

Table of Contents

Table of Contents	1
MEMS Pro Tutorial	5
Introduction	6
Creating a Schematic	7
Setting up the Probing Information	16
Probing a Waveform	17
Viewing a Waveform	18
Calculating Resonance Frequency	20
Generating a Layout	21
Viewing a 3D Model	30
Drawing Tools	36
Optical Library Tutorial	43
Introduction	44
Theory	45
Tutorial Procedures	47
Schematic Editing	48
Placing Components	49
Connecting Elements	54
Writing Simulation Commands	57
Running Simulation and Viewing Results	59
Statistical Analysis Tutorial	61
Introduction	62
Process Information	63
Resonator Example	64
AC Analysis	66
Start Simulation	68
Design Rule Checker	69
Introduction	70
Design Rule Checker Commands	71

DRC Setup	72
Design Rule Types	73
Ignore Options	81
DRC Tutorial	82
Setting Design Rules	83
MEMS Tutorial - DRC	85
Running DRC	87
Re-running DRC	94
Optimization Tutorial	95
Introduction	96
Tanner Setting up the Optimization	97
Examining the Output	105
Verification Tutorial	107
Introduction	108
Adding Connection Ports	109
Extracting Layout	111
Extracting Schematic for LVS	114
Comparing Netlists	116
External Model Generator	119
Introduction	120
Creating the Model	121
Generating the Model	124
Instantiating the Model in a schematic	125
Instantiating the Model in the circuit	127
Simulating the circuit	128
MEMS Modeler Tutorial	129
Introduction	130
Tunable Filter	131
Generating 3D Model	133
ANSYS Setup	136
Loading 3D Model	138
Loading Macro	140
MemsModeler Setup	141
FEM Model	145
Electrical Data	146

DOF Sweeping	147
Output	148
Running MemModeler	149
Saving Model	150
Simulation With Reduced Macro Model	151
Creating Filter Model	152
Creating Symbol	153
Creating Top-Level Model	154
Creating Driving Signal	155
Grounding Voltage Source	157
Setting Voltage Source Parameters	158
Wiring Schematics	159
Setup Probing Parameters	160
Generating Spice Netlist	161
Setting Analysis Type	162
Starting Simulations	164
Boundary Conditions Tutorial	167
Introduction	168
Applying Boundary Conditions	169
MultiPhysics Problem	170
Opening Layout	171
Boundary Conditions Setup	172
Additional BC Settings	173
Set Boundary Conditions	174
Create Voltage BC Tag	175
Create Temperature BC Tag	177
Create Structural BC Tag	179
Create BC Tags for the Second Anchor	180
Setting Material Database	182
Using Correct Units in MEMS Simulation	184
Create 3D Model	186
View Boundary Conditions	187
Prepare 3D Model	188
Export 3D Model	189
Start ANSYS	190
Load 3D Model	191
Set Element Type	192
Meshing	193
Check Boundary Conditions	195
Obtain Solution	196

Plot Temperature Results	197
Plot Transformation	198
Plot Voltage Results	199
Plot Displacement Results	200
Plot Stress Results	201
Animate Displacement Results	202
INDEX	203

1 MEMS Pro Tutorial

▪ Introduction	6
▪ Creating a Schematic	7
▪ Setting up the Probing Information	16
▪ Probing a Waveform	17
▪ Calculating Resonance Frequency	20
▪ Generating a Layout	21
▪ Viewing a 3D Model	30
▪ Drawing Tools	36

Introduction

In the MEMS Pro tutorial, you will follow the complete design of a MEMS resonator. The advanced portion of this tutorial includes a demonstration of layout extraction and netlist comparison using L-Edit/Extract and LVS as well as the [Optimization Tutorial on page 95](#). Those tutorials assume that you are completely familiar with the material covered in this general tutorial.

In this tutorial, you will create a schematic design, analyze system behavior, and generate device layout with the MEMS Pro tools S-Edit, T-Spice, W-Edit, and L-Edit. You will draw mask layout manually and automatically, and generate and view 3D models and cross-sections in L-Edit.

All files mentioned in this chapter are located under ***{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\Resonator***. You are recommended to follow the tutorial from the beginning; however, you may enter and exit the tutorial at several points during the design. Tutorial breakpoints occur at [Generating a Layout on page 21](#), [Viewing a 3D Model on page 30](#), and at [Drawing Tools on page 36](#).

The Design Example

The design example, appearing throughout the MEMS Pro Tutorials, is an electrostatic lateral comb-drive resonator. A resonator is a MEMS transducer that can be used as a sensor by exploiting the high sensitivity of its resonant frequency to various physical parameters. The resonator was chosen for your review because it is an easily understood coupled electro-mechanical system. The resonator will be designed using the MEMSPro library components including comb-drives, a plate and folded springs.

Creating a Schematic

In this section, you will learn how to navigate and manipulate designs using the S-Edit schematic editor.

Launching S-Edit

- ☑ Launch S-Edit by clicking from the "Start" menu: **"Start >All Programs > MEMS Pro v8.0> MEMS S-Edit"**.

Note

Make sure that you run MEMS S-Edit as administrator by right-clicking on **MEMS S-Edit > Run as administrator**.

- ☑ The S-Edit user interface consists of nine areas:
 - The *Title Bar* (at the very top of the application window) shows the name of the current cell and the view type (symbol, schematic, etc).
 - The *Menu Bar* (at the top) contains the MEMS S-Edit menu titles. The menu displayed may vary depending on the view type that is active.
 - The *Status Bar* (at the bottom) contains runtime information.
 - The *Design Area* (in the center) contains the schematic.
 - The *Standard Commands Toolbar* (below the menu bar) contains often used commands.
 - The *Draw Toolbar*, *Segment Toolbar* and *Electrical Toolbar* (next to the Standard Toolbar) contain tool icons.
 - The *SPICE Simulation Toolbar* (below the Standard Toolbar) contains simulation and probing tools.
 - The *Libraries window* (at the left) contains the list of libraries and their components.
 - The *Properties window* (at the right) contains the properties of the component selected in the schematic window.

Designs are contained in *Libraries*, each of which consists of one or more *cells*. Cells are viewed in either of two modes:

- **Symbol Mode:** a graphical representation of the module showing the cell's connections.
- **Schematic Mode:** shows the composition and connectivity of the cell.

The schematic may contain one or more pages which consist largely of two component types:

- *Primitives:* geometrical objects, wires, ports, annotation objects, and labels; all created with the S-Edit drawing tools.
- *Instance:* ì copiesî of other cells, dynamically linked to their originals. Instances are displayed in a design using the cell's symbol.

Opening the File

As part of this section of the tutorial, you will place the components of an electrostatic lateral comb-drive resonator, connect those components, and run an AC analysis on your design.

The schematic symbols for these components have two pins for each connecting side: one carrying the electrical signal (denoted with the prefix **V**, as in **VAnchor**), and the other carrying the mechanical signal (with no prefix, as in **Anchor**). Other terminals are named after their location on the component: a suffix is appended to their name. For instance, **Disp_b** is the displacement pin located at the bottom (**_b**) of the plate. The other suffixes are **_t** for top, **_l** for left and **_r** for right.

These symbols are assembled to form the resonator design and a frequency sweep (AC analysis) of the system is performed to discover the resonant frequency and the magnitude of displacement. The electro-mechanical behavior of the components is modeled by expressing the mechanical behavior in terms of electrical analogs. These models can then be used to solve for the electrical and mechanical behavior of the system as well as the coupling between the two energy domains.

The complete design is provided for your reference in the **reson.tanner** file under **{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\Resonator\reson**.

- Select **File > Open > Open Design** to open this file.

The current (visible) file, cell, page and mode are named at the top of the title bar. The schematic view of the resonator appears in [Figure 1](#).

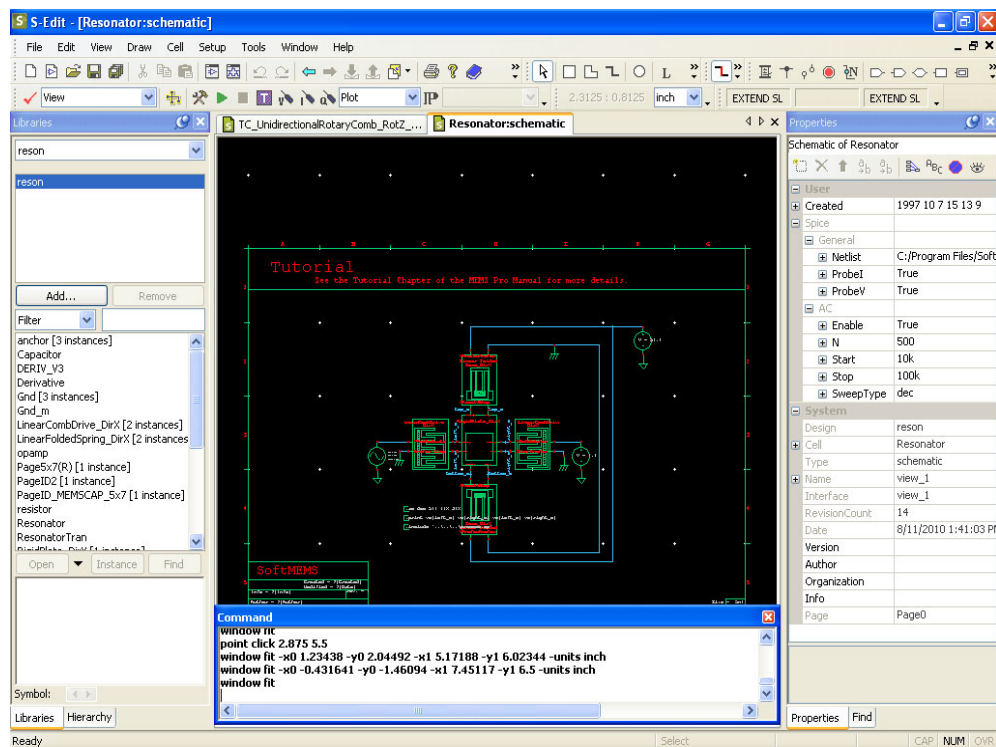


Figure 1: Schematic view of the complete resonator

Creating a New Schematic View

To initiate your new resonator design, you must first create a new schematic view (new cell).

- Select **Cell > New View** to create a new schematic view.

- ☑ In the **Cell** edit field, enter **MyResonator**, you must be sure that the **Design** edit field is kept **reson** and the **view type** field is **schematic**. Finally, click **OK**.
- ☑ Select **File > Save > Save Design "reson" and Its Libraries**.

You can compare your work to the reference design at any time by using the **Cell > Open View** command and choosing **Resonator** as the cell name to open. Use **Cell > Open View** again to return to your design, this time selecting **MyResonator** as the cell to be opened.

Instantiating Components

Instantiating a Plate

- ☑ Select **Cell > Instance** and the **Instance Cell** dialog opens as shown in [Figure 2](#).
- ☑ Select **reson** in the **Design** and **RigidPlate_DirX** in the **Cell name** field.
- ☑ Move your cursor to the schematic page. You will find that the **RigidPlate_DirX** symbol is on the tip of the cursor. Place the **RigidPlate_DirX** by clicking on the schematic page.
- ☑ Hit **<esc>** to end the operation. If you find that you have instanced too many plates you can always delete them by selecting them with the right mouse button and click on the **Delete** button which exists on the keyboard. (When an object is selected in S-Edit it turns a different color for example, it turns yellow).
- ☑ Home the view by selecting **View > Fit** or by pressing the **Home** key. The view of the plate will be resized so that the plate fills the contents of the window.

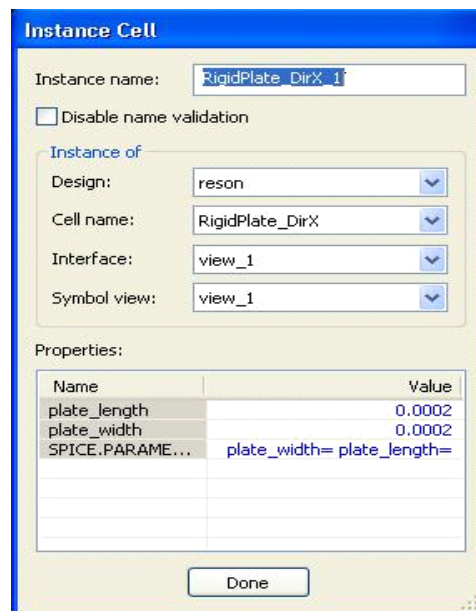


Figure 2: Instance Cell dialog

Instantiating Comb-drives

- ☑ If the **Instance Cell** dialog is still open, select the **LinearCombDrive_DirX** in the **Cell name** field. If the **Instance Cell** dialog is closed, select **Cell > Instance** to open it.
- ☑ Place two **LinearCombDrive_DirX** cells on the schematic page; one to the right side and the other to the left side of the previously instantiated plate. You can use the arrow keys to pan while placing instances. You keep placing CombDrive cells as long as you don't hit the **<esc>** key. So, you should hit the **<esc>** key to stop placing cells.
- ☑ Select the left **LinearCombDrive_DirX** cell and then flip it by choosing **Draw > Flip > Flip Horizontal**.
- ☑ Move the comb-drives so that their connection pins, represented by circles, line up with the pins on the plate (**Figure 3**).

Note Objects in S-Edit can be moved by pressing the **Alt** key and left-clicking to drag them. or by using the middle mouse button on a 3 button mouse.

Note Make sure that the **Vactuate** pins of the comb face the **Vactuate** pins of the plate.

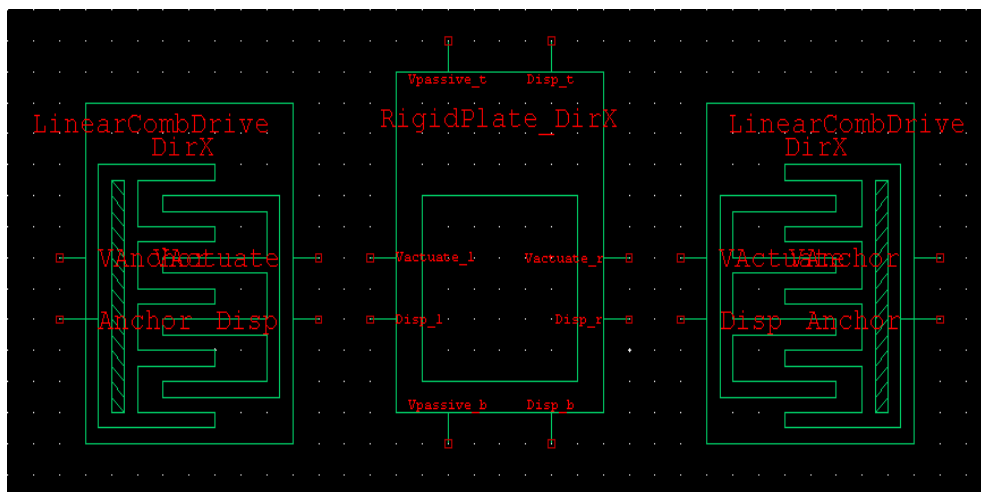


Figure 3: Aligning the comb-drives to the plate


Instantiating Folded Springs

- ☑ If the **Instance Cell** dialog is still open, select the **LinearFoldedSpring_DirX** in the **Cell name** field. If the **Instance Cell** dialog is closed, open it with **Cell > Instance**.
- ☑ Place two **LinearFoldedSpring_DirX** cells on the schematic page; one above and the other below of the instantiated plate. You may need to pan using the up and down arrow keys to make room.
- ☑ Select the **LinearFoldedSpring_DirX** cell below the plate and then flip it vertically by choosing **Draw > Flip > Flip Vertical**.

Note

Make sure that the **VPassive** pins of the spring face the **Vpassive** pins of the plate.

Wiring Objects

Wires are drawn using the **Wire** tool .

At first, you may confuse the **Wire** tool with the **Line** tool. Lines are used to graphically represent components; they are non-electrical objects used to annotate your schematic. Wires are electrical and are used to connect objects.

Zooming the View

Sensitive operations such as wiring nodes require a closer view for accuracy.

- ☑ Select **View > Zoom > Mouse** or hit the "z" key.
- ☑ Drag a box around the plate with the left mouse button. Allow enough room to see the areas between the comb-drives and the folded springs.

If you find you have zoomed in too much or too little, use the plus and minus keys to **Zoom** in and out. The arrow keys can be used to pan the view.

You will now create connections between the plate and the other schematic components with wires.

- ☑ Select the **Wire** tool from the schematic toolbar.
- ☑ Initiate the wire placement by left-clicking on the plate at the **Disp_b** pin. The pin is shown as an open circle on the bottom left terminal of the plate.
- ☑ Move the cursor down and end the wire placement by right-clicking at the pin called **Disp** on the bottom spring. This pin is shown as an open circle on the top left terminal of the bottom spring.
- ☑ Repeat this process to wire the plate with the other components (see [Figure 4](#)).
- ☑ Home the view by pressing the **Home** key.

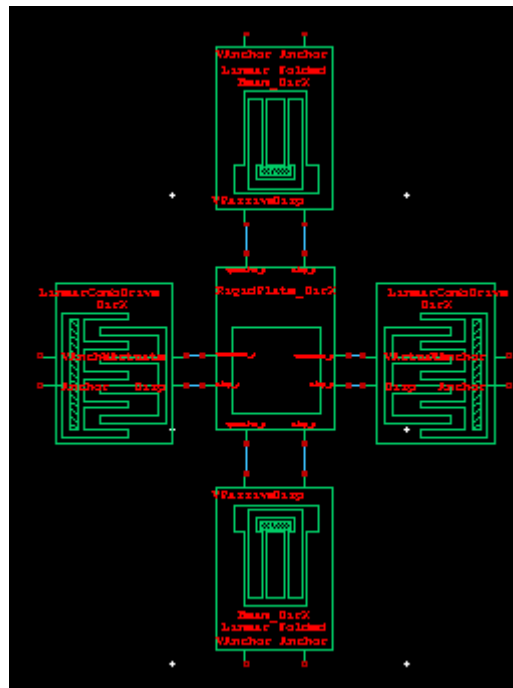



Figure 4: Schematic view of the wired elements

Instantiating Voltage Sources


- Select the selection tool  to exit wire drawing.
- Instantiate the **Source_v_ac** cell.
- Place it to the left of the left comb.
- Instantiate the **Source_v_dc** cell.
- Place it to the right of the right comb.
- Copy the instance of **Source_v_dc** by selecting it and clicking **Edit > Copy**. Then **Edit > Paste** and place it to the right of the top spring.
- Wire the upper terminal of the AC source to the **VAnchor** pin of the left comb and the positive terminals of the DC sources to the **VAnchor** pin of the right comb, and top spring. The positive terminals, in this example, are located at the top of the voltage sources, in this example.
- Compare your design to the complete design in [Figure 1](#) to make sure you have placed the voltage sources correctly or switch to the Resonator cell to compare as described previously.

Instantiating Gnd & Anchor

- In the **Libraries** window, select **Gnd** in the list of cells.
- Click on the **Instance** pushbutton that is present below the list of cells. This invokes the **Instance Cell** dialog.

- ☑ Be sure that the **Gnd** is in the **Cell name** field.
 - ☑ Move your cursor to the schematic page. You will find that the **Gnd** symbol is on the tip of the cursor. Place the three **Gnd** cells on the schematic page by clicking on the schematic page.
 - ☑ Move the three ground symbols to a place in the schematic near each voltage source.
 - ☑ Then, wire the negative (lower) terminal of each of the three voltage sources to a ground symbol.
- You have placed the three ground symbols. Now, you have to place the three anchors.
- ☑ In the **Libraries** window, select **anchor** in the list of cells.
 - ☑ Click on the **Instance** pushbutton that is present below the list of cells. This invokes the **Instance Cell** dialog.
 - ☑ Be sure that the **anchor** is in the **Cell name** field.
 - ☑ Move your cursor to the schematic page. You will find that the **anchor** symbol is on the tip of the cursor. Place the three **anchor** cells on the schematic page by clicking on the schematic page.
 - ☑ Move two anchors to a place on the schematic near each comb-drive. And, move the third anchor near the top spring.
 - ☑ Then, wire the **Anchor** pins of the top spring with one **anchor** symbol.
 - ☑ Wire the **Anchor** pins of the two comb-drives with an **anchor** symbol.
 - ☑ At this point, the three anchor symbols should be connected to the **Anchor** pins of the top spring and the two comb-drives.
 - ☑ Pins **VAnchor** (fixed electrical) and **Anchor** (fixed mechanical) of the bottom spring should be the only pins left unconnected at this point. Connect them to the **VAnchor** and **Anchor** pins, respectively, of the top spring.
 - ☑ Compare your design to the complete design presented in [Figure 1](#) to make sure the resonator has been wired correctly.

Editing Object Properties

Make sure that you are using the Selection Tool .

Now, you will edit the properties of one of the voltage sources in the schematic to set up the design for simulation.

- ☑ Left click on the voltage source next to the left comb to select it. You will have the properties of the instance selected at the right side of the schematic window.

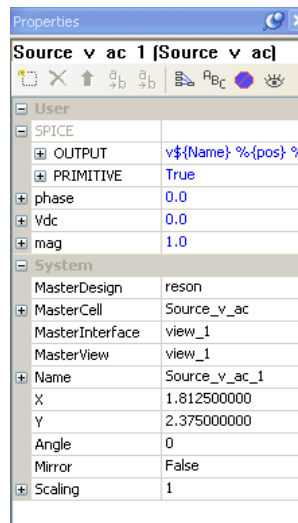



Figure 5: Properties window of Cell Source_v_ac

- Enter **1** for **mag**, **0** for **phase** and **0** for **Vdc**.
- Give the voltage source for the right comb-drive a DC value of 0 volts. Do this by editing on the properties window on the right side with this source selected. Enter **0** in the **V** field.
- Similarly, give the voltage source for the folded springs a DC value of 50 Volts.

Labeling Nodes

In S-Edit, connectivity is defined in terms of *nodes*. A node is a point on the schematic to which one or more pins or wires are connected. Nodes are defined by their names, and the *scope* of a node is normally the collection of schematic pages in a module. That is, if a node-name appears twice within a single module, both names refer to the same point of connection. If the same node-name appears within two different modules, the nodes refer to completely different points of connection. S-Edit automatically assigns a name to each node, but you may also name nodes manually. User-assigned node labels are helpful for annotating S-Edit schematics and producing more readable netlists.

- Select the **Net Label** tool  from the schematic toolbar.
- A symbol called **NetLabel** will be attached to your cursor. Click on the center of the wire between the *Disp_r* pin of the plate and the *Disp* pin on the right comb. The **Net Label Settings** dialog appears. Type *right_m* in the **Name:** field and hit **OK**. Similarly, place the node label *right_e* between the *VActuate* and *Vactuate_r* pins.
- To change the placement of the node label; select the node label. From the **Properties** window, select the **Center** entry of **Horizontal** and **Middle** entry of **Vertical Text Justification**.
- Edit the node labels orientations to look some what like the layout in [Figure 6](#).

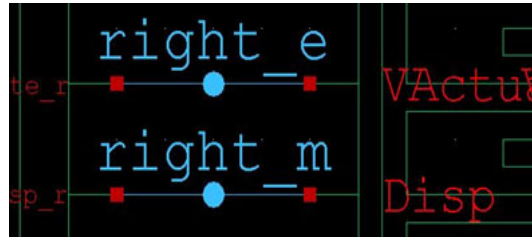


Figure 6: right_m and right_e nodes

- ☑ You may rename the rest of the nodes in your diagram to match the names used in the **Resonator** module of the *reson.tanner* file if you wish. This is an optional step. S-Edit will automatically assign names to unlabeled nodes.

Setting up the Probing Information

You will now specify the probing information for the resonator.

- ☑ Select **Setup > SPICE Simulation**.

The **Setup SPICE Simulation of cell "MyResonator"** dialog box appears, as shown in [Figure 7](#).

- ☑ Select the **General** field in the left side of the window.
- ☑ In the **Simulation Outputs** on the right side of the window, set the **Enable Waveform Voltage Probing** to **true**.
- ☑ In the **Simulation Outputs** on the right side of the window, set the **Enable Waveform Current Probing** to **true**.
- ☑ In the **SPICE File Name** on the right side of the window, type in the following: **{MEMS Pro v8.0 Installation Directory}/MEMS_Tutorials/Resonator/reson/MyResonator**.

Your netlist will be saved in this directory under the name "MyResonator".

- ☑ Change the **Show Waveforms** drop down to **Don't show**.
- ☑ In the **Include Files** field, click the **Browse** button, change the file type into **Spice Files (*.sp, *.spc, *.cir)**, select the **process.sp** file (located under the **MEMS Pro v8.0 Installation Directory**) and click **"Open"**.
- ☑ Check the **AC Analysis** check box and click on it to open the **AC Analysis**.
- ☑ Select **dec** as the **Sweep Type**.
- ☑ Set **Number of Frequencies** to **500**, **Start Frequency** to **10k** and **Stop Frequency** to **100k**.

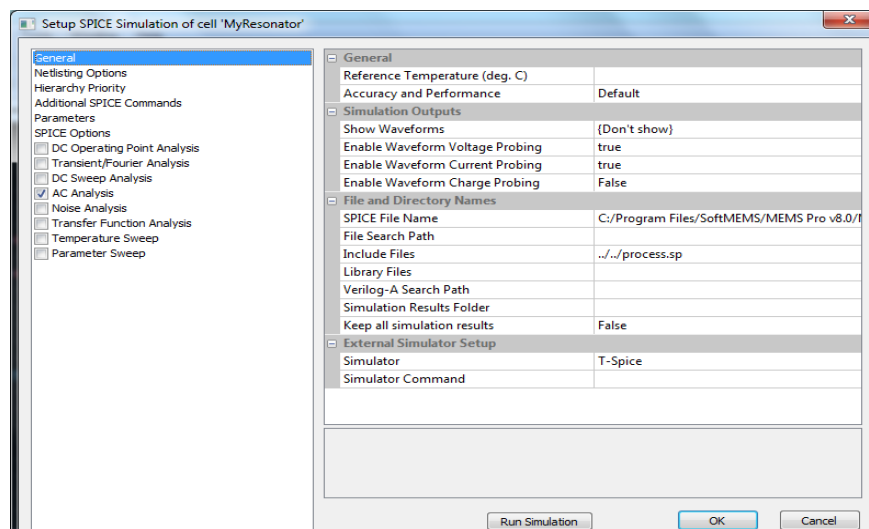


Figure 7: Waveform Probing Setup dialog box



- ☑ Click on **Run Simulation** to run the circuit.

Probing a Waveform

The waveform probe is used to browse through an S-Edit design and probe nodes to examine circuit simulation results for the specified node. When a node is probed, S-Edit invokes W-Edit, which automatically displays the waveform corresponding to the simulation results for that node. W-Edit can also be launched from T-Spice by selecting **View > Show Waveform Viewer** or by clicking the **W-Edit** button from the **T-Spice** toolbar. In this tutorial, you will invoke W-Edit from S-Edit using the probing feature.

Warning

Make sure that you have already set up the probing information using **Setup > SPICE Simulation** in S-Edit.

- ☑ Click anywhere in the S-Edit schematic window that contains the resonator schematic to re-activate S-Edit.
- ☑ Depending on which of the probes are activated in the **SPICE Simulation** window, click on the appropriate probe from the **Probe** tool  located on the **Schematic** toolbar. The appropriate probe is voltage so, you will click on . The cursor now has the shape of the **Probe** tool.
- ☑ Left-click with the **Probe** tool on the **right_m** node which is a mechanical node but, due to mapping rules we mapped displacement into voltage and force into current, that is why we used voltage probe.

During waveform probing W-Edit is launched displaying the results of the T-Spice simulation graphically.

The **W-Edit** window should display a chart containing the magnitude and phase of the displacement of node **right_m** resulting from the performed AC analysis.

Viewing a Waveform

The **W-Edit** application window can contain many child windows, each containing one or more charts.


- ☑ Maximize the window containing your results by clicking the  button at the upper right corner of the window.

Chart Setups

W-Edit allows the expansion of charts with more than one trace per chart into separate charts, each containing a single trace. Collapsing the charts causes W-Edit to show all the visible traces in one chart.

There should now be two charts ([Figure 8](#)), one displaying the amplitude information of node **right_m**, (**vm(right_m)**), and the other displaying the phase angle, (**vp(right_m)**), both plotted versus frequency.

- ☑ Now you should be able to view a peak in the amplitude at around 17 kHz.

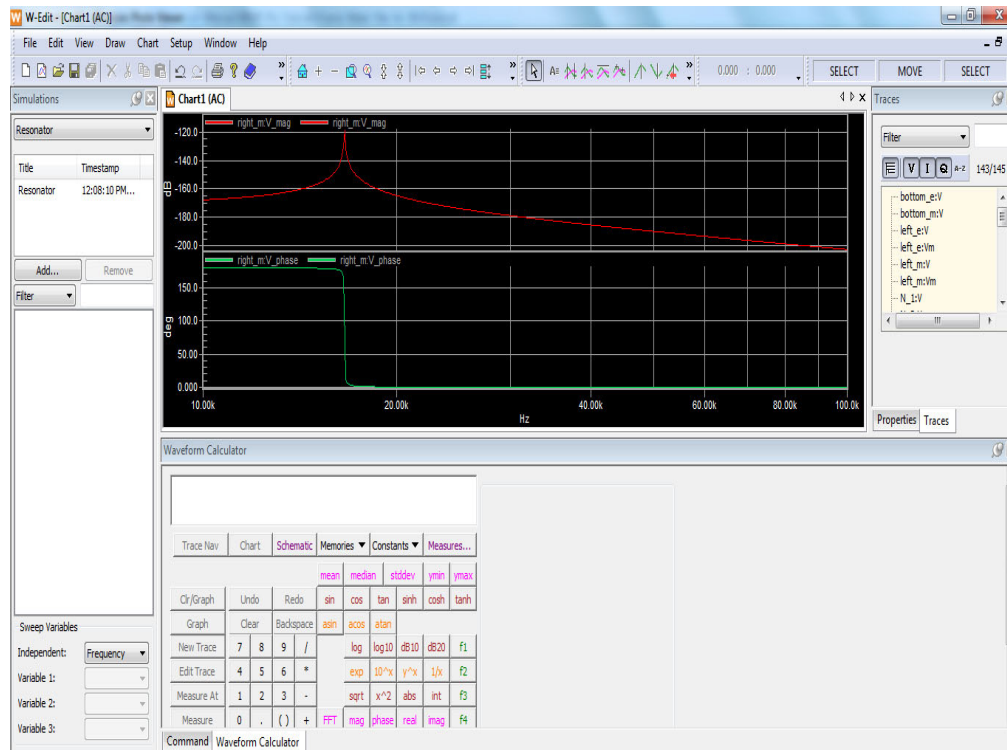


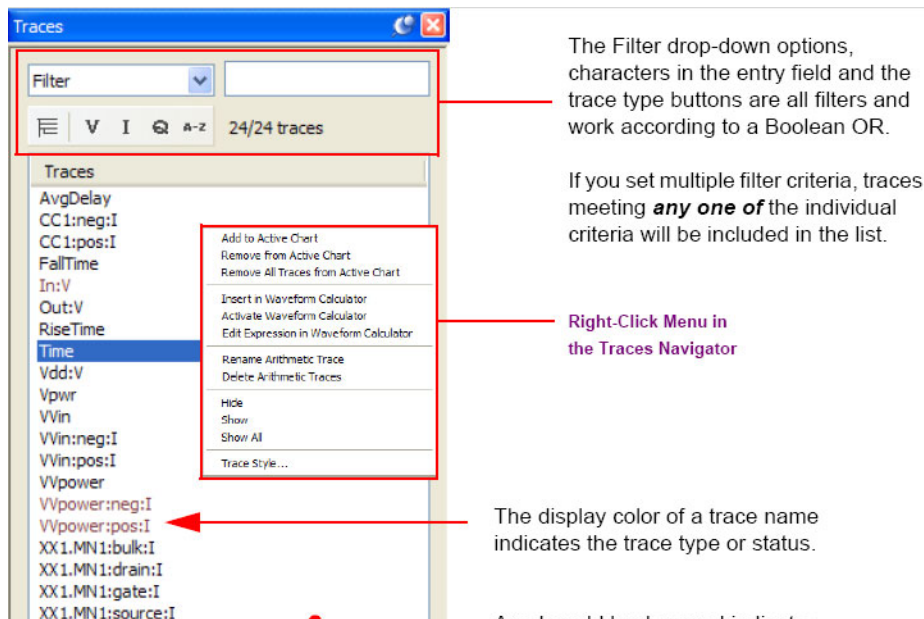
Figure 8: Charts representing the amplitude and the phase angle

Trace Manipulation

At times, you may find it necessary to hide traces in order to simplify a waveform window.

You will find the **Traces** window on the right side of W-Edit [Figure 9](#).

- ☑ In order to hide a trace, let's say the phase of **right_m**, select "**right_m:V_phase**" from the the **Traces** window, right-click on it and select **Remove from Active Chart**.



The Filter drop-down options, characters in the entry field and the trace type buttons are all filters and work according to a Boolean OR.

If you set multiple filter criteria, traces meeting **any one of** the individual criteria will be included in the list.

Right-Click Menu in the Traces Navigator

The display color of a trace name indicates the trace type or status.

Figure 9: Traces window

Calculating Resonance Frequency

In this section we will calculate the resonance frequency manually for our Resonator test circuit.

$$f_{res} = 1/2\pi*\sqrt{(K_{total}/\text{Mass of Rigid Plate})}. \quad (1.1)$$

$$K_{total} = 2*K_{spring}. \quad (1.2)$$

$$K_{spring} = E * \text{Thickness} * (\text{Width})^3 / (\text{Length})^3. \quad (1.3)$$

$$\text{Young's Modulus} = 150e9.$$

$$\text{Thickness} = T_{poly1} = 2e-6m.$$

$$\text{Width} = 3e-6m.$$

$$\text{Length} = 0.0002m.$$

Substitute by these values in Eqn. (1.3) to get Eqn. (1.4)

$$K_{spring} = 150e9 * 2e-6 * (3e-6)^3 / (0.0002)^3 = 1.0125 \text{ N/m}. \quad (1.4)$$

$$K_{total} = 2 * 1.0125 = 2.025 \text{ N/m}. \quad (1.5)$$

$$\text{Mass of Rigid Plate} = \rho * \text{Volume of Rigid Plate}. \quad (1.6)$$

$$\text{Volume of Rigid Plate} = \text{Length} * \text{Width} * \text{Thickness}. \quad (1.7)$$

$$\text{Length} = 0.0002 \text{ m}.$$

$$\text{Width} = 0.0002 \text{ m}.$$

$$\text{Thickness} = T_{poly1} = 2e-6 \text{ m}.$$

$$\rho = 2.33e3 \text{ kg/m}^3$$

Substitute by these values in Eqn. (1.6) & Eqn. (1.7) to get Eqn. (1.8)

$$\text{Mass of Rigid Plate} = 2.33e3 * 0.0002 * 0.0002 * 2e-6 = 186.4e-12 \text{ Kg}. \quad (1.8)$$

Substitute by the values of Eqn. (1.5) & Eqn. (1.8) in Eqn. (1.1) to get Eqn. (1.9).

$$f_{res} = 1/2\pi*\sqrt{(2.025/186.4e-12)} = 16.588 \text{ KHz}. \quad (1.9)$$

Generating a Layout

- ☑ You may now exit S-Edit, T-Spice and W-Edit.

Now you will learn how to create the layout of the lateral comb-drive resonator using the MEMS layout components - plate, comb-drives and springs.

Tutorial Breakpoint

You will now generate the layout of the resonator using the **MEMS Library Palette**.

Launching L-Edit

- ☑ Launch L-Edit by double-clicking the **MEMS L-Edit** icon  under **{MEMS Pro v8.0 Installation Directory}**. A default file named **Layout1** should be visible in the work area.

The L-Edit user interface is similar in appearance to the S-Edit user interface. In addition to the menu, palette and status bars, there is a *shortcut bar* that contains buttons for the most commonly-used menu commands.

As in S-Edit, L-Edit files are assembled hierarchically from discrete - usually functionally distinct - units called *cells*; these cells can be edited and instantiated.

The *current* file and cell are named at the top of the application window.

Opening the File

- ☑ Select **File > Open** to open the **reson.tdb** file located under the **MEMS_Tutorials\Resonator\Resonator** directory. The **Resonator** cell should appear as the active cell. Use this view of the **Resonator** cell, of the **reson.tdb** file, as a reference while working through this section.
- ☑ Select **File > New** and click **OK** in the open dialog box to create a new layout.

Creating Components

The mask layout for MEMS components can be created using the **Library Palette** accessed via the **Library** option of the MEMS Pro Toolbar.

The **MEMS Library Palette** contains active elements, passive elements, thermal elements, optical elements, test elements, resonator elements and fluidic elements. The resonator elements collection contains all the components you will need to create a resonator. All of these parts can be created manually, using the drawing primitives available in MEMS Pro, but that task would be tedious and time consuming.

The **MEMS Pro Toolbar**, which includes the **MEMS Library Palette**, should have been loaded during your installation setup.

- ☑ Check that the **MEMS Pro Toolbar** appears automatically in the L-Edit window.

If the **MEMS Pro Toolbar** is not automatically loaded when you launch L-Edit, you will need to load it manually, by selecting **Tools > Macro** from the L-Edit tool bar, then go to **{MEMS Pro v8.0 Installation Directory}** and select **MEMSPPhysical.dll**.

Using the MEMS Library Palette

- ☑ Select **Library > Library Palette** in the **MEMS Pro Toolbar** to invoke the **Library Palette** dialog box.

The **MEMS Library Palette** dialog box contains seven icons in the left side tree: **Active Elements**, **Passive Elements**, **Test Elements**, **Thermal Elements**, **Optical Elements**, **Fluidic Elements**, and **Resonator Elements**. You will use the **Resonator Elements** in this tutorial.

- ☑ Select **Resonator Elements** to display the resonator components.

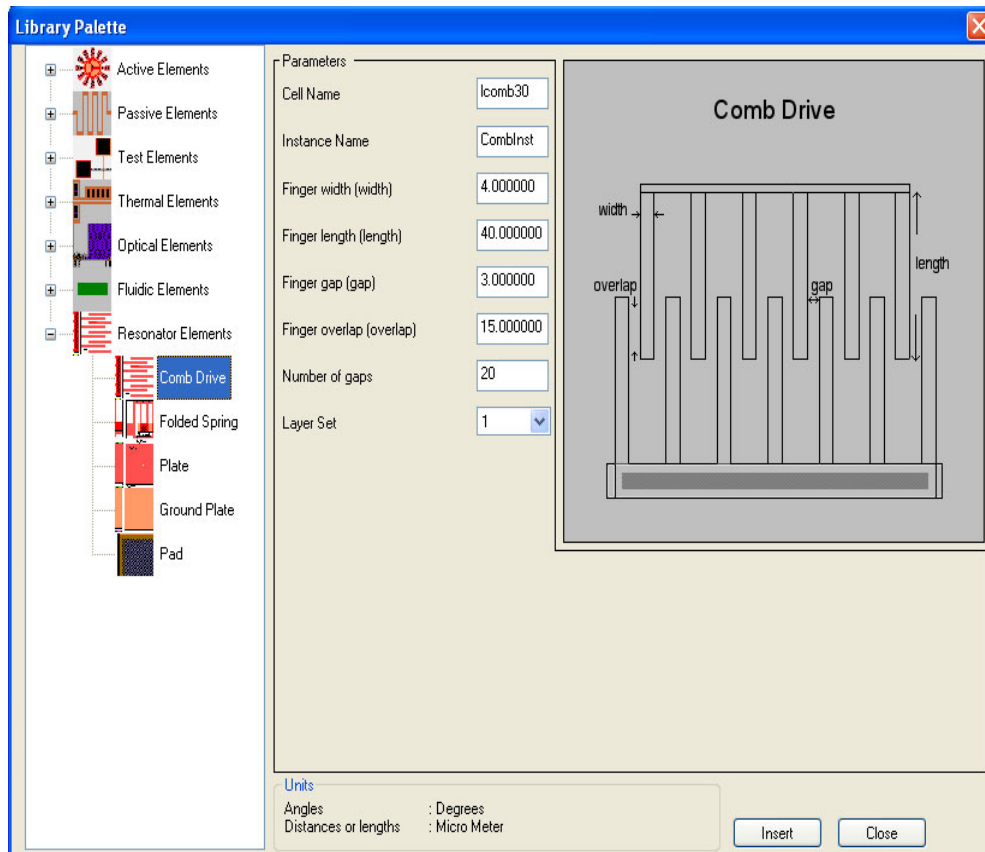



Figure 10: MEMS Library Palette

Generating the Plate

- ☑ Click the **Plate**  icon to invoke the plate generation macro.

The **Parameters** section will appear, where you will set the parameters of the plate.

- ☑ Enter **100** as the **Width** and click **Insert** accepting the default values for the other parameters.

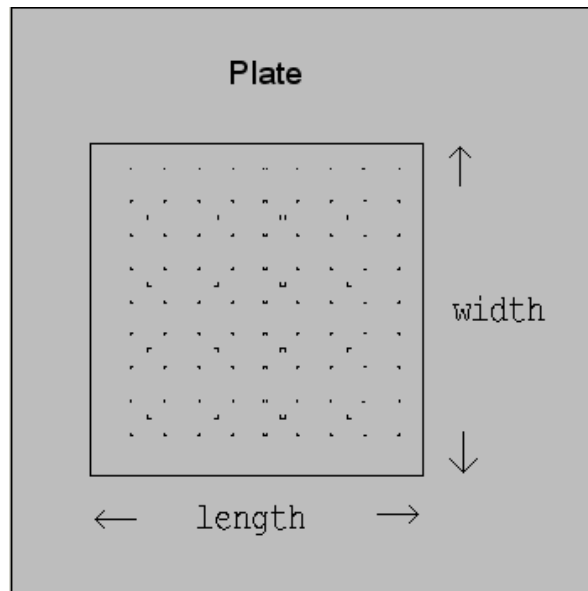



Figure 11: Plate

L-Edit/ UPI now creates a plate matching the input parameters. To see the entire plate, home the view by selecting **View > Home** or by pressing the **Home** key. The plate shown in the active window is an instance (**PlateInst**) of a newly created cell named **Plate10**.

Generating the Comb-drives

- ☑ Click the **comb-drive**  icon to create a lateral comb-drive.
- ☑ Change the **Instance Name** to **CombRight** and click **Insert**.
- ☑ Once a comb-drive appears on the screen, zoom out by pressing the minus key several times or by selecting **View > Zoom Out**.

Editing an Already Generated Component

You will now learn how to edit a component once you have created it. The component you will edit is the comb-drive you have just instantiated.

- ☑ Select the comb drive.
- ☑ Select **Library > Edit Component**.

The **Edit Component** dialog box appears.

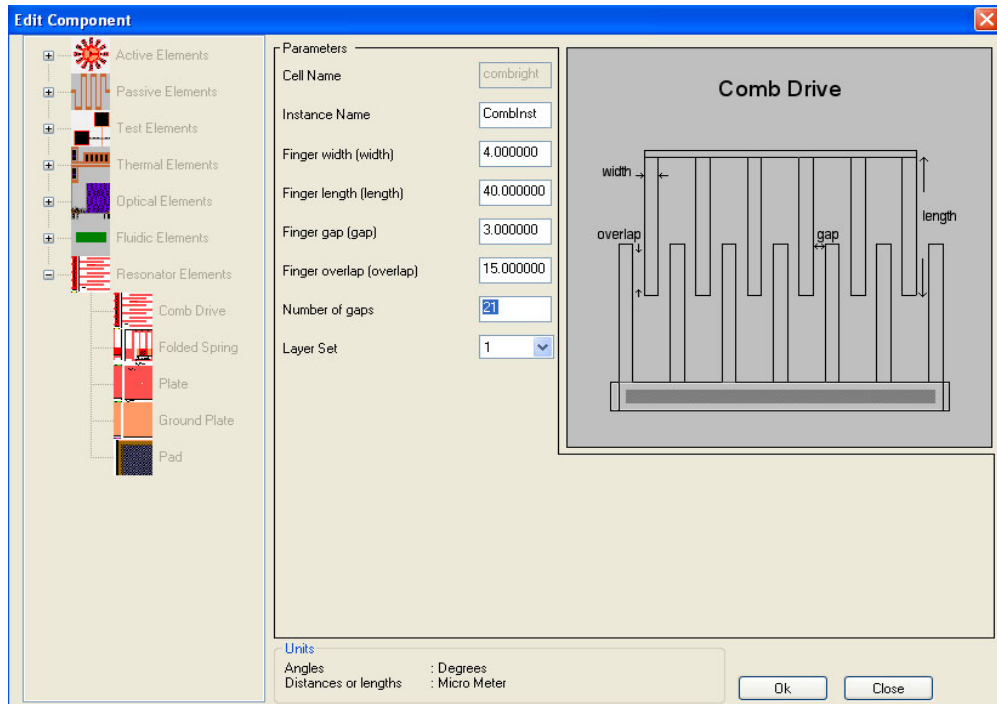


Figure 12: Edit Component dialog box

- Set the **Number of gaps** to **21** and click **OK**.

The modified comb drive appears in **L-Edit** window.

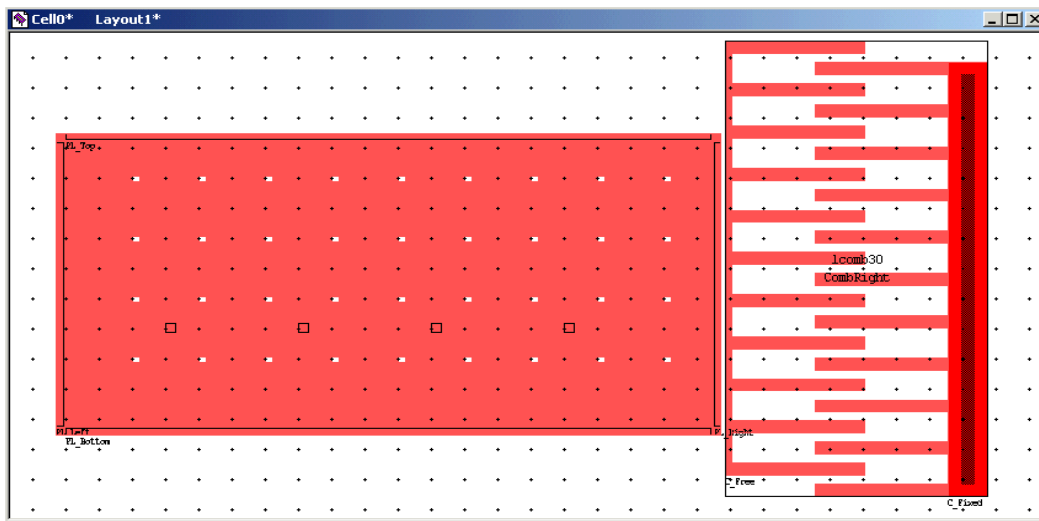


Figure 13: Viewing the modified comb-drive

This editing possibility is quite useful. Indeed, if you instantiate a component and then decide that it should be larger or longer, you can just modify one of its parameters and the newly edited component is automatically instantiated and replaces the first one.

Attaching Components

- ☑ Drag the comb-drive to the right side of the plate so that the two objects slightly overlap.
- Recall that an object can be dragged to new locations by selecting it with a click then holding down the center mouse button while moving it to the desired location on the page. For two-button mice, left-click on the object while holding down the **Alt** key to accomplish the move.
- ☑ Zoom in with the plus key. Use the arrow keys to pan the view to where the comb-drive overlaps the plate.
 - ☑ Re-align the **comb-drive** so that it looks like the layout in [Figure 14](#).

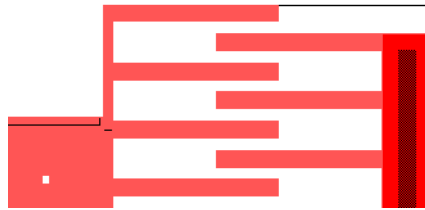


Figure 14: Aligning the comb-drive to the plate

- ☑ Copy the comb-drive by clicking it and choosing **Edit > Copy**, then **Edit > Paste**. The new comb-drive will appear in the center of the page on top of your existing drawing. Move it to the side of the other layout objects.
- ☑ Flip the second comb-drive by selecting **Draw > Flip > Horizontal**.
- ☑ Change the name of this copied instance by selecting **Edit > Edit Object** and entering **CombLeft** in the **Instance Name** field.
- ☑ Attach the second comb-drive to the left side of the plate ([Figure 15](#)).

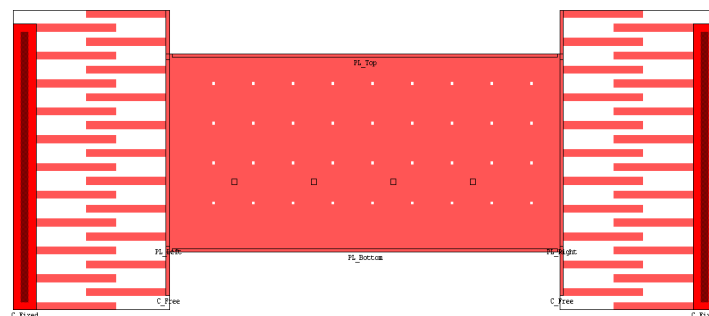


Figure 15: Viewing the incomplete design

Generating the Folded Springs

- ☑ Create a folded spring by clicking the **Folded Spring** icon in the **Library Palette**.



- ☑ Change the **Instance Name** to **SpringTop** and click **OK**.
- ☑ Position it above the center of the plate so that it overlaps (see [Figure 16](#)).

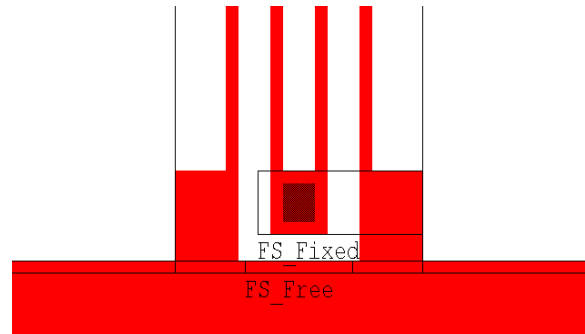




Figure 16: Positioning the folded spring

- ☑ Copy and paste **SpringTop** then flip it by selecting **Draw > Flip > Vertical**. Position the new folded spring below the plate so that it overlaps the plate.
- ☑ Change the name of this copied instance by selecting **Edit > Edit Object** and entering **SpringBottom** in the **Instance Name** field.


Generating the Ground Plate

- ☑ Create a ground plate by clicking the **Ground Plate**  icon in the **Library Palette**. Leave all the parameters at their default values and click **Insert**.
- ☑ Home the view, then move the ground plate so that it covers all the moving parts of the resonator (refer to [Figure 18](#)).

Generating the Bonding Pads

- ☑ Create a bonding pad by clicking the **Bonding Pad**  icon. Leave all parameters for the bonding pad at their default values. Click **OK**.
- ☑ Position the bonding pad slightly to the right of the right comb-drive ([Figure 17](#)).

Now, you must connect the bonding pad to the comb-drive by drawing a box on **Poly0** overlapping the two components.

- ☑ Choose the **Box** tool  by clicking it and select the **Poly0** layer from the **Layers Palette** by clicking on the first item in the first row of the **Layers Palette**. As your mouse is moved over the **Poly0** button, a tool tip will appear displaying the layer name. **Poly0** will also appear in the list box at the top of the **Layers Palette**.
- ☑ Click once to set the upper left corner, hold down the key down and drag to the opposite corner, release at the desired location of the opposite corner.

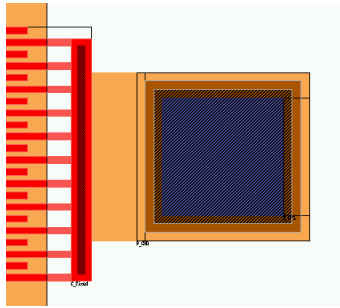


Figure 17: Attaching the first bonding pad

- ☑ Copy the bonding pad and place it to the left of the left comb-drive.
- ☑ Flip the bonding pad by selecting **Draw > Flip > Horizontal**.
- ☑ Connect the bonding pad to the comb-drive by drawing a box on **Poly0** overlapping the comb-drive and the bonding pad in a similar fashion as above.
- ☑ Make a copy of the last created bonding pad and place it to the bottom left side of the ground plate.
- ☑ Connect the ground plate to the bonding pad by drawing a box on **Poly0** overlapping the bonding pad and the ground plate in a similar fashion as above.
- ☑ Change the name of the cell you have been working on from **Cell0** to **MyResonator**. Select **Cell > Rename** and enter **MyResonator** as the new cell name.
- ☑ Save the file by choosing **File > Save**. Enter **myreson.tdb** as the file name and click **OK**.

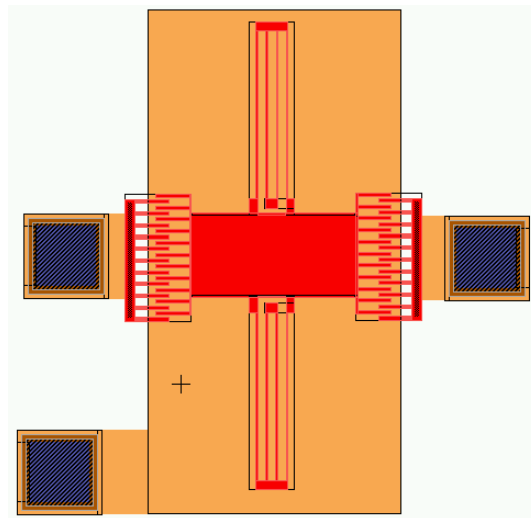


Figure 18: Final view of the lateral resonator

If you are interested in performing layout netlist extraction and layout vs. schematic comparison, refer to [Chapter 6 - Verification Tutorial](#) and use the **myreson.tdb** file.

Viewing Properties

Properties can be attached to any L-Edit object including boxes, polygons, wires, circles, ports, rulers, instances, cells and files. Properties can contain supplementary but necessary information about the object, such as what color it will appear in when 3D-modeled, or information about its constituent material.

MEMS Pro library components have a properties category called **Extract Properties**. This provides the netlist description of a design layout. The **Extract Properties** category is accessed via the **Cell Info** dialog box or the **Edit Instance** dialog box.

Properties were applied to each part of the resonator as it was constructed. You will look at those properties now.

- ☑ Select the instance of the **Plate** and choose **Edit > Edit Object(s)** from **L-Edit** tool bar.
- ☑ Click the **Properties** button. This instance should have no **Extract Properties** attached to it. If the instance does not have **Extract Properties**, L-Edit/ **Extract** pursues the hierarchy and looks for **Extract Properties** attached to the parent cell.
- ☑ Select **Cell** from the **Parent** list box and click the **View Parent** button to view the properties for the **Plate** cell.
- ☑ To view the **Extract Properties**, click the **+** sign next to the **EXTRACT** folder.

Some properties should be displayed under the **EXTRACT** folder (Figure 19).

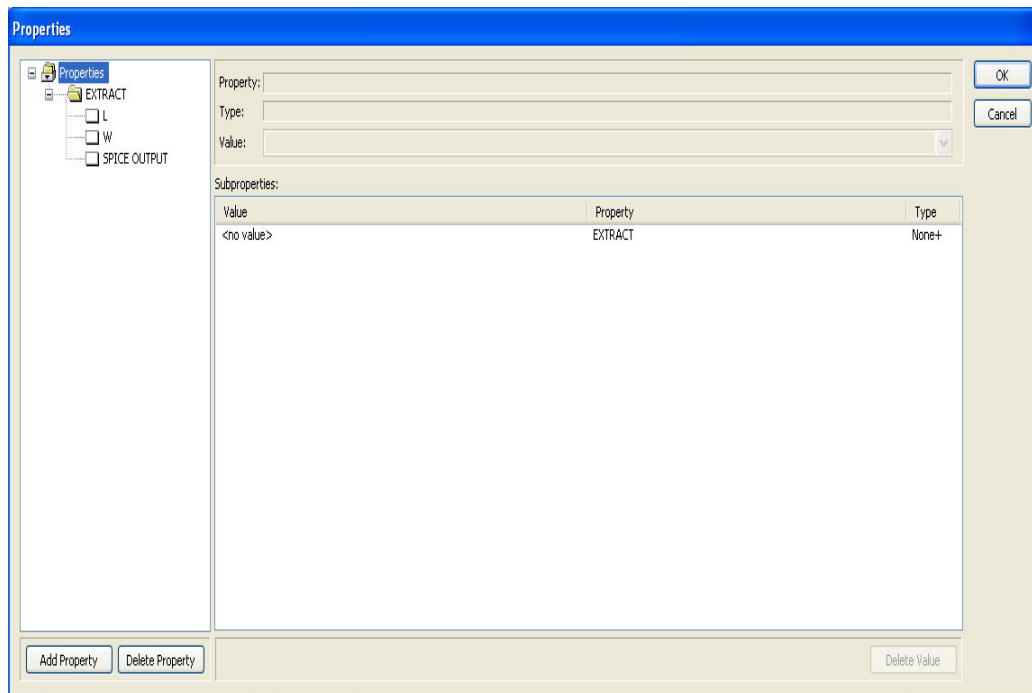


Figure 19: Properties dialog box

Try selecting different properties to view their types and values on the right side of the display.

- ☑ Click the **L** property. The value that you entered to represent the plate length is shown as the **Value** on the right side of the dialog.

To return to the layout, perform the following operations:

- ☑ Click **Cancel** to exit the parent **Properties** dialog box.
- ☑ Click **Cancel** to exit the main **Properties** dialog box.
- ☑ Click **Cancel** to exit the **Edit Object(s)** dialog box.


Viewing a 3D Model

The 3D Model Viewer generates a 3D model from a mask layout and the technology process definition the layout was constructed upon.

Tutorial Breakpoint

You will now create and view a solid model. If you are beginning the tutorial from here, open the design file you have been provided with by starting with the next section - [Launching L-Edit and Opening a File](#). If you are proceeding from the previous section, you may use your own design and skip to [Process Definition on page 31](#).

Launching L-Edit and Opening a File

- ☑ Launch L-Edit by double-clicking the **L-Edit** icon  in the *{MEMS Pro v8.0 Installation Directory}\L-Edit.exe*. A default file named *Layout1* should be visible in the work area.
- ☑ Close the *Layout1* file by selecting **File > Close**.
- ☑ Using the **File > Open** command, open the *reson.tdb* file located under the **MEMS_Tutorials\Resonator\Resonator** directory ([Figure 20](#)).

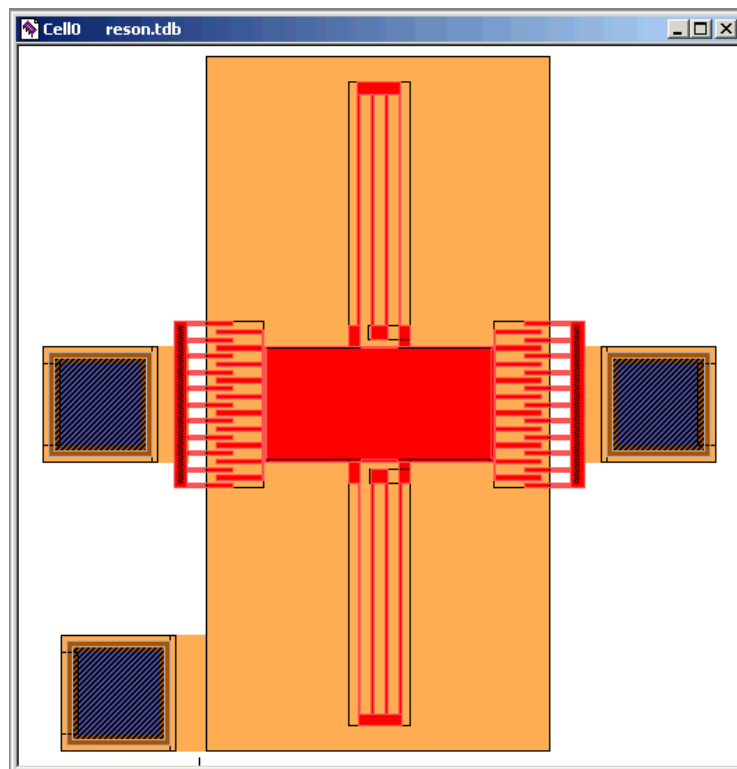


Figure 20: Layout view of the resonator

Process Definition

Importing the Process Definition

In addition to layout or mask data, the 3D Modeler needs the fabrication process description to generate 3D models. This information may already be saved within the layout, if not, it must be imported into the design file.

The following setup procedure loads the process information into the design file. Once imported, the process definition information is saved within the design file; it need not be re-imported when the file is re-opened.

- ☑ Choose **3D Tools > Edit Process Definition** in the **MEMS Pro Toolbar** to display the **Technology Manager** dialog.
- ☑ In the **3D Process Steps** tab, click the **Import** button.
- ☑ In the **Open** dialog box, select **polymumps.mpd** located under the **{MEMS Pro V8.0 Installation Directory}\Design Kits\MEMSCAP\polymumps** directory and click the **Open** button.

Note

For more information on *process definitions*, see [Process Definition on page 511](#) of the Tutorials.

Information describing the MEMSCAP PolyMUMPs process is imported into the dialog box. This information, in conjunction with the open layout, is used to build a 3D model.

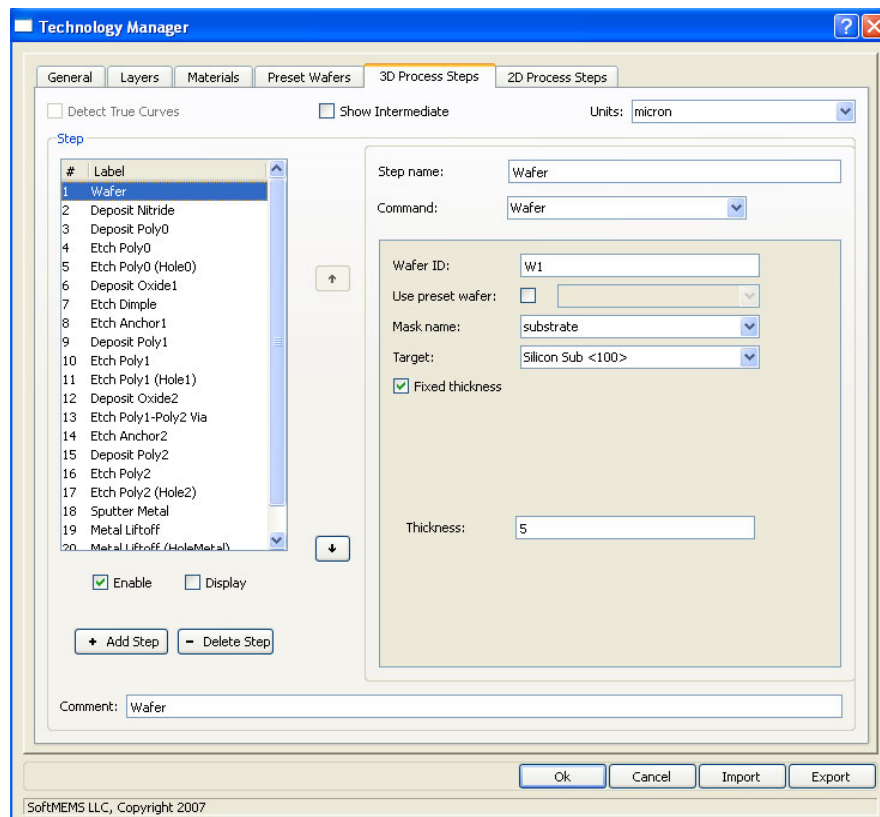


Figure 21: Process Definition dialog box

The top of the dialog box contains identifying information for the process definition, and two check boxes to check for curves and showing 3D intermediate steps. The left side of the dialog contains a list of the process steps. Beneath it and to its side are the controls for adding, deleting, moving, and enabling. The right side of the dialog contains the parameters of the selected step in the process steps list.

- ☑ To attach the PolyMUMPs process information to the design database, click **OK** to close the **Process Definition** dialog.

3D Model View

Generating the 3D Model

- ☑ Click anywhere on the title bar of the resonator layout window to make it active.
- ☑ Choose **3D Tools > View 3D Model** in the **MEMS Pro Toolbar**.

The 3D model generation will begin and the **Generating 3D Model** progress dialog will appear.

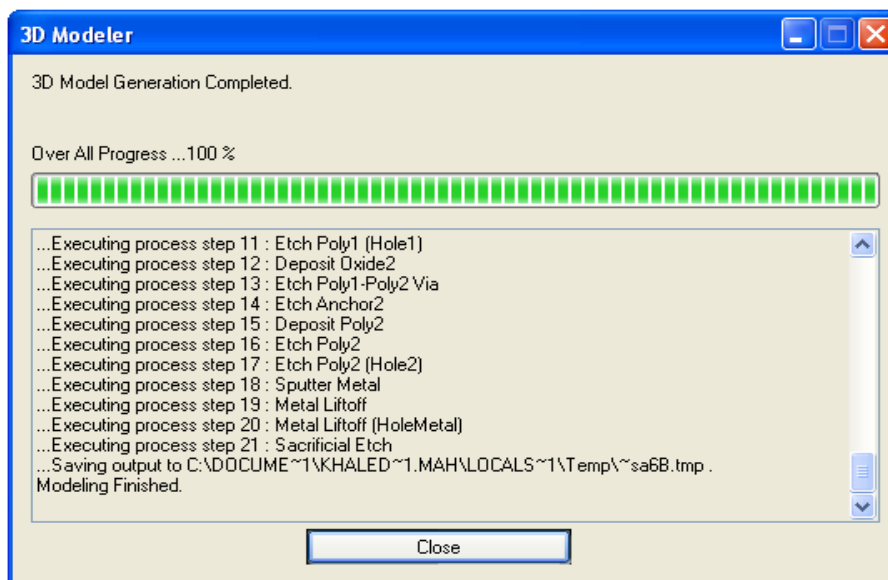


Figure 22: 3D modeler proress window

- ☑ Click the **Ok** button after completion to view the 3D model
- ☑ A **"Log Dialog"** Log Dialog window will appear, click on **"Save Log File"** if you want to save the log file.

the 3D model will appear in a new, active, **L-Edit** window. When a **3D Model View** window is active, the menu bar changes and the **3D Model View** toolbar buttons become enabled.


Manipulating the 3D Model View

To manipulate the 3D model view, use either the **3D Model View** toolbar or the menu options under the **View** menu.




Figure 23: 3D Model View Toolbar


The **Orbit View** allows you to rotate the model in order to view it from any angle.

- ☑ Select **View > Orbit** or click the **Orbit**  toolbar button.
- ☑ Click and drag over the 3D model window to orbit the model.

The 3D model view may be translated by panning the view.

- ☑ Select **View > Pan** or click the **Pan**  toolbar button.
- ☑ Click and drag over the 3D model window to pan.

You may examine the details of the model by zooming in to the area of interest.

- ☑ Select **View > Zoom > Box** or click the **Window Zoom**  toolbar button.
- ☑ Click and drag the pointer to the opposite corner of the zoom box and release the mouse button.

You may also use the **Ctrl** key and the three mouse buttons to **Orbit**, **Pan**, and **Zoom**:

- **Ctrl+Left** click and drag over the 3D model window to **Orbit**.
- **Ctrl+Right** click and drag over the 3D model window to **Zoom** in and out.
- **Ctrl+Center** click and drag over the 3D model to **Pan** the view.

Multiple Views

Multiple views of the generated 3D model may be viewed simultaneously.

- ☑ Click on the title bar of the layout window to make it active.
- ☑ In the **MEMS Pro Toolbar**, select **3D Tools > View 3D Model** twice to create two more views of the 3D model.
- ☑ Select **Window > Tile** to tile the windows. All open windows will be resized so that they fit without overlapping (Figure 24). Each 3D model view may be manipulated independently.

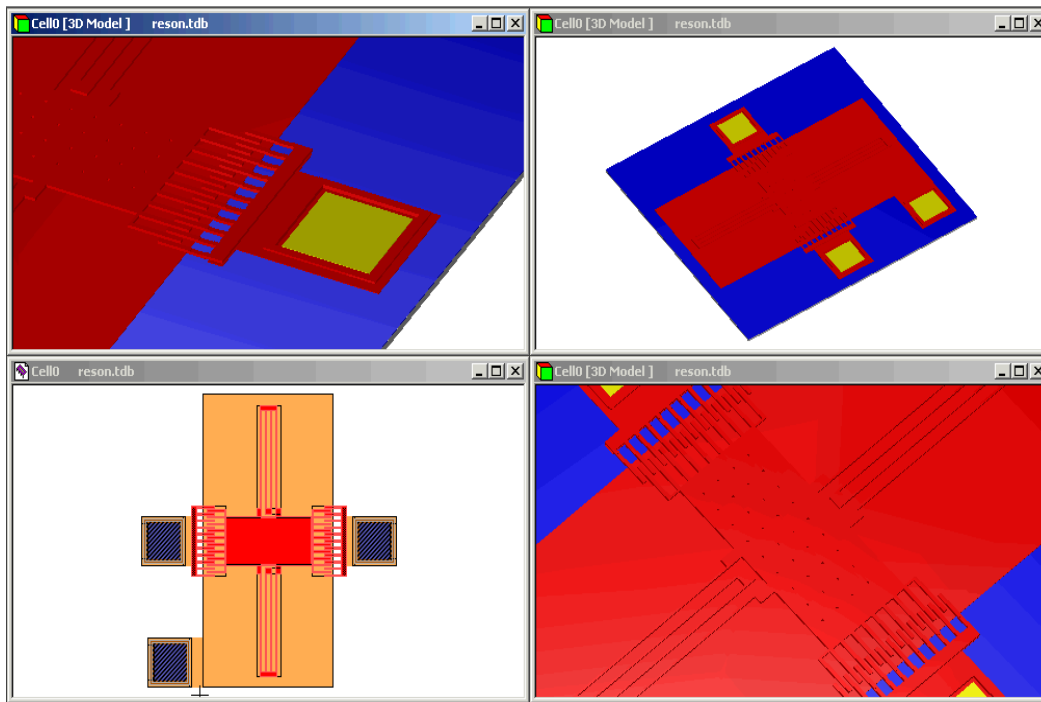



Figure 24: Tiling the windows displaying the 3D generation steps


Viewing the 3D Model

Once generated, 3D models do not need to be regenerated to be viewed again. In addition, 3D models are saved with the design information in the Tanner Database **.tdb** file.

- Keep the layout window with the original design open and active, close all 3D model windows by clicking the  button at the upper right corner of each window.
- Select **3D Tools > View 3D Model** in the **MEMS Pro Toolbar** and the 3D model will reopen without being re-generated.

3D Cross-section

Cross-sections may be taken in the 3D model using the **Cross-section** tool.

- Click on the title bar of the layout window to make it active.
- Select **3D Tools > View 3D Model** in the **MEMS Pro Toolbar** to create another view of the 3D model.
- Click the **Cross-section** tool . The **Generate 3D Model Cross-Section** dialog will appear (Figure 25).

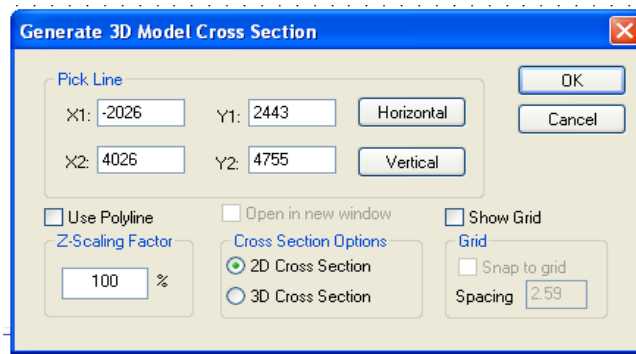


Figure 25: Generate 3D Model Cross Section dialog

- ☑ Click **OK**. A new L-Edit window will appear with a cross-section view.

The **3D Model View** window will snap to the top view and a line representing the cross-section cut plane will be displayed on top of the model. The cross-section plane is always normal to the surface of the wafer.

- ☑ Select **Window > Tile** so that you can view all the open windows simultaneously.
- ☑ Adjust the cross-section plane line to the desired location by using the left mouse button to move the end points of the line. The cross-section window will be updated after each adjustment of the line.

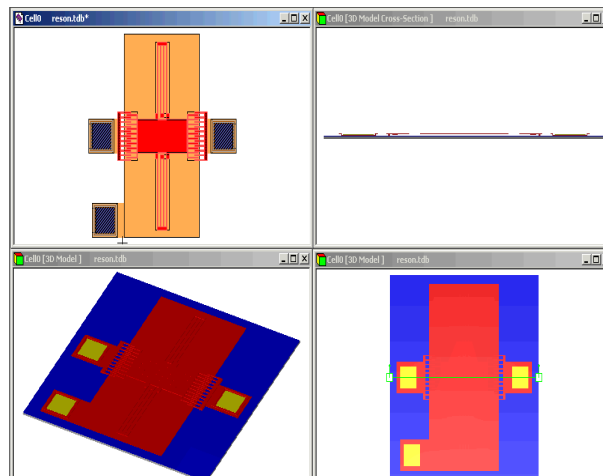


Figure 26: Tiling the windows displaying the various cross-section steps

- ☑ To exit the cross-section mode, select a different mode by clicking a toolbar button or by selecting the menu item corresponding to the next desired function.

Note

For more information on cross-sections, see [Viewing a Cross-section on page 422](#).

Drawing Tools

In this section of the tutorial, you will explore the drawing tools available with MEMS Pro. MEMS Pro supports objects such as all-angle wires / polygons, arcs, tori, circles, splines, curved polygons, arbitrary curve generator etc. We will use some of those to draw a rotary side-drive motor.

Tutorial Breakpoint

- ☑ If you are starting the tutorial here, double-click the **L-Edit** icon.

Seventeen object types are supported:

- Boxes.
- Polygons.
- Wires.
- Circles.
- Arcs.
- Tori.
- Splines.
- Ellipses.
- Parabolas.
- Hyperbolas.
- Sinusoidal Arcs.
- Spirals (archimedean, hyperbolic and logarithmic).
- Fillet.
- Arbitrary curves.
- Ports.
- Rulers.
- Instances.

- ☑ From L-Edit, open the **motor.tdb** file located under the **MEMS_Tutorials\motor** directory.

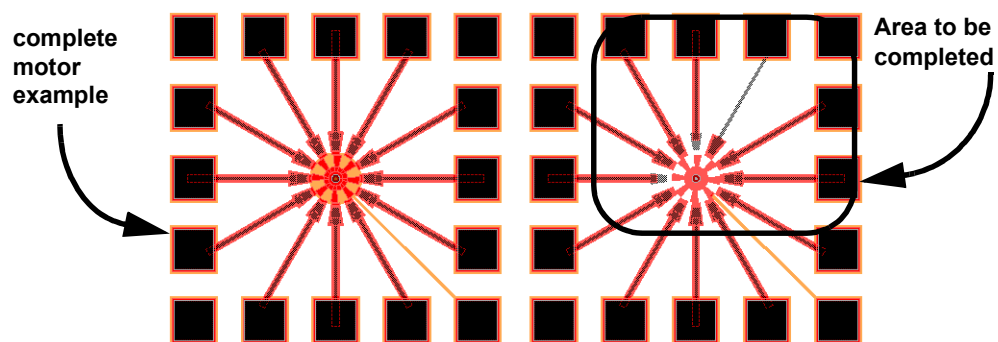


Figure 27: Viewing the motor.tdb file

Drawing a Torus

When drawing a torus, the first click of the left mouse button sets the center. The second click determines the inner radius of the torus. The third click determines the outer radius and the sweep angle.

- ☑ Select the **Torus** tool  from the **Drawing** toolbar.

Note

To display all buttons in L-Edit toolbar, right click the drawing toolbar and select "All Angles and Curves" from the popup menu.

- ☑ Left-click at the center of the incomplete motor to begin drawing the torus. Left-click again at the inner radius point, then complete the torus by right-clicking at the outer radius point (the inner and outer radius points are shown in [Figure 29](#)).

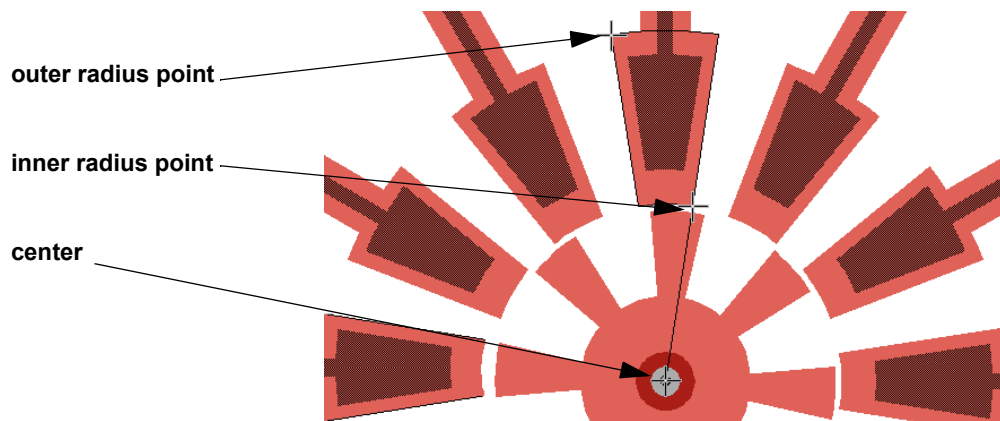



Figure 29: Creating a torus

Drawing a Curved Polygon

To create a curved polygon; a straight edged polygon must be drawn first. The straight edges can then be converted to curved edges by selecting a given edge with a **Ctrl+Right** click and then dragging it out to create the desired curve with the center mouse button.

You will use a curved polygon to draw a stator like the one you drew with the torus tool.

- ☑ Select the **Window Zoom** tool to manipulate the view so that the left-most stator is visible ([Figure 30](#)).
- ☑ Select the **All Angle Polygon** tool  from the **Drawing** toolbar.
- ☑ Left-click to set the first vertex of the polygon (vertex 1 in [Figure 30](#)). Left-click to set vertices 2 and 3, and then complete the polygon by right-clicking to determine the fourth vertex ([Figure 30](#)).

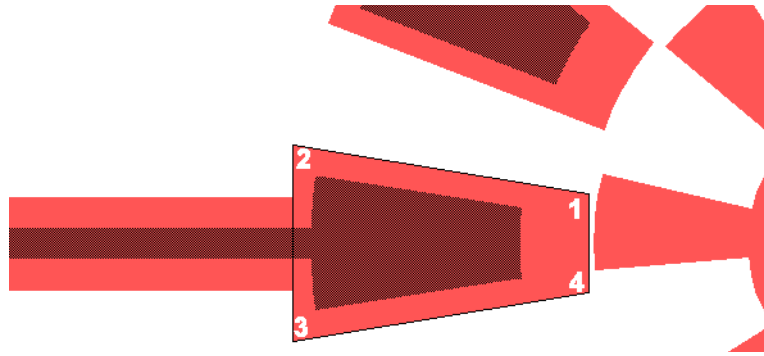


Figure 30: Creating the all angle polygon

- ☑ Select the right most edge with a **Ctrl+Right** click. Once selected, the edge will be highlighted.
- ☑ Press the **Ctrl** key, hold and drag the center mouse button (**Alt+Left** click for two button mice instead of the center mouse button) to the left to convert the straight edge into a curved edge as shown below. Release the mouse button to complete this action.

Note

Keep the All Angle Polygon drawing command selected while following the above procedure to curve the two straight edges.

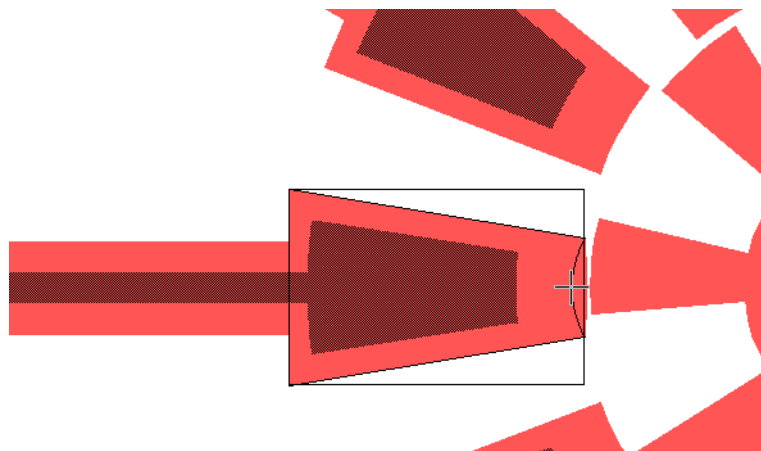


Figure 31: Curving the right most edge of the polygon

- ☑ Similarly, select the left edge with a **Ctrl+Right** click. Once selected, the edge will be highlighted.
- ☑ Press the **Ctrl** key, hold and drag the center mouse button (**Alt+Left** hold for two button mice) to the left to convert the straight edge into a curved edge. Release the mouse button to complete this action (Figure 32).

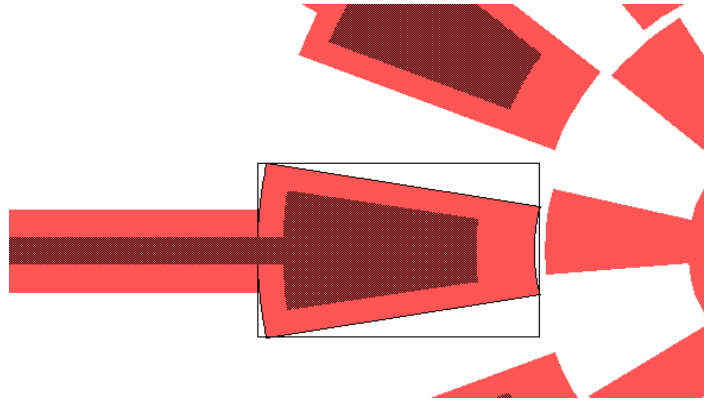




Figure 32: Curving the left most edge of the polygon

Drawing a Circle

- ☑ Select the **Circle** tool  and select the *poly0* layer.
- ☑ Place a circle at the center of the incomplete design so that it matches the complete design. Left-click to anchor the center of the circle, drag the mouse to set the radius of the circle and release.

Drawing a Box

Close inspection of the rotor reveals that there are three dimples positioned at 90° intervals around the center of the rotor. A fourth dimple must be placed on the *dimple* layer to complete the pattern.

- ☑ Choose the **Box** tool  and select the *dimple* layer.
- The box may be constructed anywhere on the layer. We will move it to the proper location after drawing it.
- ☑ Left-click to anchor the first corner of the box, drag the cursor three grid units from the anchor point to determine the opposite corner of the box, and release.
 - ☑ Left-click the newly drawn dimple box to select it.
 - ☑ Move the dimple to a different location with a center-click (**Alt+Left** click for two button mice), hold, drag and release at the desired location. The dimple should be placed to the right of the rotor's center so that it is approximately 90° away from the next nearest dimples (Figure 33).

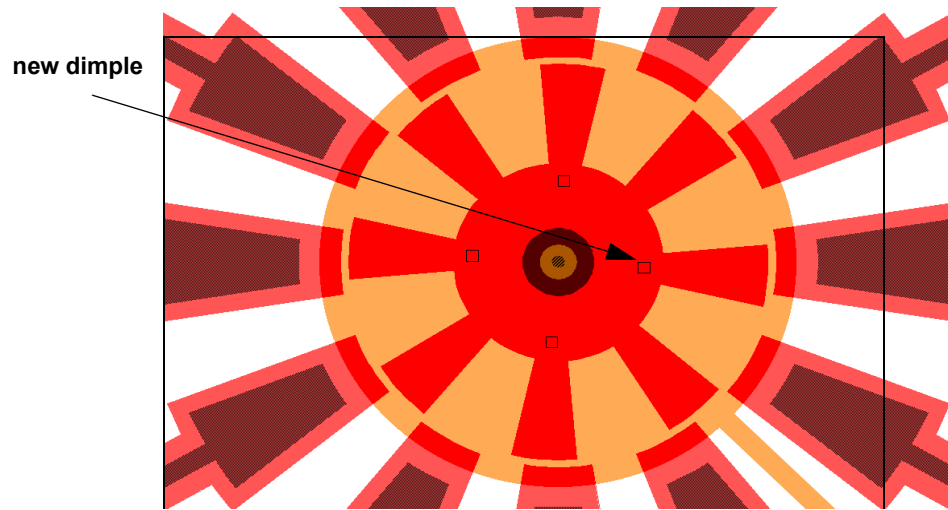


Figure 33: Creating a dimple

This concludes the general tutorial of MEMS Pro.

2 Optical Library Tutorial

▪ Introduction	44
▪ Theory	45
▪ Tutorial Procedures	47
▪ Schematic Editing	48
▪ Placing Components	49
▪ Connecting Elements	54
▪ Writing Simulation Commands	57
▪ Running Simulation and Viewing Results	59

Introduction

This tutorial aims at displaying the abilities of the Gaussian optical library, as well as the ability of integrating multi-domain components. Having quite a good knowledge of **MEMS Pro**, it will be easy for you to run the tutorial and view the results successfully. However, It is preferred that you run the Resonator tutorial before getting into this one.

Theory

A switch works by moving a mirror that changes the state of the switch from the "Cross" state to the "Through" state.

In order to differentiate between the two input signals, the magnitudes of their power are made different (i.e. the first has an input power of 2 mW, and the second has an input power of 1 mW).

Cross State

The beam is reflected by the mirror to the fiber receiver as shown in (Figure 34).

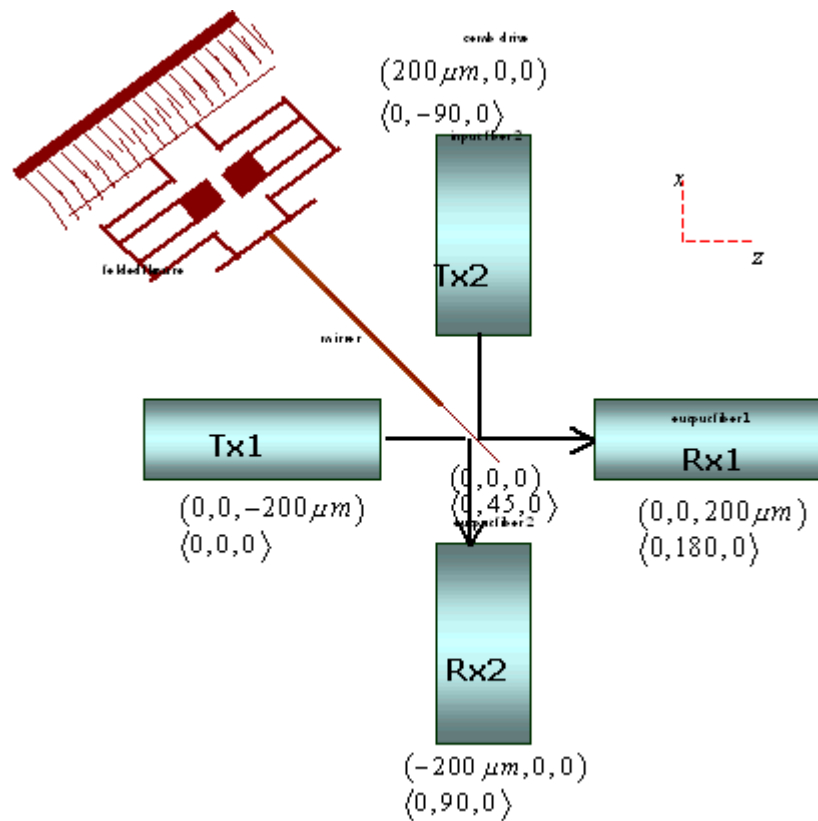


Figure 34: Cross state

Through State

The mirror is moved by the combdrive and the beam passes without reflection, as shown in (Figure 35).

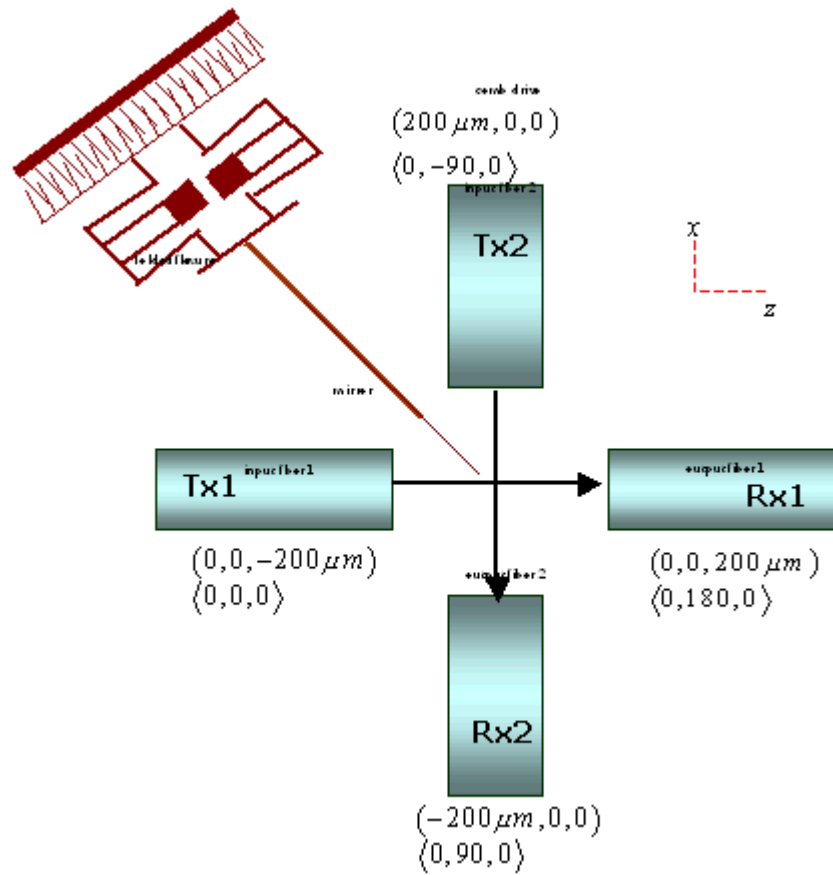


Figure 35: Through state

Tutorial Procedures

This tutorial is composed of the following main steps:

- Placing the components.
- Connecting the elements.
- Writing the simulation commands.
- Running the simulation and viewing the results.

Schematic Editing

- ☑ Launch **S-Edit** from the start menu through the following path:
Start >All Programs > MEMS Pro V8.0 > MEMS S-Edit.
- ☑ Click **File > Open > Open Design** and select the file **{MEMS Pro installation directory}\MEMS_Tutorials\Optical\Switch\Switch.tanner.**
- ☑ Create a new view by clicking **Cell > New View.**
The **New View** dialog is displayed (Figure 36), where you can add the details of the new view in MySwitch.

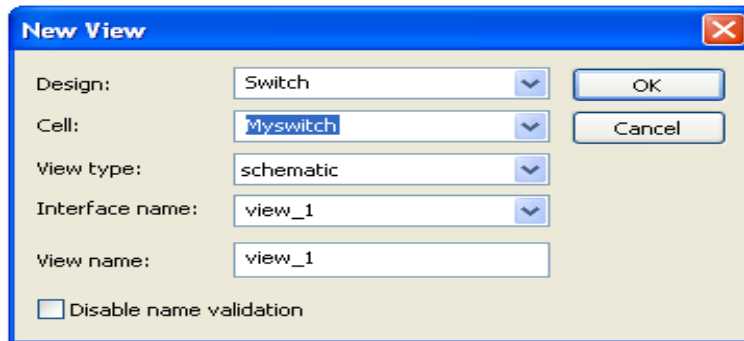


Figure 36: New View dialog

Placing Components

Optical Components

- ☑ From the **Libraries** dialog that exists on the left of schematic page select LensedFiberEmitterGs and click on **Instance** to place this instance in the schematic page. Do the same step so that we will have the following components from the **Libraries** dialog ([Figure 37](#)):

- LensedFiberEmitterGs (2 instances).
- PlaneMirrorFiniteThick_Gs (1 instance).
- LensedFiberReceiverGsTwoMode (2 instances).

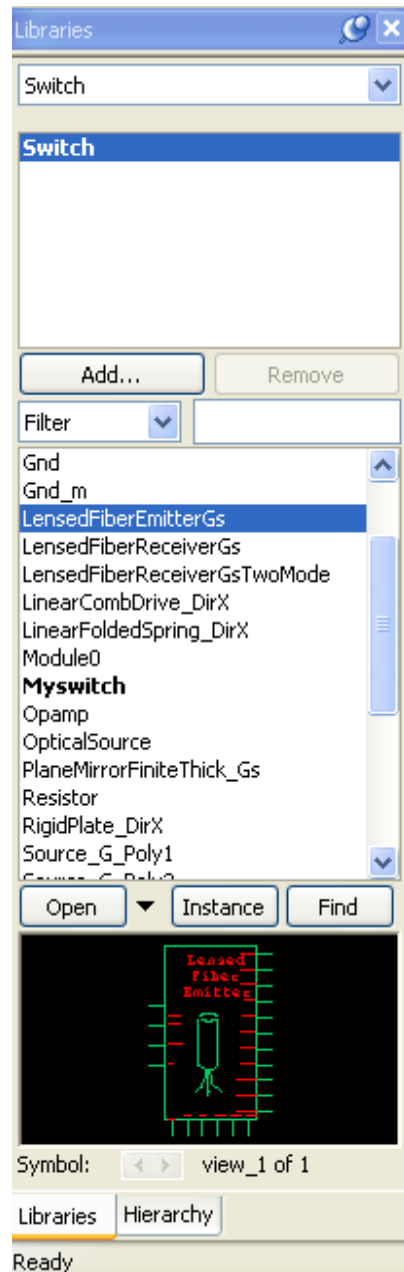


Figure 37: **Libraries** window

- ☑ Edit the properties of the instance by selecting the instance and you will find the **Properties** dialog will be on the right of schematic page.

The **Properties** dialog is displayed (Figure 38), where you can edit in the values of the properties.

- LensedFiberEmitterGs (1).

Place it on the left side of the PlaneMirrorFiniteThick_Gs.

ï Fib_z0=-200e-6.

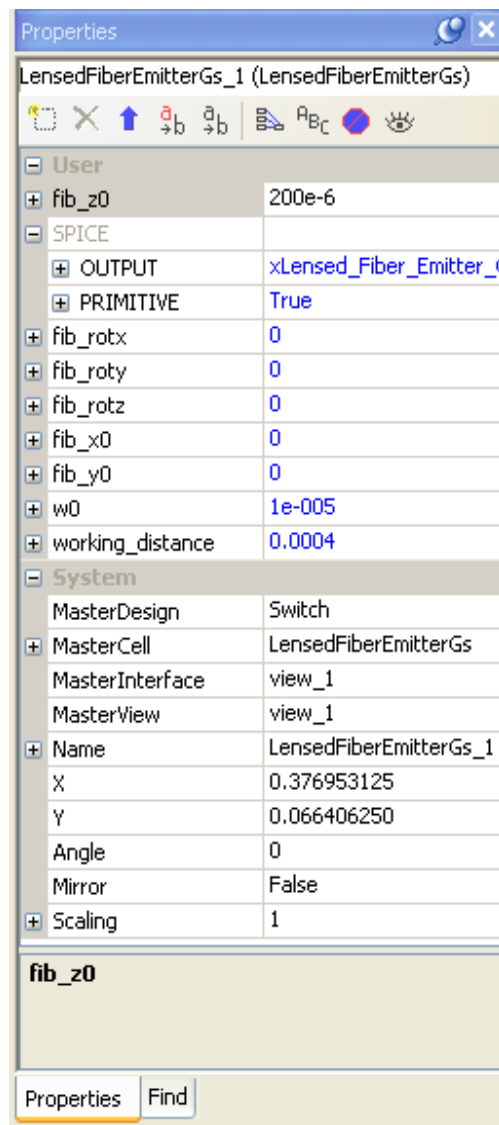


Figure 38: **Properties** dialog.

- Similarly, create instances of the following modules and set their properties as specified below:

- LensedFiberEmitterGs(2).

Place it on top of the PlaneMirrorFiniteThick_Gs.

ï Fib_x0= 200e-6.

ï Fib_roty= -90.

- PlaneMirrorFiniteThick_Gs.

ï Mir_roty= 45.

ï Refl1= 0.95.

- ï Refl2= 0.95.
- ï Thickness= 2e-6.
- ï Width= 50e-6.
- ï Height= 0.0002.

- LensedFiberReceiverGsTwoMode (1).

Place it below the PlaneMirrorFiniteThick_Gs.

- ï Fib_rotty = 90.
- ï Fib_x0 = -200e-6.
- ï working_distance = 400e-6.

- LensedFiberReceiverGsTwoMode (2).

Place it on the right side of the PlaneMirrorFiniteThick_Gs.

- ï Fib_rotty = 180.
- ï Fib_z0 = 200e-6.
- ï working_distance = 400e-6.

Place two instances of "Optical Source" cells:

- The First

- ï Power = 2e-3.

- The Second

- ï Power = 1e-3.

Mechanical Components

- From the **Libraries** dialog select **Instance** to place an instance of the following components in the schematic page:

- LinearFoldedSpring_DirX (2 Similar Instances).

- ï flexure_length = 400e-6.
- ï flexure_width = 2e-6.

- LinearCombDrive_DirX.

- ï finger_length = 300e-6.
- ï finger_overlap = 150e-6.
- ï number_of_gaps = 150.
- ï tStruct = 20e-6.

- RigidPlate_DirX .

- ï plate_length = 200e-6.
- ï plate_width = 40e-6.
- ï tSacrif=0.5e-6.

- VCVS.

- ï K=-1.

- Source_V_Pulse.

- ï delay = 100e-6.
- ï falltime = 1e-6.
- ï period = 2e-3.
- ï risetime = 1e-6.
- ï vhigh = 15.
- ï width = 1e-3.

Connecting Elements

Mechanical Elements

- Connect the mechanical parts as shown in (Figure 39).

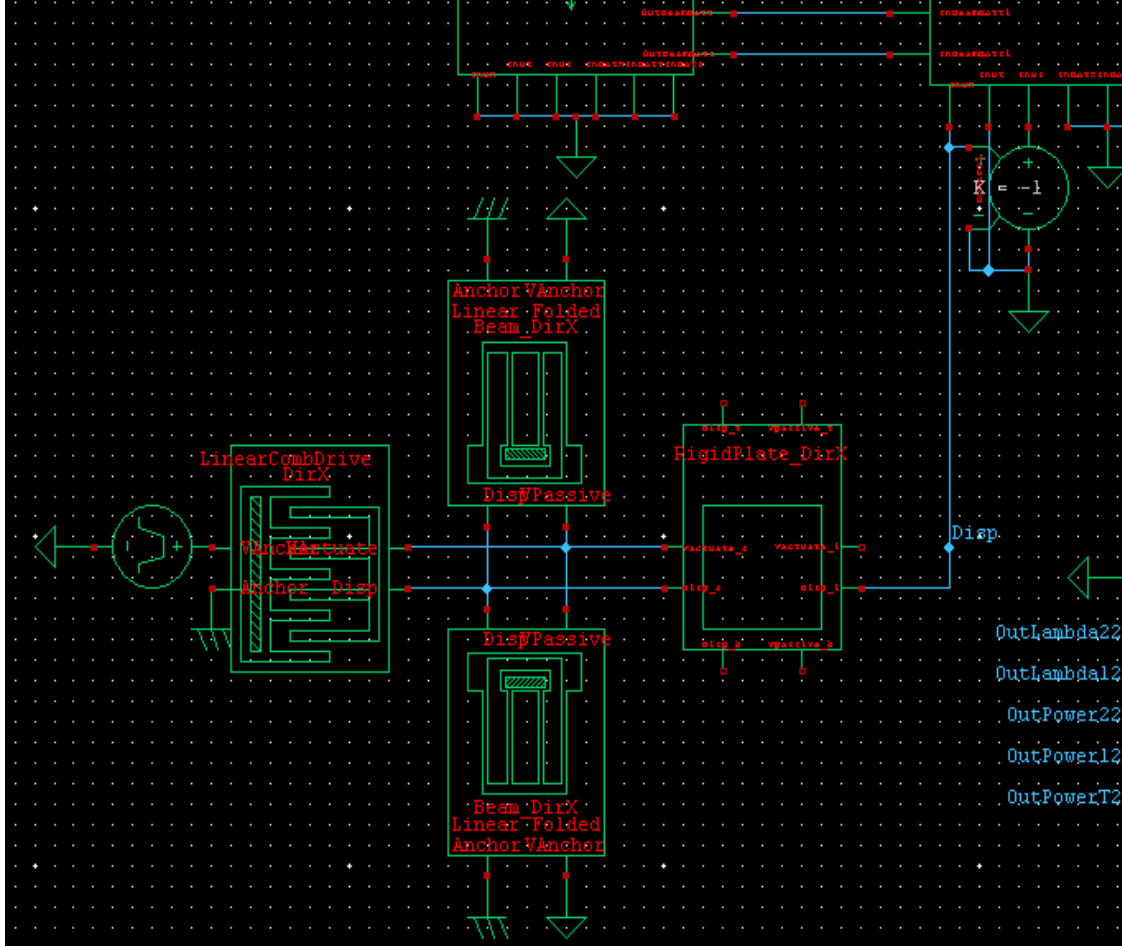


Figure 39: Connecting mechanical elements

Optical Elements

- The **Lensed Fiber Emitter** is connected to the optical power source of OutPower 2e-3 through its inputs, while its outputs are connected to the double-sided mirror, as shown in (Figure 40).

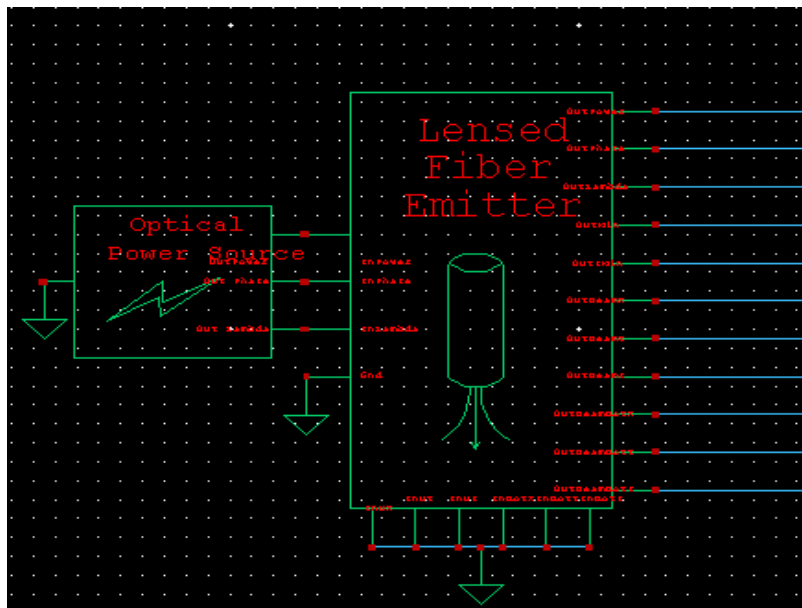


Figure 40: Connecting emitter

Each output pin in the emitter is connected to its corresponding input pin in the mirror.

- When connecting the receiver, note that the output of each port of the mirror is connected to both receivers as shown in (Figure 41).

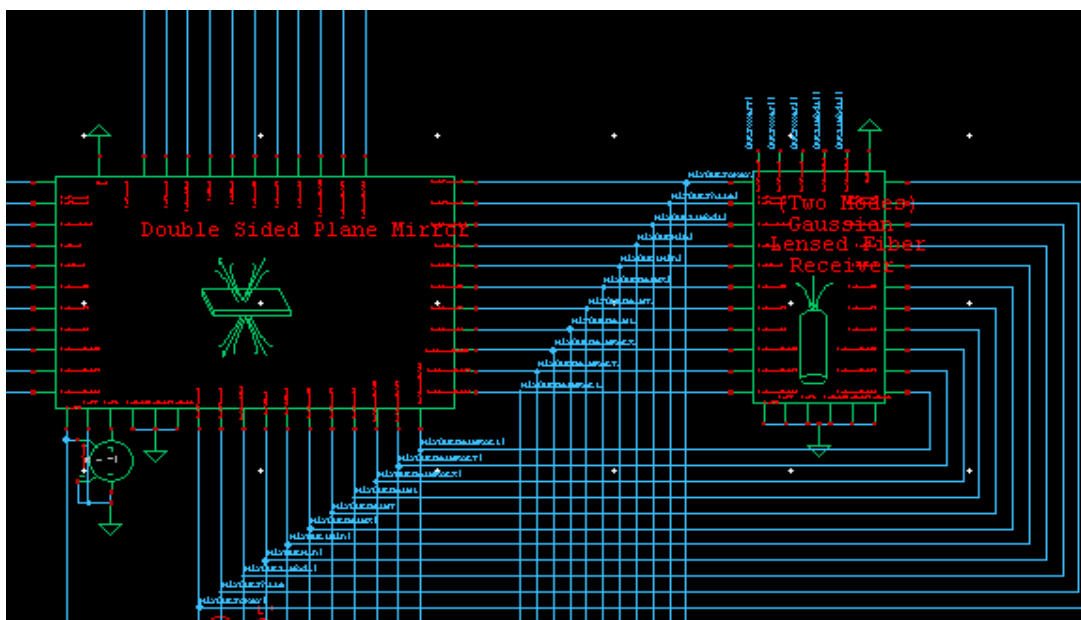


Figure 41: Connecting receivers

Coupling

- Connect the VCVS as shown in (Figure 42).

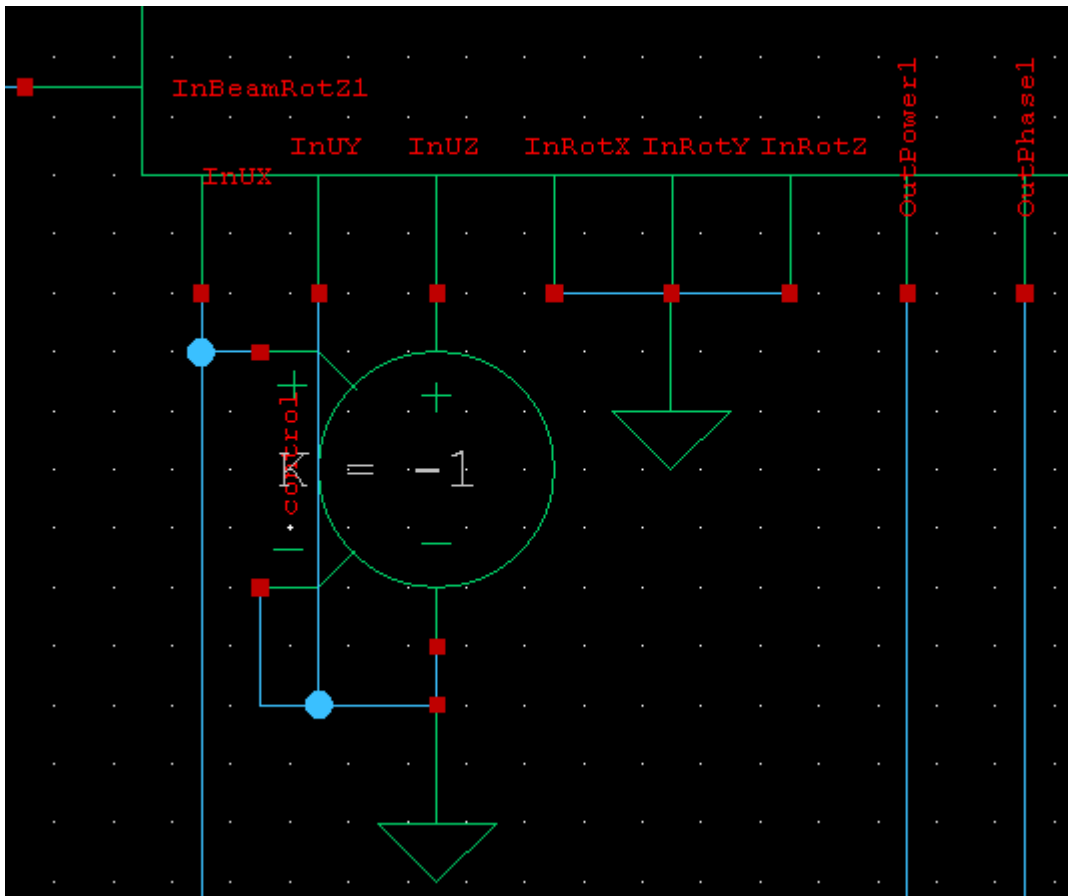


Figure 42: Mechanical coupling

For the complete schematic connections, please see (Figure 43) or refer to "TC_switch" in the optical tutorial design in **MEMS_Tutorials**.

Writing Simulation Commands

- To add a spice command:

- Select "TSpiceCommand" from **Libraries** dialog.
- Click on "TSpiceCommand" that is instanciated in the schematic page.
- Enter the following command in the **SPICE OUTPUT** field in the **Properties** dialog:

```
.include "..\..\..\process.sp"
```

- Repeat the previous 3 steps for the following commands:

A. Model declarations:

```

i .model lensedFiberEmitterGauss external winfile="..\..\..\memslib.dll"
i .model lensedFiberReceiverGauss external win-
file="..\..\..\memslib.dll"
i .model planeMirrorGaussFiniteThickInteg external win-
file="..\..\..\memslib.dll"

```

B. Simulation commands & options:

```

i .tran 8u 4m
i .options abstol=1e-11 numnt=100 method=gear maxord=1

```

C. Output desired waveforms:

```

i .print tran V(OutPowerT2) V(OutPowerT1)
i .print tran Displacement<M>='v(Disp)'

```

You can print the waveform of any other node at your choice by using the ".print" command as shown above.

The final step is to make sure that the SPICE output files will be created in the right directory path, which is "**{MEMS Pro installation directory} \MEMS_Tutorials\Optical\Switch\MySwitch**". You must write down this path in "**SETUP > SPICE Simulation > SPICE File Name**".

The final view will be as shown in (Figure 43):

Running Simulation and Viewing Results

Note

If you do not start the tutorial from the beginning you can

- Open the file {MEMS Pro installation directory}\MEMS_Tutorials\Optical\Switch.tanner.
- Open the cell TC_switch.



- Click  to create the netlist.
- Run the transient simulation by clicking the **Run Simulation** command  in the generated T-Spice netlist.

Figure 44 shows the waveforms of the output from the first receiver, the second receiver and the comb displacement:

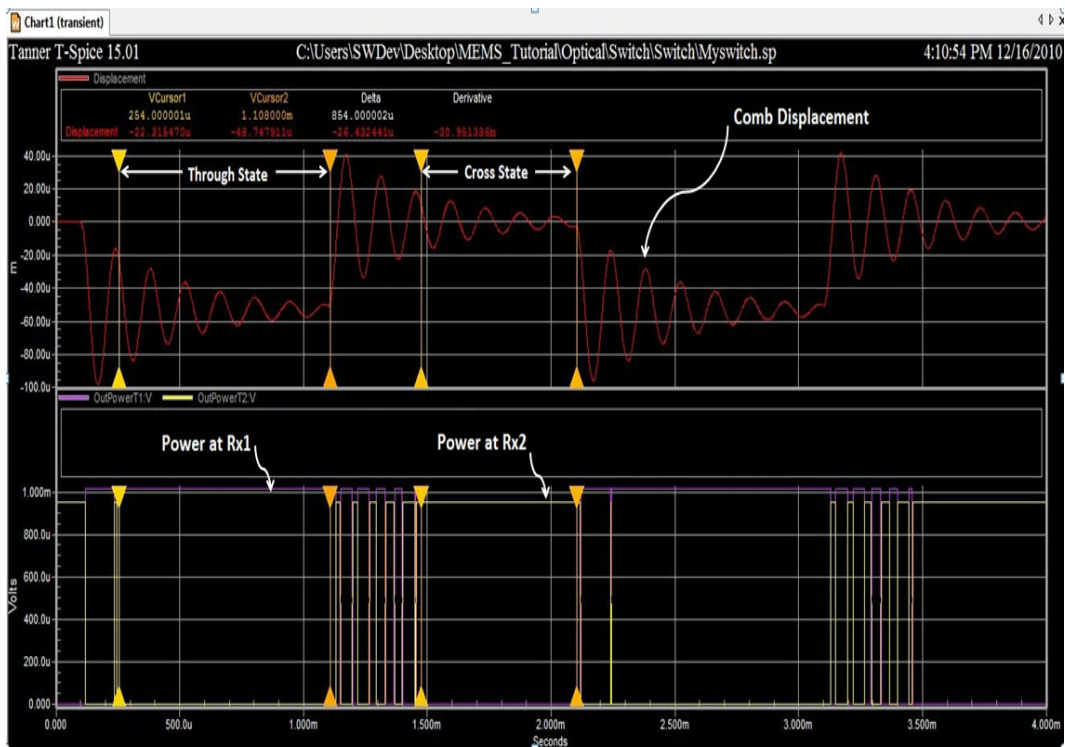


Figure 44: Simulation results

3 Statistical Analysis Tutorial

▪ Introduction	62
▪ Process Information	63
▪ Resonator Example	64
▪ AC Analysis	66
▪ Start Simulation	68

Introduction

In the Statistical Analysis tutorial the following is tackled:

- Statistical Analysis based on process/mask variations.
- Incorporates statistical data from foundries into Spice model.

Statistical Analysis can be used in the following applications:

- Develop process corners for simulation.
- Design centering: choose mask/process designs which maximize yield, and are the most tolerant to variation.
- Sensitivity calculation: parameters contribution to design goal changes.
- Yield analysis.

Process Information

Process information can be obtained from MEMSCAP.

- Example: Thin film properties from MUMPSrun #68 (Figure 45).

Run #68					
150mm wafers					
	Thickness (Å)	Standard Deviation (Å)	Sheet Resistance (ohm/sq)	Resistivity (ohm-cm)	Stress (MPa)
Nitride	6,008	292			95 T
Poly0	4,999	51	35.6	1.78E-03	16.33 C
Oxide1	19,810	316			
Poly1	19,922	220	13.9	2.77E-03	6.33 C
Oxide2	7,582	72			
Poly2	14,605	240	28	4.09E-03	6.00 C
Metal	5,245		0.048	2.52E-06	17.13 T

Figure 45: Process Information.

Resonator Example

In this tutorial we will study the effect of process variation (represented in the tolerance of the beam width of the folded beam flexure) on the resonator frequency. The variation of the flexure beam width is considered to be of a random gaussian distribution with a nominal value of 3um and a standard variation of 0.25um.

- ☑ Create a new folder (*C:\MonteCarlo*) and copy the folder "**reson**" and the file **process.sp** found in **{MEMS Pro installation directory}\MEMS_tutorials\Resonator** into it.
- ☑ Open *reson.tanner*.
- ☑ Select the top flexure instance, then go to the **Properties** dialog on the right of schematic page.
- ☑ Change the value of the *flexure_width* from a constant value to a parameter name (let it be *wf_monte*) as shown in the figure below. Note that the parameter name should be written between quotation marks.

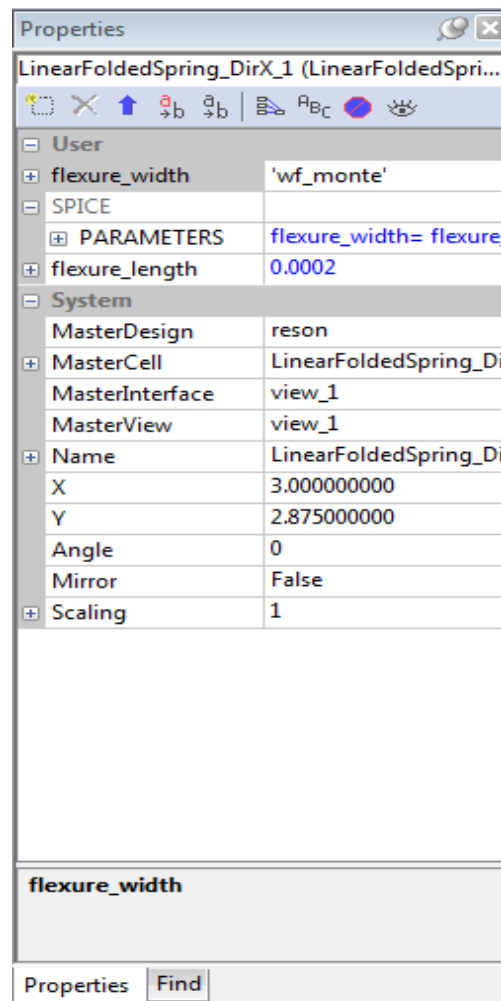


Figure 46: **Properties** Dialog.

- ☑ Repeat the previous step with the bottom flexure instance.

- ☑ Select **Setup >SPICE Simulation** as shown in the figure below.
- ☑ Select the file **process.sp** from the **"Include Files"** entry, and write **"C:\MonteCarlo\reson\MyResonator"** in the **"SPICE File Name"**.
- ☑ Set **Enable Waveform Voltage Probing** to **True**. Also check **AC Analysis**. Double click on **AC Analysis** and set the values of **AC Analysis** as mentioned below.
- ☑ Start Frequency: 10K.
- ☑ Stop Frequency: 100K.
- ☑ Number of Frequencies: 500.
- ☑ Sweep Type: dec.

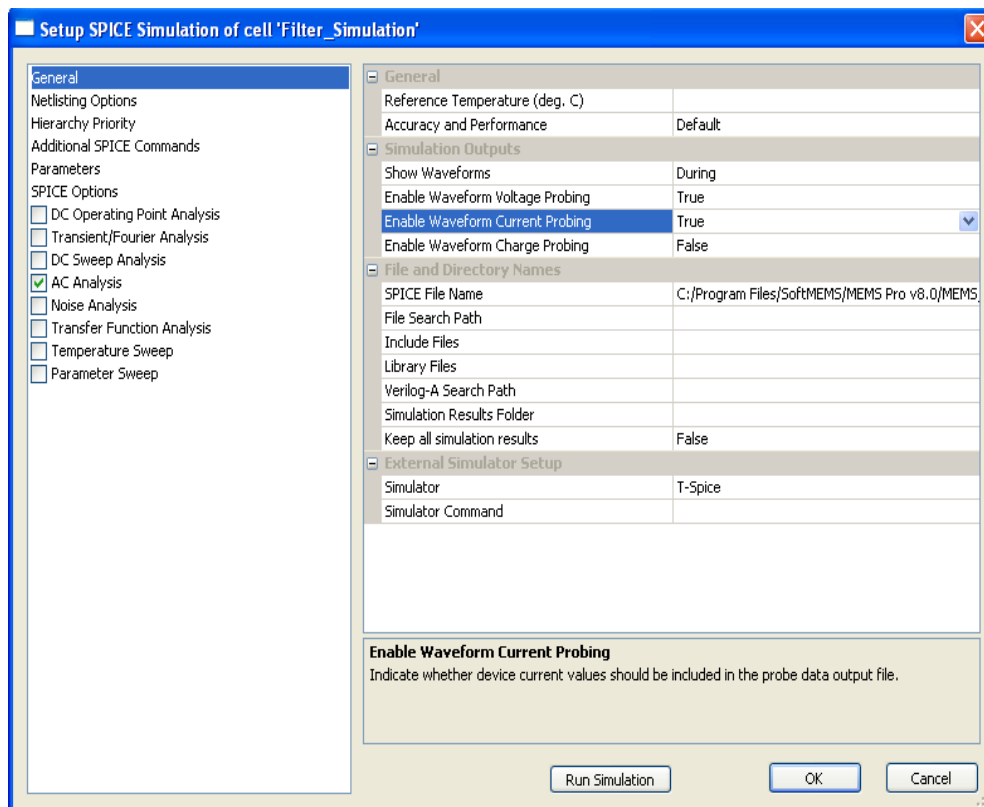


Figure 47: Setup SPICE Simulation

Now we will set the parameter type, nominal and the standard deviation values of $\acute{w}f_monte\acute{i}$.

AC Analysis

- ☑ In the **Setup SPICE Simulation** window, select **Additional SPICE Command** from the right then click the **Insert command** button.
- ☑ In **SPICE Command Wizard** window select **Settings** from the right then click on **Parameters**.
 - From the **Parameter** drop down list, select **Monte Carlo**.
 - Enter *wf_monte* in the **Parameter** name field.
 - Select **Gaussian (absolute variation)** from the **Probability distribution type** drop down list.
 - Enter "3e-6" in the **Nominal value** field.
 - Enter "0.25e-6" in the **Absolute variation** field.
 - Enter "1" in the **Sigma Level** field.
- ☑ Click **Add**. The parameter is added to the **List of** area. The window should look like the figure below. After this click **Insert Command**.
- ☑ The inserted command should be:

```
.param wf_monte=agauss(3e-6,0.25e-6,1).
```

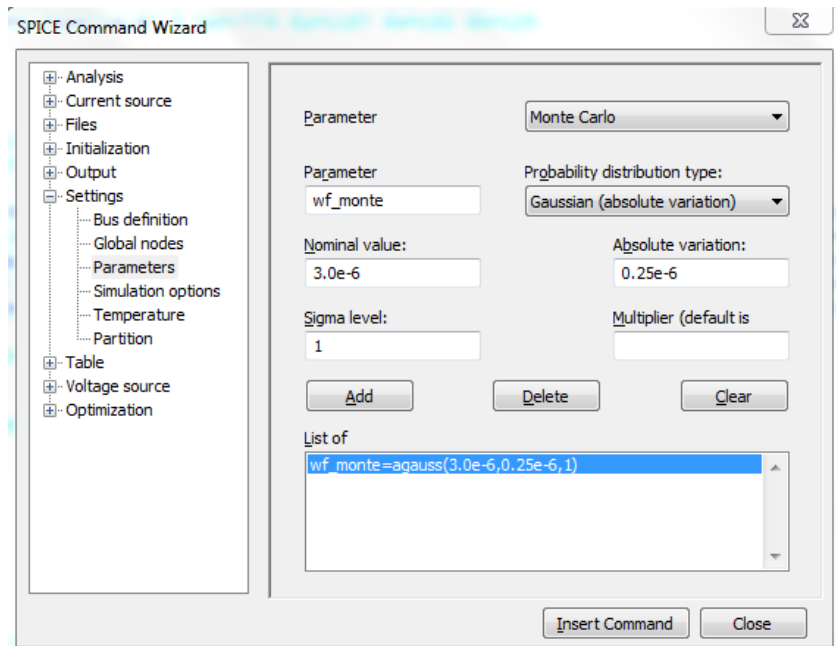


Figure 48: T-Spice Command Tool

Now, we will set the analysis type and determine the number of Monte Carlo points.

- ☑ In **SPICE Command Wizard** window select **Analysis** from the right then click on **AC**. Choose **"decade"** for Frequency sampling drop down list. Set frequencies per decade to **500**. Set frequency range from **10k** to **100k**. Click the **Sweep** button. Set **Sweep** to **Monte Carlo**. Set the **Number of Monte Carlo analysis runs** to **30**.

- Click **Accept** and then click **Insert Command**. The inserted command should be:
.ac dec 500 10k 100k sweep monte=30.
- Close the Command Tool.

Start Simulation

- Select **Probe Voltage** tool and click the **left_m** node. The results should appear as shown in the figure below.

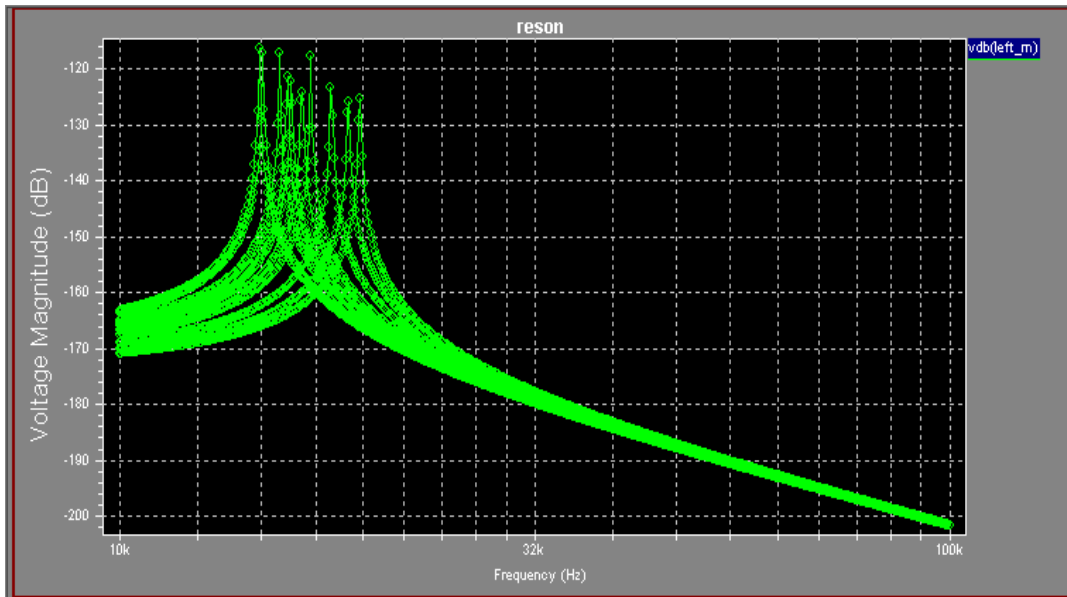


Figure 49: Simulation

The results show that the variation of the beam width due to process variation lead to a large variation in the resonance frequency of the resonator.

4 Design Rule Checker

▪ Introduction	70
▪ Design Rule Checker Commands	71
▪ DRC Setup	72
▪ Design Rule Types	73
▪ Ignore Options	81
▪ DRC Tutorial	82
▪ Setting Design Rules	83
▪ MEMS Tutorial - DRC	85
▪ Running DRC	87
▪ Re-running DRC	94

Introduction

The Design Rule Checker (DRC) provides you with a number of features that facilitate the compatibility checking between layout and fabrication capability. Layout designs have geometric rules or constraints, limited by fabrication capabilities. Therefore, there are sets of design rules, provided by foundaries, that are determined for specific fabrication processes.

To check a layout for design rule violations you have to define the design rules at first, then run the Design Rule Checker (DRC).

Through the Design Rule Checker (DRC), you can define mixed MEMS and IC design rules sets. It supports all-angle checking, which serves MEMS. Furthermore, you can run the DRC checking on full-chip or on a selected area in the layout.

Design Rule Checker Commands

You can access the DRC commands by selecting **Tools** from the **L-Edit** menu.

- **DRC Setup: Tools > DRC Setup.**
 - Sets up the design rules.
- **DRC : Tools > DRC.**
 - Runs DRC on entire layout.
- **DRC Box: Tools > DRC Box.**
 - Runs DRC on a selected (rectangular) region.
- **Clear Error Layer: Tools > Clear Error Layer.**
 - Removes error flags created by previous DRC.
- **DRC Error Navigator: Tools > DRC Error Navigator.**
 - Steps through and displays DRC errors in the active layout.

DRC Setup

In order to setup the DRC, you have to define the design rules in the DRC setup:

- ☑ Open **Tools > DRC Setup** in the L-Edit menu.
The **Setup DRC** dialog is displayed, where you can create a new design rule set, load a set from a text file, or import an exported rule set.
- ☑ You can import either DraculaÆ or CalibreÆ files.

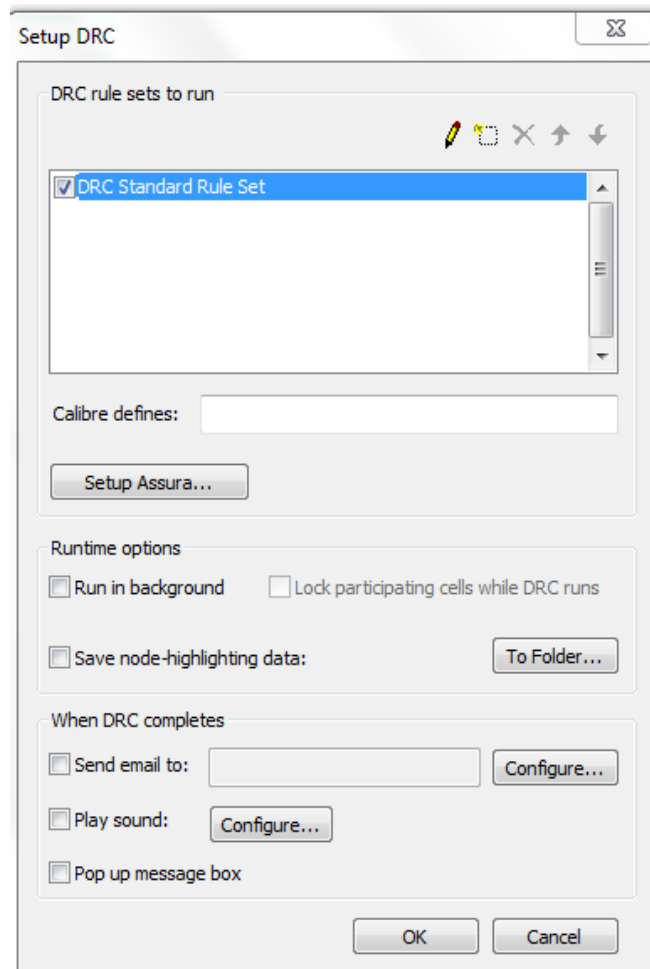


Figure 50: Setup DRC

Design Rule Types

The following are the types of design rules (Figure 51):

- Minimum width.
- Exact width.
- Not exist.
- Overlap.
- Spacing.
- Surround.
- Extension.
- Density.

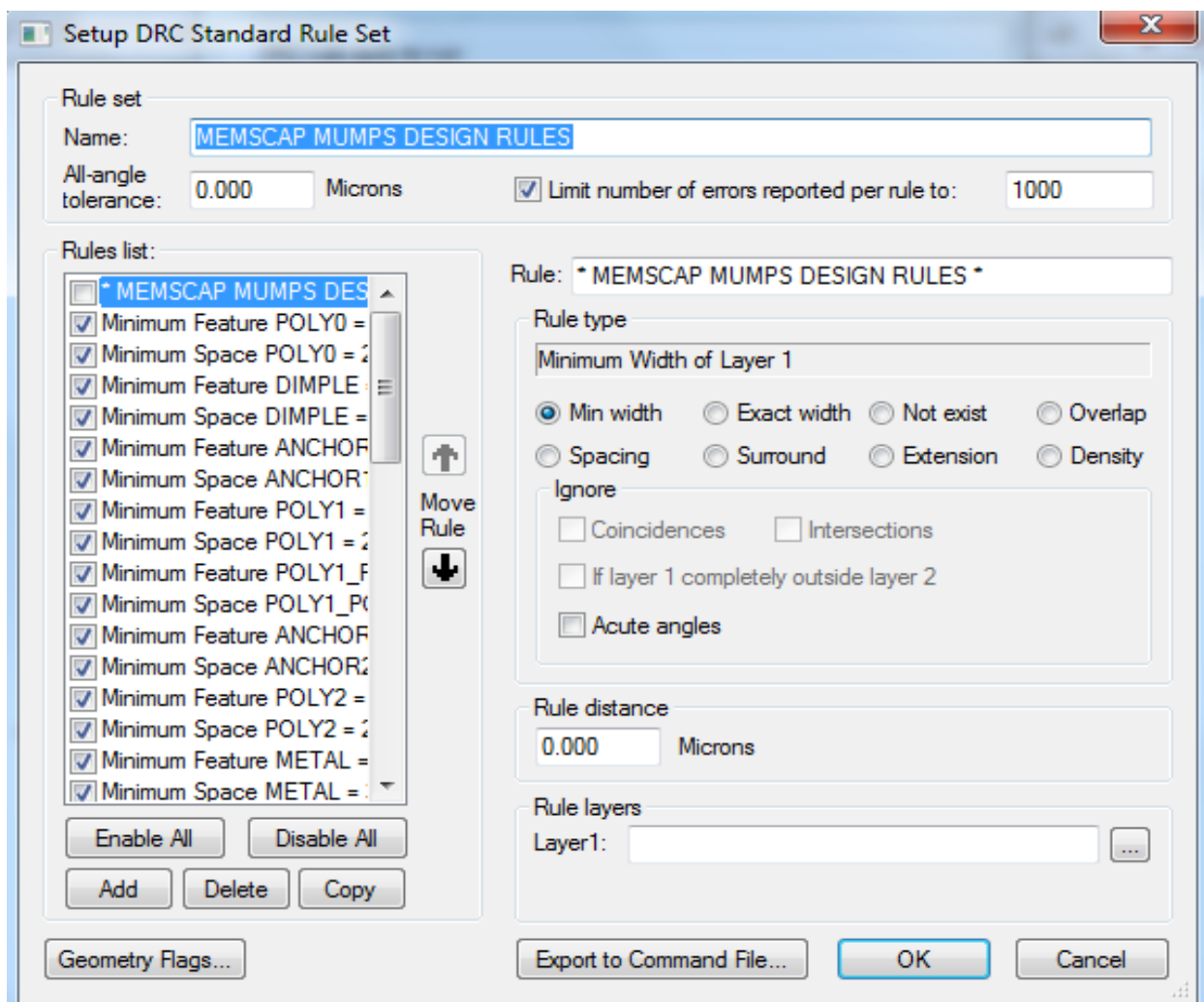


Figure 51: Setup DRC Standard Rule Set

Minimum Width Rule

- The Minimum Width rule specifies the minimum width of all objects, in any direction, on the specified layer, as shown in the figure below.

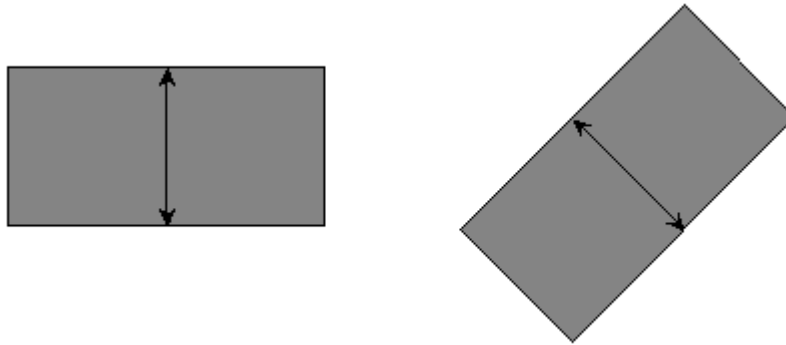


Figure 52: Minimum width of objects.

Exact Width Rule

- The Exact width rule specifies the exact width of all objects on the named layer as shown in the figure.
 - The width of octagons is measured between parallel sides.

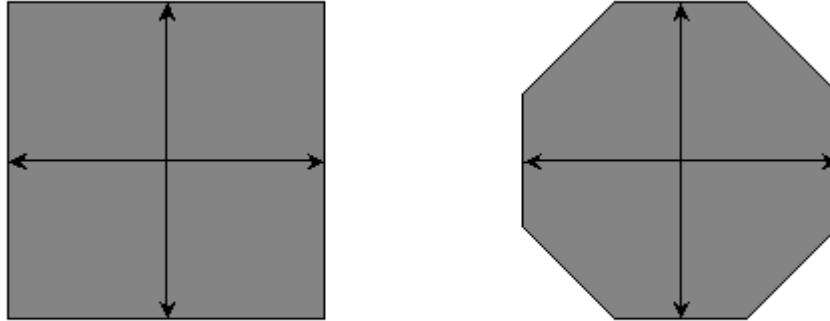


Figure 53: Exact width of objects.

Overlap Rule

- The Overlap rule specifies the minimum amount that an object on one layer must overlap an object on another layer (when there is an overlap).
 - Objects with coincident edges are not considered a violation of overlap rule.

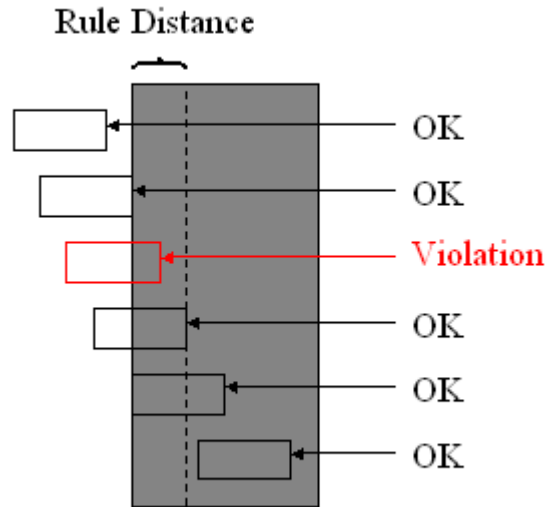


Figure 54: Overlap rule of objects

Spacing Rule

- The Spacing rule specifies the minimum distance that should separate all pairs of objects, either on the same layer or on two different layers (Figure 55).

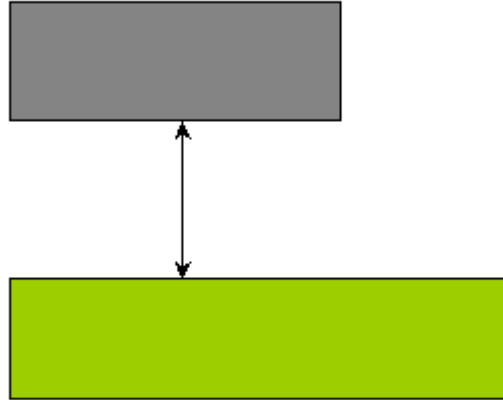


Figure 55: Spacing rule of objects.

Surround Rule

The Surround rule specifies that objects on one layer must be completely surrounded by objects on another layer by a minimum of the specified distance ([Figure 56](#)).

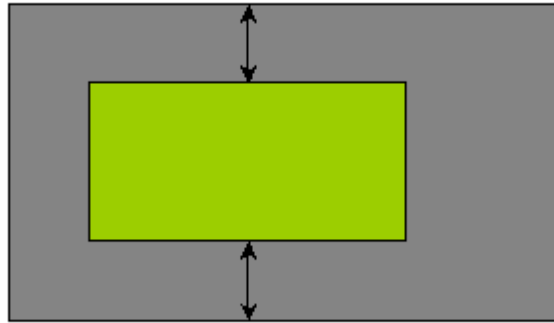


Figure 56: Surround rule of objects

Extension Rule

- The Extension rule specifies the minimum amount that an object on one layer must extend beyond the edge of an object on another layer (Figure 57).
- Objects not considered as violations
 - Have a coincident edge but are otherwise outside.
 - Are entirely surrounded.

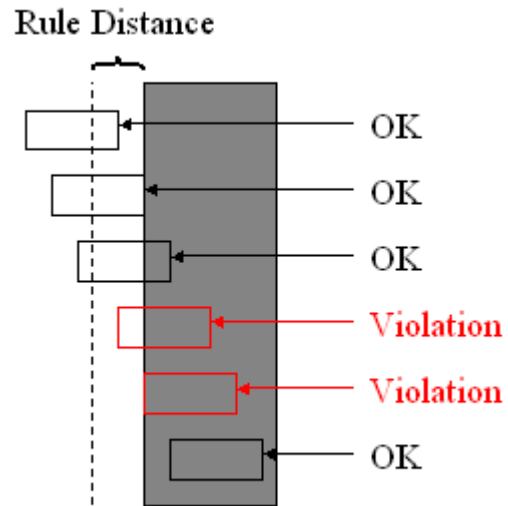


Figure 57: Extension rule of objects

Other Rules

- **Not Exist** rule: specifies that no objects should exist on the named layer.
 - Has no associated distance.
- **Density** rule: checks the percentage of area coverage by objects on a certain layer.

Ignore Options

- Ignore options: conditions where rule violations are NOT reported (rule exceptions).

Ignore Options	Description	Applicable Rules
Coincidences	Rule is ignored if objects have coincident edges	Surround
Intersections	Rule is ignored if objects intersect	Surround
If layer 1 is completely outside layer 2	Rule is ignored if an object on one layer is completely outside another layer	Spacing
acute angles	Portions of objects containing acute angles (<45) are ignored	Minimum width Spacing Surround

DRC Tutorial

- ☑ Launch L-Edit by clicking **Start >All Programs >MEMS Pro V8.0> MEMS L-Edit**.
- ☑ From the L-Edit menu, click **File > Open**.
- ☑ Browse to **{MEMS Pro Installation Folder}\ MEMS_Tutorials\DRC\DRC.tdb**.
- ☑ Click **OK** to open it.

The layout for a pressure sensor is displayed (Figure 58).

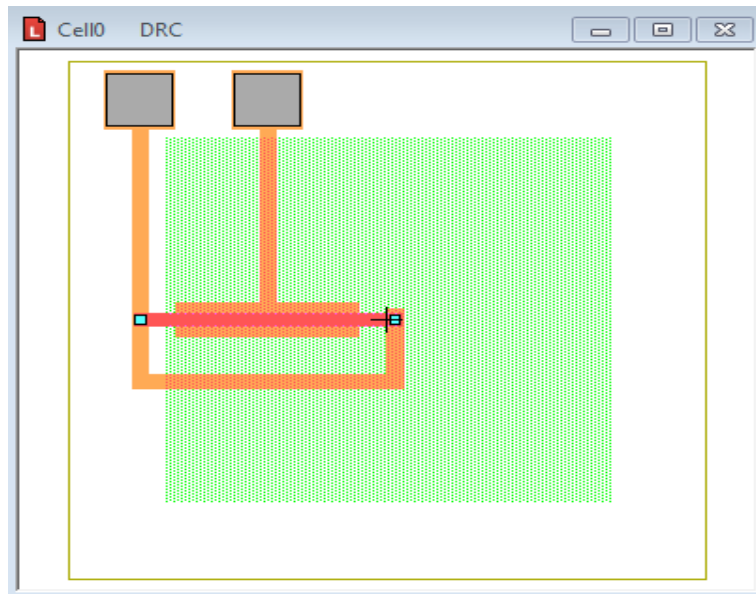


Figure 58: Layout for pressure sensor

Setting Design Rules

- ☑ From the L-Edit menu, click **Tools > DRC Setup**. The **Setup DRC** dialog is displayed.
- ☑ Select the rule set and click the pencil button to open the **Setup DRC** rule set dialog.

The initial step contains only one DRC rule: A. Poly1 encloses anchor (Figure 59).

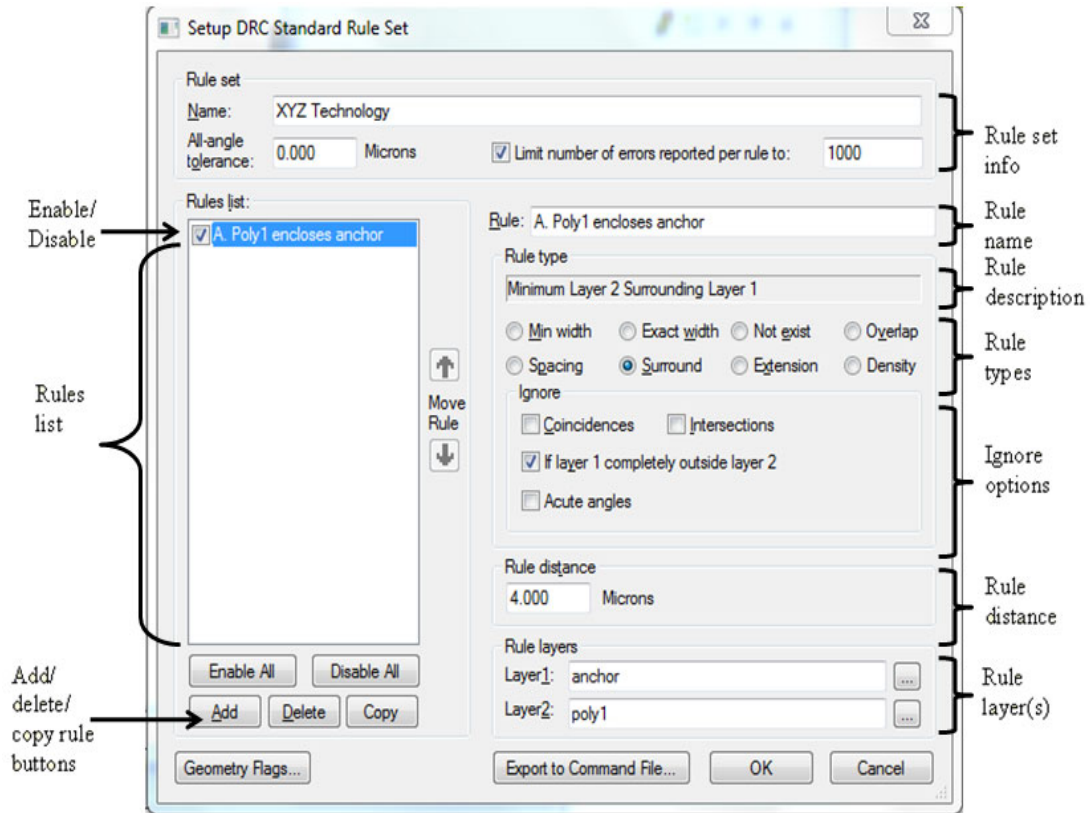


Figure 59: Setup DRC Standard Rule Set window

Rule A: Poly1 Encloses Anchor

- The initial setup contains only one DRC rule: A. Poly1 encloses anchor.
- Anchor layer must be completely surrounded by the Poly1 layer by a minimum of 4 um (Figure 60).

This is to make sure that the structural patterns on the Poly1 layer are anchored correctly, even if the layers are mis-aligned.

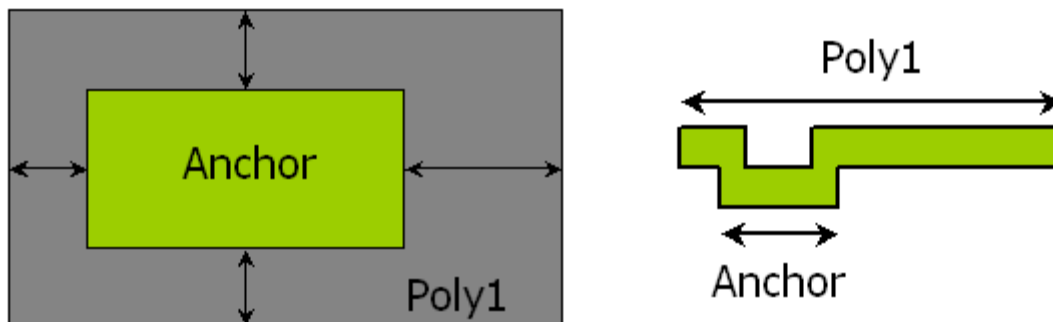


Figure 60: Anchor enclosed by Poly1 layer

- The rule gets ignored if the anchor is completely outside Poly1.

MEMS Tutorial - DRC

- Add rule B to the list.
 - Click the "Add" button, then type in the new values.

	A	B
Rule name	A. Poly1 encloses anchor	B. Poly0 encloses metal
Enable	X	X
Rule Type	Surround	Surround
Coincidences		
Intersections		
Layer1 outside layer2	X	
acute angle		X
Rule distance (ums)	4 (ums)	6 (ums)
Layer1	anchor	metal
Layer2	poly1	poly0

Rule B: Poly0 Encloses Metal

- The metal layer must be completely surrounded by the Poly0 layer by a minimum of 6 μm (Figure 61).

This is to make sure that the contact pads do not extend outside of Poly0.

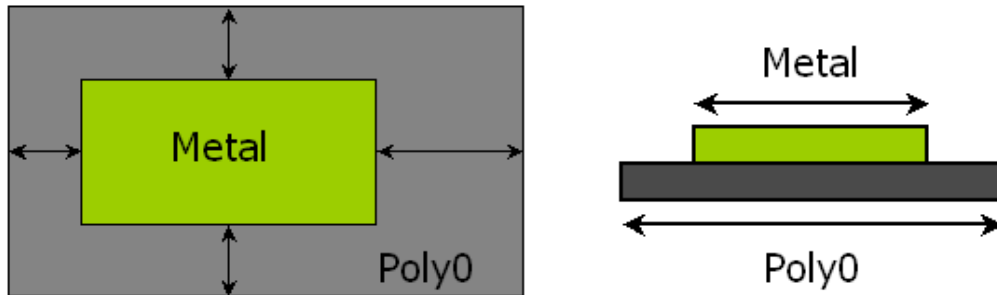


Figure 61: Metal enclosed by Poly0 layer

- The rule becomes ignored in geometries with acute angles.

Running DRC

To run the DRC, perform the following steps:

- ☑ Click **OK** to exit the **DRC** setup.
- ☑ Open **Tools > DRC** from the **L-Edit** menu.
- ☑ The following dialog will be displayed while the **DRC** is running (Figure 62).

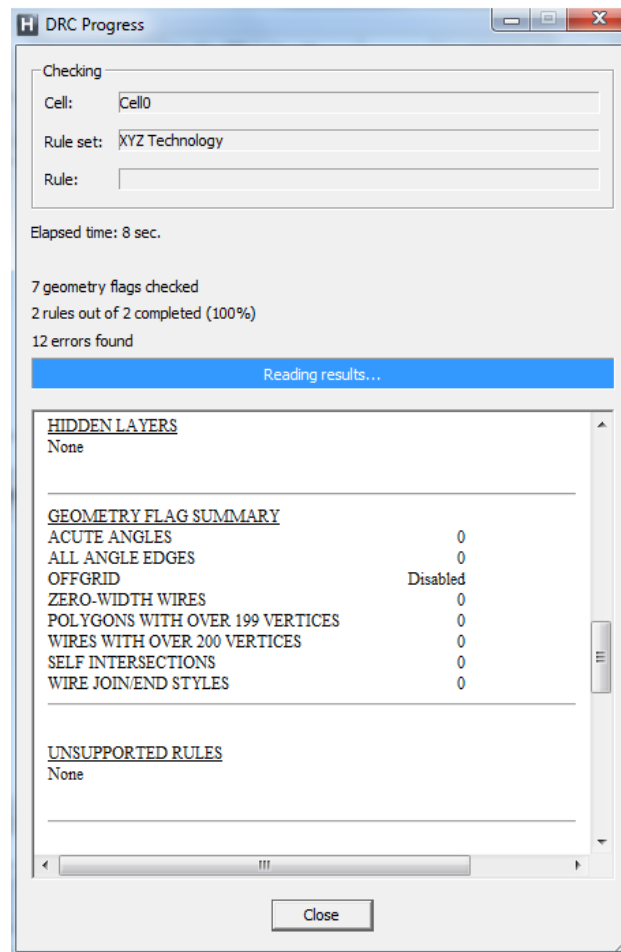


Figure 62: DRC Progress dialog

DRC Navigator

- The **Verification Error Navigator** dialog displays all errors (Figure 63).
 - Standard rules and user-defined rules.

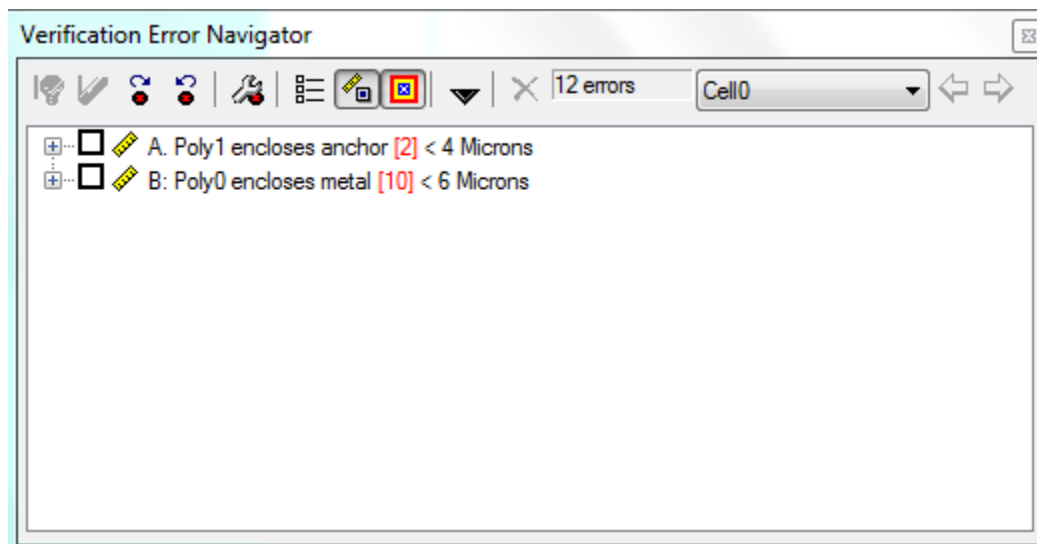


Figure 63: **DRC Error Navigator** dialog.

- The types of errors are listed with the number of occurrences.

Error Navigator

- ☑ To locate errors, expand the tree in the **Verification Error Navigator** dialog (Figure 64).
- ☑ In the rule A errors expand all the way to Error 1 and Error 2.

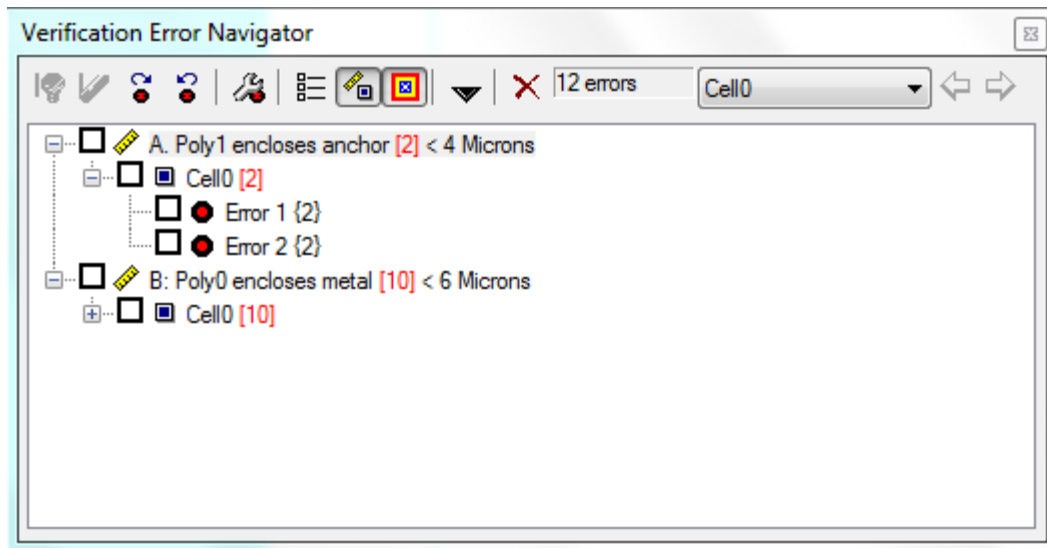


Figure 64: Expanding the tree in **DRC Error Navigator**.

Locating Error A

To locate error A, check the following :

- Rule A: the anchors need to be enclosed by poly1 by more than 4 ums (Figure 65).
 - Layers.
 - Poly1: red.
 - Anchor: light blue.
 - Double click Error 1.
 - Screen zooms to error.
 - Anchor is too close to Poly1.
- Current distance: 2um.
- Rule distance: > 4um.

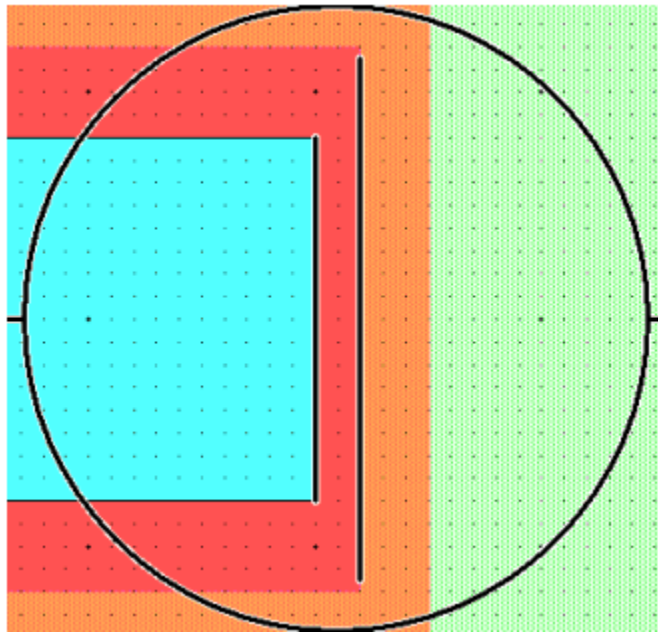


Figure 65: Locating errors

Fixing Error A

To fix error A, perform the following steps:

- ☑ Select the anchor.
- ☑ Drag the edge of anchor 2um to the left, away from edge of poly1, using the middle mouse button.

The final distance should be 4um (Figure 66).

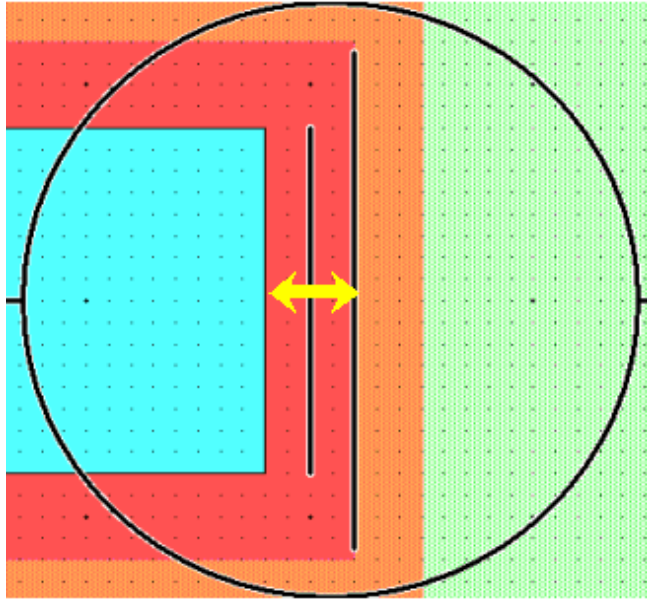


Figure 66: Fixing A errors.

- ☑ Repeat these steps for the other A errors.

Locating Error B

To locate error B, perform the following steps:

- ☑ Expand the rule B errors in the **Verification Error Navigator** dialog.
 - Rule B: the metal pads need to be enclosed by poly0 by more than 6 ums.
 - Layers.
 - Poly0: orange.
 - Metal: gray.
- ☑ Expand rule B.
- ☑ Double click on Error 1.
 - Screen zooms to error.
 - Metal is too close to Poly0.
 - Current distance: 4um.
 - Rule distance: > 6um.

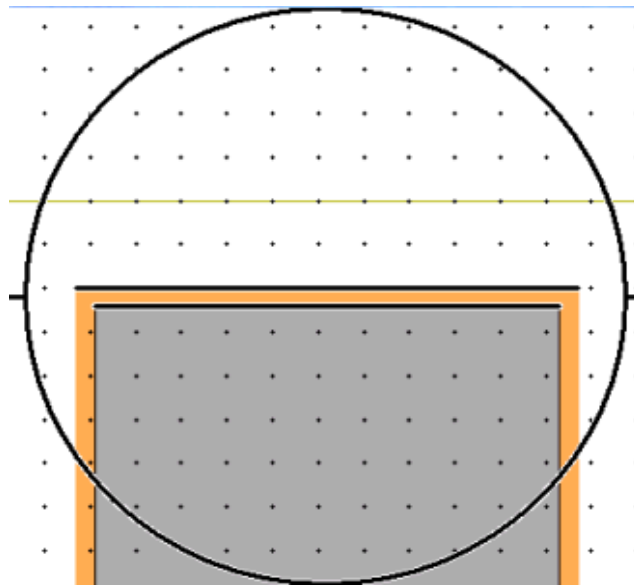


Figure 67: Locating error B

Fixing Error B

To fix error B, perform the following steps:

- ☑ Zoom into the error area.
- ☑ Select the metal.
- ☑ Drag the edge of metal 2um down, away from edge of Poly0, using the middle mouse button.

The final distance should be 6um (Figure 68).

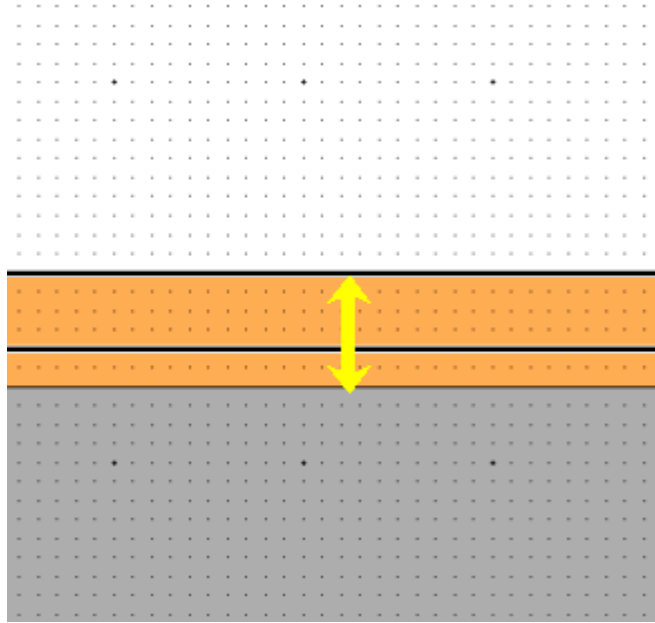


Figure 68: Fixing B errors

- ☑ Repeat these steps for the other B errors.

Re-running DRC

After correcting errors, you need to re-run the DRC.

- From the L-Edit menu, click **Tools > DRC**.

The DRC should return no errors.

5 Optimization Tutorial

- Introduction 96
- Tanner Setting up the Optimization 97
- Examining the Output 105

Introduction


Optimization is a critical tool for the MEMS engineer. The MEMS Pro optimization engine lets you tune the parameters of your system to achieve its best possible performance. Optimization is achieved by running iterative simulations over a constrained set of selected parameters. In order to run an optimization, you must specify a list of parameters, the optimization goal and the analysis and optimization algorithms. Further, you decide which measurements will be used to determine if the optimization has been successful.

Once you have successfully run an optimization, the optimized parameter values can be used in subsequent analyses of the same model. This allows for incremental optimization: some parameters can be optimized while others are held fixed; later, other parameters can be optimized based on the results of the earlier optimization. If multiple analyses are requested, DC analyses will be performed first, then AC analyses and then transient analyses. Multiple analyses of the same type are performed in the order they appear in the input file.

The optimization process is most easily explained by going through some examples. The provided example is a simple optimization with a single parameter, single measure, single analysis and a single goal.

Tanner Setting up the Optimization

If you recall, in the [MEMS Pro Tutorial on page 5](#), you followed the construction and analysis of a resonator. Now you will be exploring that model further. The MEMS Pro optimizer will help you find the value of the **springlength** parameter that will most closely achieve the optimization goal: a resonant frequency of 40 kHz.

- Launch MEMS S-Edit by double-clicking on its icon .
- Select **File > Open > Open Design** and choose the **resonator.tanner** file located under the **{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\Optimize\Resonator\Resonator.tanner**.

The schematic view of the resonator appears in the S-Edit window ([Figure 69](#)).

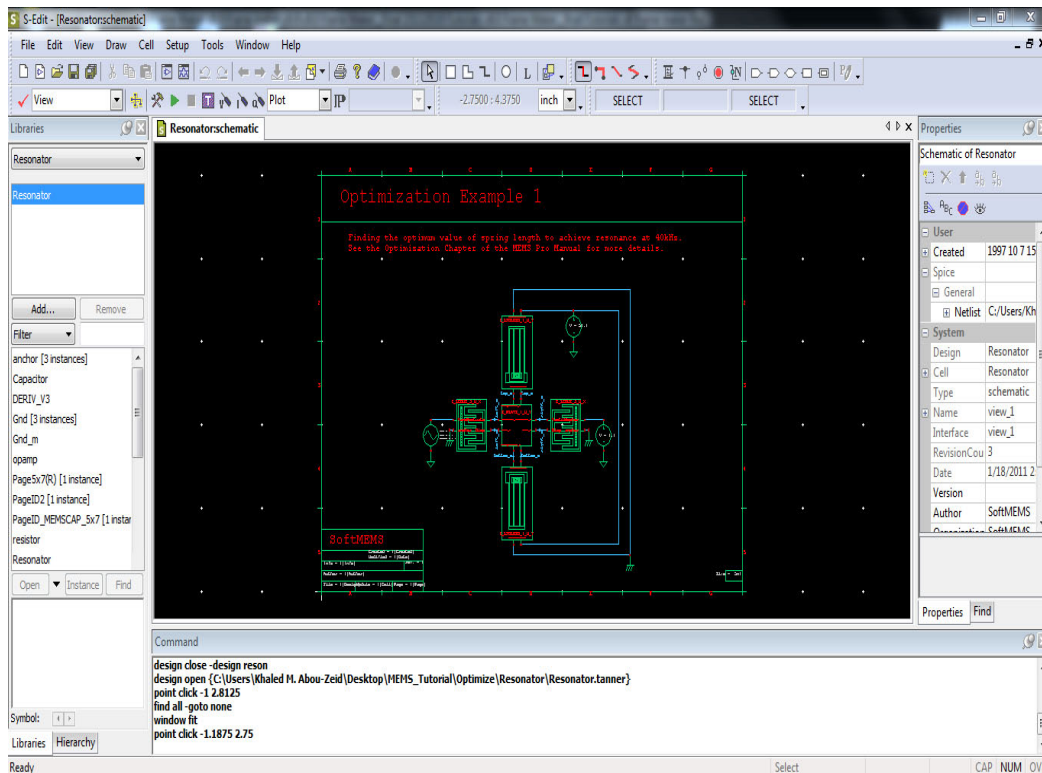


Figure 69: Viewing the resonator in S-Edit

- Verify that the correct path is written in the **SPICE File Name** under **Setup > SPICE Simulation**, which is **"{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\Optimize\Resonator\TC_Reson_optimize"**
- Click the **T-Spice** icon to create the netlist corresponding to the resonator and open it in T-Spice.

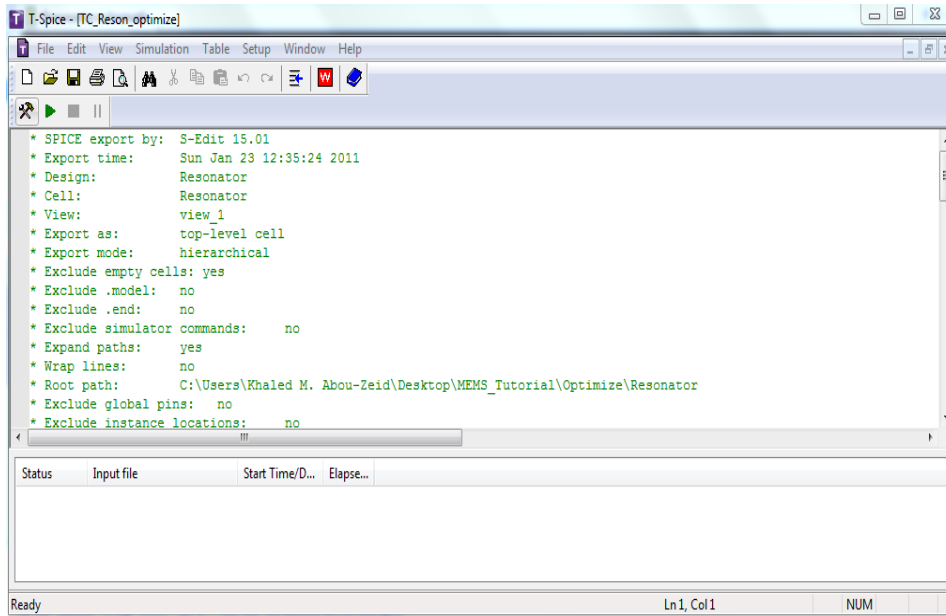


Figure 70: Viewing the generated netlist

Now, you need to associate process and material properties to the netlist.

- ☑ Select the **Insert Command** button to invoke the **T-Spice Command Tool** dialog (Figure 71).

Note Before inserting commands, make sure that your cursor is located at the end of the netlist.

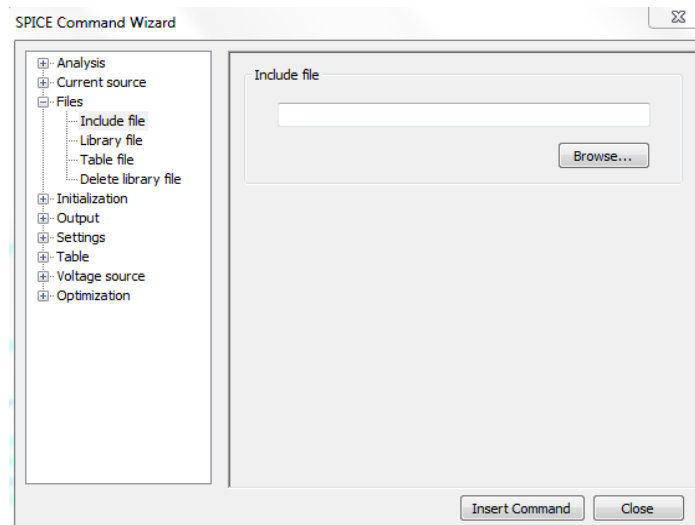


Figure 71: T-Spice Command Tool dialog

- ☑ In the left-hand tree, select **Files > Include file**. Browse for the *process.sp* file located under **<MEMS Pro v8.0 Installation Directory>** and click **Insert Command**.
- ☑ Click **Close** to close the **Insert Command** dialog, and press the **Enter** key to add a new line.

You must specify the analysis you will use for the optimization engine to determine whether you have reached an optimization goal.

- ☑ Select the **Insert Command** icon again.
- ☑ In the left-hand tree, select **Analysis > AC**.
- ☑ Choose **decade** for **Frequency sampling type**, set **Frequencies per decade** to **500**, **Frequency range From** to **10k** and **Frequency Range To** to **100k** (Figure 72).
- ☑ Click **Insert Command**.
- ☑ Click **Close** to close the dialog, then press **Enter**.

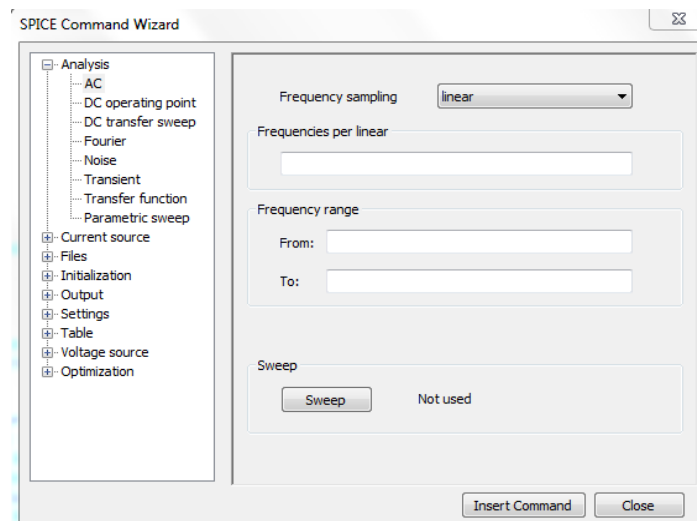


Figure 72: Customizing the AC analysis

Next, you need to specify the parameters of interest in the model.

- ☑ Invoke the **T-Spice Command Tool** dialog box again.
- ☑ In the left tree, double-click **Settings > Parameters**. Leave the "Parameter Type" field set to "General". Set **Parameter name** to **springlength**, **Parameter value** to **100e-6**, then click **Add** (Figure 73).

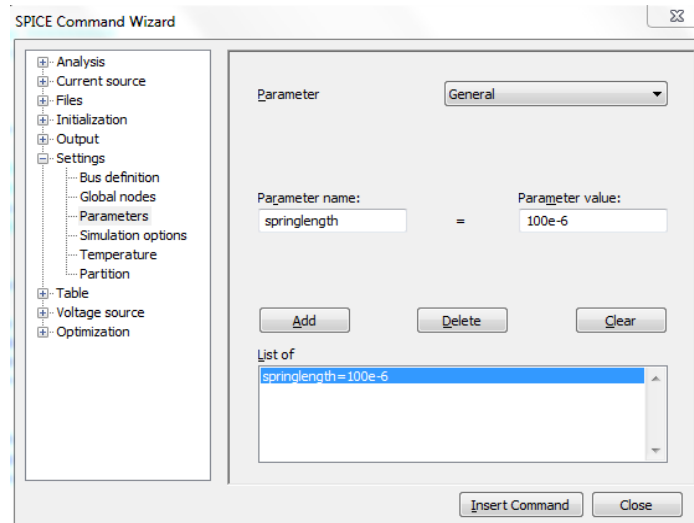


Figure 73: Customizing the setting parameters

- ☑ Click **Insert Command**.

Now, you will define the quantities that will be measured during the simulation.

- ☑ Click **Close** to close the dialog, then press **Enter**.
- ☑ Invoke the **T-Spice Command Tool** dialog box.
- ☑ In the left tree, double-click **Output** and then **Measure**.
- ☑ Set **Analysis type** to **AC** and **Measurement type** to **Find-when**. Enter **res_freq** in the **Measurement result name** field. Under **When**, select **Signal**. Then, under **Find**, click the **x-value** radio button. Then, under **When** again, enter **vp(right_m)**. Set **equals value** to **-90**. From the drop-down menu next to **on** select **crossing**. For **number**, select **1** (Figure 74) and click **Insert Command**.
- ☑ Click **Close** to close the **Insert Command** dialog and press the **Enter** key to add a new line.

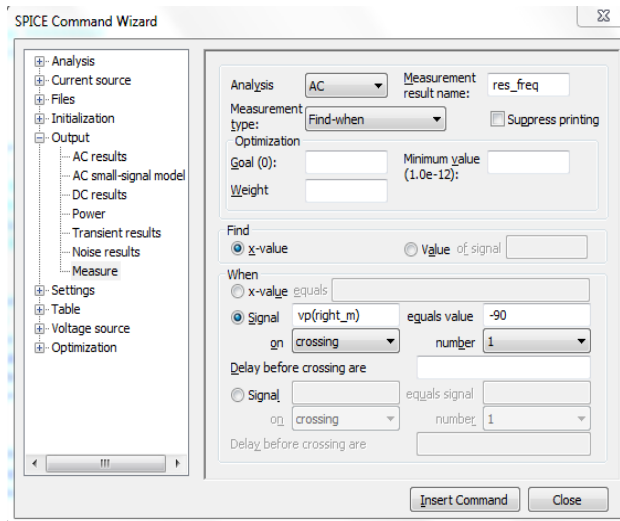


Figure 74: Customizing the quantities to measure

Now, you are ready to set up the optimization.

- ☑ Click the **Insert Command** icon again.
- ☑ In the left-hand tree, double-click **Optimization** (Figure 75).

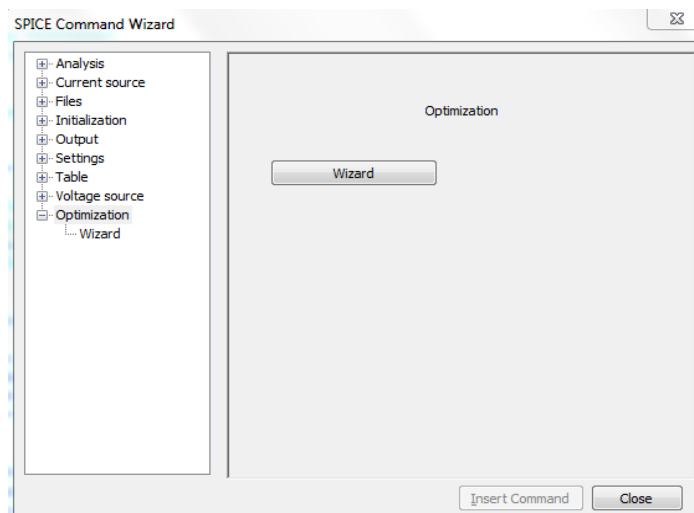


Figure 75: Customizing the optimization

- ☑ Click **Wizard** in the left tree or the **Wizard** button on the right to bring up the **Optimization setup** dialog.
- ☑ Enter **opt1** in the **Optimization name** field and select **First AC Analysis** as the **Analysis name** (Figure 76).

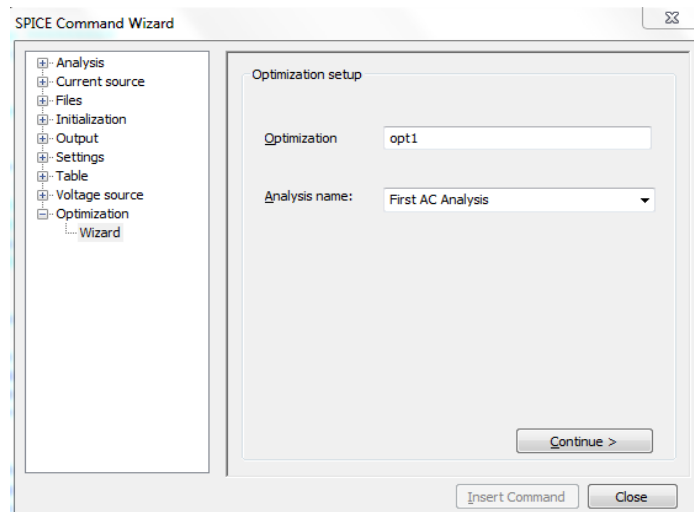


Figure 76: Customizing the optimization setup

- ☑ Click **Continue** to access the next dialog, **Set optimization goals**.
- ☑ Set **Measurement** to **res_freq** and **Target value** to **40e3**. Click **Add** to add these values to the **List of optimization goals** (Figure 77).

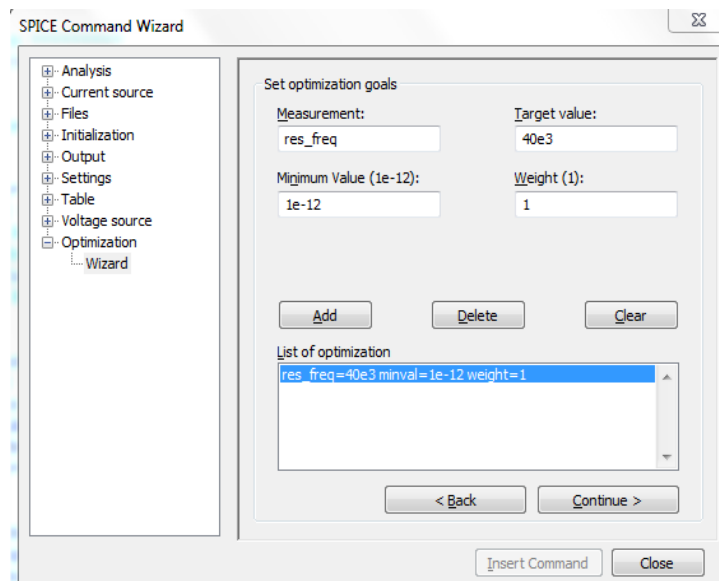


Figure 77: Customizing the optimization goals

Note

This target value will overwrite the one set earlier during the measurement setup.

- ☑ Click **Continue** to access the next dialog, **Set parameter limits**.
- ☑ Set the **Parameter name** to **springlength**, **Minimum value** to **10e-6**, **Maximum value** to **200e-6**, **Delta (Optional)** to **0.25e-6** and **Guess value (Optional)** to **100e-6** (Figure 78).
- ☑ Click **Add** to add the values to the **List of parameters**.

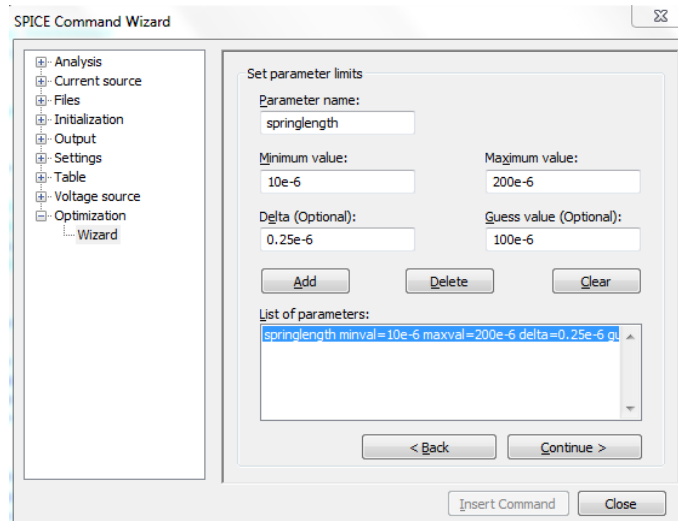


Figure 78: Customizing the parameters limits

- ☑ Click **Continue** to access the next dialog, **Set optimization algorithm**.
- ☑ In the **Name** field, type **optmod**. Accept the default values for the other parameters (Figure 79).

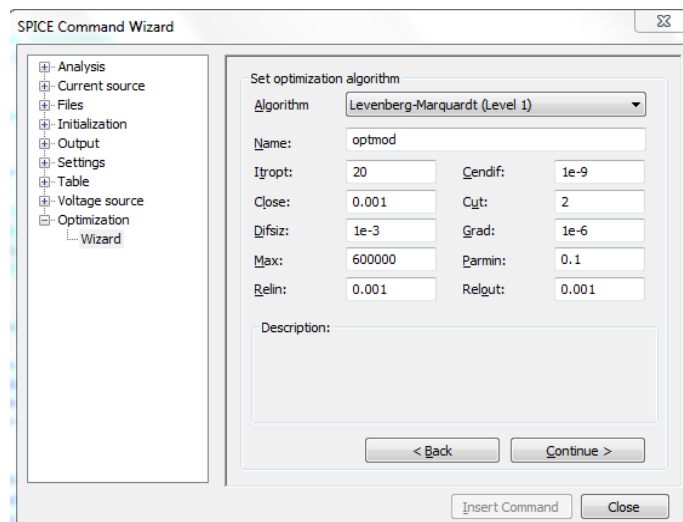


Figure 79: Customizing the optimization algorithm

- ☑ Click **Continue** to access the next dialog, **Insert command**.

The optimization commands you have specified are displayed in the dialog. Make sure that your entries are correct. If you need to change a line, click **Back** to make changes. If the commands are correct, click **Insert Command**, then **Cancel** to close the dialog and press **Enter**.

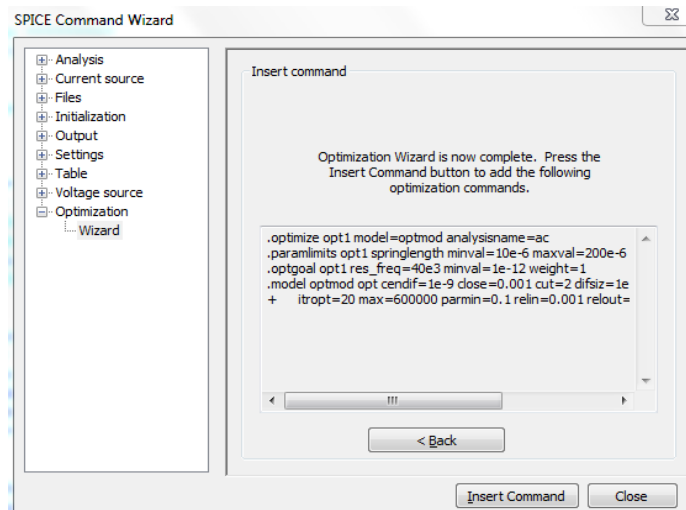


Figure 80: Finalizing the optimization

- ☑ Click the **Save** icon to save your netlist.
- ☑ Run the optimization by selecting **Simulation> Run Simulation**.


Examining the Output

The optimization engine output file contains the results of each simulation iteration. The value of each optimization parameter (submitted for a given run) appears followed by the gradient of the objective function at that parameter value. In this example, only one parameter, **springlength**, is of interest. The next line contains the residual, or the difference between the output measure and your goal. The Levenberg-Marquardt algorithm is used for the optimization; the Marquardt value is an artifact of that algorithm.

Once the optimization engine produces a result that falls within the tolerance you have set, it desists. The final parameter estimate, **Optimized parameter values: springlength = 7.425e-005** and the goal measure, **Measurement result summary - OPTIMIZE=opt1 res_freq = 3.9906e+004**, appear in the output file.

The optimization output file appears below:

Optimization
Results



```

Optimization parameters:
  springlength = 0.0001           derivative = 2.42866

Optimization initialization: resid=0.36174 grad=2.42866 Marquardt=0.001

Optimization parameters:
  springlength = 6.65e-005       derivative = -2.20566

Optimization iteration 1: resid=0.17744 grad=2.20566 Marquardt=0.0005

Optimization parameters:
  springlength = 7.425e-005      derivative = 0.0498307

Optimization iteration 2: resid=0.00235139 grad=0.0498307
  Marquardt=0.00025
Optimization iteration 3: resid=0.00235139 grad=0.0498307 Marquardt=0.001
Optimization iteration 4: resid=0.00235139 grad=0.0498307 Marquardt=0.004

Optimization parameters:
  springlength = 7.425e-005      derivative = 0.0498307

Optimization iteration 5: resid=0.00235139 grad=0.0498307 Marquardt=0.002

Optimized parameter values:
  springlength = 7.4250e-005

Measurement result summary - OPTIMIZE=opt1
  res_freq = 3.9906e+004

```

```

Parsing                1.64 seconds
Setup                  0.01 seconds
.AC Sweep              0.87 seconds
-----
Total                  2.52 seconds

```


6 Verification Tutorial

▪ Introduction	108
▪ Adding Connection Ports	109
▪ Extracting Layout	111
▪ Extracting Schematic for LVS	114
▪ Comparing Netlists	116

Introduction

This chapter explains how to verify a mixed technology layout by showing the processes of layout extraction and layout vs. schematic comparison. This chapter contains a tutorial on these important design steps. The tutorial is a continuation to the main MEMS Pro tutorial of [Chapter 1 - MEMS Pro Tutorial](#).

Adding Connection Ports

We will go through this tutorial using the design you completed in [Chapter 1 - MEMS Pro Tutorial](#). Please, open it in L-Edit.

- ☑ Launch L-Edit by double clicking the **L-Edit** icon and select **File > Open** to load the design file you created in [Chapter 1 - MEMS Pro Tutorial](#), called *myreson.tdb*.

As long as geometrical objects on the same layer touch or overlap, they will be fabricated as connected. However, for SPICE netlist extraction to work correctly, the connection must be explicitly stated. These connections are called ports, and they define connectivity for a cell. Ports are used specifically for interconnection and they are objects drawn with the **Port** tool on a layer.

Ports allow L-Edit's **Extract** command to recognize connectivity at a block or circuit level. The cells that were generated automatically for this design (the **plate**, **comb-drive**, **folded spring**, **ground plate** and **bonding pad**) already have properly drawn ports in place. You can examine the port connections in the example *Resonator* by choosing **Edit > Find** to find objects of type port.

The location of ports in each of the layout cells is described and illustrated in [Figure 81](#).

- The **plate** has 4 ports (*PL_Left*, *PL_Right*, *PL_Top*, *PL_Bottom*). They look like long rectangles (2 units thick) stretching across the length of each of the four sides of the plate.

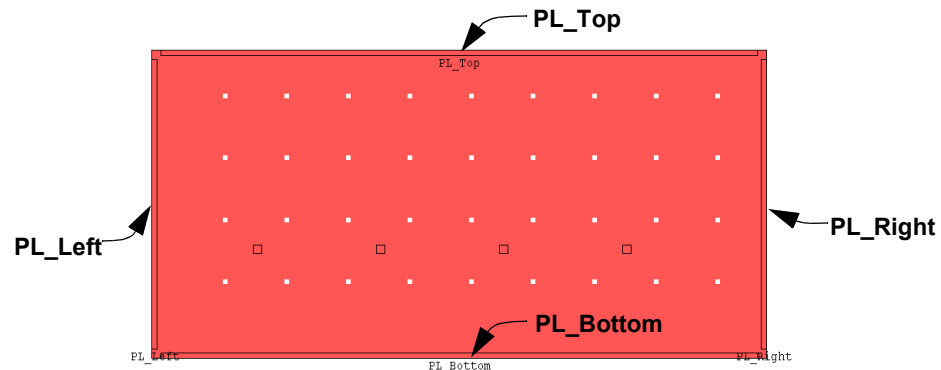



Figure 81: Ports of the plate element

- The **comb-drive** has 2 ports (*C_Free*, *C_Fixed*). They look like long rectangles (2 units thick) stretching across the left and right sides of the element.
- The **folded spring** has 2 ports (*FS_Free*, *FS_Fixed*). *FS_Free* looks like a rectangle stretching across the bottom of the element. *FS_Fixed* looks like a rectangle attached to the right side of the element overlapping the anchor point.
- The **ground plate** has 1 port (*GP_GND*). It looks like a rectangle overlapping the entire ground plate.
- The **bonding pad** has 2 ports (*P_GND*, *P_MTL*). *P_GND* looks like a long rectangle (2 units thick) stretching across the left side of the pad. *P_MTL* looks like a long rectangle stretching across the right side of the pad.

Begin by connecting the *PL_Left* and *PL_Right* ports of the plate to the *C_Free* ports of the two combs.

- ☑ Choose the **Box** tool  and select the *Poly1* layer from the **Layer Palette** by clicking it.

- ☑ Draw a box covering the **PL_Right** port of the plate and the **C_Free** port of the right comb. Click once to set the lower left corner, hold the mouse button down, drag to the opposite corner, and release.
- ☑ Choose the **Port** tool  and click anywhere on the box drawn in the previous step.
- ☑ In the **Edit Object(s)** window that appears, select the **Poly1** layer in the **On Layer** drop-down menu, enter **Right** in the **Port Name** field, and click **OK**.
- ☑ Select the **Box** tool and the **Poly1** layer from the **Layer Palette** to draw another box covering the **PL_Left** port of the plate and the **C_Free** port of the left comb. Click once to set the lower left corner, hold the mouse button down, drag to the opposite corner of the box, and then release.
- ☑ Using the **Port** tool, place a port on the **Poly1** layer anywhere on the box drawn in the previous step.
- ☑ In the **Port Name** field, enter **Left** and click **OK**.

You will now connect the **PL_Top** and **PL_Bottom** ports of the plate to the **FS_Free** ports of the two springs.

- ☑ Connect the top folded spring to the plate by drawing a box on **Poly1** covering the **FS_Free** port of the spring and the **PL_Top** port of the plate.
- ☑ Place a port on the **Poly1** layer anywhere on the box you drawn and name it **Top**.
- ☑ Connect the bottom folded spring to the plate by drawing a box on **Poly1** covering the **FS_Free** port of the spring and the **PL_Bottom** port of the plate.
- ☑ Place a port on the **Poly1** layer anywhere on the new box and name it **Bottom**.

Now, you will connect the **FS_Fixed** ports of the two springs to the **GP_GND** port of the ground plate.

- ☑ Connect the **FS_Fixed** ports of the two folded springs to the ground plate by drawing a box on the **Poly0** layer covering the **FS_Fixed** port of each spring and the **GP_GND** port of the ground plate. This box should cover all of the ground plate.
- ☑ Place a port on the **Poly0** layer anywhere on the newly drawn box and name it **SpringAnchor**.

Finally, you will place nodes on the fixed ports of the comb drives:

- ☑ Place a port on the **Poly0** layer anywhere on the **Poly0** box drawn connecting the **C_Fixed** port of the right comb to the right pad. Name it **Rfixed**.
- ☑ Place a port on the **Poly0** layer anywhere on the **Poly0** box drawn connecting the **C_Fixed** port of the left comb to the left pad. Name it **Lfixed**.

Now, all the connections will be properly recognized by L-Edit/ Extract.

Extracting Layout

Layout extraction produces a SPICE netlist consisting of device and connectivity information used for comparing layout vs. schematic (LVS) or SPICE simulations. Design rule checking (DRC) ensures that a layout conforms to fabrication process restrictions, but it does not verify that the layout actually implements what was intended, nor does it assist in determining whether the system will perform according to the specifications.

Extracting MEMS designs involves the use of the MEMS Pro feature called *subcircuit extract*. Subcircuit extract involves the extraction of subcircuit cells as *black boxes* with connection ports and cell properties.

To run **Extract** you must first load and select the extract definition files you want to run.

- ☑ Select **Tools > Extract Setup** to invoke **Setup Extract** dialog.

An **extract definition file** must be loaded to provide technology information about your design. It contains a list of the connections and devices to be extracted.

- ☑ Click the **Browse** button and use the windows browser to select **polymumps.ext** from the **MEMS_Tutorials\Resonator** directory and enter **MyResonator.spc** as the output file name.

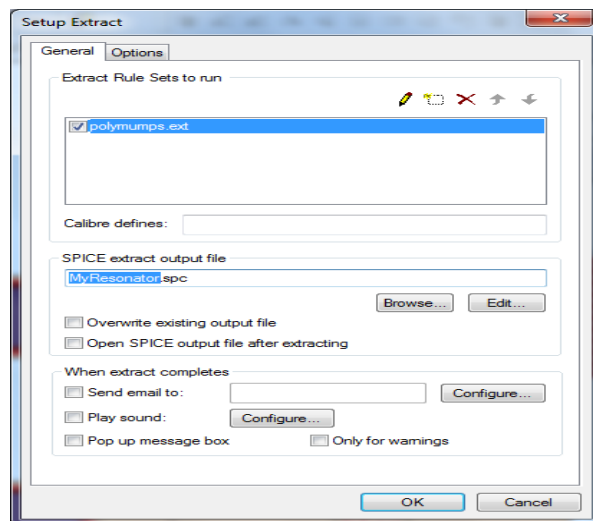


Figure 82: Selecting the extract definition file

- ☑ Click the **Options** tab.

Note

You may need to refer to [Viewing Properties on page 28](#) to change the extracted properties of each subcircuit (model names, pin order) in order for it to be compatible with S-Edit.

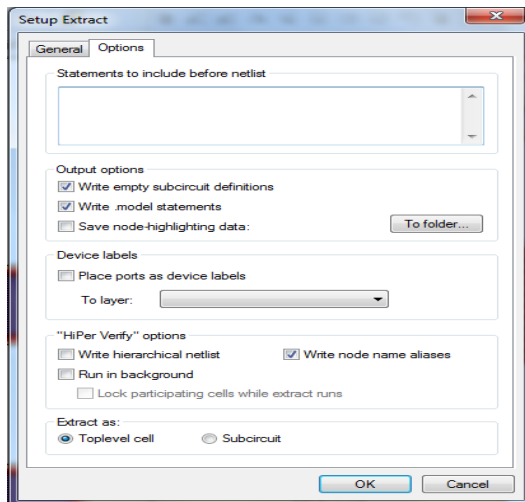
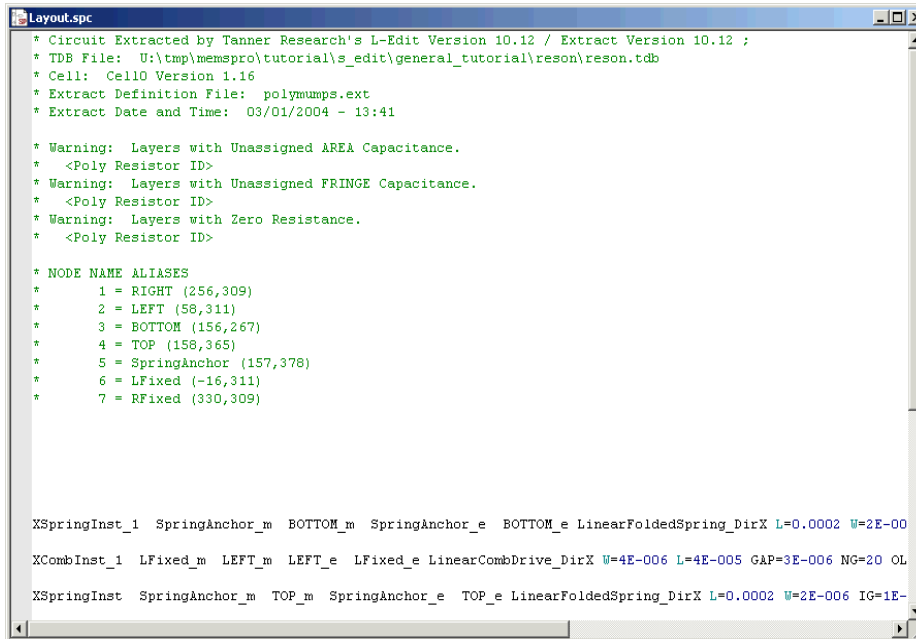


Figure 83: **Options** tab of the **Extract** dialog box

- ☑ Select the **Subcircuit** in **Extract as**.
- ☑ Click on **OK** button.
- ☑ Select **Tools > Extract** to begin the layout extraction.

After selecting **Extract**, a netlist file called **MyResonator.spic** will be created. This is a text file in SPICE format containing the devices extracted, their connectivity and device geometrical parameter information. The netlist file can be used to run T-Spice simulations or to perform layout versus schematic verification (LVS).

- ☑ When the **L-Edit Warning** dialog appears, click the **Ignore All** button.
- ☑ Open the **MyResonator.spic** file by selecting **File > Open**. In the **Open** dialog box, change the **File Type** to **Spice Files (*.sp, *.spic)**, select **MyResonator.spic** from the file list, then click **OK**. A text file is opened in L-Edit containing the file contents. Examine the extracted file by using the scroll bar on the text window.



```

* Circuit Extracted by Tanner Research's L-Edit Version 10.12 / Extract Version 10.12 ;
* TDB File:  U:\tmp\memspro\tutorial\s_edit\general_tutorial\reson\reson.tdb
* Cell:  Cell10 Version 1.16
* Extract Definition File:  polymumps.ext
* Extract Date and Time:  03/01/2004 - 13:41

* Warning: Layers with Unassigned AREA Capacitance.
* <Poly Resistor ID>
* Warning: Layers with Unassigned FRINGE Capacitance.
* <Poly Resistor ID>
* Warning: Layers with Zero Resistance.
* <Poly Resistor ID>

* NODE NAME ALIASES
*   1 = RIGHT (256,309)
*   2 = LEFT (58,311)
*   3 = BOTTOM (156,267)
*   4 = TOP (158,365)
*   5 = SpringAnchor (157,378)
*   6 = LFixed (-16,311)
*   7 = RFixed (330,309)

XSpringInst_1 SpringAnchor_m BOTTOM_m SpringAnchor_e BOTTOM_e LinearFoldedSpring_DirX L=0.0002 W=2E-00
XCombInst_1 LFixed_m LEFT_m LEFT_e LFixed_e LinearCombDrive_DirX W=4E-006 L=4E-005 GAP=3E-006 NG=20 OL
XSpringInst SpringAnchor_m TOP_m SpringAnchor_e TOP_e LinearFoldedSpring_DirX L=0.0002 W=2E-006 IG=1E-

```

Figure 84: Viewing the extracted file

- Select **File > Exit** to exit L-Edit.

Extracting Schematic for LVS

To export a schematic netlist for use in LVS, the schematic must contain only the device components and be free of all stimuli and simulation commands. You will now re-open the tutorial file.

- ☑ Open **MEMS S-Edit**.
- ☑ Select **File > Open > Open Design** to open the *reson.tanner* file located under {MEMS Pro installation directory}\MEMS_Tutorials\Resonator\reson directory.
- ☑ Select **Cell > Open view** to open the *schemlvs* cell.

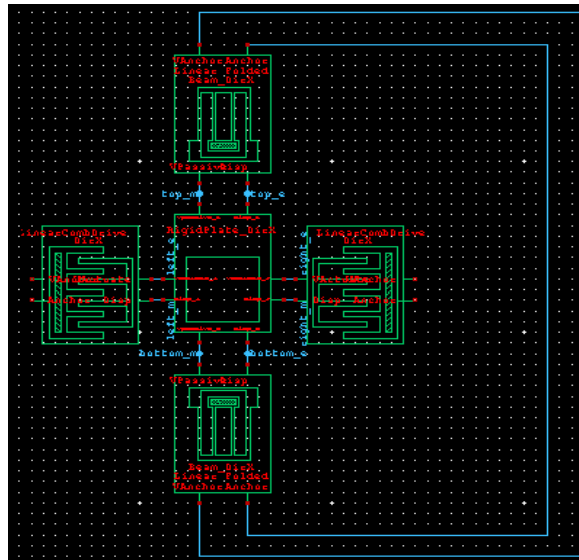


Figure 85: Schematic view of the resonator

- ☑ Select **File > Export > Export SPICE** to invoke the **Export Netlist** dialog.

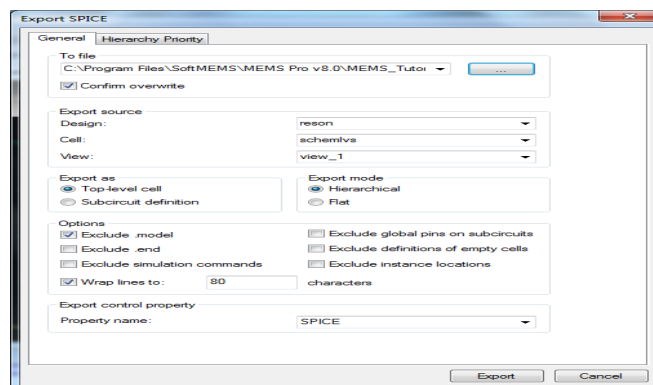


Figure 86: **Export Netlist** dialog


Clicking **Export** will save the setup information, run the extraction and create an output netlist file that is preloaded with the module name. In this case, the file name will be *schemlvs.sp*. This is a text file in SPICE format that contains device descriptions, their connectivity and geometrical

parameter information. The netlist file can be used to perform layout versus schematic verification (LVS).

- Select **File > Exit** to exit S-Edit.

Comparing Netlists

An important step in the MEMS design process involves comparing the layout and the schematic to ensure that they describe the same system. This is *layout versus schematic comparison*, performed by comparing two *netlists* – one derived from the layout and one from the schematic.

- ☑ Double-click the **LVS** icon  to launch LVS.
- ☑ Select **File > Open** to open the **reson.vdb** file under the **MEMS_Tutorials\Resonator** directory. The file contains predefined parameters to compare the SPICE files you created.

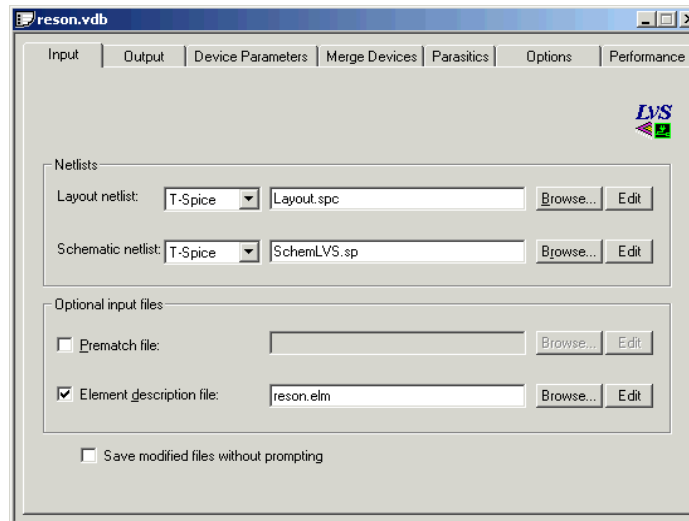



Figure 87: Viewing the preset parameters of the reson.vdb file

- ☑ Click the **Run** button  located on the toolbar to launch the comparison. The verification window will appear displaying the results.

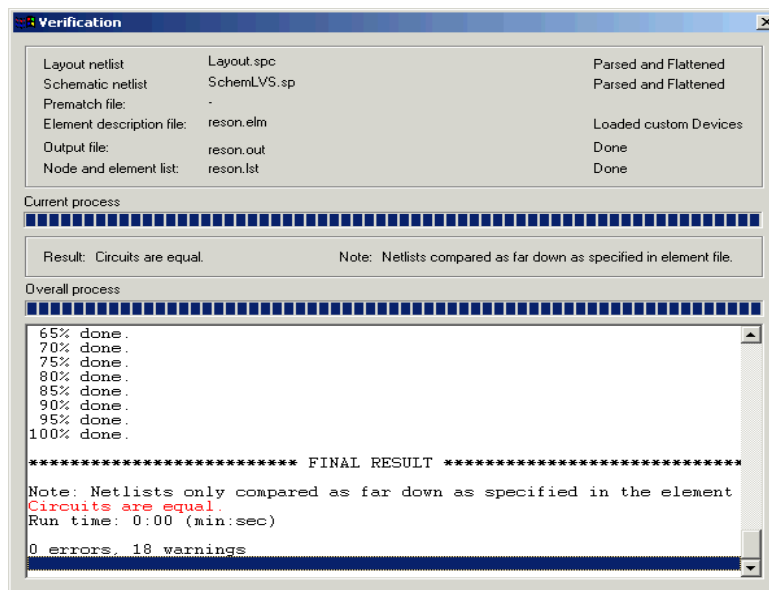


Figure 88: Verification window

The netlist generated from the schematic and the netlist generated from the layout are identical.

- Select **File > Exit** to exit LVS.

7 External Model Generator

- Introduction 120
- Creating the Model 121
- Generating the Model 124
- Instantiating the Model in a schematic 125
- Instantiating the Model in the circuit 127
- Simulating the circuit 128

Introduction

The aim of this tutorial is to create and simulate a model for a comb drive actuator generated using the **External Model Generator** tool.

Start by launching the External Model Generator by selecting **Start > All Programs > MEMS Pro v8.0 > External Model Generator**.

You can either enter the model information or load it from the

{**MEMS Pro v8.0 Installation Directory**}\MEMS_Tutorials\ModelGen\resonEMG\combDrive.emg and follow the tutorial from section **Generating the Model** on page 124

Creating the Model

- ☑ On the **Pins/Parameters** tab click on the **Add** button under the Pin list and type in the name of the first pin "anchor". Set the pin type to "Gnd" by double clicking the pin type field.
- ☑ Similarly add the following pins:

Pin Name	Pin Type
vactuate	Input/Output
actuate	Input/Output
vanchor	Gnd
anchor	Gnd

Note

Make sure to enter the pins in the same order listed above.
You can drag and drop the pins to rearrange them.

- ☑ Under the **Parameters** list click the **Add** button and add the following parameters with their corresponding parameter type and default values.

Parameter Name	Parameter Type	Default Value
combFingers	Integer	20
fingerGap	Double	3E-06
fingerThickness	Double	2E-06
fingerOverlap	Double	3E-05

- ☑ Click on the **Equation Editor** tab to start entering the governing equations:

The pins and parameters entered in the previous step should be listed in the corresponding lists (Figure 89).

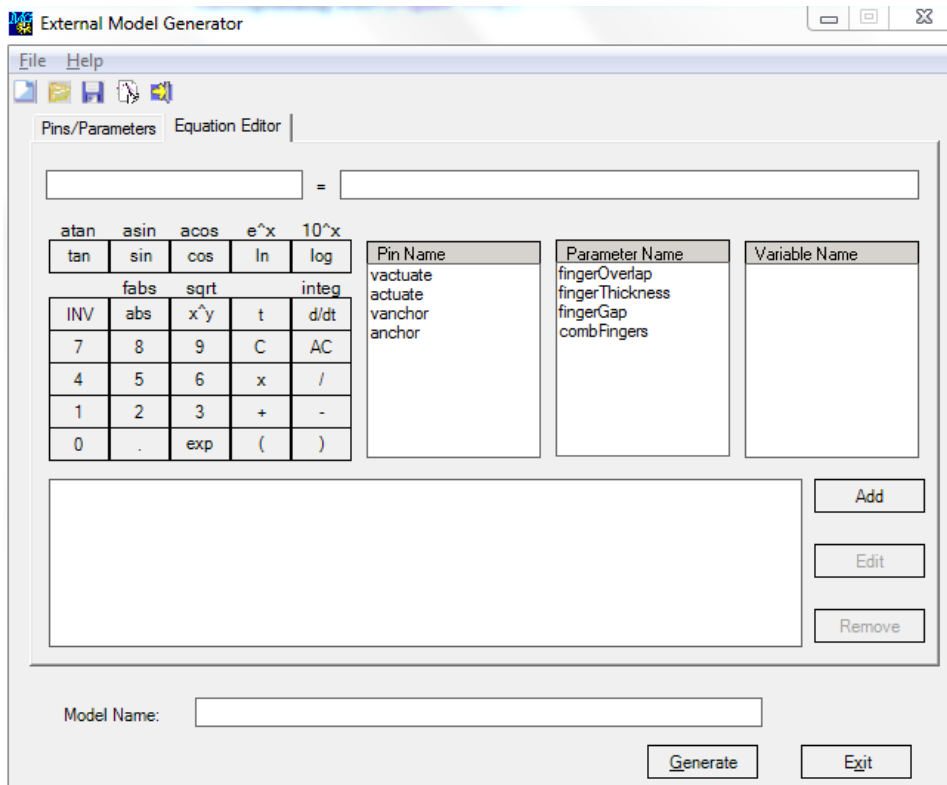


Figure 89: Pins and Parameters in the Equation Editor

We will start by calculating the initial capacitance between two opposite fingers:

$$c0 = \text{fingerThickness} * 8.854e-12 / \text{fingerGap}.$$

- Start by typing on the left hand side of the equation the variable "c0".
- From the parameter list double click on the "fingerThickness parameter". The parameter will be automatically placed to the right hand side of the equation.
- Type in the expression "*8.854e-12/" or click on the corresponding symbols.
- Double click on the "fingerGap" parameter in the parameters list to add it to the right hand side of the equation.
- Click on the Add button to add the equation to the equation list.
- A confirmation message will appear indicating that a new variable has been added to the left hand side of the equation. Accept the message and the equation will be inserted in the list.

Now we need to calculate the displacement of the movable part of the comb:

$$\text{disp} = \text{Voltage}(\text{actuate}) - \text{Voltage}(\text{anchor})$$

- Start by typing on the left hand side of the equation the variable "disp".
- Right click on the "actuate" parameter in the parameters list, a pop up menu will show up, select "Voltage" another pop up menu will appear select right hand side "RHS". The expression "Voltage(actuate)" will appear on the right hand side of the above equation.
- Click the minus "-" sign.
- Similarly add the "Voltage(anchor)" expression.

- Click the Add button to add the equation to the equation list.

Similarly we will need to add the following equations (Figure 90):

- To calculate the voltage between the movable part and the anchored part of the comb.
 $vact = Voltage(vactuate) - Voltage(vanchor)$
- To calculate the electrostatic force between the movable part and the anchored part of the comb.
 $Current(actuate) = -0.5*c0*combFingers*pow(vact,2)$
 $Current(anchor) = -Current(actuate)$
- To calculate the current passing through the comb.
 $Current(vactuate) = combFingers*((fingerOverlap-disp)*c0*diff(vact)-c0 *diff(disp)*vact)$
 $Current(vanchor) = -Current(vactuate)$

In the **Model Name** field type in "combDrive".

Note

The diff () function is the differentiation over time (d/dt).

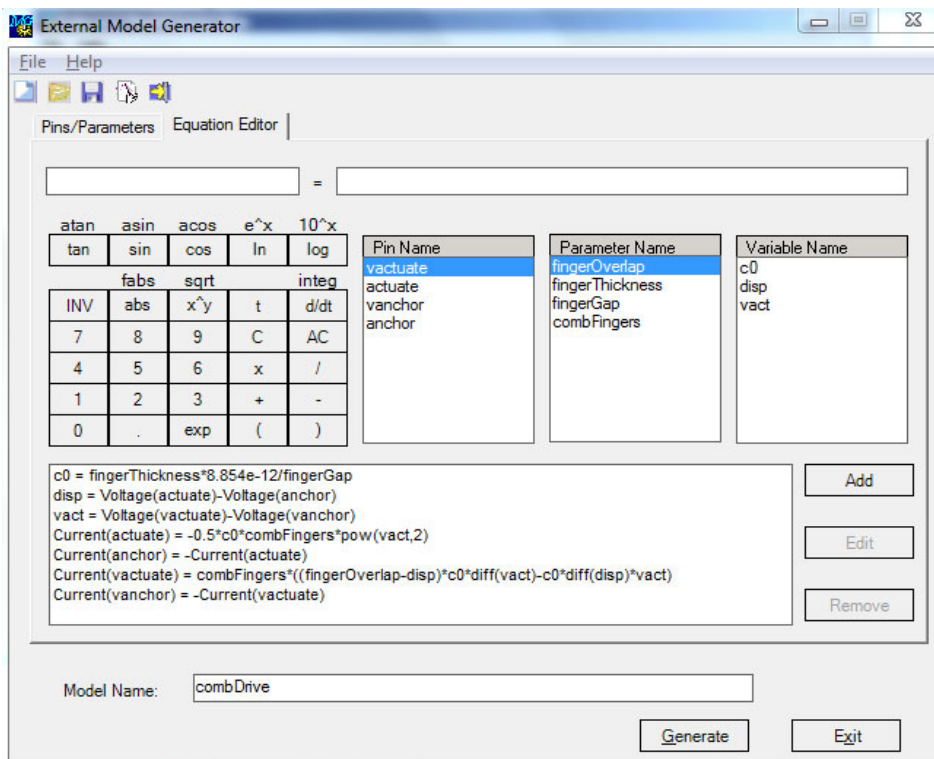


Figure 90: Equation Editor after adding all of the comb drive equations

Generating the Model

- ☑ Click on the generate button, the save as dialog box will show up with the name of the model as default value.
- ☑ Click the **Ok** button. The following message will appear (Figure 91).

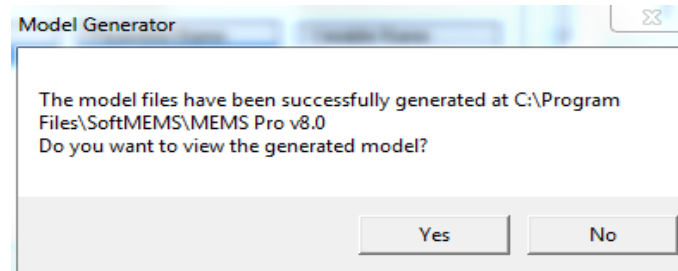


Figure 91: C-Model successfully generated

- ☑ Click the "Yes" button the C-model will be displayed in a text editor.



Instantiating the Model in a schematic

- ☑ Open the following schematic file **{MEMS Pro v8.0 Installation Directory}/MEMS_Tutorial/ModelGen/resonEMG/resonEMG.tanner** from **MEMS S-Edit**.

The resonEMG.tanner file contains the schematic of a resonator. The resonator is composed of two springs, one plate and two comb drives. The comb drives are missing and will be created and instantiated in the schematic.

Creating the symbol

To create the symbol of a comb drive we will first need to create a new cell.

- ☑ Select **Cell > New View** from the menu and enter "CombDrive" in the cell name.
- ☑ Select Symbol in the **View Type** then click the **OK** button.
- ☑ From the left palette, select the **Box** tool  and draw a rectangle.
- ☑ Then select the **Bidirectional Port** tool  and click on the left hand side of the rectangle to place it. A property sheet will show up asking about the name of the port. Enter "VActuate".
- ☑ Similarly add the remaining 3 ports (*Actuate*, *VAnchor* and *Anchor*) as shown in [Figure 92](#).

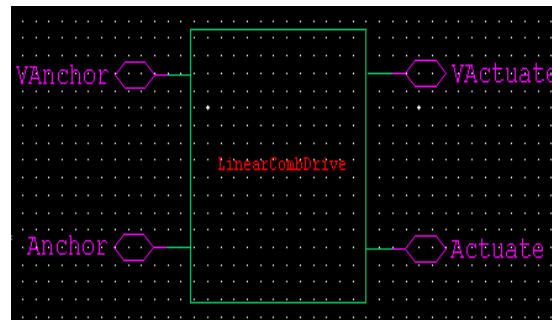





Figure 92: Symbol of the comb drive

- ☑ Connect the ports to the rectangle using the **Path** tool  as shown in [Figure 92](#).
- ☑ Select the **Label** tool  and then click in the center of the rectangle. A Test Label Settings window will show up. In **Name** field enter "LinearCombDrive".
- ☑ Add the pins' names by clicking the **Comment** tool.
- ☑ Now we will start to add properties to the symbol. Click on the Properties tool  and then click under the comb drive symbol. A property sheet will show up. Enter the values as shown in [Figure 93](#).

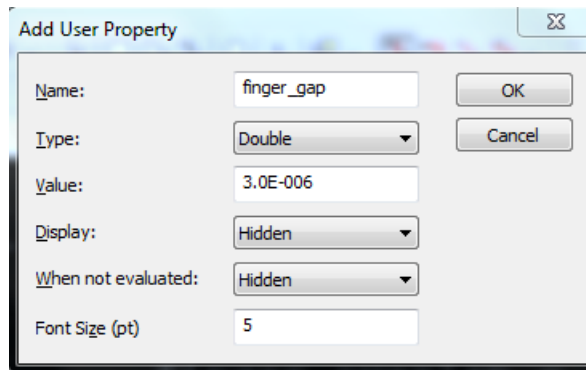


Figure 93: Creating properties

- ☑ Similarly add the following properties

Name	Value	Font Size	Type	Display	When not evaluated
finger_gap	3E-006	5	Double	Hidden	Hidden
finger_overlap	3E-005	5	Double	Hidden	Hidden
comb_fingers	20	5	Integer	Hidden	Hidden
finger_thickness	2E-006	5	Double	Hidden	Hidden
SPICE.OUTPUT	see below	5	String	Hidden	Hidden

The value of the SPICE.OUTPUT property should be:

```
xcombEMG${Name} %VActuate} %Actuate} %VAnchor} %Anchor} combDrive
fingerGap=${finger_gap} fingerOverlap=${finger_overlap} fingerThickness=${finger_thickness}
combFingers=${comb_fingers}
```

Note

The value of the SPICE OUTPUT property should be written in one line, with no spaces before or after the "%" and "\$" signs.

The SPICE OUTPUT property indicates the command used in the Spice file to instantiate an instance of this model. It is composed of:

- The name of the instance followed by the "\${Name}".
- The list of the ports. The order of the pins should be the same as the order used to create the behavioral model in the external model generator.
- The name of the model "combDrive".
- The list of the parameters of the model with their default values.

Now we need to instantiate the model in the resonator circuit.

Instantiating the Model in the circuit

- ☑ Open the resonator circuit by clicking on **Cell > Open View** then select "Resonator" and select **View Type** to be **Schematic**.
- ☑ In the **Libraries** window select "combDrive" and click on **Instance**.
- ☑ Place the first comb drive to the right side of the plate and connect its pins as shown in [Figure 94](#).
- ☑ Do the same previous steps to instantiate another comb to the left side of the plate as shown in [Figure 94](#).

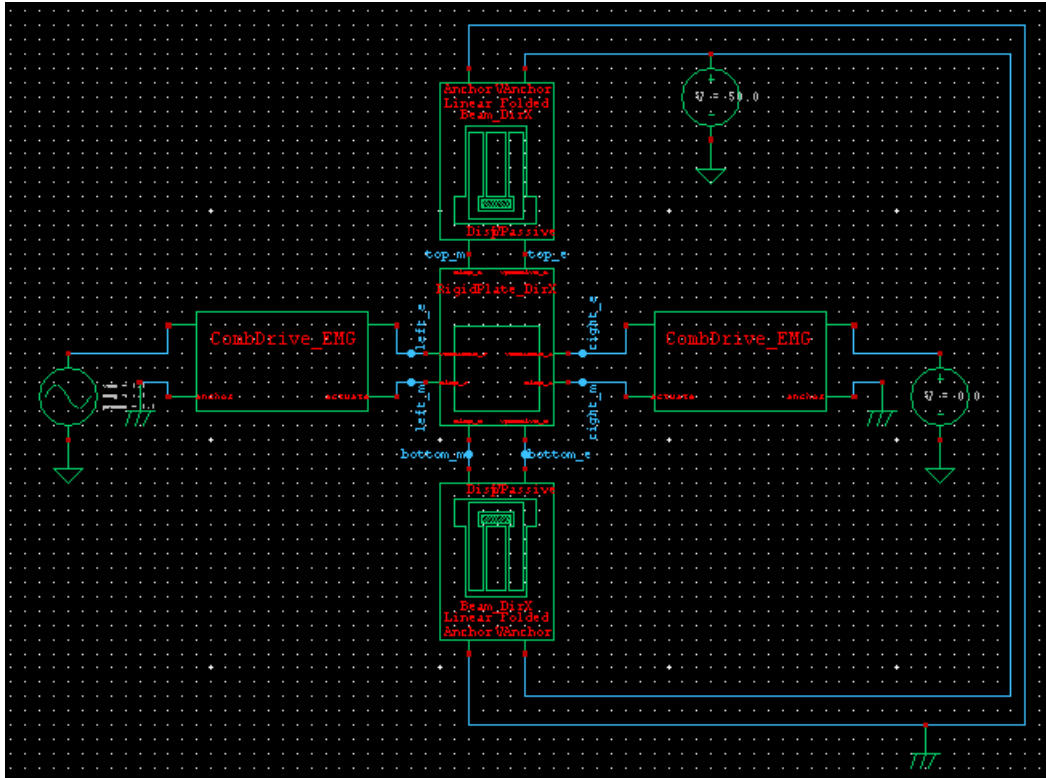


Figure 94: Resonator Circuit

- ☑ Modify the value for "finger_thickness" of the twocomb drives to be "TPOLY1".
- Now we'll start to add some Spice commands for the simulation.

Simulating the circuit

- To add the Spice commands select TSPICECOMMAND from the Libraries window and click on **Instance**.
- Click on the placed **Tspice command** and go to the **Properties** window to edit the command.
- In the **OUTPUT** field enter *.model combDrive external winfile="combDrive.c"*
 The following command should appear on your schematic
SPICE OUTPUT = .model combDrive external winfile="combDrive.c"
 This command specifies the name of the C file containing the behavioral model of the comb drive.
- Similarly add the following commands
SPICE OUTPUT = .include process.sp
 The process.sp file contains the values of some parameters used in the circuit (e.g. the value of TPOLY1 used to specify the thickness of the fingers of the comb drive).

SPICE OUTPUT =.ac dec 500 10k 50k
SPICE OUTPUT =.print ac left_m left_e
- Verify that you entered the correct path for SPICE simulation in **Setup > SPICE Simulation**.
- Click on the T-Spice button from the S-Edit toolbar to launch the simulator.
- Once T-Spice is launched with the corresponding Spice netlist, click on the **Run Simulation** button. The simulator will start the simulation and display the resultant waveform (**Figure 95**). The resonant frequency should be around 16.5 KHz.

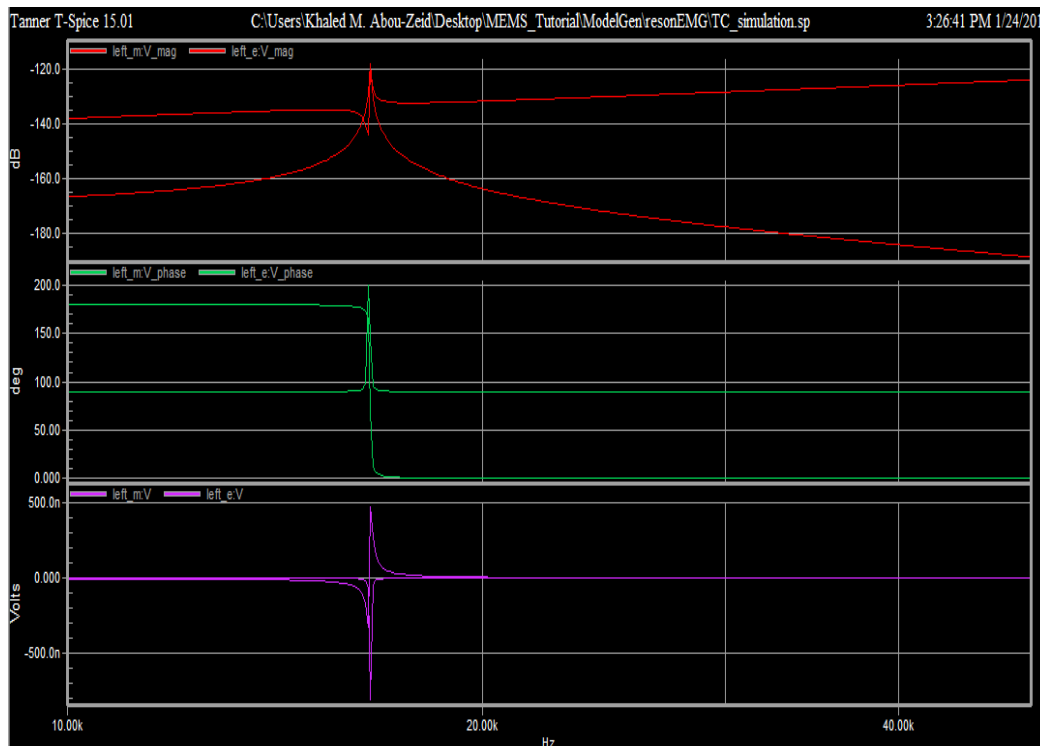


Figure 95: Resultant waveform

8 MEMS Modeler Tutorial

▪ Introduction	130
▪ Tunable Filter	131
▪ Generating 3D Model	133
▪ ANSYS Setup	136
▪ Loading 3D Model	138
▪ Loading Macro	140
▪ MemModeler Setup	141
▪ FEM Model	145
▪ Electrical Data	146
▪ DOF Sweeping	147
▪ Output	148
▪ Running MemModeler	149
▪ Saving Model	150
▪ Simulation With Reduced Macro Model	151
▪ Creating Filter Model	152
▪ Creating Symbol	153
▪ Creating Top-Level Model	154
▪ Creating Driving Signal	155
▪ Grounding Voltage Source	157
▪ Setting Voltage Source Parameters	158
▪ Wiring Schematics	159
▪ Setup Probing Parameters	160
▪ Generating Spice Netlist	161
▪ Setting Analysis Type	162
▪ Starting Simulations	164

Introduction

Throughout the following tutorial, you will be working on the model of a tunable filter, as shown in (Figure 96).

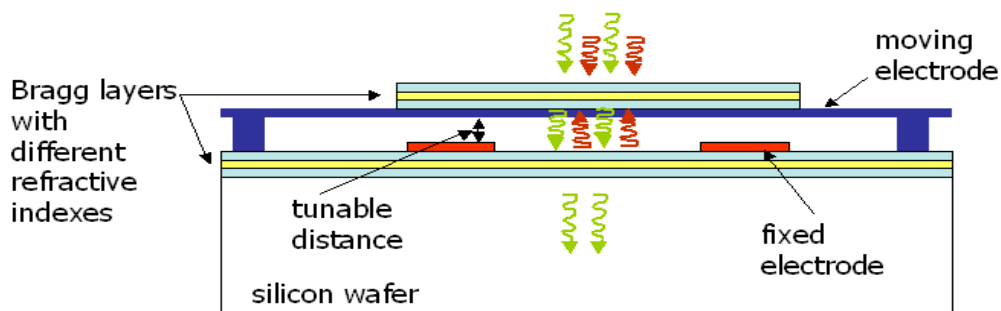


Figure 96: Tunable Filter

In the tunable filter, the distance between the two mirrors determines the range of frequencies that passes through. This distance is controlled by electrostatic actuation.

You will be studying the electrostatic behavior of the tunable filter using FEA, in this tutorial.

Note

This tutorial was run on a PC with the following specifications:

- Intel Core2 Duo CPU E8400 @ 3.00GHz.
- RAM: 4.00 GB.
- Operating System: Windows 7 Ultimate 32-bit.

The run time on such a machine for this tutorial is 11 minutes.

Tunable Filter

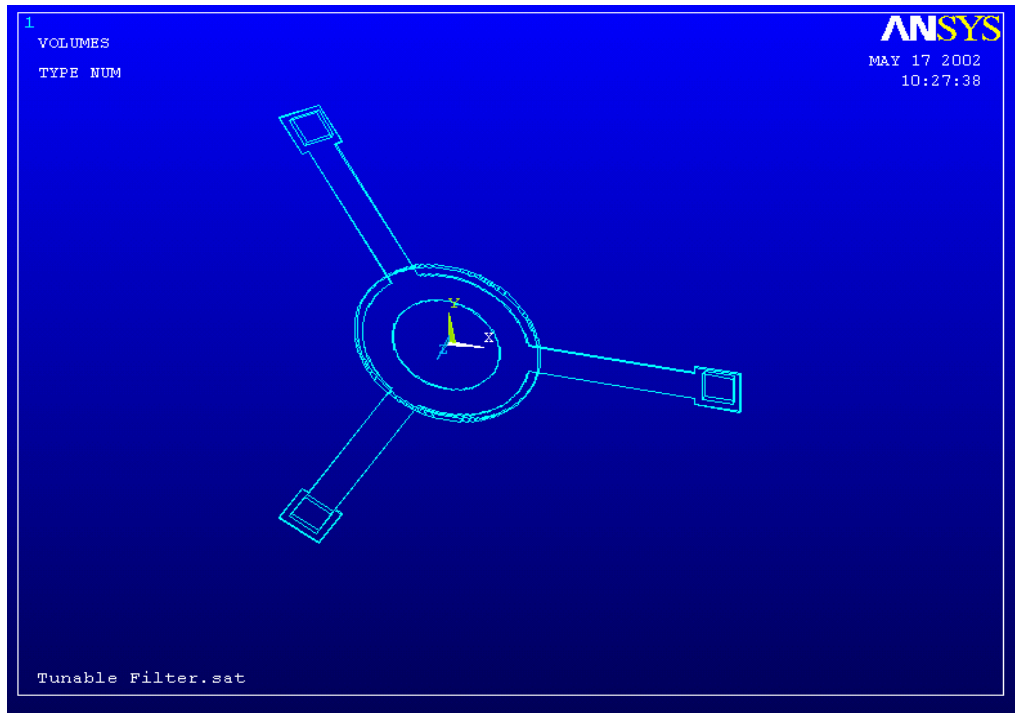


Figure 97: Complete tunable filter

The model will be symmetrically cut into three parts, hence, the size of the FEA model will be reduced. It will only include a top electrode with one support arm and an air volume.

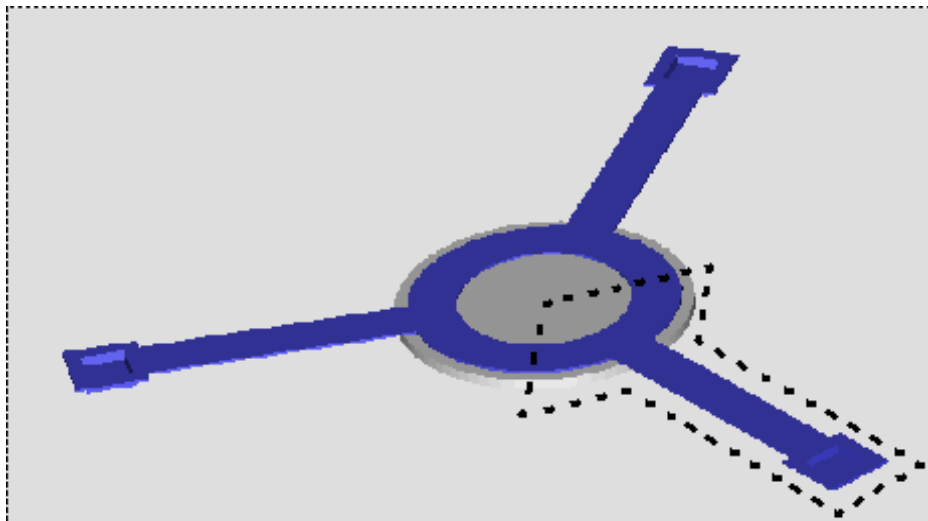


Figure 98: Complete tunable filter

First you have to load the 3D model into ANSYS, then mesh the model and run **MemsModeler**.

Procedures

The tutorial is composed of the following main steps:

- Generating the 3D Model
- ANSYS steup
- Loading 3D model
- Meshing
- Running MemModeler
- System-level simulation with Reduced Macro Model

Generating 3D Model

- ☑ Open **MEMS Pro** launcher by clicking **Start> All Programs> MEMS Pro v8.0> MEMS Pro Launcher**.
- ☑ Click **Layout Editor** in the **Tools Selection** window.
- ☑ Select the file **{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\MemsModeler\layout\filter.tdb**.
- ☑ Click the **Add** button to add the file to the **Layout Files** list.
- ☑ Select the file from the list, then click **Run** to launch **L-Edit**.
L-Edit will open with the tunable filter as displayed in (Figure 99).

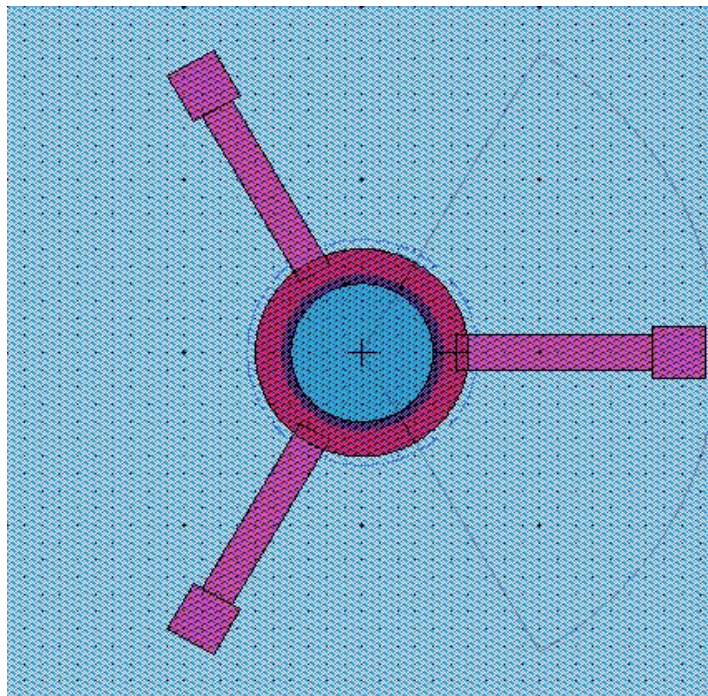


Figure 99: Tunable Filter

- ☑ Click **3D Tools> View 3D Model** to generate the 3D model. After generation the 3D model is displayed as in (Figure 100).

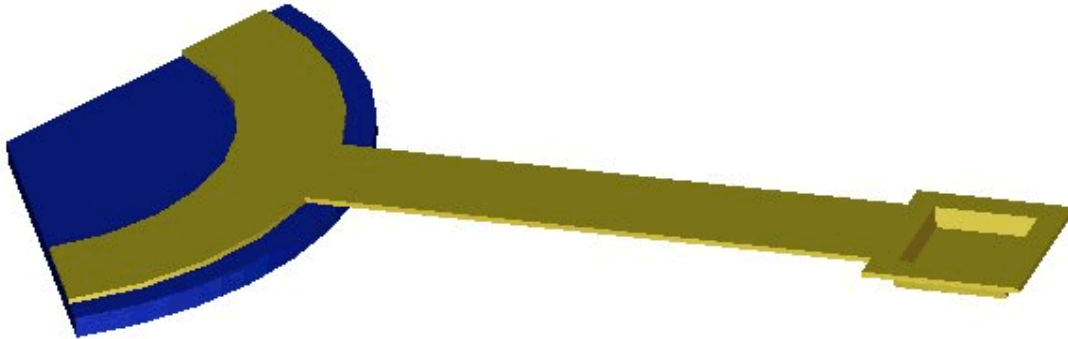



Figure 100: 3D Model

Note

To generate a cross section, set the substrate at the part you want to generate.

- ☑ To view the boundary conditions on the 3D model, click the map BC button .

The following components are displayed:

- Volume Components (Figure 101).
 - Air volume.
 - Metal electrode (ring suspended by support arm).

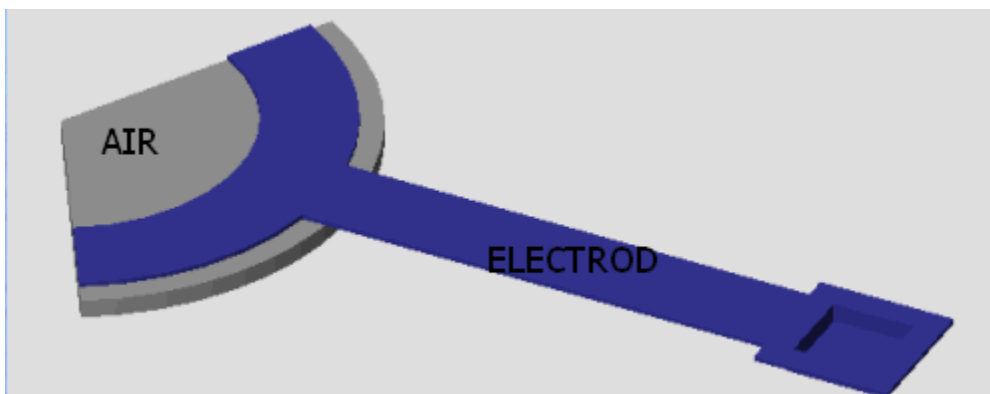


Figure 101: Volume Components

- Area Components (Figure 102):
 - ANCHOR: anchor of support arm.
 - AIR_BOTTOM: bottom of air volume.
 - SYMMETRY: planes of symmetry.

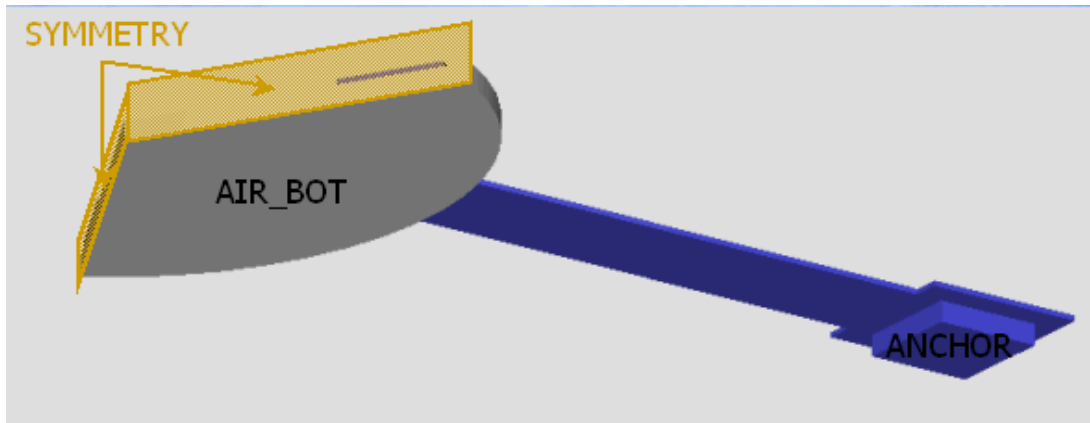
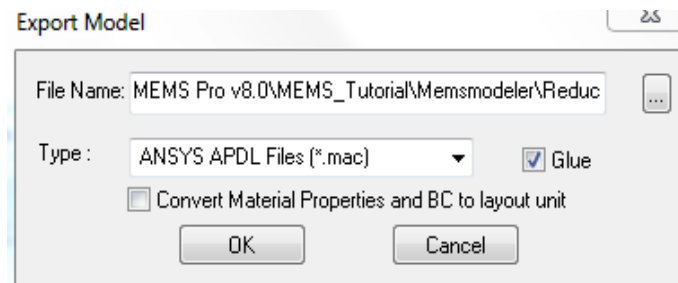


Figure 102: Area Components

- ☑ To export the model to ANSYS, select **3D Tools>Export 3D Model** to open the **Export Model dialog** (Figure 103).

Figure 103: **Export Model** dialog.

- ☑ Set the file name to **{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\MemsModeler\Reduction\Filiter.mac**.
- ☑ Select the **Glue** option to glue adjacent volumes, so that they will be correctly meshed.

ANSYS Setup

- ☑ Open **MEMSPro** launcher by clicking **Start> All Programs> MEMS Pro v8.0> MEMS Pro Launcher**.
- ☑ Click ANSYS in the **Tools Selection** window. The information tab is related to the ANSYS setup as shown in (Figure 104). By default, the workspace and the database space are filled.

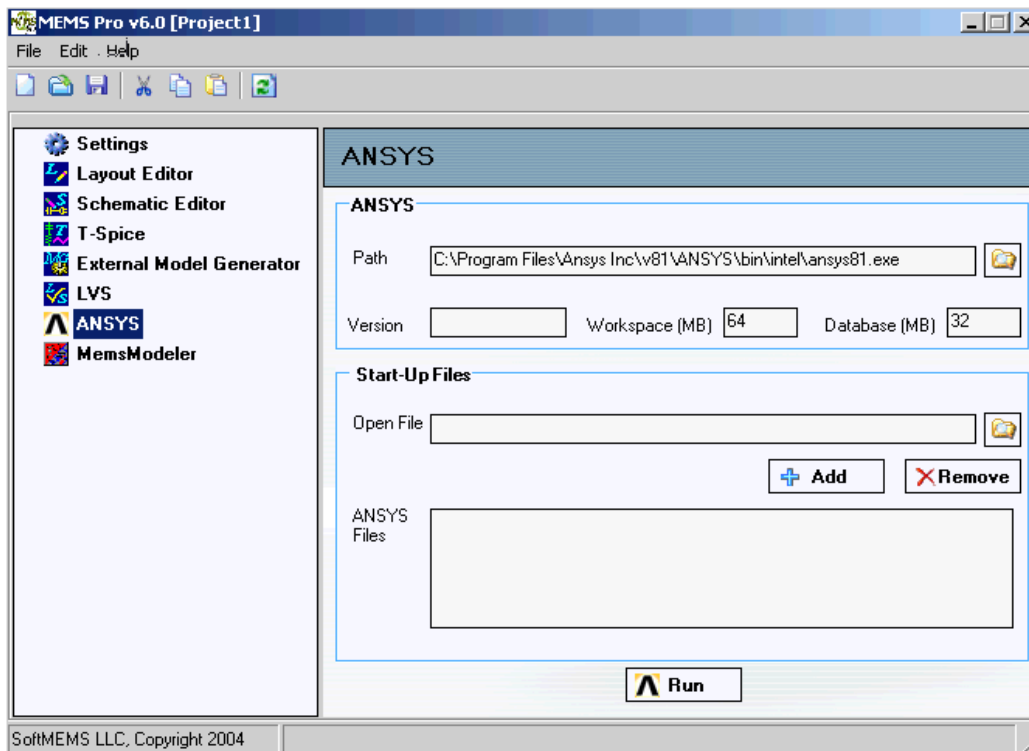


Figure 104: ANSYS Setup

- ☑ To launch ANSYS, click **Run**. The **ANSYS Graphical Interface** appears (Figure 105).

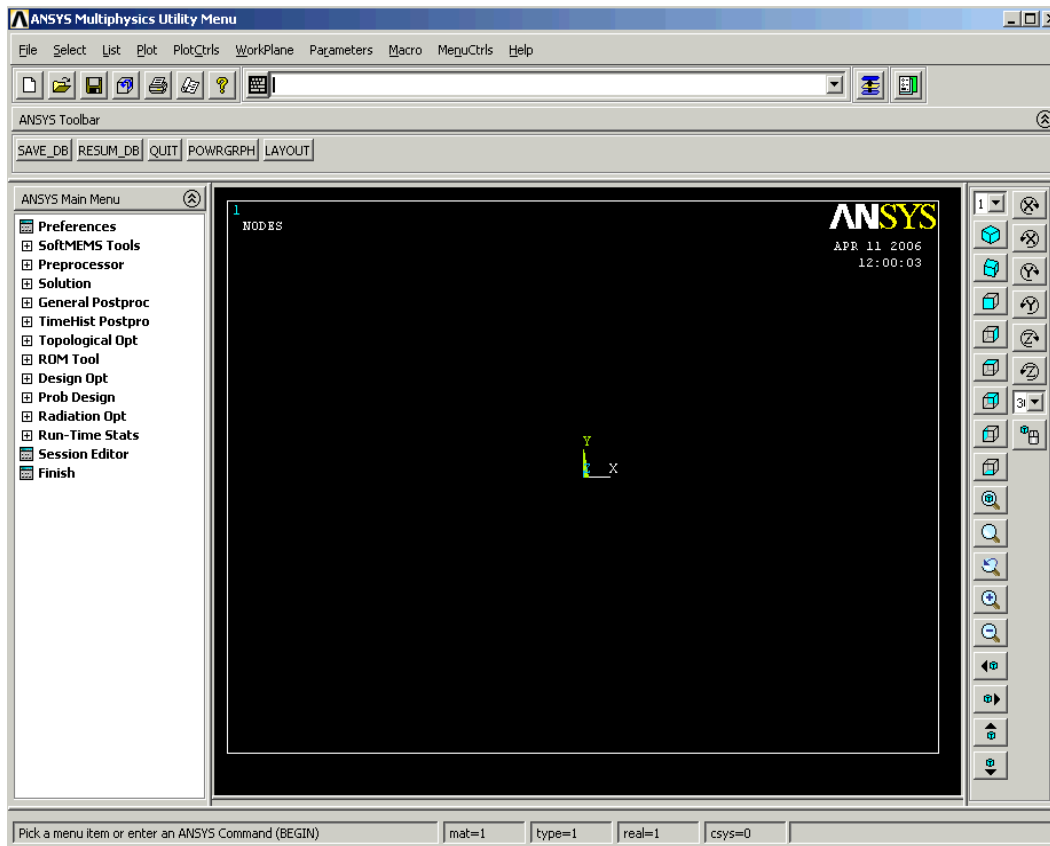


Figure 105: ANSYS Graphical Interface

- To change the working directory, click **File -> Change directory.**
- Select **{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\MemsModeler\Reduction.**
- Change the Job name to Filter by using **File -> Change Jobname.**

Loading 3D Model

- Navigate to **File->Read Input From** and select **Filter.mac** (Figure 106).

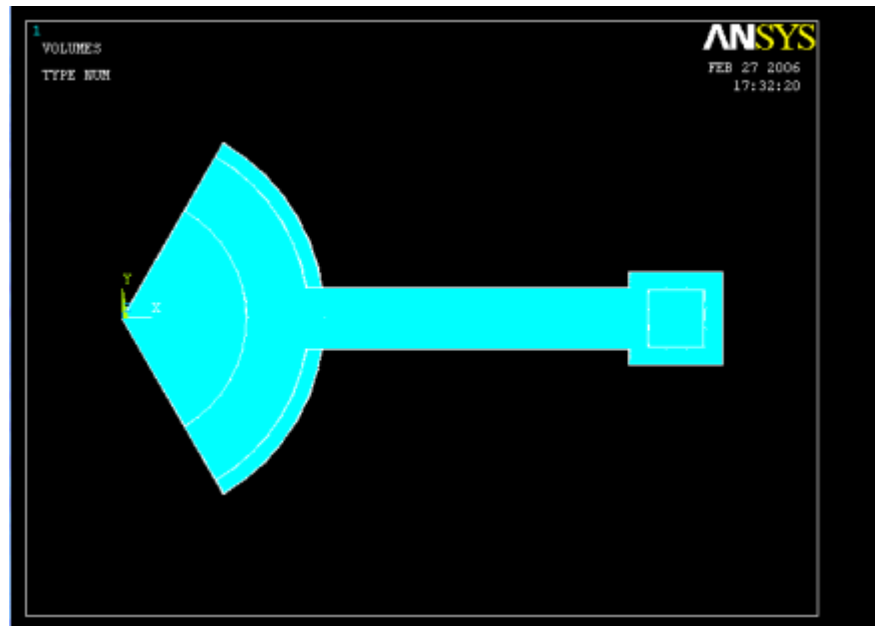


Figure 106: 3D Model loaded

After loading the 3D model you now need to :

- Mesh model.**
- Create electrostatic physics file:**
 - Assigning node and element components.
 - COND 1: Conductor #1, includes all nodes in metal electrode.
 - COND 2: Conductor #2, includes all nodes in bottom air surface.
 - Assigning element type and material properties.
- Create structural physics file:**
 - Assigning element type, boundary conditions and material properties.

- ☑ **Assign master degree of freedom** (Figure 107).
 - This is where displacement output will be distracted.

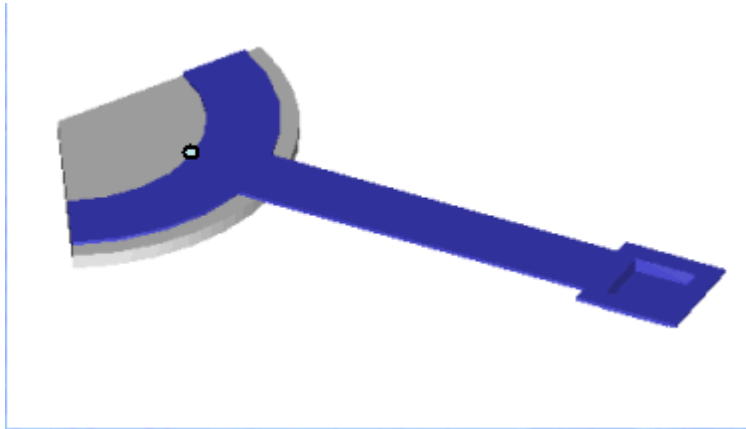


Figure 107: Master degree of freedom

Loading Macro

In order to mesh the model then create the necessary structural physics and electrostatic physics file, you need to perform the following steps:

- ☑ From the ANSYS Menu, click **File > Read Input From**.
- ☑ Select the **Filter.mdl** file from the Reduction directory.
- ☑ Click **OK**.
- ☑ Close ANSYS without saving once the macro is complete (Figure 108).

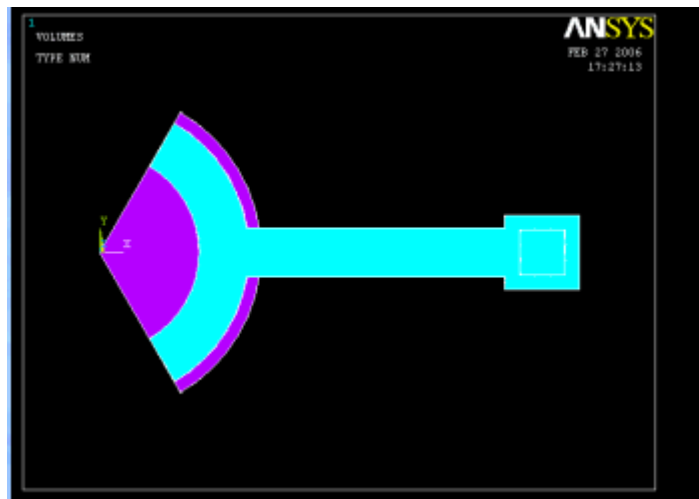


Figure 108: Loading Macro in ANSYS.

MemsModeler Setup

- ☑ Launch **MEMS Pro** Launcher from **Start>All Programs>MEMSPro v8.0> MEMS Pro Launcher**.
- ☑ Click MemsModeler in the **Tools Selection** window (Figure 109).
The information tab is related to **MemsModeler** setup.

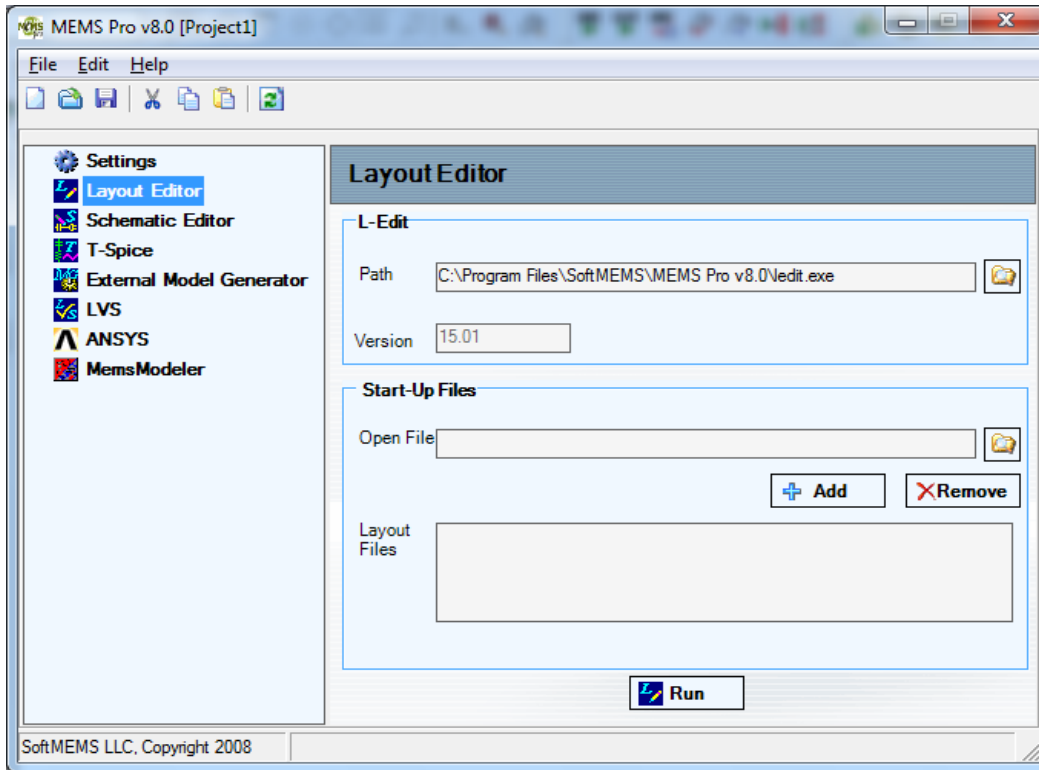


Figure 109: MEMS Pro launcher

- ☑ To launch **MemsModeler**, click Run.
The **MemsModeler** interface is displayed (Figure 110).

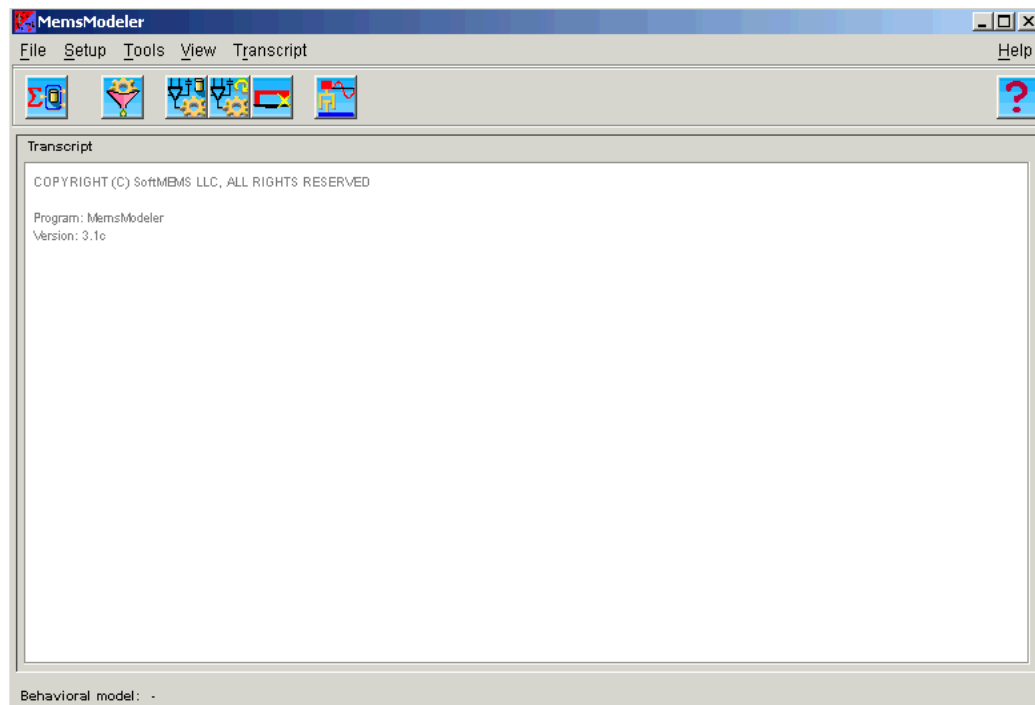


Figure 110: **MemsModeler** interface

- ☑ Set the working directory by clicking **Setup -> Working directory**.
- ☑ Select **{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\MemsModeler**.
The transcript should display the information.
- ☑ To configure ANSYS settings, click **Setup->ANSYS** to open the **ANSYS setup** dialog (Figure 111).

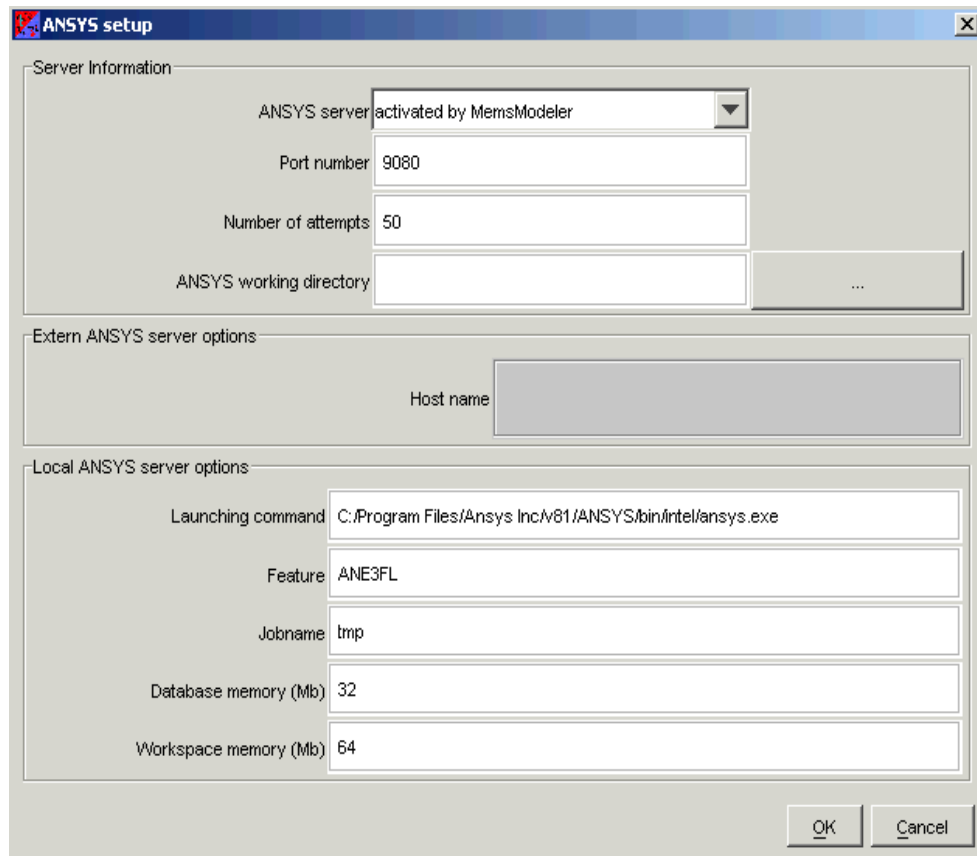


Figure 111: ANSYS Setup

- Set the ANSYS working directory as **{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\MemsModeler\Reduction**.
- Click **OK** to validate. ANSYS parameters are displayed in the transcript.
- Set the ANSYS database memory as 32, and the Workspace memory as 64.

Selecting the Reduction Algorithm

- Select the Electro-Static algorithm by clicking **Tools -> ES-STR, Model Deformation** (Figure 112).

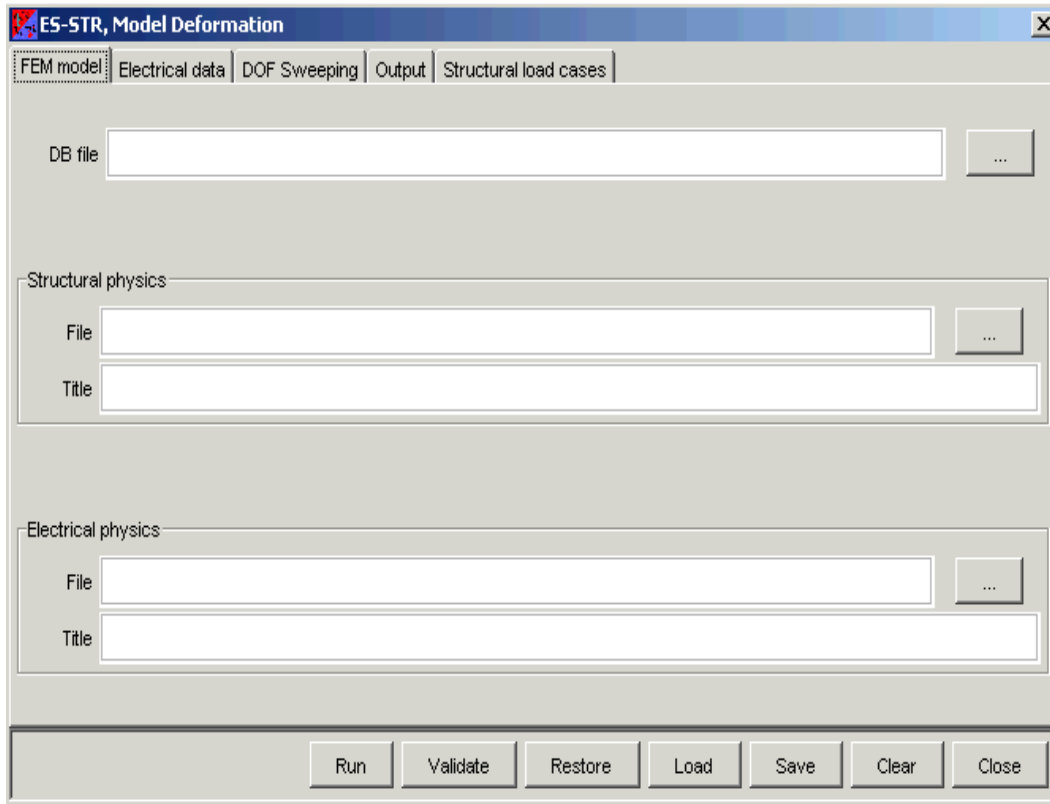


Figure 112: Selecting electro-static algorithm.

FEM Model

- In the **FEM Model** tab, set the ANSYS related information, as shown in (Figure 113).

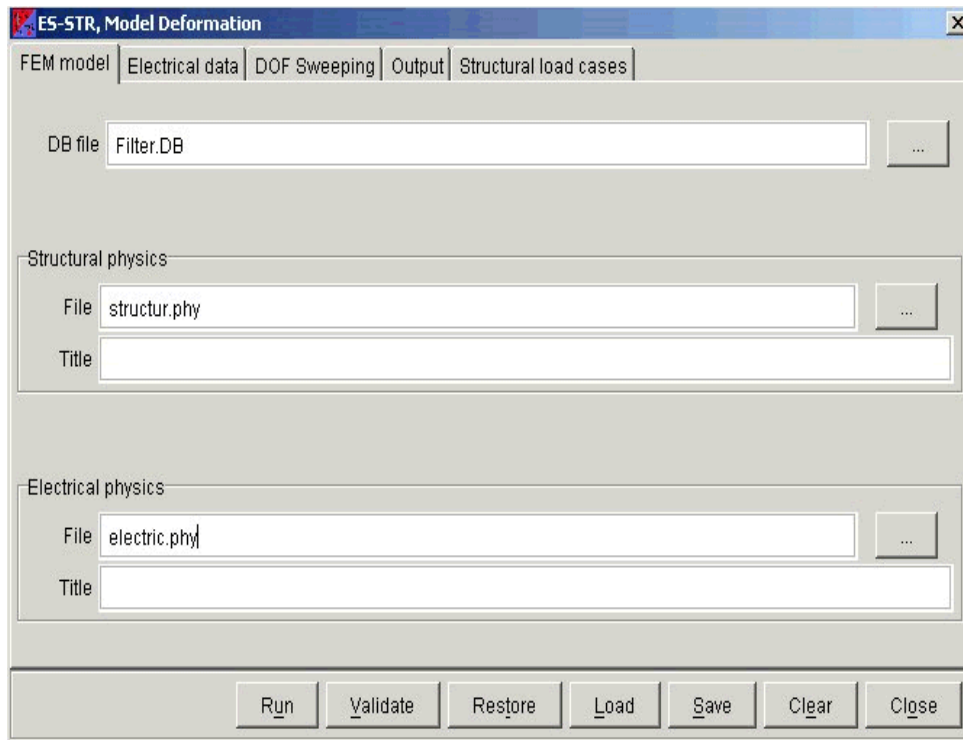


Figure 113: **FEM model** tab

Electrical Data

- ☑ In the **Electrical Data** tab, set the number of conductor and the used radical, as shown in (Figure 114).

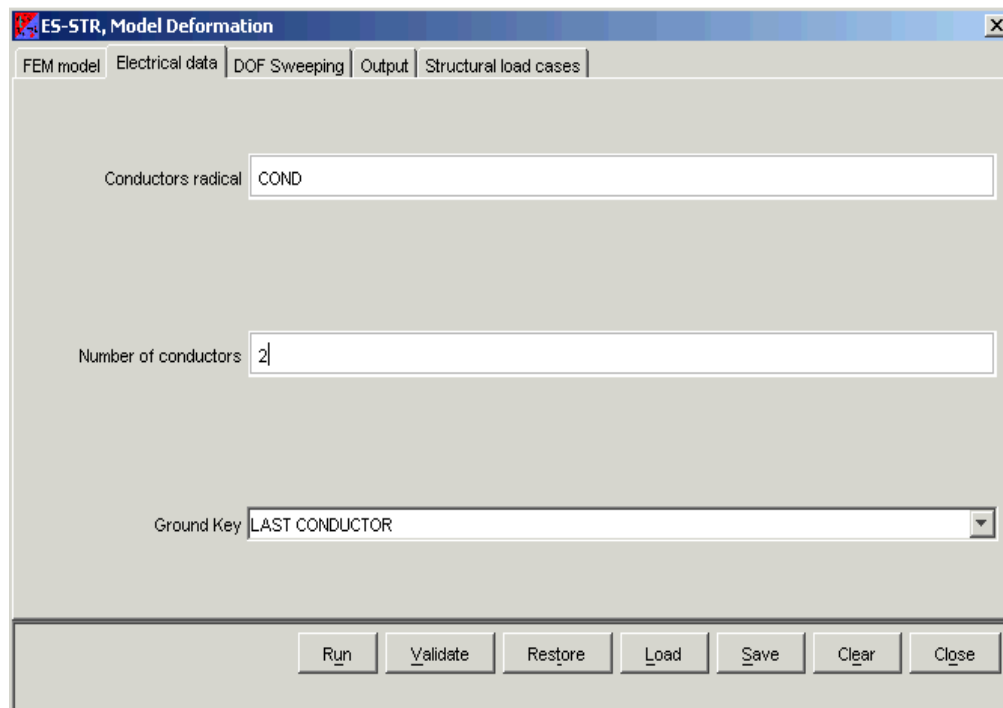


Figure 114: **Electrical data** tab

DOF Sweeping

- In the **DOF Sweeping** tab, set the name of the selected degree of freedom and the operating values, as shown in (Figure 115).

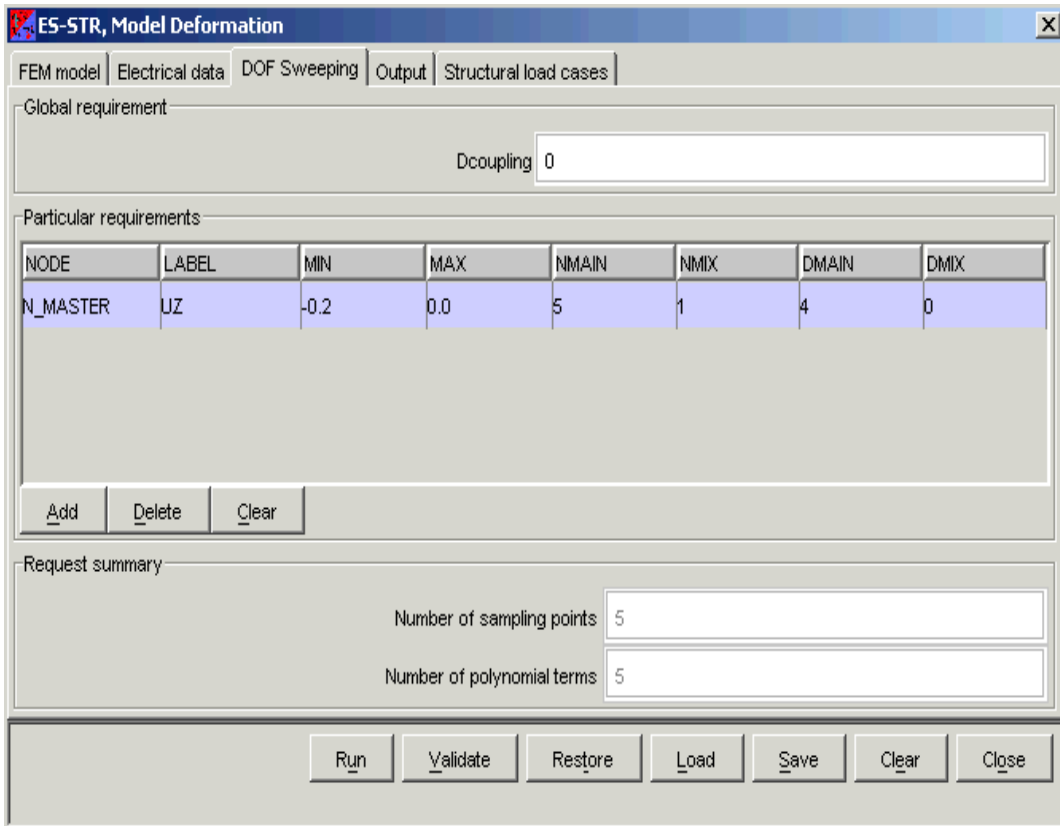


Figure 115: **DOF Sweeping** tab

Output

- In the **Output** tab, set the output model name and the behavior you need in the model, as shown in (Figure 116).

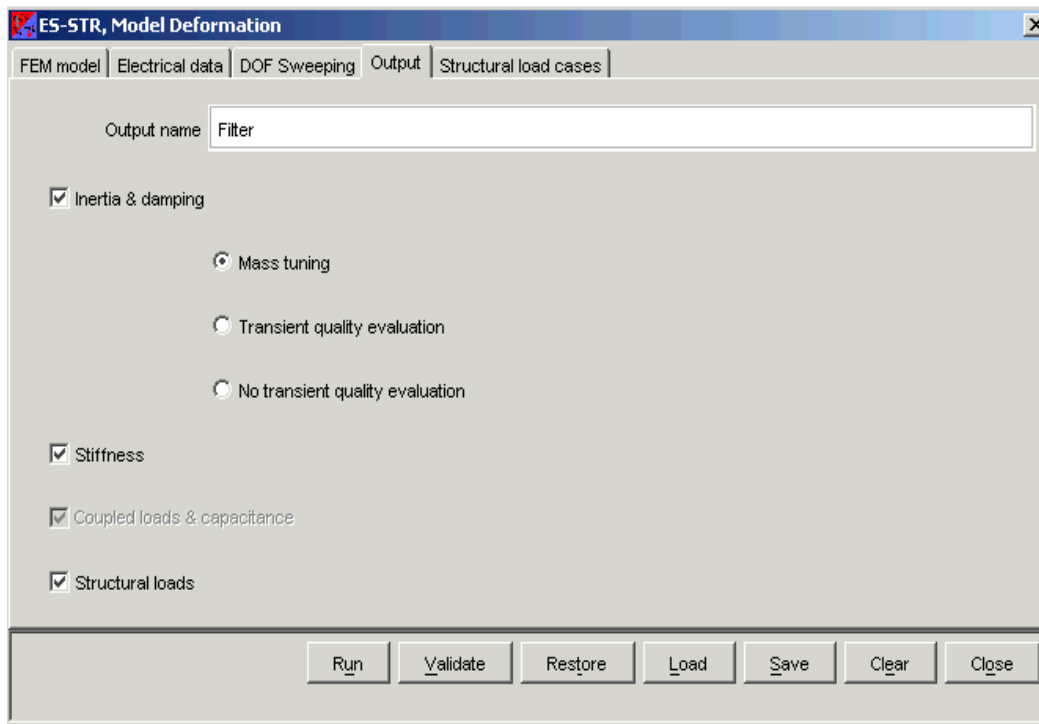


Figure 116: **Output** tab

Running MemsModeler

- ☑ Once you set all the parameters, click **Run** to start the reduction process.
 - This simulation takes approximately 10 minutes to run.
 - The transcript displays the current status.

Saving Model

- ☑ After the reduction is complete, click **File-> Save**.
The **Save Model** dialog is displayed as shown in (Figure 117).

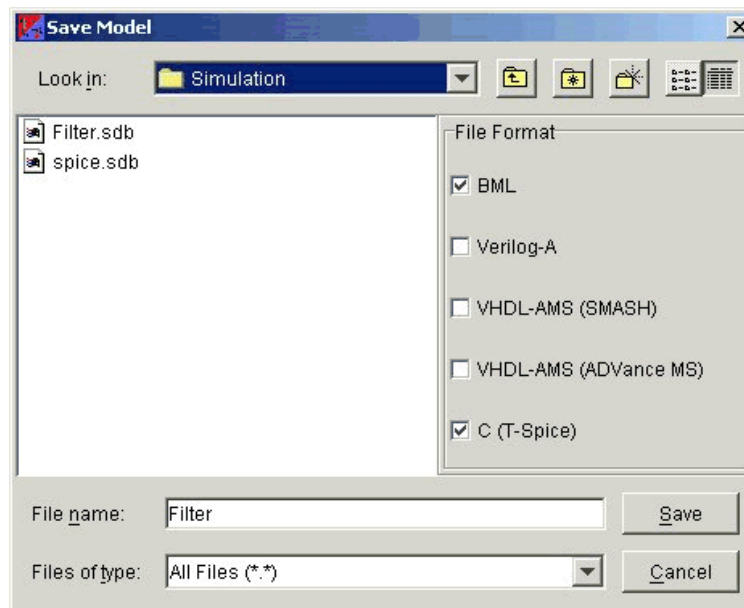


Figure 117: Save Model dialog.

- ☑ Select the Output as **C (T-Spice)**.
- ☑ Make sure that the path points to **{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\MemsModeler\Simulation**, then save the model as filter.

Simulation With Reduced Macro Model

You will now use the generated Spice model to run a simple simulation.

- Launch **MEMS Pro Launcher** from **Start> All Programs> MEMS Pro v8.0> MEMS Pro Launcher**.
- Select the **S-Edit** **Schematic Editor** tab.
- Select the file" **{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\MemsModeler\Simulation\filter.tanner**".
- Click the **Add** button, then click **Run** to launch **S-Edit**.

Creating Filter Model

- ☑ In S-Edit select **Module > New**.
- ☑ Set the module name to "**Filter Model**".
- ☑ Select **View > Symbol Mode**.
- ☑ Select the Text tool **L** .
- ☑ Click in the center of the work area. The **Place Text** dialog box opens (Figure 118).
- ☑ Enter "Filter Model" in the **Name** field.

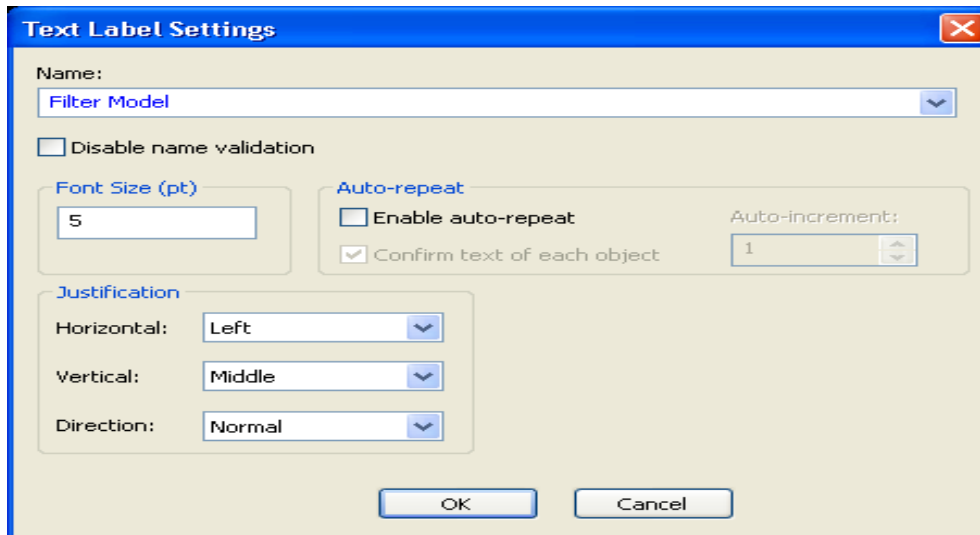





Figure 118: **Place Text** dialog.

- ☑ The text is inserted in the center of the work area.

Creating Symbol

- ☑ Select the "Box" tool .
- ☑ Draw a box around the "Filter Model" text just placed.



- ☑ Select the "Input Port" tool .
- ☑ Click the left side of the box, and type in "CONDO_Vi".
- ☑ Select the "Output Port" tool .
- ☑ Click the right side of the box, and type in "CONDO_Vo".

Note

Make sure the names of the ports are typed in correctly. They correspond to ports in the C netlist produced by **MemsModeler**.



Figure 119: Viewing the model and its attached ports

- ☑ Add the mechanical pin (output port) "N_MASTER_UZ".

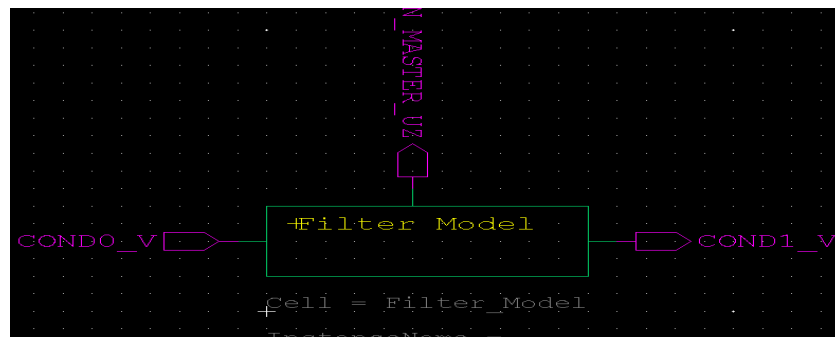


Figure 120: Viewing the model and its attached ports and pin

Creating Top-Level Model

- ☑ Select **Module > New** and enter "Filter Simulation".
- ☑ Select **View > Schematic Mode**.

To instantiate the Filter Model that you have just created, perform the following:

- ☑ Select **Module > Instance**.
- ☑ Select **Filter Model** and click **OK**.

The module instance is created ([Figure 121](#)).



Figure 121: Instantiated module

Creating Driving Signal

You will now create a voltage source as the electrostatic driving signal. The voltage source is a standard IC model, and you will load it from the Spice library.

- ☑ Select **Module > Symbol Browser**.
- ☑ Click **Add Library**.
- ☑ Select **{MEMS Pro v8.0 Installation Directory}\MEMS_Tutorials\MemsModeler Simulation\Filter.tanner**.
- ☑ Click **Open**.
- ☑ Select **Spice** at the bottom of the library list (Figure 122).

All the modules available in the Spice library are displayed.

- ☑ Select **Source_v_bit** from the **Modules** list (binary waveform source).

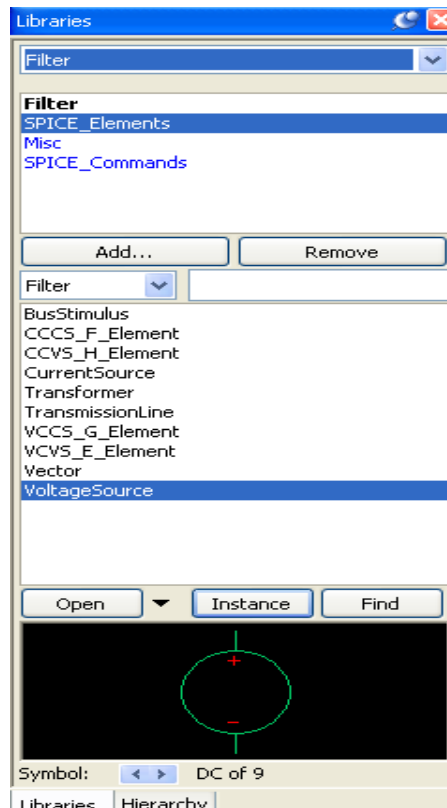


Figure 122: **Symbol Browser** dialog

- ☑ Click **Place**.

- Click **Close**.

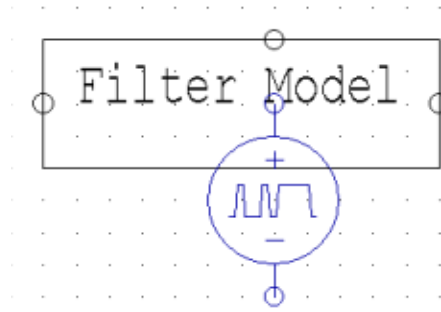


Figure 123: Instantiated voltage source

- Select the voltage source instance, use the middle mouse button to move it to the left of the filter model module (Figure 124).

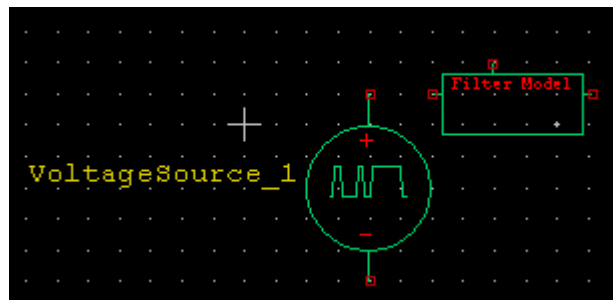


Figure 124: Viewing the module and its voltage source

Grounding Voltage Source

You need to provide a voltage reference to the source:


- ☑ Select the **Global Symbol Instance** tool  .
- ☑ Click anywhere in the work area.
- ☑ Select 'Gnd' from the module list.
- ☑ Move the 'Gnd' symbol so that it is connected to the bottom node of the voltage source.



Figure 125: Grounded Voltage Source

- ☑ Similarly, instantiate a ground module and place it to the right of the filter model (Figure 126).

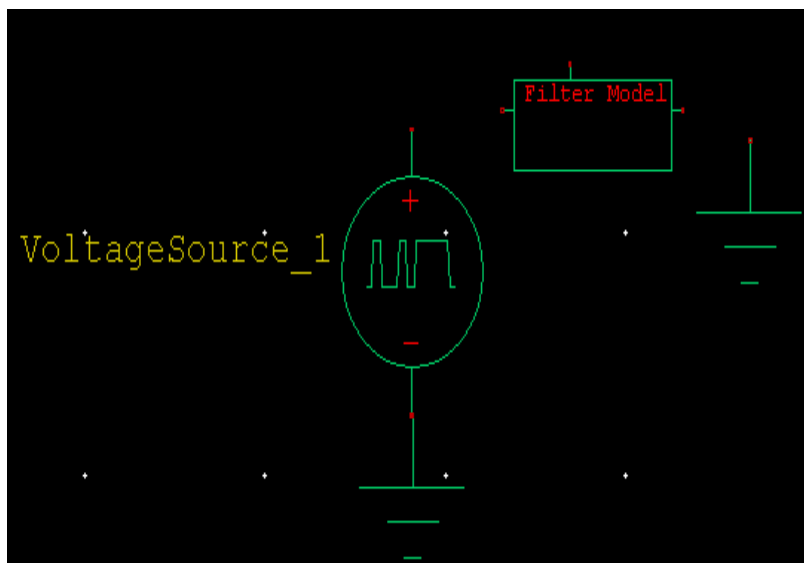



Figure 126: Instantiated ground module

Setting Voltage Source Parameters

- ☑ Right-click on the voltage source to select it.
- ☑ Select **Edit > Edit Object** (or press **Ctrl + E**).
- ☑ In the **Edit Instance** dialog box, set the parameters as described in the following table to specify the shape of the driving waveform.

Parameters	Value
Delay	0
Falltime	5e-4
Hightime	2e-3
Lowtime	2e-3
Pattern	010100111
Risetime	5e-4
Spice Output	Default
Vhigh	20
Vlow	0
Width	4e-3

Wiring Schematics

- ☑ Select the **Wire** tool  .
- ☑ Connect the voltage source (top node) to the Filter Model input (left node).
- ☑ Connect the ground pin to the output of the filter model (right node), as shown in (Figure 127).

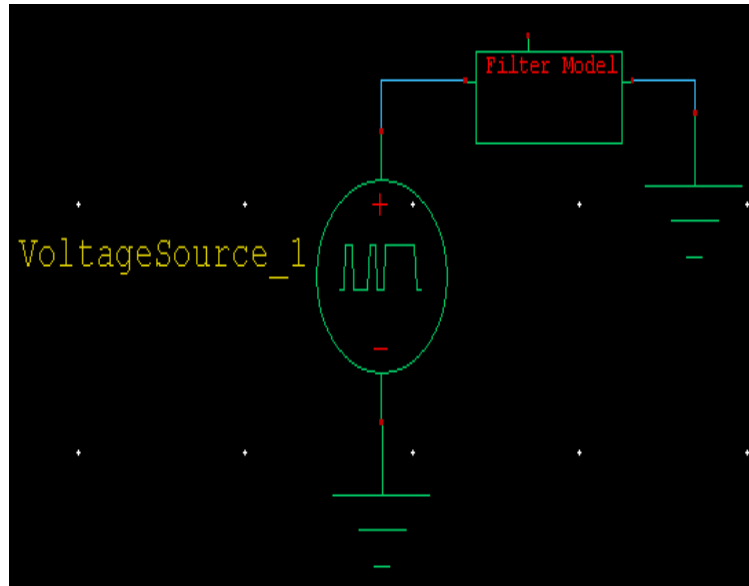



Figure 127: Wired instance

- ☑ To probe the results at the output node of the Filter Model, place a wire segment to the top of the model passing through the upper pin.

You will now label a couple of nodes that you will probe.

- ☑ Select the **Node Label** tool  .
- ☑ Select the left wire segment of the filter and enter **Vin**.
- ☑ Select the top wire segment of the filter and enter **Disp**.

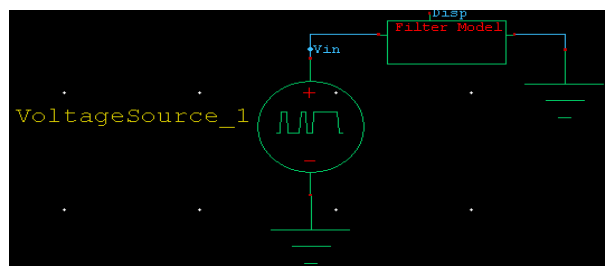


Figure 128: Labelled nodes

Setup Probing Parameters

You need to specify the location where the simulation data will be stored.

- ☑ Select **Setup > Probing**.
- ☑ For probing data file location, browse to your current working directory and enter **Filter.dat**.
- ☑ Click **OK**.

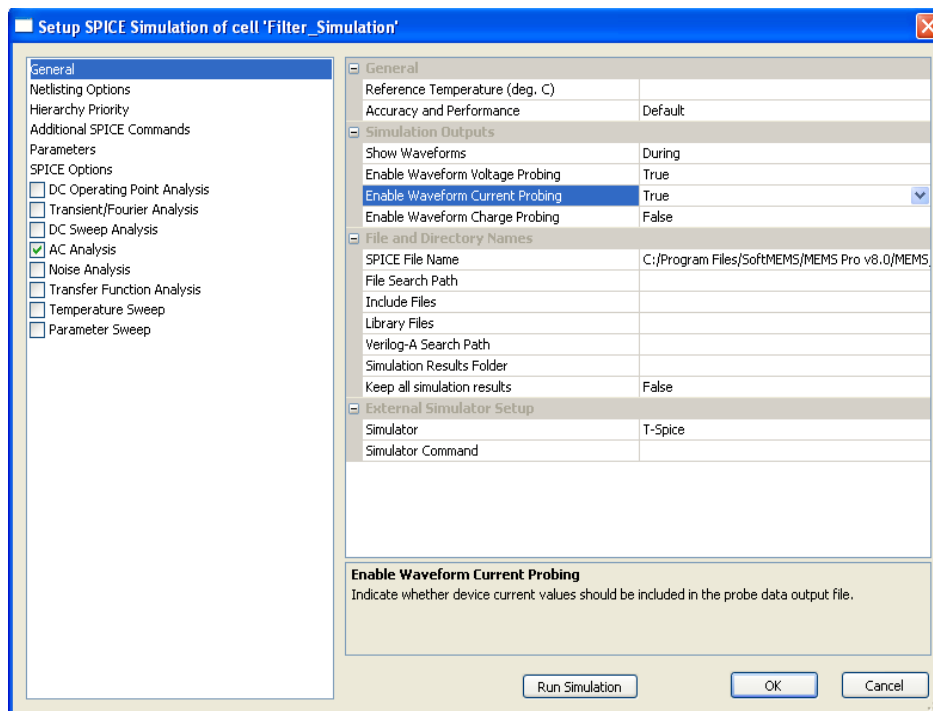

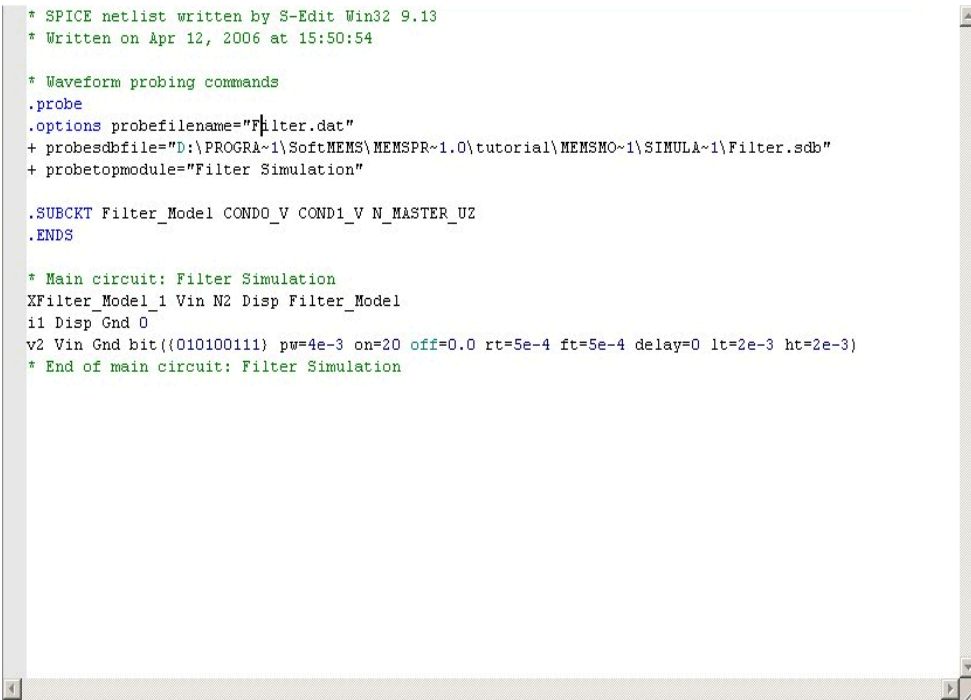


Figure 129: Waveform Probing Setup dialog.

Generating Spice Netlist

- To generate the Spice netlist of the current schematics, click the **T-Spice** button .

The **T-Spice** window displays the generated netlist (Figure 130).



```
* SPICE netlist written by S-Edit Win32 9.13
* Written on Apr 12, 2006 at 15:50:54

* Waveform probing commands
.probe
.options probefilename="Filter.dat"
+ probesdbfile="D:\PROGRA~1\SoftMEMS\MEMSPR~1.0\tutorial\MEMSMO~1\SIMULA~1\Filter.sdb"
+ probetopmodule="Filter Simulation"

.SUBCKT Filter_Model CONDO_V COND1_V N_MASTER_UZ
.ENDS

* Main circuit: Filter Simulation
XFilter_Model_1 Vin N2 Disp Filter_Model
i1 Disp Gnd 0
v2 Vin Gnd bit((010100111) pw=4e-3 on=20 off=0.0 rt=5e-4 ft=5e-4 delay=0 lt=2e-3 ht=2e-3)
* End of main circuit: Filter Simulation
```


Figure 130: T-Spice window.

- Remove or comment the `.SUBCKT Filter_Model CONDO_V COND1_V N_MASTER_UZ` and the `.ENDS` lines.
- Replace them by the **.model** statement as shown below.

```
*.SUBCKT Filter_Model CONDO_V COND1_V N_MASTER_UZ
*.ENDS
.model Filter_Model external winfile="..\Reduction\Filter|.c"
```

Figure 131: **.model** statement

Setting Analysis Type

- ☑ Insert a blank line at the end of the netlist and before the `ì * End of main circuit: Filter simulationì` line.
- ☑ Select the **Command Tool** button .
- ☑ Select **Analysis > Transient** in the opening dialog box (Figure 132).

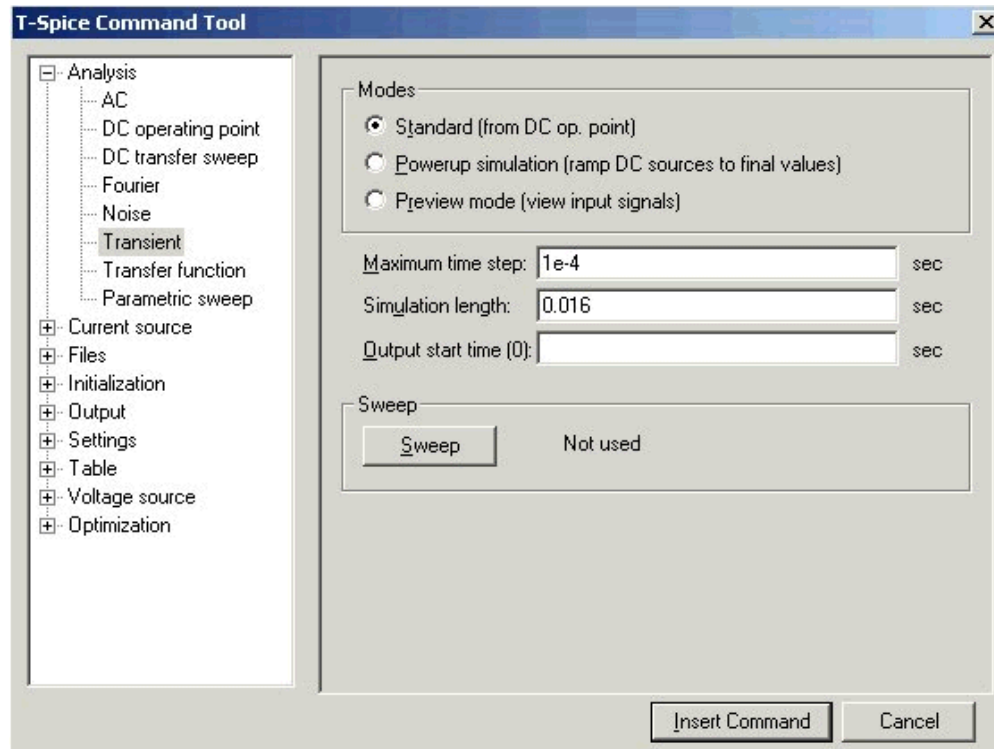





Figure 132: T-Spice Command Tool dialog

- ☑ Check the **Standard** radio button to specify the **Modes**.
- ☑ Set the **Maximum** time step to 1e-4, the **Simulation** length to 0.016.
- ☑ Click **Insert Command**.
- ☑ Close the Command tool.
The inserted command is : `.tran 1e-4 .016`
- ☑ Add a new line.
- ☑ Select the **Command Tool** .
- ☑ Select **Settings > Simulation Options**.
- ☑ Set the **Option Name** to `numnt (itl4)` and the **Value** to `500`.

- ☑ Click **Insert Command**.
- ☑ Close the Command tool.
- ☑ Add a new line.
- ☑ Select the **Command Tool** .
- ☑ Select **Settings > Simulation Options**.
- ☑ Set the **Option Name** to *absi (abstol)* and the **Value** to *1e-12*.
- ☑ Click **Insert Command**.
- ☑ Close the Command tool.
- ☑ Add a new line.
- ☑ Add the last Simulation Option: Set the **Option Name** to *Method* and the **Value** to *Gear*.

- ☑ Select the **Command Tool** button .
- ☑ Select **Output > Transient Results**.
- ☑ Set the **Plot Type** to **Voltage**, the **Node Name** to **Vin**.
- ☑ Click **Add**.
- ☑ Set the **Node Name** to **Disp** and click **Add**.
- ☑ Click **Insert Command**.

The newly inserted command reads : `.print tran v(Vin) v(Disp)`

Note


With the **.print** command, the simulation results at the specified nodes will be automatically displayed.

Starting Simulations

The simulation toolbar can be used to start, stop, and pause a simulation (Figure 133).



Figure 133: Simulation toolbar

- ☑ To start the simulation, press the **Start** button .
- ☑ During the simulation, a text window opens displaying relevant information (Figure 134).

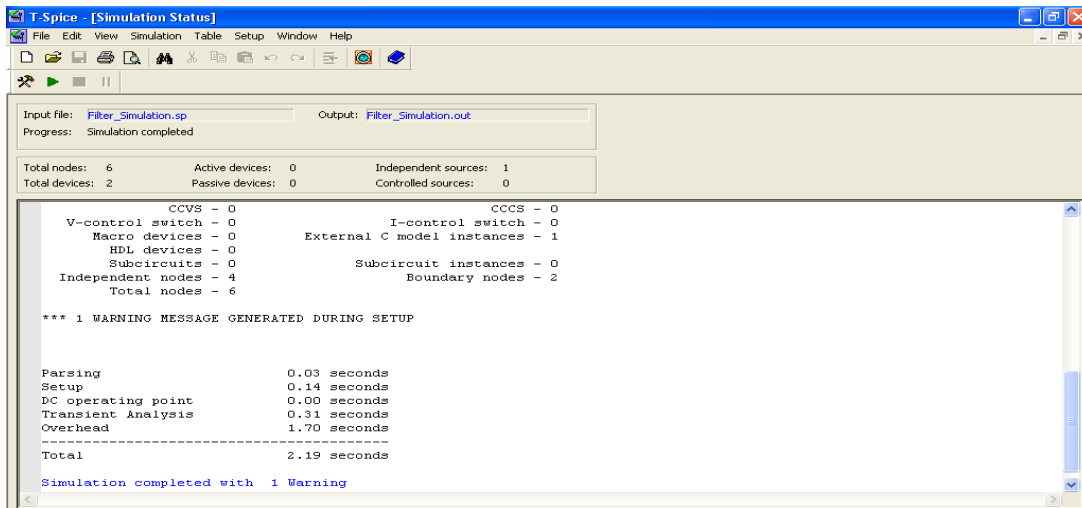
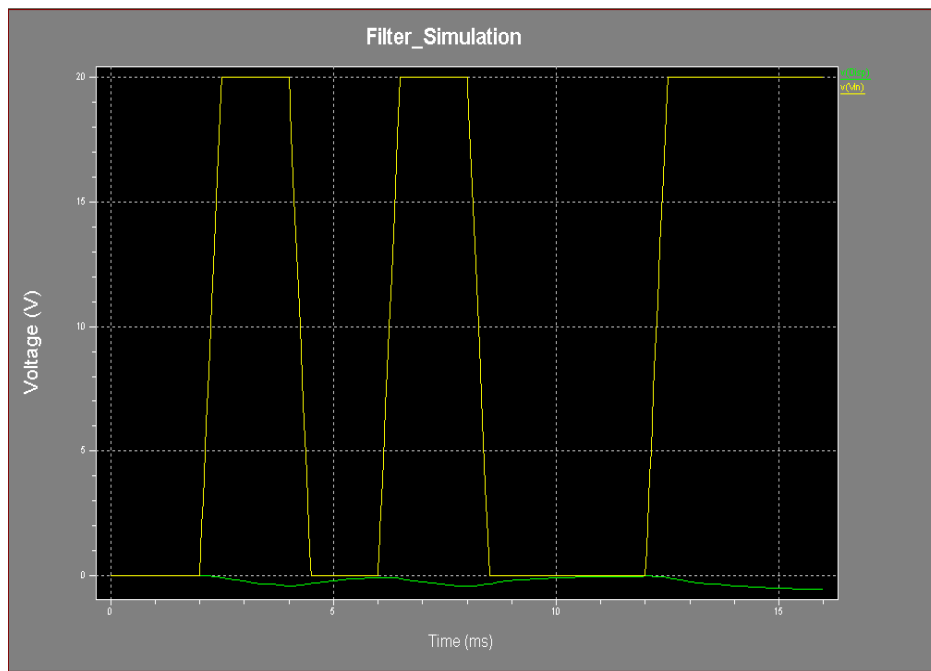


Figure 134: Simulation Output window

Note

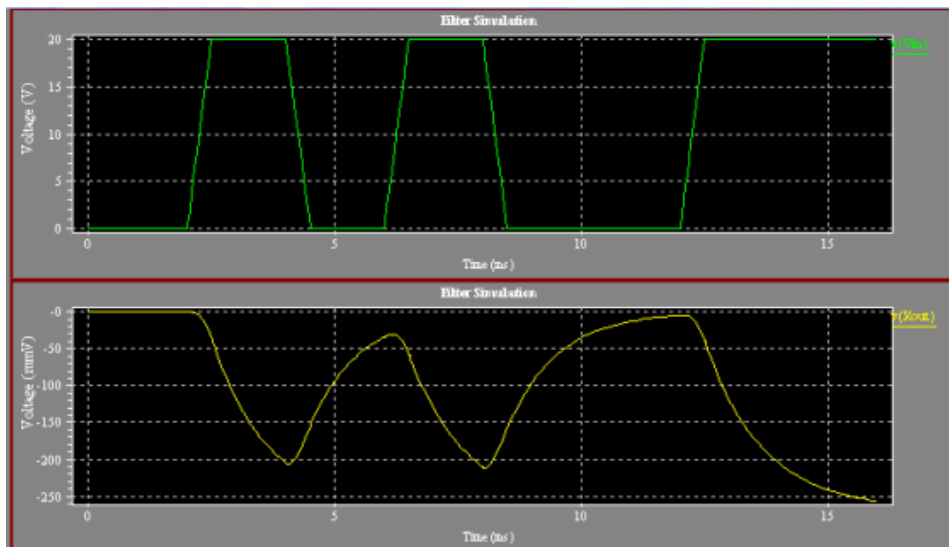
Setting the output option (**.print** statement) this time, the specified results will be displayed after simulation is finished.

The waveform viewer (W-Edit) opens and displays the simulation results at the probed nodes (Figure 135).

Figure 135: **Waveform Viewer** window

- Click the **Expand Charts** icon.

The **Input** and **Output** plots are displayed in two separate charts (Figure 136).

Figure 136: **Input** and **Output** plots

9 Boundary Conditions Tutorial

▪ Introduction	168
▪ Applying Boundary Conditions	169
▪ MultiPhysics Problem	170
▪ Opening Layout	171
▪ Boundary Conditions Setup	172
▪ Additional BC Settings	173
▪ Set Boundary Conditions	174
▪ Create Voltage BC Tag	175
▪ Create Temperature BC Tag	177
▪ Create Structural BC Tag	179
▪ Create BC Tags for the Second Anchor	180
▪ Setting Material Database	182
▪ Using Correct Units in MEMS Simulation	184
▪ Create 3D Model	186
▪ View Boundary Conditions	187
▪ Prepare 3D Model	188
▪ Export 3D Model	189
▪ Start ANSYS	190
▪ Load 3D Model	191
▪ Set Element Type	192
▪ Meshing	193
▪ Check Boundary Conditions	195
▪ Obtain Solution	196
▪ Plot Temperature Results	197
▪ Plot Transformation	198
▪ Plot Voltage Results	199
▪ Plot Displacement Results	200
▪ Plot Stress Results	201
▪ Animate Displacement Results	202

Introduction

MEMS Pro includes a new mechanism that allows you to assign ANSYS loads, or Boundary Conditions (BCs) from within MEMS Pro. This includes BCs and externally or internally applied forcing functions. These BCs will be transferred to the exported 3D model to be read by ANSYS.

The BCs can be assigned from L-Edit through the **BC Toolbar**.

BCs are assigned using a special BC Layer. Drawing objects created on this layer become BC Tags once you have assigned BC properties to them. Ports can be used to assign points and wires can be used to assign lines. Then, these BC Tags are mapped into the 3D model as boundary conditions that are exported to the ANSYS file.

The use of the BCs confers the following advantages:

- BCs are not associated with any drawing object, so they can be applied anywhere.
- Any shape supported by the standard L-Edit drawing interface can be drawn. Points and lines can be drawn using the port and wire objects.
- By using the standard L-Edit interface, you can easily hide / show and edit objects on this layer as any other L-Edit layer.

The only inconvenience is that since the BCs are not dynamically linked to any layout objects, you will have to manually change them when you change the layout.

Applying Boundary Conditions

Electric Thermal Actuator

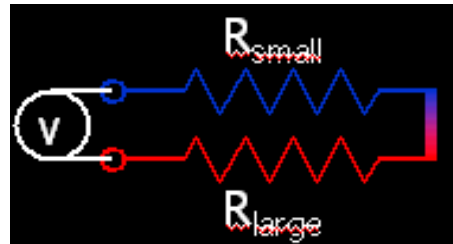


Figure 137: Electric Thermal Actuator.

$$P = V \cdot I = I^2 R$$

$$R \propto 1/w \rightarrow P \propto 1/w$$

- Wider beam: colder.
- Narrower beam: hotter.



- Result: differential thermal expansion

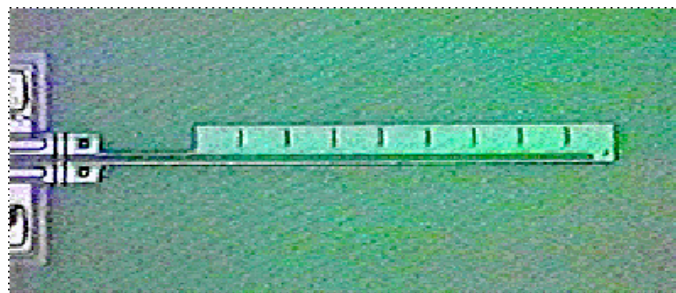
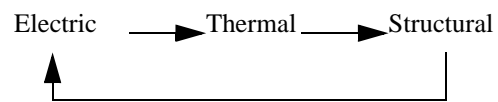


Figure 138: Differential Thermal Expansion.

MultiPhysics Problem

ANSYS can be used to solve complex problems in multiple domains. These problems can be solved indirectly or directly.

- Indirectly: By iterating between the disciplines.



- Directly: By using elements that support all disciplines.

Opening Layout

- ☑ Create a folder named "C:\BCTutorial".
- ☑ Copy the layout file from "*{MEMS Pro installation directory}\MEMS_Tutorials\BC\heat.tdb*" to the newly created folder "BCTutorial".
- ☑ Launch **L-Edit** by clicking **Start > All Programs > MEMS Pro v8.0 > MEMS L-Edit**.
- ☑ Click **File > Open**, and open the file **C:>BCTutorial>heat.tdb**.

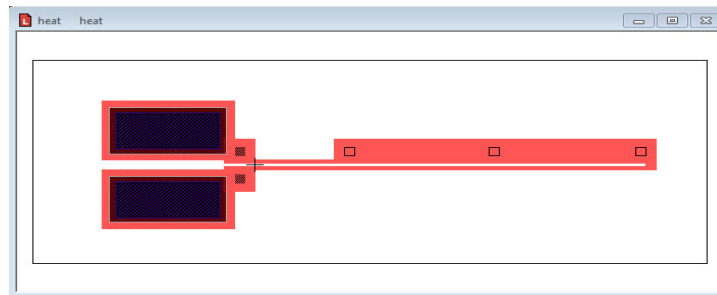


Figure 139: Thermal Actuator Layout File.

Boundary Conditions Setup

- ☑ In the MEMS Pro floating toolbar, select **Settings** from the **BC Tags** menu.
- ☑ Enable the boundary conditions settings by checking the **Include Boundary Conditions** box.
- ☑ Create a layer for boundary conditions by clicking the **Create BC Layer** button, and enter "*myBCLayer*" as the layer name in the **Create BC Layer** dialog.

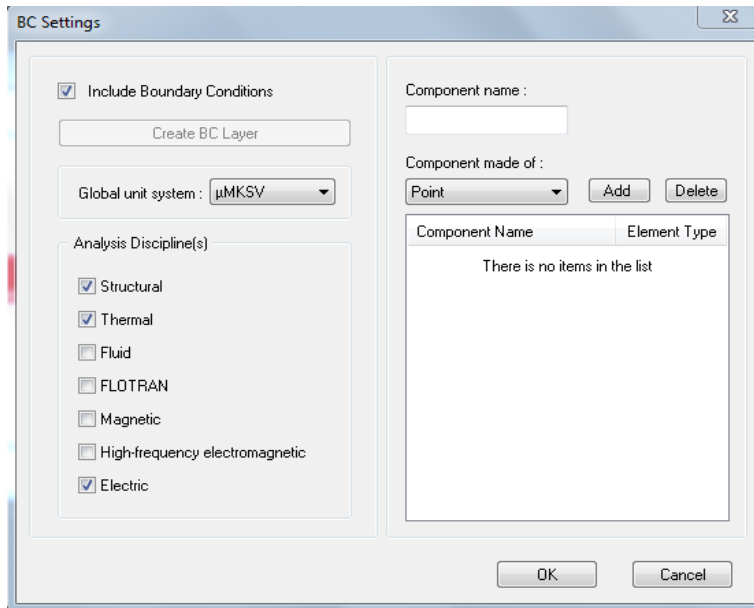


Figure 140: Create BC Layer dialog- Include BC.

- ☑ Use **µMKS** as the default unit settings.
- ☑ Uncheck physics that are not used: Fluid, Fltotran, Magnetic, and High-Frequency Electromagnetic.

Additional BC Settings

- You can add component names to the 3D model, so that they can be easily identified in **ANSYS**.
 - ☑ In the **BC Settings** dialog, enter the name of the component (*Anchor1*) in the **Component name** field.
 - ☑ Set the element type to "**Area**" from the **Component made of** drop down list.
 - ☑ Repeat the previous steps to add (*Anchor2*).

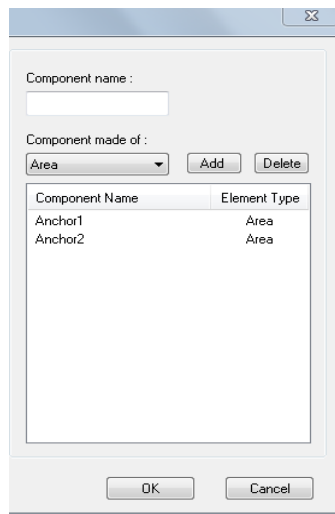


Figure 141: BC Settings dialog.

- ☑ Click **OK** to exit the **BC Settings** dialog.

Set Boundary Conditions

- Electrical boundary conditions: voltage

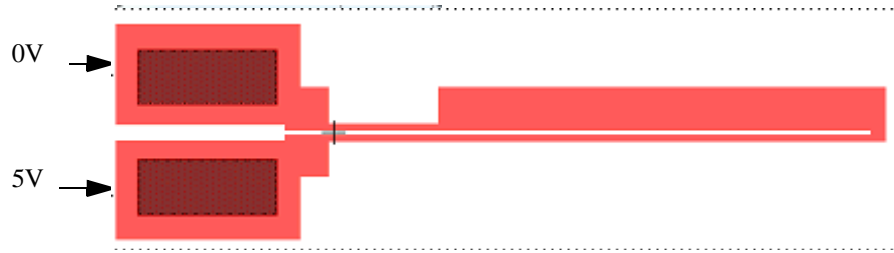


Figure 142: Electric Boundary Conditions-Voltage

- Thermal boundary conditions: temperature



Figure 143: Thermal Boundary Conditions-Temperature.

- Structural boundary conditions: displacements.

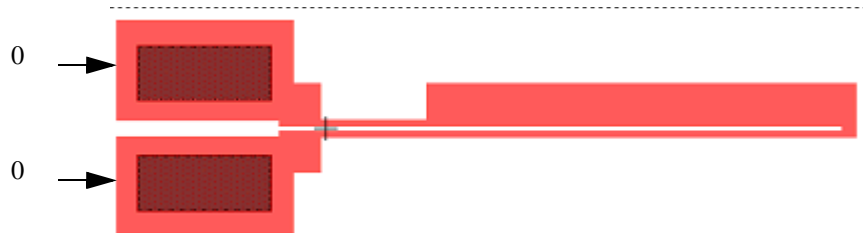


Figure 144: Structural Boundary Conditions-Displacements.

- Boundary conditions are specified by creating tags on the layout using the BC layer.
- Zoom in to the top anchor as shown in (Figure 145).

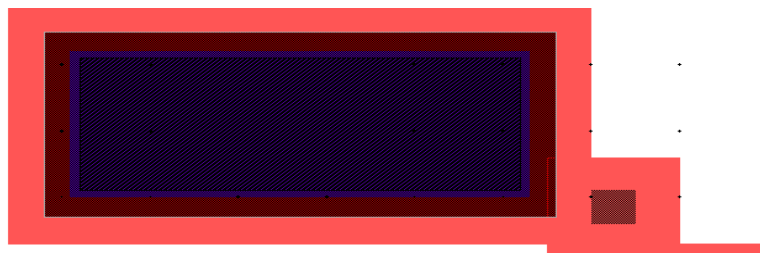


Figure 145: Preparing the Creation of BC Tags.

Create Voltage BC Tag

- ☑ Select the box tool from the drawing toolbar.
- ☑ Select the "myBCLayer" from the layer palette. This should be the last layer in the palette.
- ☑ Draw a box over the anchor, covering it completely as shown in (Figure 146).

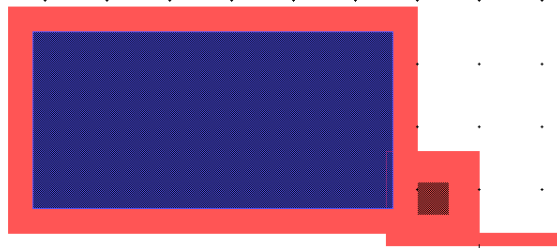


Figure 146: Creating Voltage BC Tags

- ☑ With the BC box selected, go to the **MEMS Pro** floating toolbar and select **Create/Modify** from the **BC Tags** menu.
- ☑ To specify the voltage BC tag as shown in (Figure 147):
 - Enter *Voltage1* in the **BC Label** field.
 - Select *Anchor1* from the **Component Name** drop down list.
 - Select *PolySilicon* from the **Material Name** drop down list.
 - Select *Top* from the **Placement** drop down list.
 - Select any color from the **Color Display** drop down list.
 - Select *Electric* from the **Type** drop down list.
 - Select *Electric Scale Potential* from the list box.
 - Enter *0* in the **VOLT** field.
 - Select the **Enable BC Tag** check box.

Click **OK** to exit.

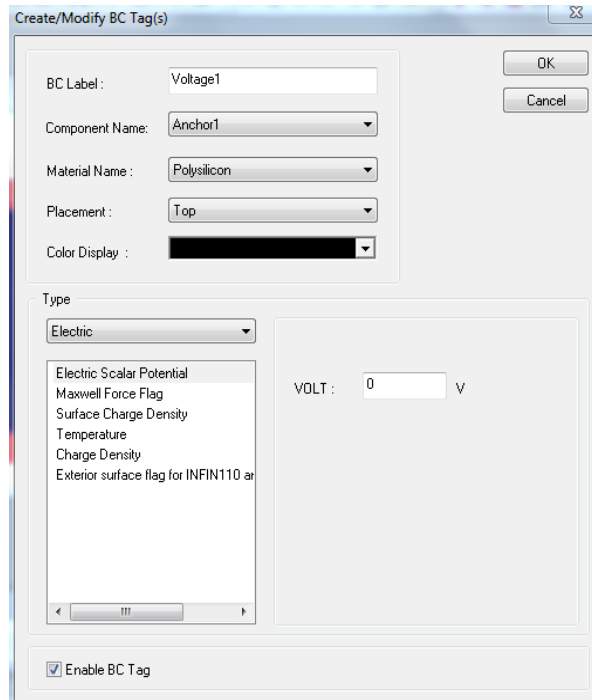


Figure 147: Specifying Voltage BC Tags.

Create Temperature BC Tag

- ☑ Draw another box on the "myBCLayer" layer, similar in size and location to the first one, as shown in (Figure 148).

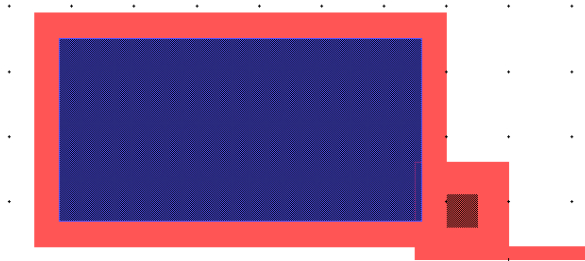


Figure 148: Creating Temperature BC Tag.

- ☑ With the BC box selected, go to the **MEMS Pro** floating toolbar and select **Create/Modify** from the **BC Tags** menu.
- ☑ To specify the temperature BC tag as shown in (Figure 149):
 - Enter *Temp1* in the **BC Label** field.
 - Select *Anchor1* from the **Component Name** drop down list.
 - Select *PolySilicon* from the **Material Name** drop down list.
 - Select *Bottom* from the **Placement** drop down list.
 - Select any color from the **Color Display** drop down list.
 - Select *Thermal* from the **Type** drop down list.
 - Select *Temperature* from the list box.
 - Enter *303* in the **TEMP** field.
 - Select the **Enable BC Tag** check box.

Click **OK** to exit.

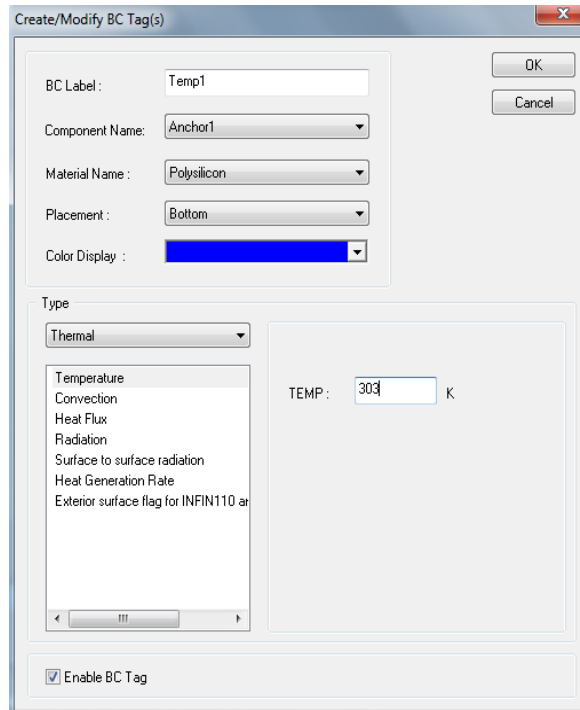


Figure 149: Specifying Temperature BC Tag.

Create Structural BC Tag

- ☑ Draw another box on the "myBCLayer" layer, similar in size and location to the previous one, as shown in (Figure 150).

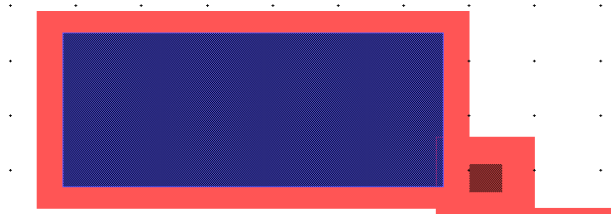


Figure 150: Creating Structural BC Tag.

- ☑ With the BC box selected, go to the **MEMS Pro** floating toolbar and select **Create/Modify** from the **BC Tags** menu.
- ☑ To specify the structural BC tag, as shown in (Figure 151):
 - Enter *Struct1* in the **BC Label** field.
 - Select *Anchor1* from the **Component Name** drop down list.
 - Select *Polysilicon* from the **Material Name** drop down list.
 - Select *Bottom* from the **Placement** drop down list.
 - Select any color from the **Color Display** drop down list.
 - Select *Structural* from the **Type** drop down list.
 - Select *Displacements_Rotations* from the list box.
 - Set all the values as 0 in the following fields: **UX**, **UY**, **UZ**, **ROTX**, **ROTY**, **ROTZ**.
 - Select the **Enable BC Tag** check box.
- ☑ Click **Ok** to exit.

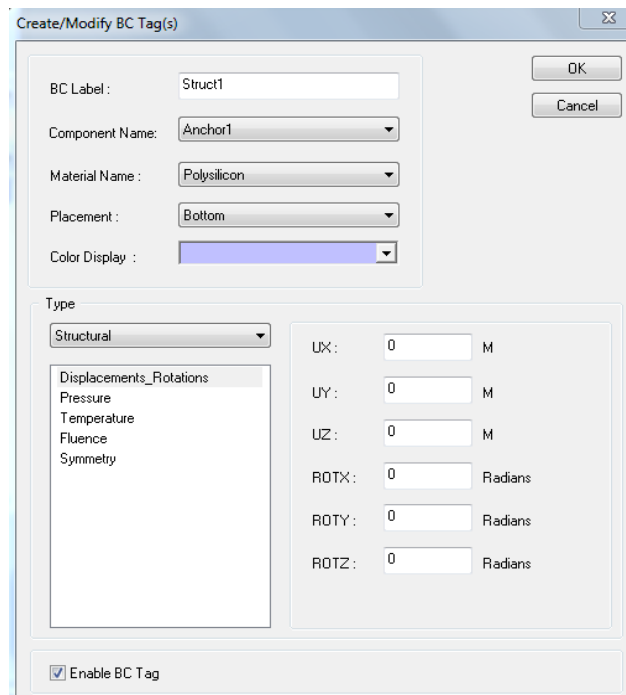


Figure 151: Specifying Structural BC Tag.

Create BC Tags for the Second Anchor

After performing the previous steps, copy the 3 BC tags created for the second anchor.

- As the 3 BC tags overlap the anchor layer, hide the anchor layer to avoid selecting it.
 - On the layer palette, find the icon for layer "Anchor1".
 - Middle click the icon to hide "Anchor1".
- Select the 3 BC tags:
 - Select the **Selection** tool from the drawing toolbar.
 - Draw a box around the 3 BC tags.
 - The status bar should display, *Selections: 3boxes*.
- Select **Copy** from the **Edit** menu.
- Zoom out to the whole layout (press the **Home** key).
- Zoom into the second (bottom) anchor. The grey anchor box in the middle should be hidden.
- Select **Paste** from the **Edit** menu.
- Show the "Anchor1" layer.
 - On the layer palette, middle click the "Anchor1" layer icon.
- Drag the pasted BC tags, so that they cover the "Anchor1" box completely, as shown in (Figure 152).
 - Middle click outside the selected objects and drag.

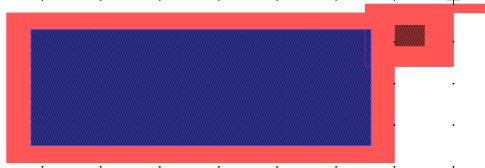


Figure 152: Pasted BC Tags.

- An identical set of BC tags has been created for the second anchor. Yet, they should be slightly different from the first anchor.
- List all BC tags:
 - In the floating **MEMS Pro Toolbar**, select **Browse** from the **BC Tags** menu to open the **BC Tags Browser** dialog, as shown in (Figure 153)

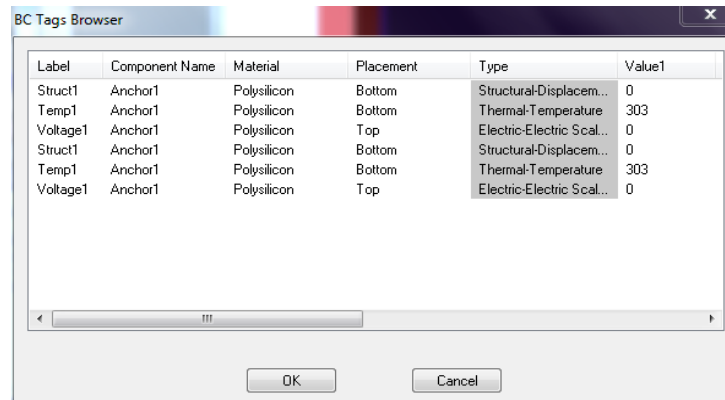


Figure 153: Listed BC Tags.

- ☑ Scroll to the end of the browser, and ensure that all tags have **Enabled** set to **On**.

A total of 6 BC tags are displayed.

- The first 3 tags are for the first (top) anchor.
- The last 3 tags are for the second (bottom) anchor.

- ☑ For the last 3 BC tags:
 - Change the labels to: *Struct2*, *Temp2*, and *Voltage2*.
 - Change the component names to "*Anchor2*".

- ☑ Change the value of "*Voltage2*" to 5.

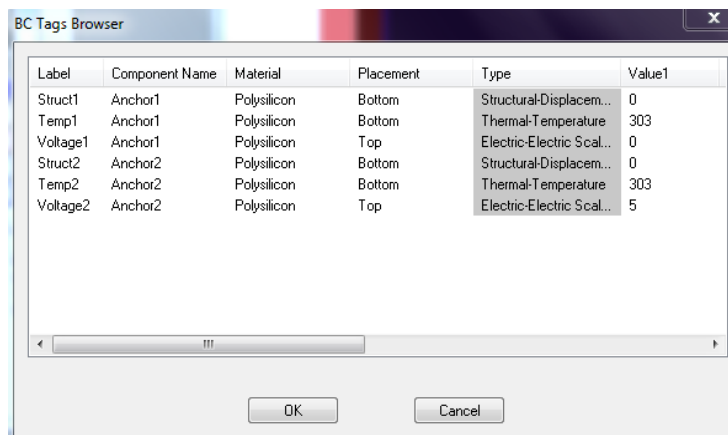


Figure 154: Modifying BC Tags.

- ☑ Click **OK** to exit.

Setting Material Database

- ☑ From the **MEMS Pro Toolbar**, select **«Edit Material Database»** from the **«3D Tools»** menu to display the **Material Database** dialog, as shown in (Figure 155).

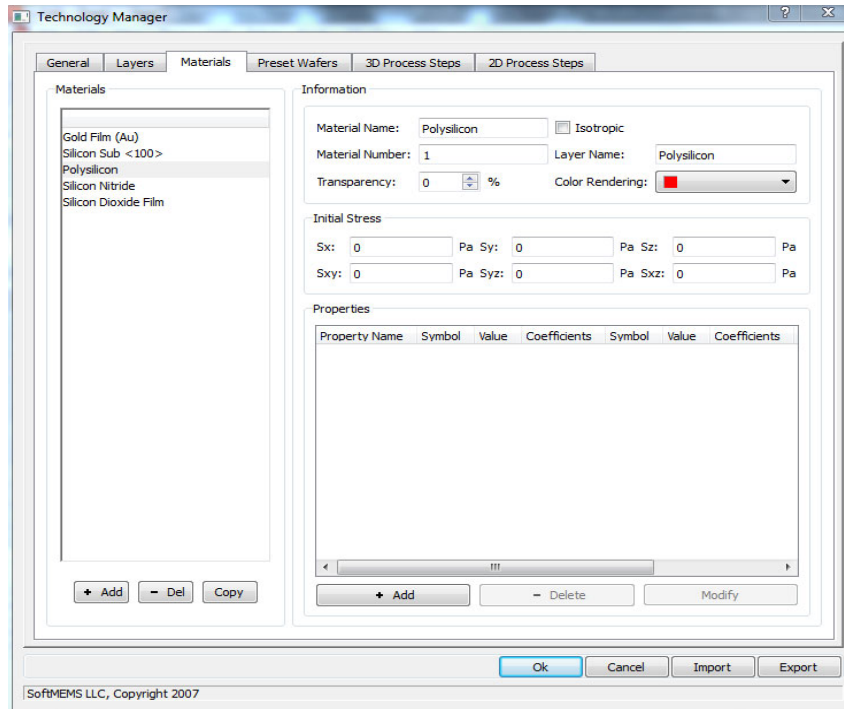


Figure 155: Material Database dialog.

- ☑ Select **«Polysilicon»** from the **«Materials»** list.
- ☑ Check the **Isotropic** option. By selecting this option, the material will have the same properties in all directions.
- ☑ Select the red color from the **«Color Rendering»** drop down list.
- ☑ In the **Material Database** dialog, click **«Add»** under the **«Properties»** list to open the **Material Properties** dialog, as shown in (Figure 156).

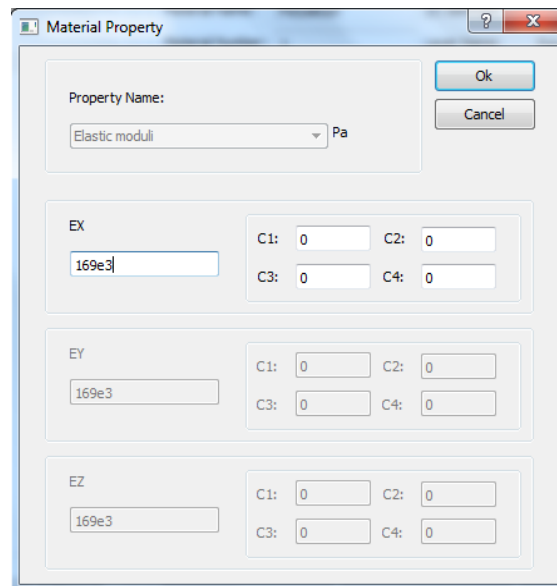


Figure 156: Material Properties dialog.

- ☑ Select **Elastic moduli** from the **Property Name** drop down list.
- ☑ Set **EX**= $169e3$.
- ☑ Click **Add**.
- ☑ Set **"Coefficients of Thermal Expansion"** **ALPX** = $2.9 e-6$.
 Set **"Thermal Conductivities"** **KXX** = $150 e6$.
 Set **"Electrical Resistivities"** **RSVX** = $2.3 e-11$.
 Set **"Major Poisson's Ratio"** **PRXY**= 0.22 .

Using Correct Units in MEMS Simulation

Some Finite-Element Modeling (FEM) tools are referred to as "unitless". This is true in a sense, yet the term is somewhat misleading. A 3D model in a FEM tool is simply a set of *numbers* defining the model dimensions, and the FEM system also has *numbers* corresponding to material properties and boundary conditions. Finally, the simulation output is generated, ultimately, as *numbers* also. The whole problem of using the "correct" units in a FEM simulation revolves around making sure that all of these numbers are compatible. The situation is made more confusing by the fact that there are a variety of ways to achieve this compatibility.

Consider a simple simulation program that allows the user to:

- 1) Draw a line of arbitrary length on the screen.
- 2) Manually enter a number, and,
- 3) Press the **Solve** button.

The length of the line drawn on the screen is used to generate a distance number in the simulation. The manually entered number is used as a rate number in the simulation. When you press the **Solve** button, the solver simply divides the distance number by the rate number. This will give a time number as a resulting output. The user can draw a line of length 10, enter a rate value of 2, and get an output time value of 5. The physical problem being solved may correspond to 10 km and 2km/hr, but the input is just 10 and 2. The units of the input would then indicate that the output time corresponds to 5 hours. The user might choose to do the *same* problem in a new system of units, for example meters and meters/minute, but the input numbers must change to reflect this. The distance would be 10000, the rate would be 2000/60, and the output result would be 300 (minutes, the same as 5 hours). The simulation in this case *is* unitless, but the user is imparting units to the model dimension, imparting units to a simulation parameter, and interpreting the output in the context of these units. **The actual values of the numbers used in the simulation serve to define the system of units.** The situation in a FEM tool is analogous.

If your 3D model is based on a length unit of meters (hence making the typical numerical dimensions of a MEMS model be on the order of 1e-3 to 1e-6), then your material property values and boundary conditions should be in the SI (or MKSV- meter, kilogram, second, volt) system of units that is used in the macroscopic world. If, however, your 3D model is based upon the more MEMS-friendly length unit of microns (making the typical numerical dimension of a MEMS model be on the order of 1000 to 1), then clearly a different system of units will be required. That system is called μ KSV (micron, kilogram, second, volt); the length is measured in microns, mass in kilograms, time in seconds, and electric potential in volts. This system of *fundamental* units allows to develop a system of *derived* units that is compatible with a micron-based 3D model, and FEM simulations can be done in the relevant physical domains: statics, dynamics, thermal (degrees Kelvin are still used), electrostatics...etc.

The unit of pressure (used for the Young's Modulus material property as well as applied boundary conditions) serves as a good example of a derived unit. Recall that the MKSV unit of pressure is the Pascal, where

$$1 \text{ Pa} = 1 \text{ N/m}^2 = 1 (\text{Kg} \cdot \text{m/s}^2) / \text{m}^2$$

By analogy, the μ KSV unit of pressure is:

$$1 (\text{Kg} \cdot \mu\text{s}^2) / \mu^2 = 1 \text{e}6 (\text{Kg} \cdot \text{m/s}^2) / \text{m}^2 = 1 \text{ MPa}$$

Hence, if the derived unit of pressure is the MPa and the material property value (for example for polysilicon) is supposed to be 169 GPa or 169e9 Pa, then the numerical value of the material property should be 169000. In terms of the value *and* the units, 169000 MPa *is* actually the correct material property value of 169 GPa. The FEM solver does not, in any sense, "know" that the correct unit of pressure is MPa in this case. Simply by typing in a numerical value of 169000 for the Young's Modulus, the user is asserting that the unit of pressure is the MPa, and that the 3D model is in microns. This is the hazard- incorrectly using the SI value of 169e9, for the material property in this case would assert that the dimensions of the 3D model are in meters.

For completeness, it should be mentioned that the electrostatic simulations do bring in the issue of a new "choice" of parameters (voltage units vs. charge units), and as seen above, the choice is made to be the Volt unit (since this is typically an experimentally applied unit). The consequence of this choice is that the unit of charge is NOT the Coulomb. This can be seen by considering the Volt as a Joule/Coulomb. As $1 \text{ J} = 1 \text{ N} \cdot \text{m} = 1 (\text{Kg} \cdot \text{m}/\text{s}^2) \cdot \text{m}$, in μKSV this would be

$$1 (\text{Kg} \cdot \mu/\text{s}^2) \cdot \mu = 1\text{e-}12 \text{ J}$$

In order to keep the Volt as the standard unit of potential, the unit of charge needs to become 1e-12 C or a pC.

More derived units and additional content regarding units in MEMS can be found on pages 20-21 in the file **MemsModeler_UserManual_v3.1c.pdf**, which can be found in the following directory: **{MEMS Pro installation directory}\MEMS Model Reduction\doc\MemsModeler\v3.1\pdf**.

Create 3D Model

- ☑ From the floating **MEMS Pro Toolbar**, select **View 3D Model** from the **3D Tools** menu.
- ☑ Click **"Regenerate"** when asked to View or Regenerate the 3D model. The created 3D model will be displayed as shown in [\(Figure 157\)](#).

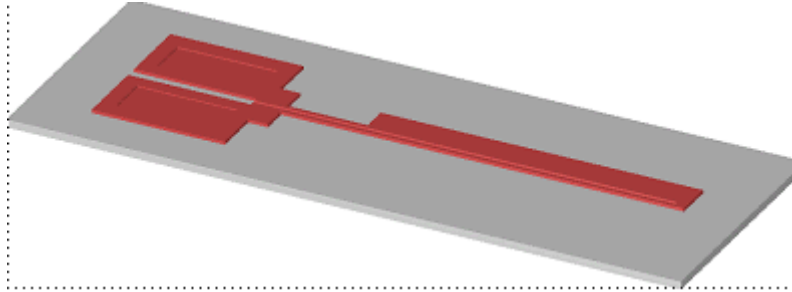


Figure 157: Created 3D Model.

View Boundary Conditions

- ☑ Hide the **Gold Film (Au)** material to better view the upper boundary conditions.
- ☑ At the right end of the **3D Model View** toolbar, click the **Show/Hide Boundary Conditions** button



You should see the top surface areas of the two anchors changing color, as shown in (Figure 158). These are the two voltage boundary conditions on the top surface.

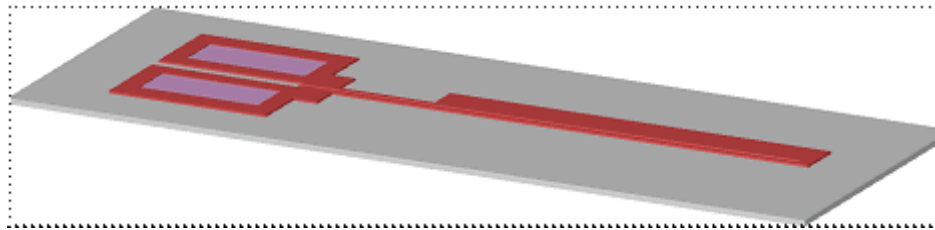


Figure 158: Boundary Conditions on a 3D Model.

Prepare 3D Model

- ☑ You do not need to export the substrate and the silicon nitride materials into **ANSYS**.
-On the 3D layer palette, middle click the first substrate and silicon nitride icons.
- ☑ Press the **Ctrl** button while dragging with the mouse left button to rotate.
You should see the two structural or temperature BCs on the bottom of the surface, as shown in (Figure 159).

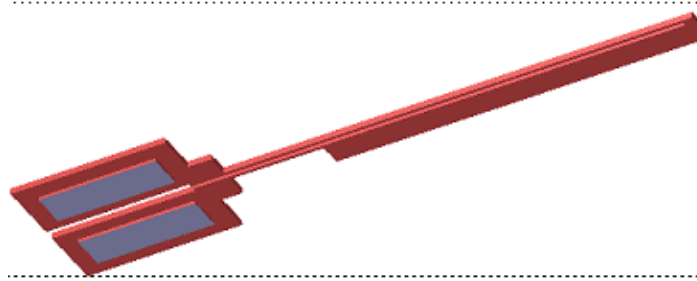


Figure 159: BCs on the bottom of the 3D surface.

Export 3D Model

Note

The boundary conditions should be visible on the 3D model to be included in the model export.

- ☑ In the floating **MEMS Pro Toolbar**, select **Export 3D Model** from the **3D Tools** menu to open the **Export 3D Model** dialog, as shown in (Figure 160).
- ☑ Make sure that the file name is pointing to the current *C:\BCTutorial* directory, with the file name as *Actuator.mac*.
- ☑ Select the file type **ANSYS APDL Files (*.mac)**.
- ☑ Click **OK**.

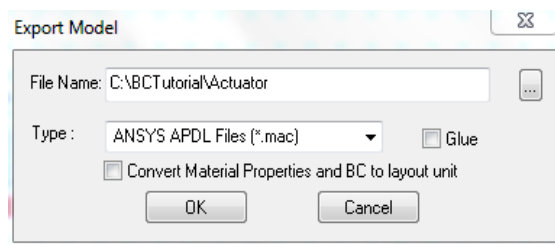


Figure 160: Export 3D Model dialog.

Before starting **ANSYS**, close the 3D model view, save the layout file and close **L-Edit**.

Start ANSYS

- ☑ Launch **ANSYS** by clicking **Start > All Programs > ANSYS 12.1 > Mechanical APDL (ANSYS)**. The main window will get displayed as shown in (Figure 161).

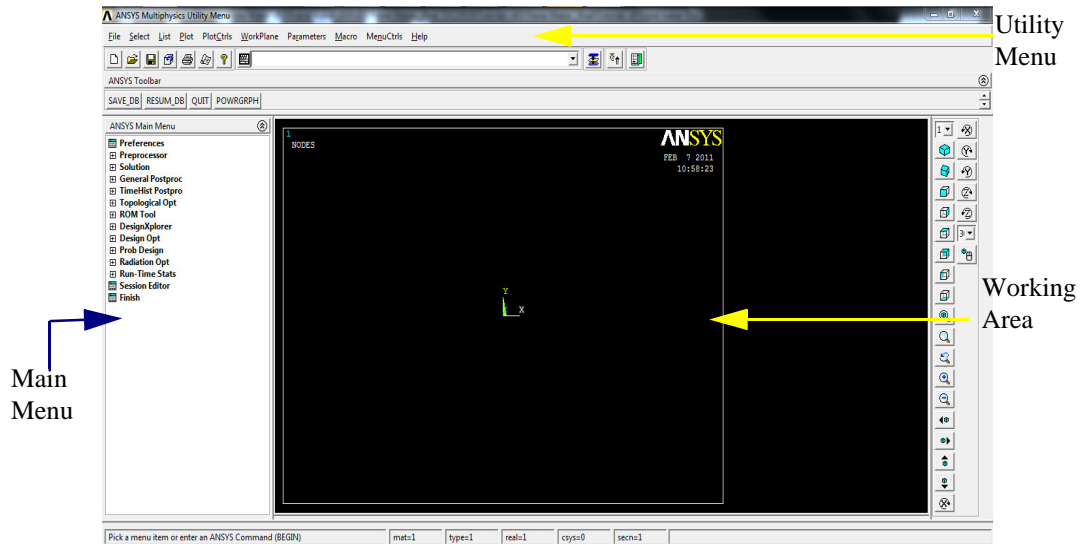


Figure 161: ANSYS Interface.

Load 3D Model

- ☑ Select **Change Directory** from the **File** menu in **ANSYS** and browse to C:\BCTutorial.
- ☑ Select **Read Input From** from the **File** menu in **ANSYS** and select **"Actuator.mac"**.

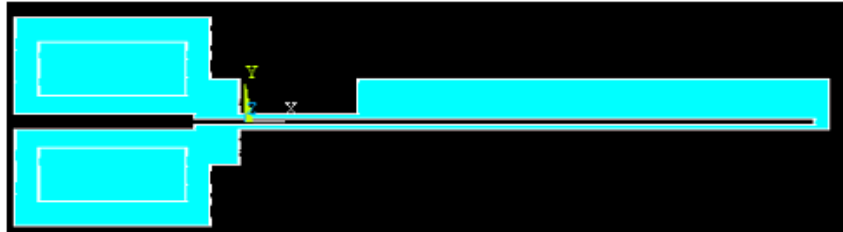


Figure 162: Loaded 3D Model.

- ☑ Click **"OK"**. The design gets loaded as shown in (Figure 162).

Note

ANSYS is unitless. You can use any unit system as long as it is consistent.

Set Element Type

- ☑ In the main menu select **Preprocessor**.
- ☑ From **Element Type**, select **Add/Edit/Delete** to open the **Element Types** dialog.
- ☑ Click **Add** to open the **Library of Element Types** dialog, as shown in (Figure 163).
- ☑ Select **Coupled Field**.
- ☑ Select **Scalar Tet 98**.

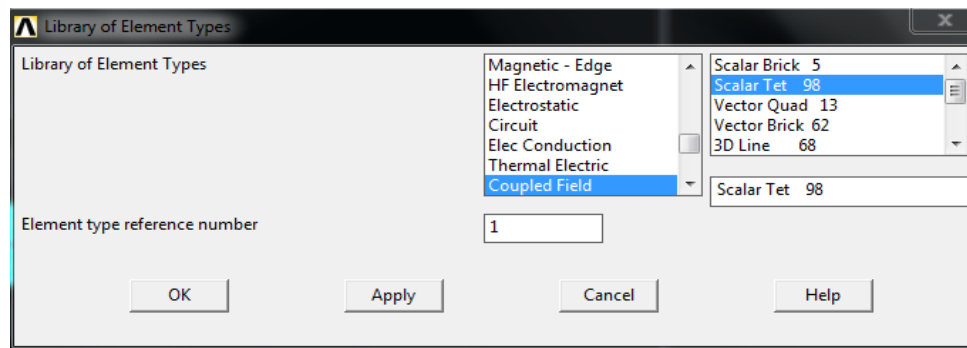


Figure 163: Library of Element Types dialog.

- ☑ Click **Ok**, then **Close** in the **Element Types** dialog.

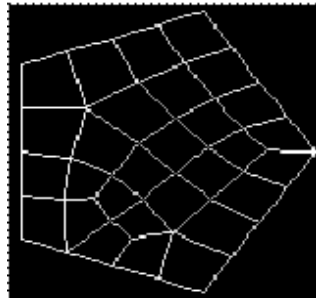
Meshing

Meshing Options

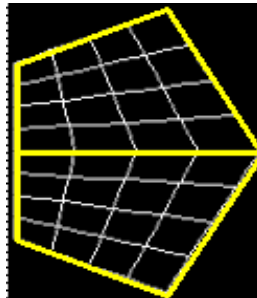
You can use Free Mesh as Mapped Mesh. Free Mesh generates irregular element shapes/sizes but it is easier and faster. On the other hand, Mapped Mesh generates regular element shapes/sizes, but the complicated shapes have to be subdivided into 3 or 4 sided polygons.

Generating the Mesh

- Free Meshing.



- Mapped Meshing:



- In the main menu select **Preprocessor**.
- From **Meshing**, select **Mesh Tool** to open the **Mesh Tool** dialog, as shown in [\(Figure 164\)](#).
- Check **Smartsize**.
- Slide bar to **10** - coarsest mesh.
- Click **Mesh**.

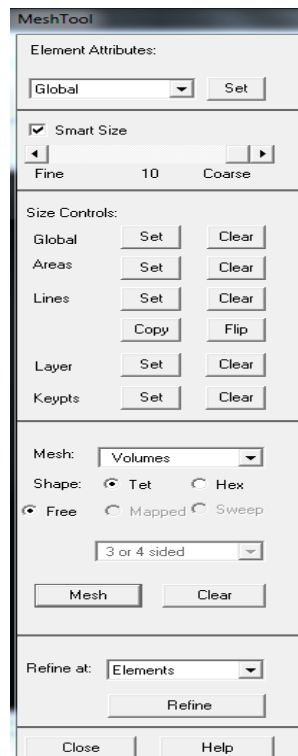
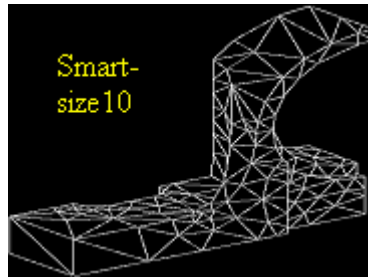
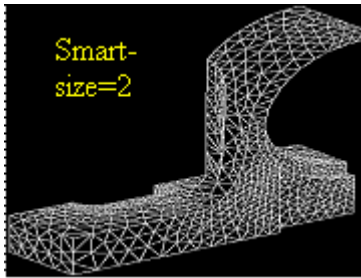


Figure 164: Mesh Tool dialog.

- Click **Pick All** and the volume meshing begins.

Check Boundary Conditions

- In the utility menu, select **Loads** from the **List** menu.
- Select **DOF Constraints > On All Areas**. The list of constraints is displayed as shown in (Figure 165).

```

LIST CONSTRAINTS ON ALL SELECTED AREAS
AREA      LOAD LABEL      VALUE(S)
   73      UOLT          0.0000      0.0000
   74      UOLT          5.0000      0.0000
  159      UX           0.0000      0.0000
  159      UY           0.0000      0.0000
  159      UZ           0.0000      0.0000
  159      ROTX        0.0000      0.0000
  159      ROTY        0.0000      0.0000
  159      ROTZ        0.0000      0.0000
  159      TEMP        303.00      0.0000
  161      UX           0.0000      0.0000
  161      UY           0.0000      0.0000
  161      UZ           0.0000      0.0000
  161      ROTX        0.0000      0.0000
  161      ROTY        0.0000      0.0000
  161      ROTZ        0.0000      0.0000
  161      TEMP        303.00      0.0000

```

Figure 165: Listed Constraints on Selected Areas.

- Click **Close** from the **File** menu.

Obtain Solution

- ☑ In the main menu select **Solution**.
- ☑ From **Solve** select **Current LSI** to open the **Solve Current Load Step** dialog, as shown in (Figure 166).

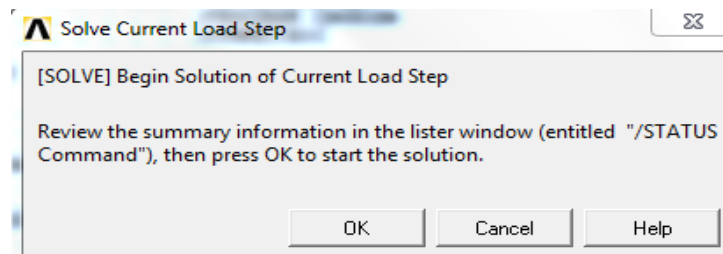


Figure 166: Solve Current Load Step dialog.

- ☑ Click **OK**.
- ☑ Ignore and bypass all warning messages.
- ☑ The solution begins.
This takes approximately 10-20 minutes depending on memory and processor speed.
The simulation status is displayed in an output window.

Plot Temperature Results

- ☑ In the main menu select **General Postproc** .
- ☑ From **Read Results** select **Last Set** .
- ☑ In the main menu select **General Postproc** .
- ☑ From **Plot Results** select **Contour Plot** then **Nodal Solu** .
- ☑ Select **DOF Solution** .
- ☑ Select **Nodal Temperature** .
- ☑ Click **Ok** .
- ☑ Temperature Profile is displayed, as shown in (Figure 167).
- Temperature range: 30 to 939 degree Celcius.

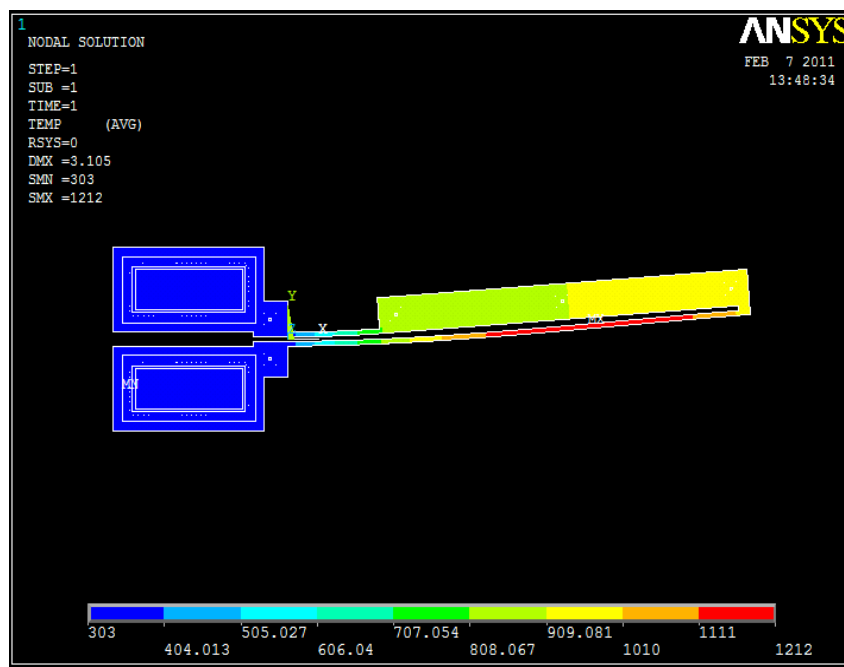


Figure 167: Plotting Temperature Results.

Plot Transformation

- ☑ In the utility menu select **PlotCtrls**.
- ☑ Select **Pan Zoom Rotate** to open the **Pan-Zoom-Rotate** window, as shown in (Figure 168).

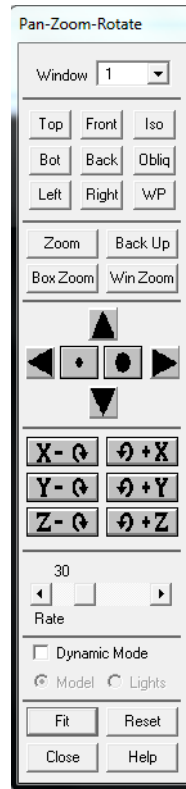


Figure 168: Pan-Zoom-Rotate dialog.

- ☑ Click **Front**. The corresponding graph is displayed as shown in (Figure 169).

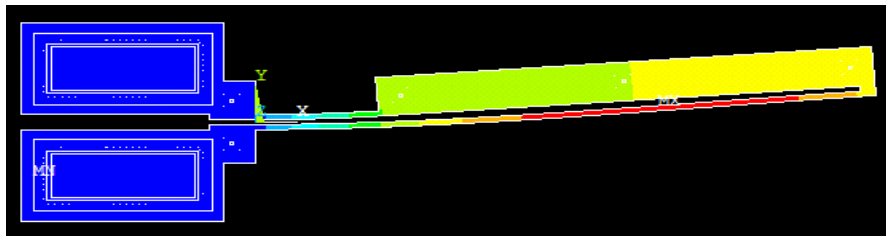


Figure 169: Plotting Transformation.

- Two pads are at the same temperature: 30.
- The narrow beam is at much higher temperature than the wider beam.

Plot Voltage Results

- In the main menu select **General Postproc** .
- From **Plot Results** select **Contour Plot** then **Nodal Solu** .
- Select **DOF Solution** .
- Select **Electric Potential** .
- Click **Ok** .

Voltage distribution profile is displayed, as shown in (Figure 170).

Voltage range: 0 to 5.

Greater voltage drop across narrow beam.

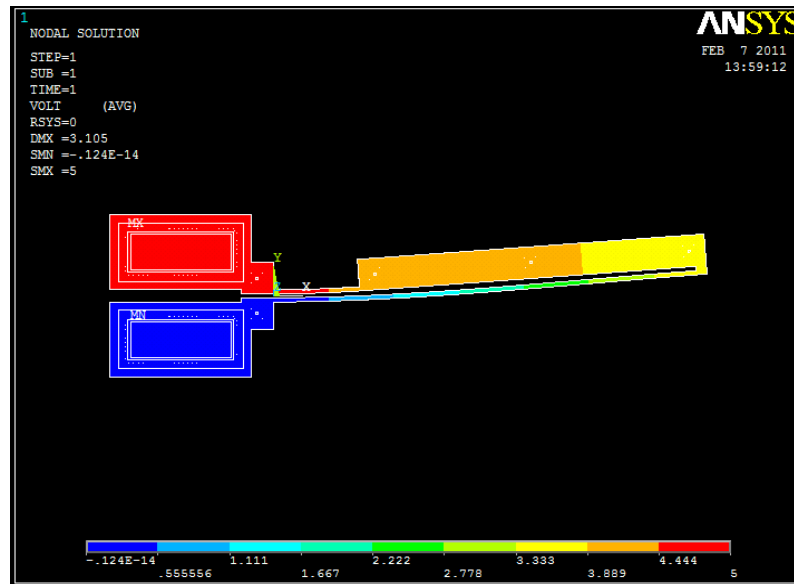


Figure 170: Plotting Voltage Results.

Plot Displacement Results

- ☑ In the main menu select **General Postproc** .
- ☑ From **Plot Results** select **Contour Plot** then **Nodal Solu** .
- ☑ Select **DOF Solution** .
- ☑ Select **Y-Component of Displacement** .
- ☑ Click **Ok** .

Displacement is displayed, as shown in (Figure 171).
Maximum displacement: 3.047 um at tip.

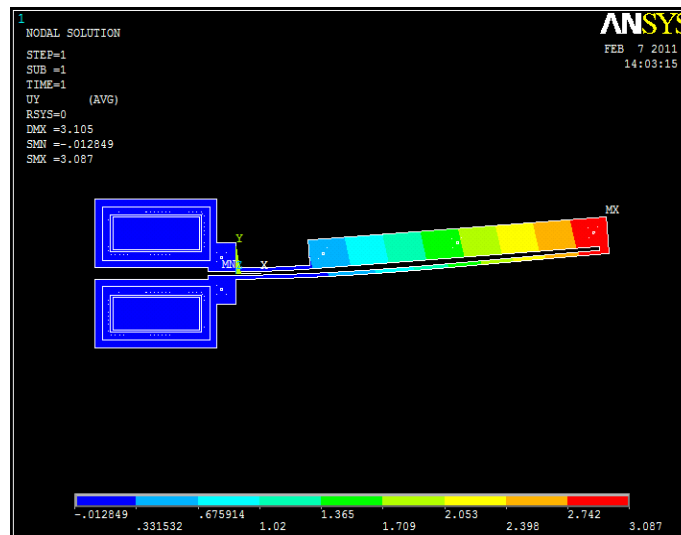


Figure 171: Plotting Displacement Results.

Plot Stress Results

- In the main menu select **General Postproc** .
- From **Plot Results** select **Contour Plot** then **Nodal Solu** .
- Select **Stress** .
- Select **von Mises stress** .
- Click **Ok** .

Highest stresses occur on thin arm and anchors, as shown in (Figure 172).

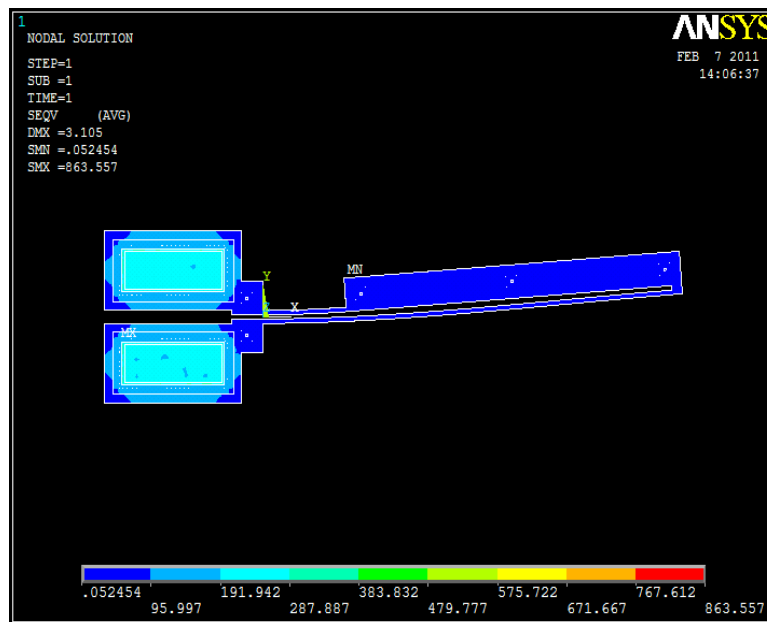


Figure 172: Plotting Stress Results.

Animate Displacement Results

- ☑ In the utility menu select **PlotCtrls** .
- ☑ Select **Animate** then **Deformed Results** .
 - Number of frames: 10
 - Delay: 0.5 sec
 - DOF Solution: TEMP

The Animation Controller is displayed as shown in (Figure 173).

- ☑ Click **"Stop"** to stop animation.
- ☑ Click **"Close"** to close controller.

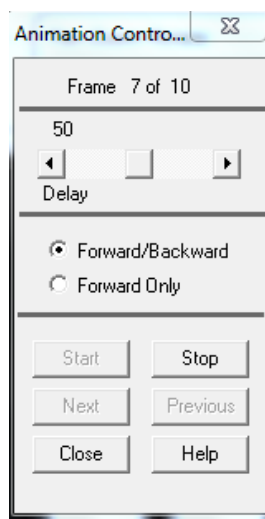


Figure 173: Animation Controller dialog.

- ☑ From the **File** menu click **Exit**: **Quit, no save.**

INDEX

Numerics

3D model	
generation	32
view	30, 34
3D Modeler	
multiple views	33

A

All-angle wire	37
----------------	----

B

Bonding pad	
generation	26
Box	
drawing	40

C

Circle	
drawing	40
Comb-drive	
generation	23
instantiation	10
Connecting	
global nodes	12
Creating	
a module	8
a schematic	7
Cross-section	34

D

Design	
optimization	95
Drawing	36
a box	40
a circle	40
a curved polygon	38
a torus	38
a wire	37

E

Editing	
object properties	13
Extracting	
a netlist	28

F

Folded spring	
generation	25
instantiation	10

G

Generating	
a 3D model	32
a comb-drive	23
a ground plate	26
a plate	22
bonding pads	26
folded springs	25
Global nodes	
connection	12
Ground plate	
generation	26

I

Instantiating	
a plate	9
comb-drives	10
components	9
folded springs	10
voltage sources	12

L

Layout	
extraction	111
Layout Versus Schematic	
See LVS	
LVS	116
extracting schematic	114
launching	116

M

MEMS Library Palette	
using	22
Module	
creation	8

N

Netlist
 comparison 116
 extraction 28

Nodes
 labeling 14

viewing 18

Wire
 drawing 37

Wire tool 11

O

Object properties
 edition 13

Optimization 95
 examining the output 105
 setting up 97

Optimizing
 a design 95

P

Plate
 generation 22
 instantiation 9

Polygon
 drawing (curved) 38

Probing 16

Process definition
 import 31

Properties
 view 28

S

Schematic
 creation 7
 extraction 114

T

Torus
 drawing 38

V

Viewing
 a 3D model 30, 34
 a waveform 18
 properties 28

Voltage sources
 instantiation 12

W

Waveform
 probing 17