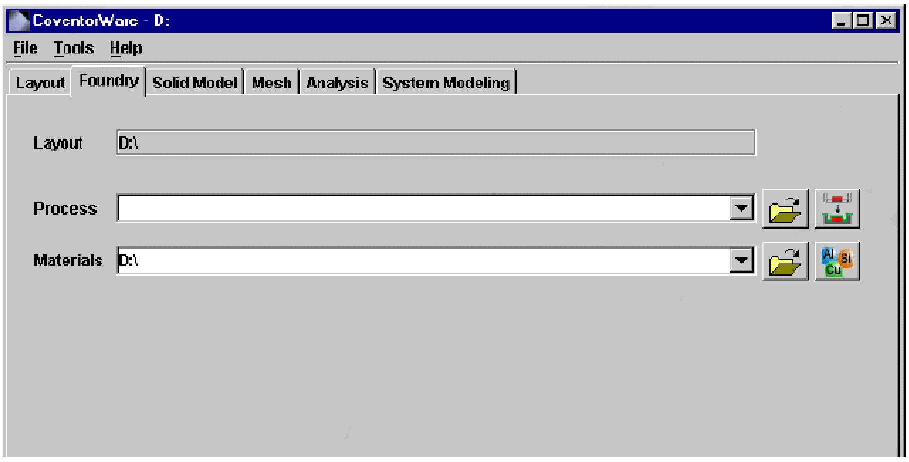
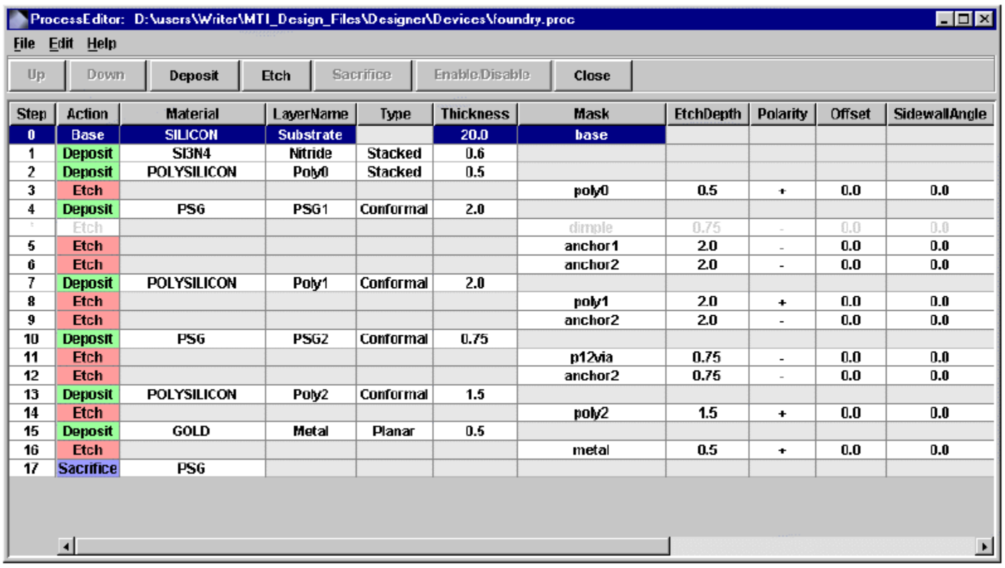


Fig. 3.80 Foundry tab



Step	Action	Material	LayerName	Type	Thickness	Mask	EtchDepth	Polarity	Offset	SidewallAngle
0	Base	SILICON	Substrate		20.0	base				
1	Deposit	SI3N4	Nitride	Stacked	0.6					
2	Deposit	POLYSILICON	Poly0	Stacked	0.5					
3	Etch					poly0	0.5	+	0.0	0.0
4	Deposit	PSG	PSG1	Conformal	2.0					
	Etch					dimple	0.75	-	0.0	0.0
5	Etch					anchor1	2.0	-	0.0	0.0
6	Etch					anchor2	2.0	-	0.0	0.0
7	Deposit	POLYSILICON	Poly1	Conformal	2.0					
8	Etch					poly1	2.0	+	0.0	0.0
9	Etch					anchor2	2.0	-	0.0	0.0
10	Deposit	PSG	PSG2	Conformal	0.75					
11	Etch					p12via	0.75	-	0.0	0.0
12	Etch					anchor2	0.75	-	0.0	0.0
13	Deposit	POLYSILICON	Poly2	Conformal	1.5					
14	Etch					poly2	1.5	+	0.0	0.0
15	Deposit	GOLD	Metal	Planar	0.5					
16	Etch					metal	0.5	+	0.0	0.0
17	Sacrifice	PSG								

Fig. 3.81 Process Editor window

To build the 3-D model click on Build Model option. The Preprocessor opens automatically after a successful build. To view the 3-D click on the *View Model* option. At the top of the Preprocessor window, click on the *Isometric View* and *Normalize* icons to adjust the view.

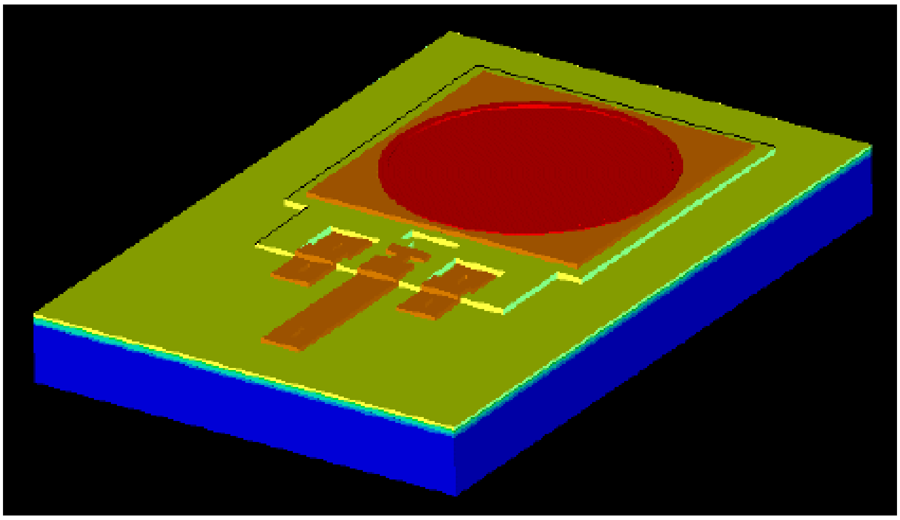


Fig. 3.82 Mirror Assembly

To repeat build and view with Clip layer first in the Function Manager, *Solid Model* Tab, check the box next to *Clip Device with Mask* and verify that the layer name field is set to *clip*. Verify that the Top Cell is still *popup*. Click on Build Model. Click on *Isometric View* and *Normalize*. In the Designer Setup window, uncheck the box next to *Clip layer*.

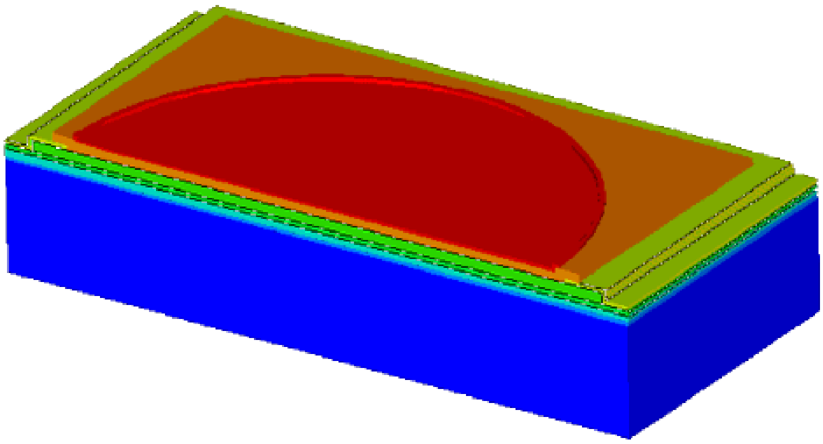


Fig. 3.83 Clip layer view

The next step is to draw a heatuator using the values from the table below.

Layer	Color	Fill	Location	Description	1st coord.	2nd coord.
poly1	green	solid	Lower left	Anchor pad	0, 0	11, 11
			Bottom half	Long, thin arm	11, 9	219, 11
anchor1	lightorange	none	On lower anchor pad	Small anchor	4, 4	7, 7
poly1	green	solid	Upper left	Anchor pad	0, 13	11, 24
			Upper half	Long, thin arm	11, 13	219, 15
anchor1	lightorange	none	On upper anchor pad	Small anchor	4, 17	7, 20
poly1	green	solid	Upper half	Long, thick arm	51, 13	219, 27
			Extreme right	Vertical arm connection	214, 9	219, 27
dimple	red	none	On thick arm	Left dimple	74, 19	77, 22
			On thick arm	Middle dimple	127, 19	130, 22
			On thick arm	Right dimple	180, 19	183, 22

Fig. 3.84 Heatuator object summary

Along the top icon row, click on the arrow next to *New File* and select *New Cell*. To build the lower anchor pad in poly1 select *poly1* and then select the *Rectangle* icon. Place coordinates *0, 0* and *11, 11*. To build in poly1 the lower long arm place coordinates *11, 9* and *219, 11*. To build the lower anchor select anchor1 and then place the coordinates *4, 4* and *7, 7*. Right click to end the Rectangle command. To create a mirrored copy of the three objects, click on one of the three components and then press and hold Shift while you select the other two objects. From the lower icon row, next to the *Mirror* icon, click on the arrow. Select *Mirror Up*. From the lower icon row, to the far right, select the *Nudge* icon. Click twice in the top middle section of the canvas.

Fig. 3.85 Screen Illustrations

To build the long thick arm on the poly1 layer select *poly1 a*nd then select the *Rectangle* icon. Place coordinates *51, 13* and *219, 17*. To build the vertical connect place the coordinates *214, 9* and *219, 27*. To build the dimple, who acts as a standoff for the sliding actuator arm, select the *dimple* layer and then place the coordinates *74, 19* and *77, 22*. Right click to end the Rectangle command. To copy and paste the remaining dimples click on the dimple to select it and then from the upper horizontal icon row select *Copy*. Click on the lower left corner of the dimple as the FROM point. If this is difficult, click when the coordinate readout is at *74, 19*. From the upper horizontal icon row, select *Paste*. Place two copies of the dimple at coordinates *127, 19* and at *180, 19*. Right click to end the Paste mode.

Save the cell as *heatuator*.

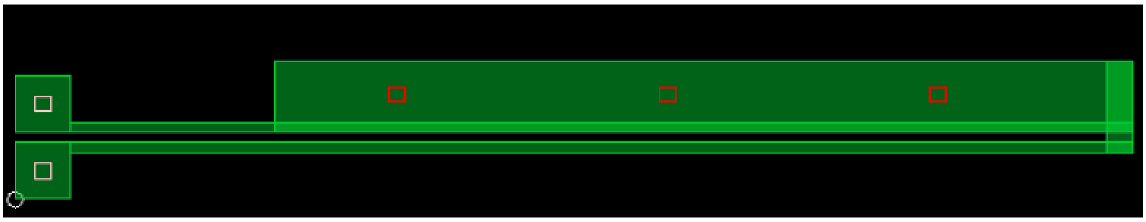


Fig. 3.86 The heatuator

The next step is to complete the heatuator assembly by adding power connections to the anchor pads and a linkage arm to the assembly.

Start a new cell by selecting *New Cell* from the upper icon row. Add heatuator as a cell reference at origin 0, 0 by selecting the *Reference* icon and then in the dialog window that opens, in the Cell field, select *heatuator* and in the origin field enter *0, 0*. Add connector as a cell reference at origin -39, 0, by creating another cell reference (*connector*) and placing it as a cell reference at origin *-39, 0*. Add contact_pad as a cell reference at origin -9, 22, by creating another cell reference (*contact_pad*) and placing it as a cell reference at the origin *-9, 22*. Draw the linkage mechanism in poly1 as a path of width 2 by selecting first *poly1* and from the menu bar *Object→Path* and the dialog window that opens set a width of *2*. From the vertical icon column select *Path* and start it at coordinate *217, 26* and move right, then click at coordinate *224, 26* and move down, then click at coordinate *224, -8* and move right, then click at coordinate *230, -8* and move down and then click at coordinate *230, -55*. Right click to end. Save the cell as *heatuator_assy*.

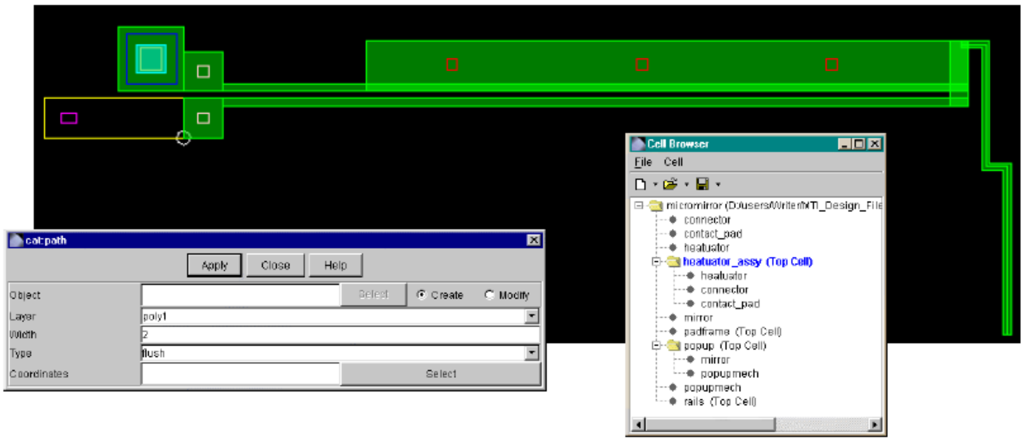


Fig. 3.87 The heatuator assembly

The next step is to create an array of ten heatuator assemblies and connect the array to a set of power rails.

Create a new cell by selecting *New Cell* from the top icon row. From the vertical icon column click on the *Reference* icon and in the Cell field enter *heatuator_assy*. Set the Origin to *0, 0*. Place an Array Reference of heatuator_assy cells (see figure 3.88) by selecting *Array Reference* and then in the Cell field click on *Browse…* button and select *heatuator_assy*. Set the Array for a 40 unit Row distance (-46 unit Row spacing) by setting the *Row distance* field to 40. Set the Columns to *1* and Rows to *5*. Set Origin to *-1, 0*. Create a cell reference using the *rails* cell. Align anchor2 polygons with an origin of *-39, -10* (see figure 3.88). Mirror Right to create a ten-cell array with full rail connections by clicking on the arrow next to the *Select objects* icon and then choosing *Select rectangular area* and then draw a box around the entire assembly to select it and from the horizontal icon menu select *Mirror Right*. Shift the mirrored portion two units to the left so the common poly is lined up. Save the cell as *heatuator_array*.

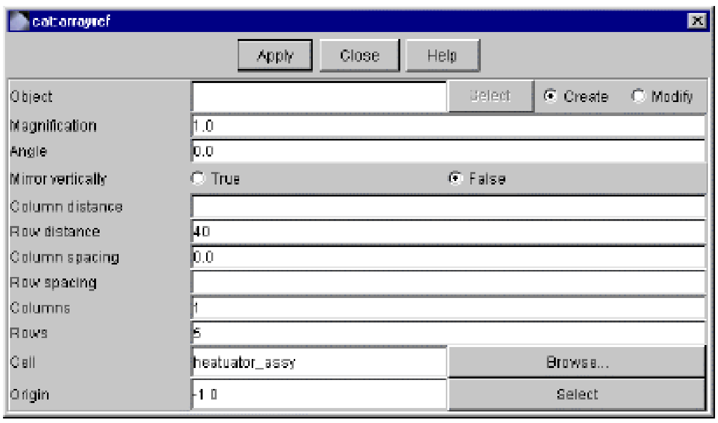


Fig. 3.88 Array Reference dialog window

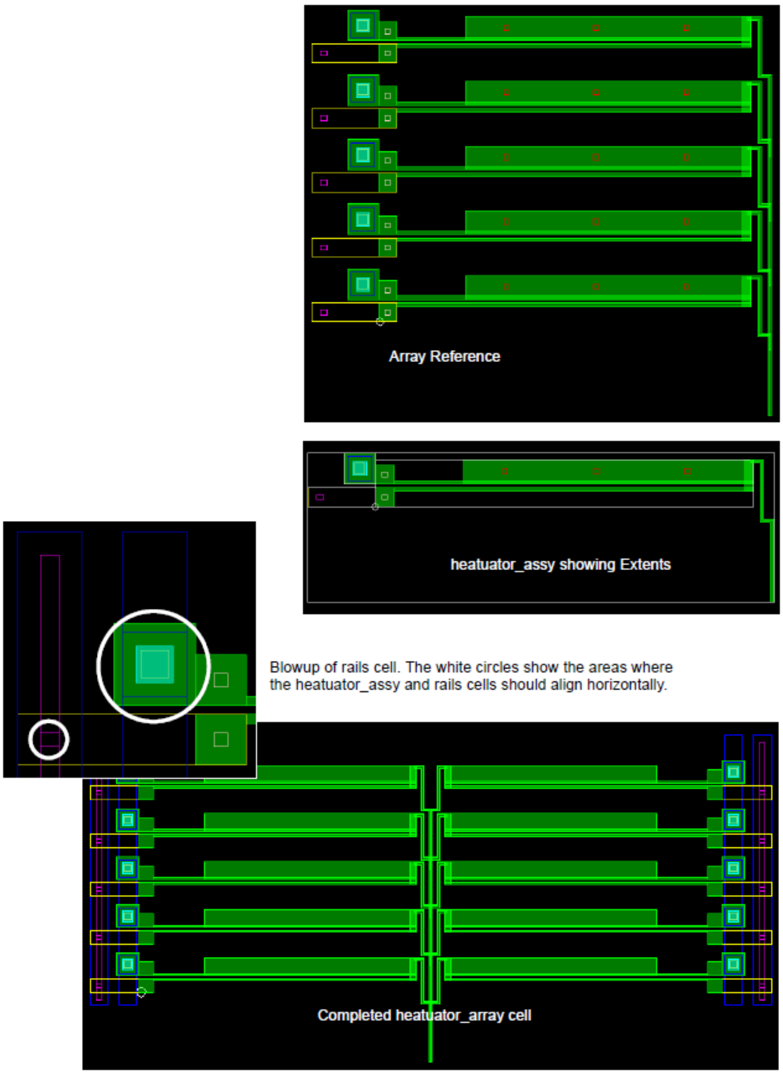


Fig. 3.89 Heatuator_array cell

The next step is to place the completed heatuator_array and popup cells and connect them with poly2.

Create a new cell by selecting *New Cell* from the top icon row. Create a cell reference using the heatuator_array cell by selecting the *heatuator_array* cell in the *Reference* window and setting the Angle at *180* degrees and the Origin to *229, -30*. Create a cell reference using the popup cell by selecting the *popup cell* in the *Reference* window and setting the Origin to *0, 120*. Connect the *poly1* linkage arms of the heatuator array and the long center connector of the popup mirror hinge assembly with a new poly1 rectangle. For exact placement, the coordinates are *-7, 0* and *7, 48*. Save the cell as *mirror_array*. Also save *micromirror.cat*.

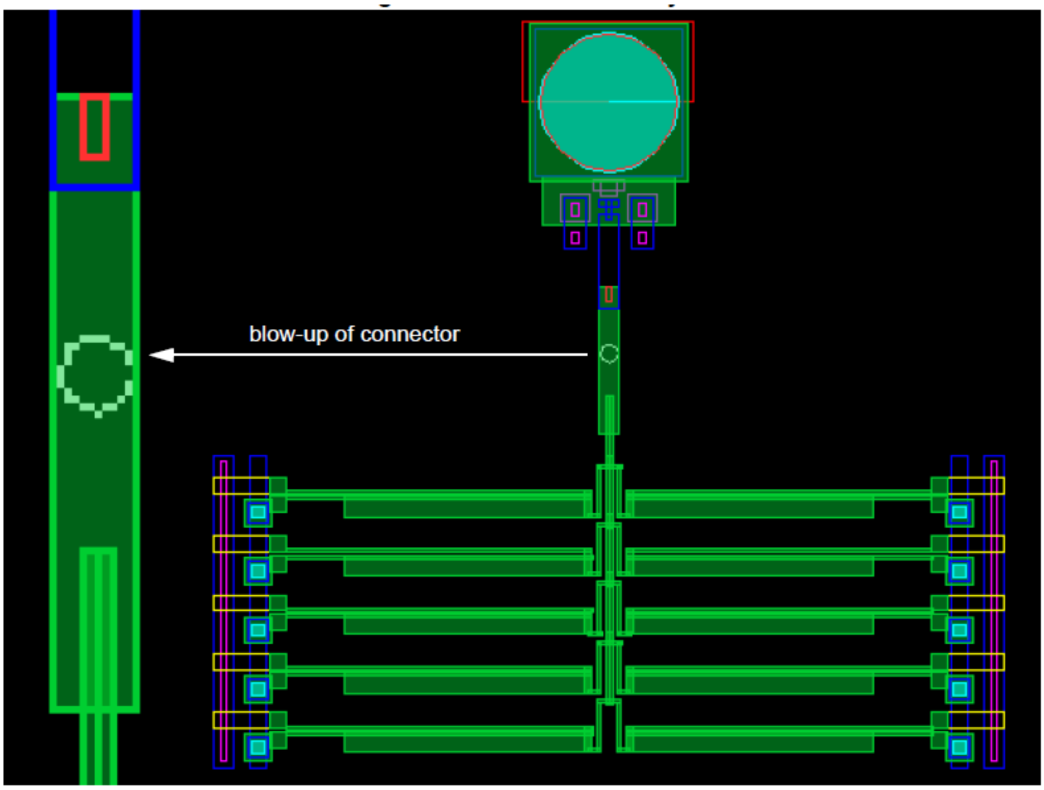


Fig. 3.90 Mirror Array

For finalizing the project place the completed mirror_array cell, create a pad frame of bonding pads around the periphery of the layout and connect the bond pads to the power rails of the heatuator assembly.

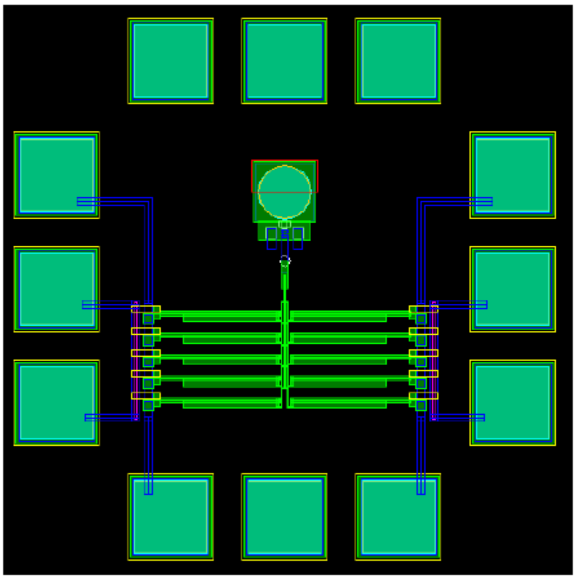


Fig. 3.91 Mirror Array with Padframe

Create a new cell by selecting *New Cell* from the top icon row. Create a cell reference using the padframe cell by selecting *padframe* in the *Reference* window and setting the Origin to *0, 0*. Create a cell reference using the mirror_array cell by selecting *mirror_array* in the *Reference* window and setting the Origin to *0, -50*. Draw connector wires 14 units thick in poly2 from the power rails to 8 of the pad cells by connecting the poly2, after selecting it, of the two right side rails to the poly2 of the pads and using a path object of width *14*. Connect rail tops and bottoms to different pads for optimal power distribution. Save the cell as *scan_assy*.

To build the 3-D Model on the *Layout* tab of the Function Manager select *scan_assy* as the Top Cell and then click on the *Solid Model* tab of the Function Manager. Click then on the Build Model button.

To view the 3-D Model select *Edit→Edit* Model from the Preprocessor menu bar. Then increase the *Z Scale* field value to see the layer depth in more detail. Select *Part Display* in the *Edit Model* window and remove some layers by highlighting layers in the left column and clicking on the *Remove* arrow icon to see only the active layers. When finished, from the Preprocessor menu bar, select *File→Quit*.

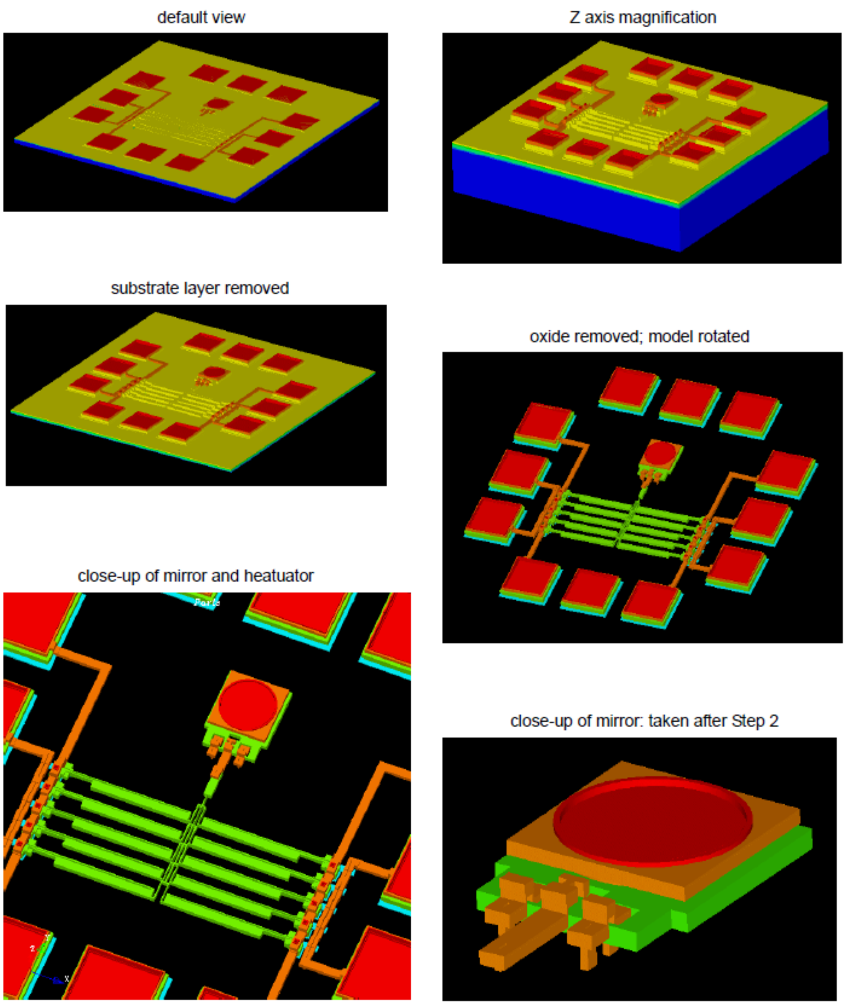


Fig. 3.92 Thermally-actuated pop-up mirror views

3.5. REFERENCES

[1] “Section 3: Tutorial”, MEMS-II Course, Department of Mechanical Engineering, University of Colorado Boulder, Boulder, Colorado, United States of America

[2] “Lecture #5 – CAD Tools for MEMS”, Department of Instrumentation & Control Engineering, Manipal Institute of Technology, Manipal, India

[3] “CAD Applications to MEMS Technology”, College of Engineering, The University of Utah, Salt Lake City, Utah, United States of America

[4] “Microsystems Design”, MEMS and Microsystems Design and Manufacture, Department of Mechanical Engineering, San José State University, San José, California, United States of America

[5] “Section 2: Beam Design”, National Chio Tung University, Hsinchu City, Taiwan