

Dalhousie University
Department of Electrical and Computer Engineering
ECED 6240 – CMOS - MEMS

Lab #1 MEMS Pro Tutorial

* Lab report due: Feb. 27th

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.

The design example 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.

- Before you start, create a folder in the C:\ECED6240 directory and save all the files you will generate into your folder.
- Your lab report should include a summary of the lab, the schematic of the resonator, the displacement waveform and the layout of a simple mechanical structure.
- Write a few lines regarding your layout, e.g. the dimension, materials used etc.

1. 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 >Programs > MEMS Pro v6.0> S-Edit".
- ✂ The S-Edit user interface consists of six areas:

- ✎ The *title bar* (at the very top of the application window) contains the file name, the module name, the page name, and the mode
- ✎ The *menu bar* (at the top) contains commands
- ✎ The *palette bar* (on the left) contains tool icons
- ✎ The *status bar* (at the bottom) contains runtime information
- ✎ The *display area* (in the center) contains the schematic
- ✎ The *standard commands toolbar* (below the menu bar) contains often used commands

Designs are contained in *files*, each of which consists of one or more *modules*. Modules are viewed in either of two modes:

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

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.
- ✎ *Instances:* "copies" of other modules, dynamically linked to their originals. Instances are displayed in a design using the module's symbol page.

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.sdb** file under **C:\Program Files\SoftMEMS\MEMS Pro v6.0\Tutorial\Resonator**.

- ✂ Select **File > Open** to open this file.
- The current (visible) file, module, page and mode are named at the top of the title bar.
The schematic view of the resonator appears in [Figure 1](#).

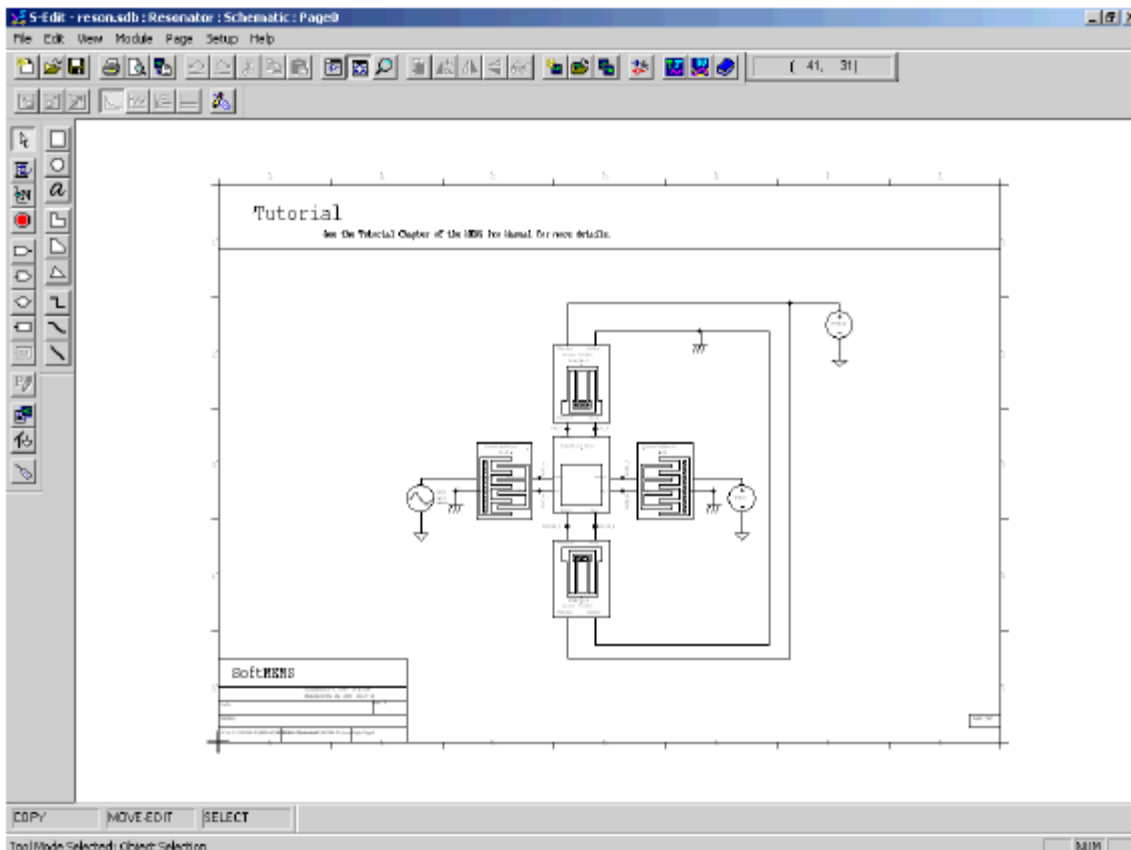


Figure 1: Schematic view of the complete resonator

Creating a New Module

To initiate your new resonator design, you must first create a new module.

- ✂ Select **Module > New** to create a module.
- ✂ In the **Module Name** edit field, enter **MyResonator** and click **OK**. Now would be a good time to save a copy of the file.
- ✂ Select **File > Save As** to invoke the **Save As** dialog.
- ✂ Select the C:\ECED6240\Your_own_folder, enter **myreson.sdb** as the filename, and click the **Save** button.

Instantiating Components

Instantiating a Plate

- ✂ Select **Module > Instance** to invoke the **Instance Module** dialog.
- ✂ Select **RigidPlate_DirX** as the **Module to Instance** and click **OK**.

RigidPlate_DirX, a four-sided plate with eight points of connection (pins), will appear at the center of the schematic page.

- ✂ Home the view by selecting **View > Home** or by pressing the **Home** key. The view of the plate will be resized so that the plate fills the contents of the window.

Instantiating Comb-drives

- ✂ Instantiate the **LinearCombDrive_DirX** module as you instantiated the plate. The newly instantiated **LinearCombDrive_DirX** will appear above the plate in the middle of the schematic window. You will have to move it to a new location using the S-Edit click and drag feature.

Note Objects in S-Edit can be moved by pressing the **Alt** key and left-clicking to drag them.

- ✂ Place the comb-drive to the right side of the previously instantiated plate.
- ✂ Place a second comb-drive into the design by copying the instance. With the first comb instance selected, select **Edit > Copy**, then **Edit > Paste**.
- ✂ Select the left comb and then flip it by choosing **Edit > Flip > Horizontal**.
- ✂ Move the comb-drives so that their connection pins, represented by circles, line up with the pins on the plate (**Figure 2**).

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

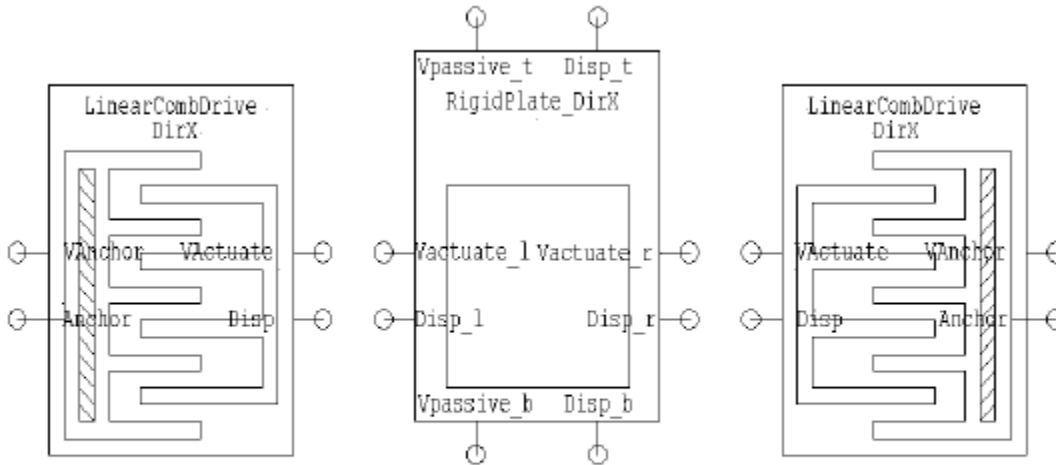


Figure 2: Aligning the comb-drives to the plate

Instantiating Folded Springs

- ✂ Instantiate the **LinearFoldedSpring_DirX** module and place it above the plate.
- ✂ Create a copy of the folded spring and place it below the plate.
- ✂ Flip the second instance of the folded spring by selecting **Edit > 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**.
- ✂ 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 3](#)).
- ✂ Home the view by pressing the **Home** key.

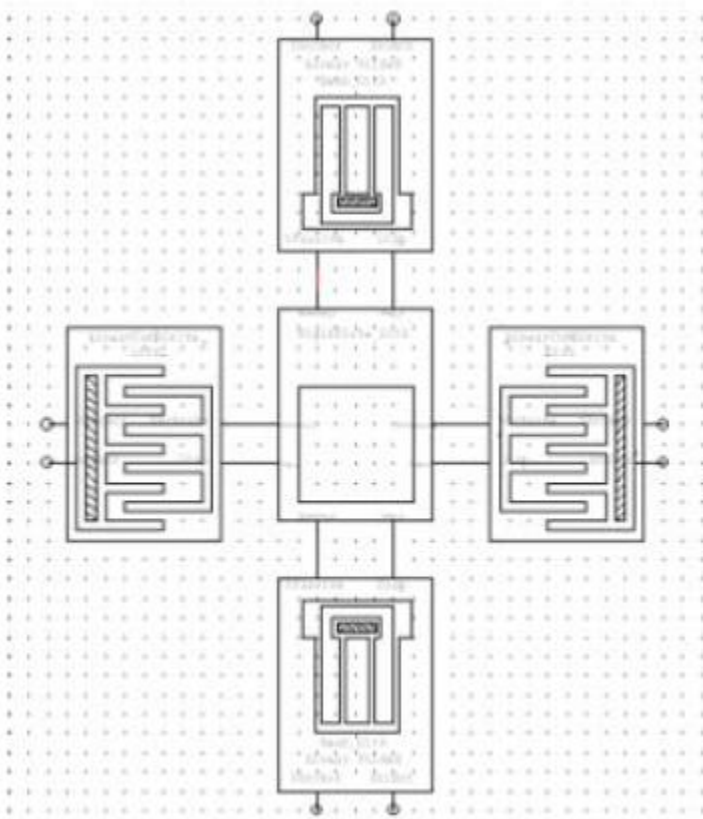


Figure 3: Schematic view of the wired elements

Instantiating Voltage Sources

- ✂ Instantiate the **Source_v_ac** module.
- ✂ Place it to the left of the left comb.
- ✂ Instantiate the **Source_v_dc** module.
- ✂ Place it to the right of the right comb.
- ✂ Copy the instance of **Source_v_dc** and place it to the right of the top spring.
- ✂ Select the wire tool.
- ✂ 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.

Placing Global Nodes

Global nodes simplify the drawing and maintenance of schematics. They allow nodes throughout a design to be connected to each other without the need to draw or delete wires. Global nodes are especially useful for power, ground, anchor, clock, reset and other system-wide nodes that require routing throughout the hierarchy of the design.

To create a global node, you must place a global symbol on the design with the **Global Symbol** tool.



Global symbols are special instances that function as wireless connectors. When you attach a node to a global symbol, you connect that node to all other nodes on every page and module in the design file that are attached to the same global symbol. Such nodes then become global nodes.

You will add three global ground symbols and three anchors to the schematic. The global grounds will be connected to the negative terminals of the voltage sources to set electrical grounds. The anchors will be connected to the fixed mechanical terminals to signify mechanical anchors.



- ✂ To place a ground symbol in the design, click the **Global Symbol** tool on the left side of the schematic window .
- ✂ Left-click on the schematic page. The **Instance Module** browse box will appear, with a list of the available global nodes.
- ✂ Select the ground (**Gnd**) symbol and click **OK**. The first ground symbol will be placed where you left-clicked in the previous step.
- ✂ Copy and paste the ground symbol twice.
- ✂ 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.
- ✂ To place an anchor symbol onto the design, click the **Global Symbol** tool on the left side of the schematic window .
- ✂ Left-click on the schematic page. The **Instance Module** browse box will appear, with a list of the available global nodes.
- ✂ Select the anchor (**anchor**) symbol (see [Figure 4](#)).

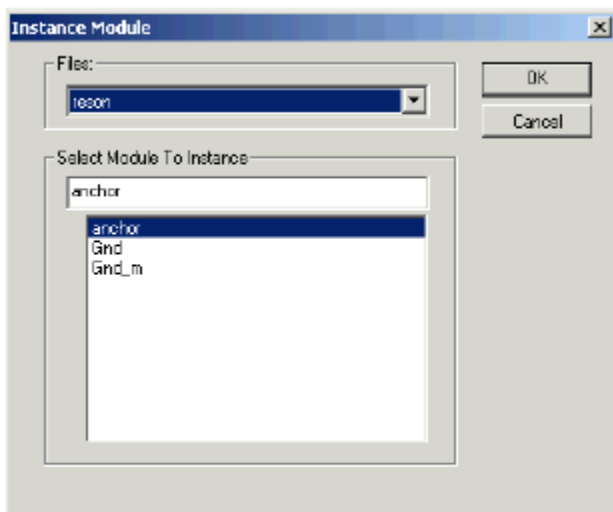


Figure 4: Instance Module browse box

- ✂ Click **OK**. The anchor will be placed where you left-clicked in the previous step.
- ✂ Copy and paste the anchor symbol twice.
- ✂ Move two anchors to a place on the schematic near each comb-drive. And, move the third anchor near the top spring.
- ✂ At this point, the three anchor symbols should be connected to the **Anchor** pins of the top spring and the two comb-drives.
- ✂ Compare your wiring to the completed schematic presented in [Figure 1](#).
- ✂ 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

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

- ✂ Select the voltage source next to the left comb. Invoke the **Edit Instance of Module Source_v_ac** dialog by selecting **Edit > Edit Object**.

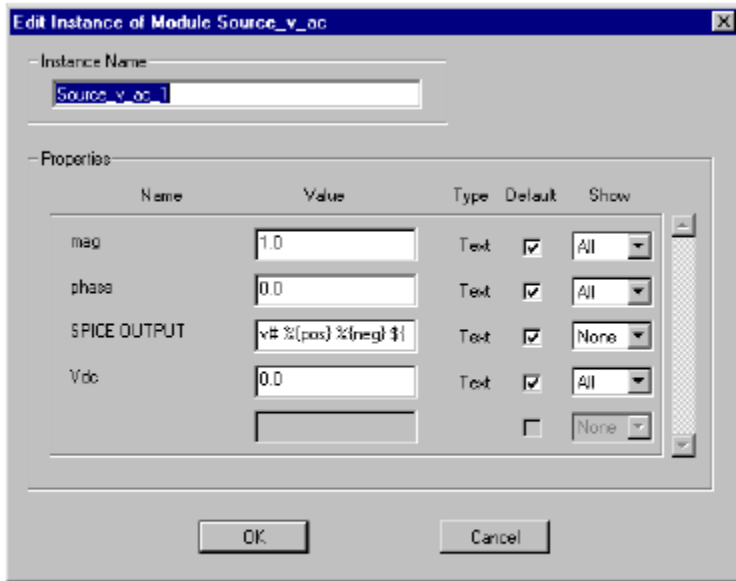



Figure 5: Edit Instance of Module Source_v_ac dialog

- ✂ Enter **1** for **mag**, **0** for **phase** and **0** for **Vdc** and click **OK**.
- ✂ Give the voltage source for the right comb-drive a DC value of 0 volts. Do this by choosing **Edit > Edit Object** 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 **Node Label** tool from the schematic toolbar. 
- ✂ Label the two wires connecting the plate to the right comb, **right_m** and **right_e**. To label a node, click it and enter the new node name in the **Place Node Label** dialog box. The **right_e** node label should be placed on the wire between the **VActuate** and **Vactuate_r** pins and the **right_m** node label should be placed on the wire between the **Disp** and **Disp_r** pins.
- ✂ To change the orientation of the node label; click the **Selection** button, click the node label then select **Edit > Edit Object**. From the **Edit Node Label** dialog box, click one of the eight radio buttons representing the location of the label origin.
- ✂ Edit the node labels orientations to look somewhat like the layout in [Figure 6](#).
- ✂ You may rename the rest of the nodes in your diagram to match the names used in the **Resonator** module of the **reson.sdb** file if you wish. This is an optional step. S-Edit will automatically assign names to unlabeled nodes.

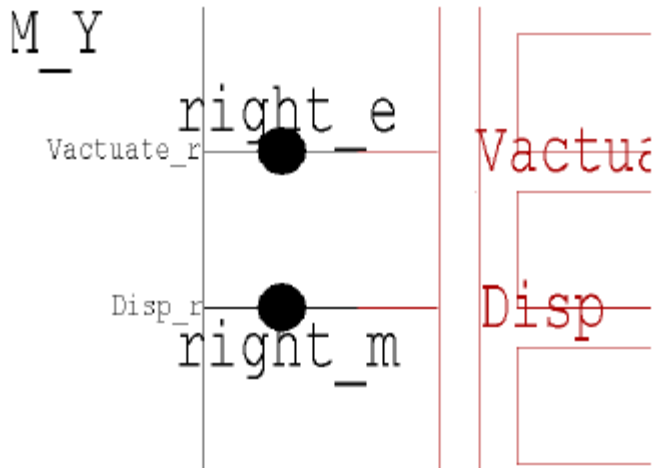


Figure 6: right_m and right_e nodes

Setting up the Probing Information

You will now specify the probing information for the resonator.

- ✂ Select **Setup > Probing**. The **Waveform Probing Setup** dialog box appears.
- ✂ In the **Waveform Probing Options** area, make sure that the **Enable waveform probing** box is checked.
- ✂ In the **Probing data file location** area, set the path to the .dat file that will be created (*myreson.dat*).
- ✂ In the **AC Analysis Probing Options**, check the **Current Magnitude (dB)**, **Current Phase**, **Voltage Magnitude** and **Voltage Phase** boxes as shown in [Figure 7](#).

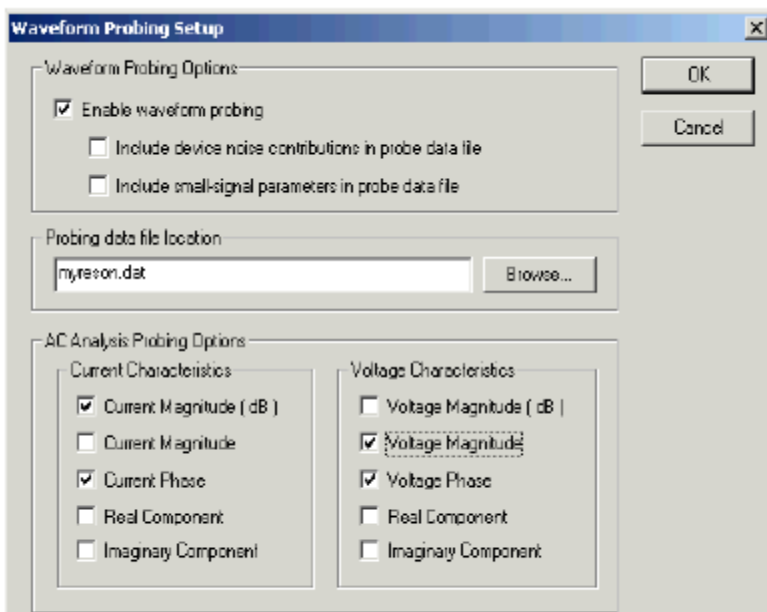


Figure 7: Waveform Probing Setup dialog box

- ✂ Click **OK** to accept the probing information.
- ✂ Click the **Save** icon to save your settings.

2. Adding Simulation Commands



You will now use T-Spice to add the simulation commands in the circuit.

✂ Click the **T-Spice** button located in the **Standard Commands** toolbar of S-Edit.

T-Spice will launch with the exported netlist (*MyResonator.sp*) open. T-Spice contains a full-featured editor that includes search and replace of strings and regular expressions; it also includes incremental find. There are four areas on the T-Spice user interface:

- ✎ The *menu bar* (at the top) contains menu commands
- ✎ The *toolbar* (beneath the menu bar) contains tool icons
- ✎ The *status bar* (at the bottom) contains runtime information
- ✎ The *work area* (in the center) contains input file and runtime information windows

The T-Spice window (Figure 8) allows you to view a SPICE file (.sp) in which the exported resonator components, their simulation parameters and connectivity are displayed.



The **Insert Command** feature provides an easy, convenient means of entering device and model statements, stimuli, simulation commands and simulation options within T-Spice. You will use the **Insert Command** feature to add two commands. One instructs the simulator to run an AC analysis. The other instructs the simulator to include a file (*process.sp*) in the simulation netlist; this file contains fabrication process parameters for the resonator components.

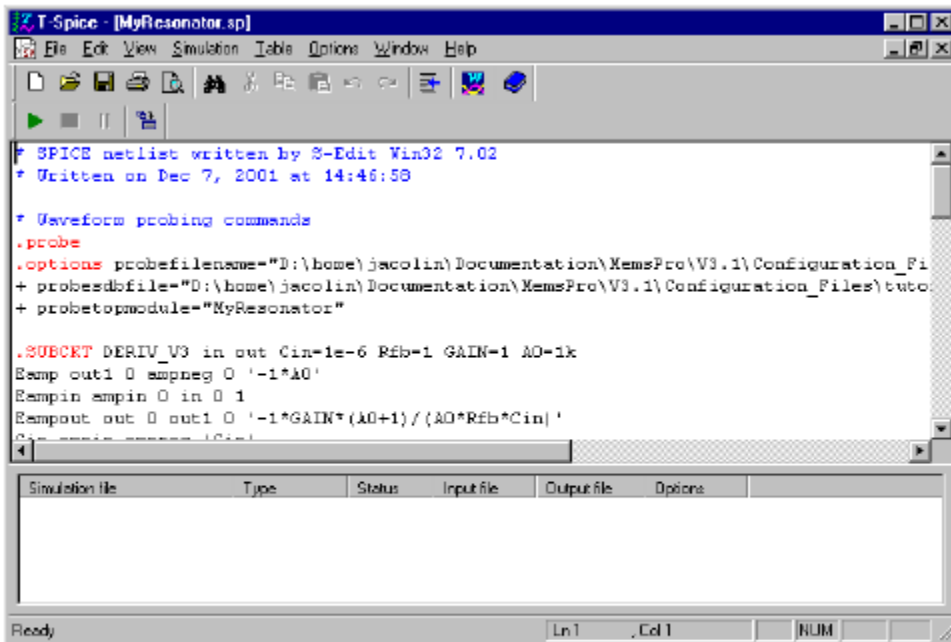


Figure 8: T-Spice window

✂ Place the cursor on a blank line before the *End of main circuit: MyResonator line.

✂ Select the **Insert Command** tool from the **Command** toolbar.

The **T-Spice Command Tool** dialog box appears (Figure 9).

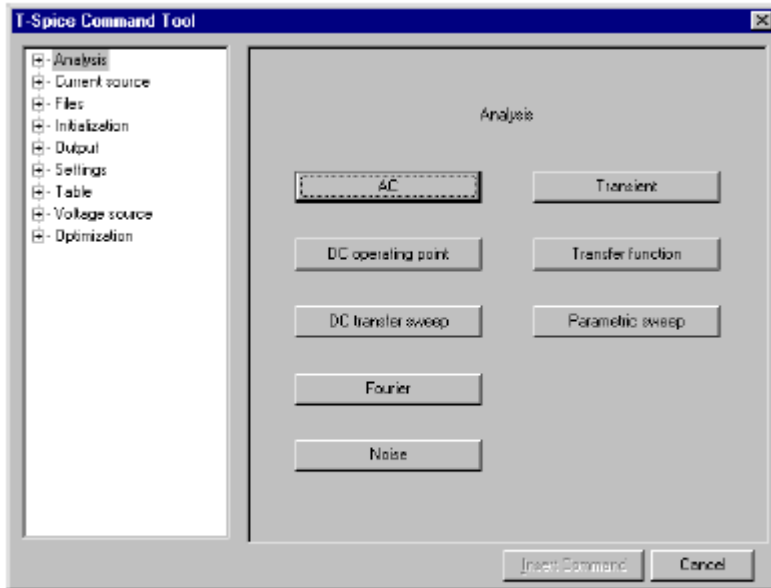


Figure 9: T-Spice Command Tool dialog

The **T-Spice Command Tool** dialog lists command categories on the left. By default, the **Analysis** category is selected. The right side of the dialog contains buttons listing the commands within the selected category; this command list may also be viewed by clicking the **+** sign next to each category. When a command is selected, the right side of the dialog changes to contain the parameters of the selected command.

✂ Add an AC analysis command by clicking the **AC** button on the right side of the **T-Spice Command Tool** dialog. The directory tree on the left side of the **T-Spice Command Tool** dialog will open up to list the commands available under **Analysis**. The right side of the **T-Spice Command Tool** dialog will contain parameters specific to the **AC Analysis** command.

✂ Select **decade** as the **Frequency sampling type**.

✂ Set **Frequencies per decade** to **500**, **Frequency range From** to **10k** and **Frequency range To** to **100k**.

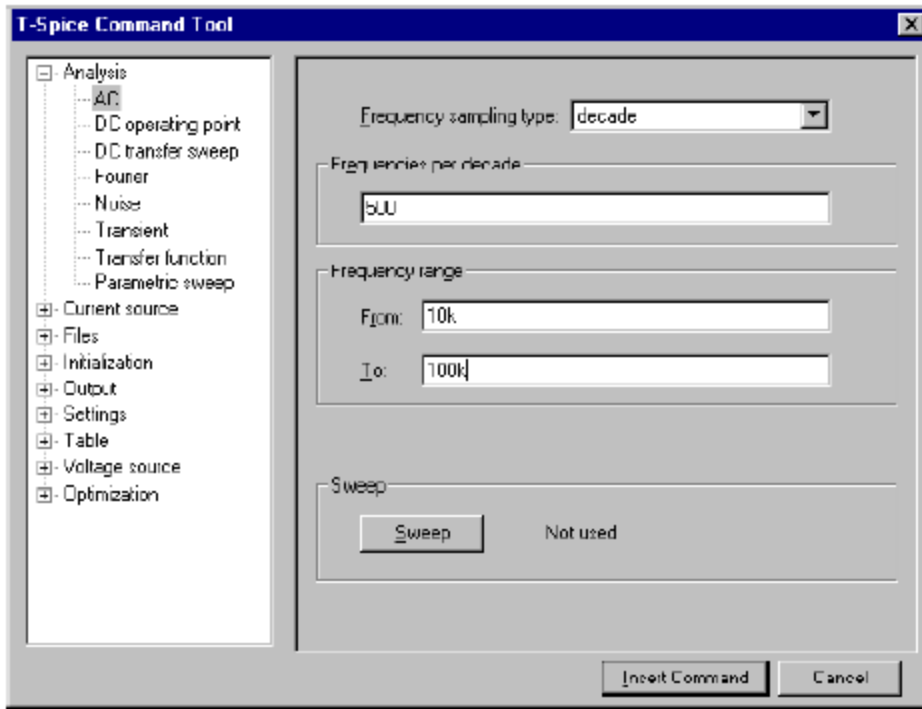


Figure 10: Customizing the AC analysis

- ✂ Click the **Insert Command** button and then cancel (close) to dismiss the **Command Tool** dialog.
- Once the AC analysis is set up, you need to include fabrication process information into the netlist. The steps below will guide you through this task.
- ✂ Place the cursor on a blank line before the End of main circuit MyResonator line.
 - ✂ Access the **T-Spice Command Tool** dialog by clicking the **Insert Command** icon.
 - ✂ Click the **+** next to the **Files** category on the left side of the dialog.
 - ✂ Click **Include file** under **Files**.
 - ✂ The right side of the dialog box displays the parameters corresponding to your selection.
 - ✂ In the **Include file** field, click the **Browse** button, change the file type into Spice Files (*.sp, *.spc, *.cir), select the **process.sp** file (located under the **Tutorial\Resonator** directory) and click "Open".

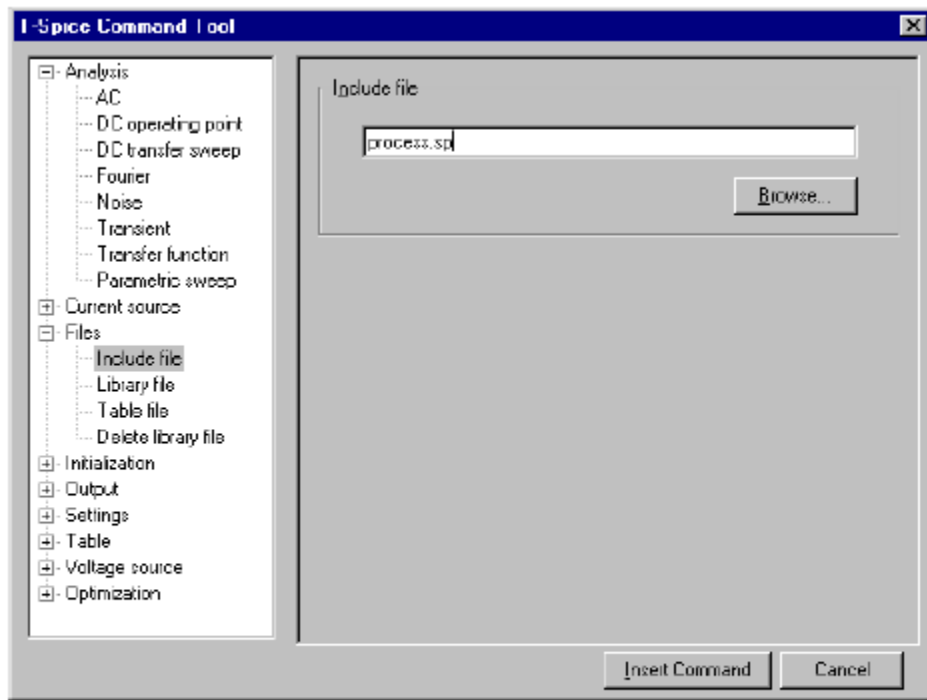


Figure 11: Selecting the technology process file

- ✂ Click the **Insert Command** button and then cancel (close) to dismiss the **Command Tool** dialog. You have just inserted the simulation commands into your netlist. You will now run the simulation.
- ✂ Click the **Save** icon on the toolbar to save the changes you made to the netlist.

Simulating from a Netlist

Using T-Spice and W-Edit, SPICE netlists can be simulated, and the simulation results can be displayed graphically. In this example, the coupled electro-mechanical behavior of the resonator is simulated using T-Spice.



Simulating with T-Spice

- ✂ Click the **Run Simulation** button located in the **Simulation** toolbar to run the simulation. The **Run Simulation** dialog opens.
- ✂ Check the **Do not show** check box.
- ✂ Click **Start Simulation** in the **Run Simulation** dialog. The AC analysis that you had set up earlier using the **Insert Command** will now be performed. The **Simulation Output** window will appear. It displays simulation statistics and progress information, as well as any warning or error messages. During the AC analysis, the simulation results are being recorded. Once the simulation is complete, you may examine the analysis results using the S-Edit **Probe** tool.

3. 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 **Window > 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 > Probing** in S-Edit.

- ✂ Click somewhere in the S-Edit schematic window that contains the resonator schematic to re-activate S-Edit.

- ✂ Click the **Probe** tool located on the **Schematic** toolbar. The cursor now has the shape of the **Probe** tool.
- ✂ Left-click with the **Probe** tool on the *right_m* node. 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 12**), 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.

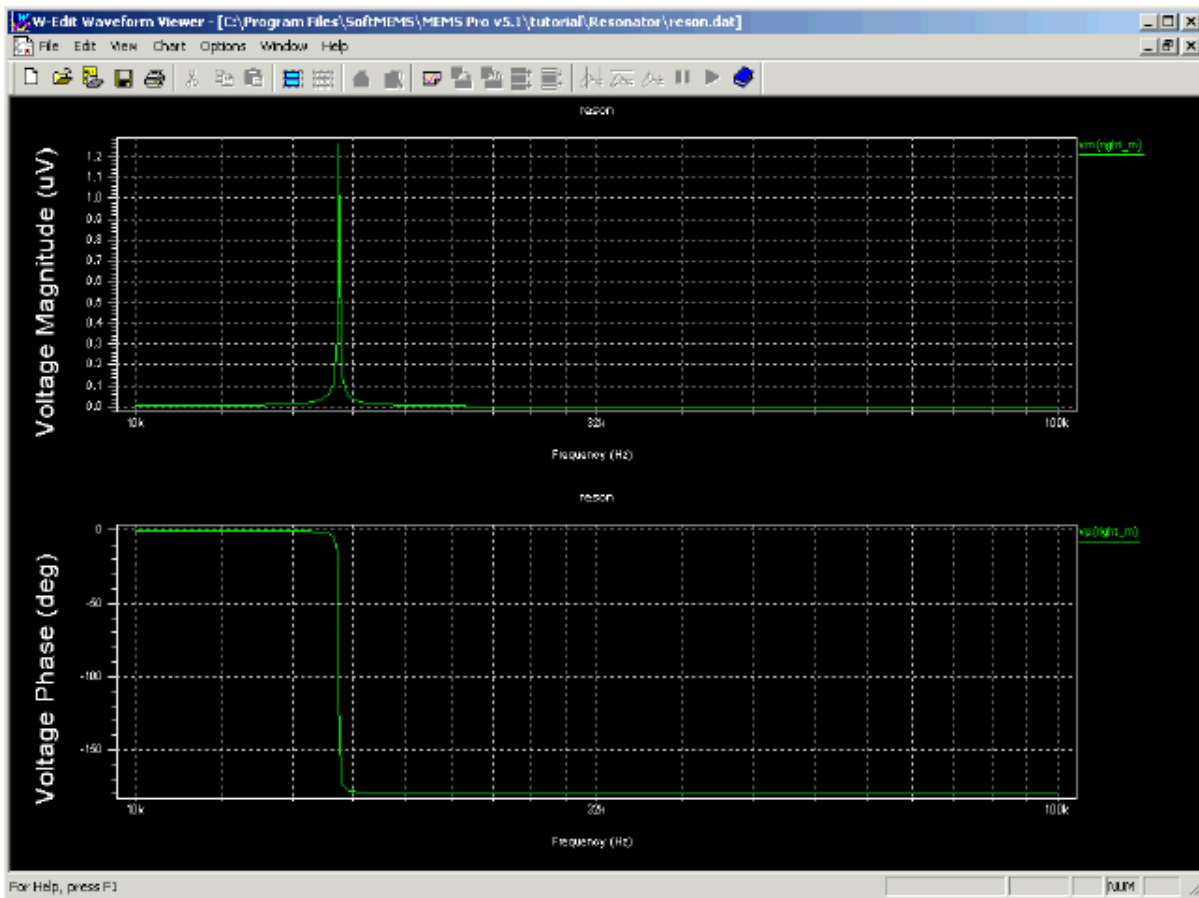


Figure 12: Charts representing the amplitude and the phase angle

Trace Manipulation

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

- ✂ Select both charts by choosing **Edit > Select All (Traces)**.
- ✂ Choose **Chart > Collapse Charts**.
- ✂ With the (now single) chart selected, choose **Chart > Traces**.
- ✂ In the **Traces** dialog, click the box beneath the **Show** label to *uncheck* *vp(right_m)*. The checkmark will disappear. The trace information will still be available, but the trace will not appear the next time you view the chart.

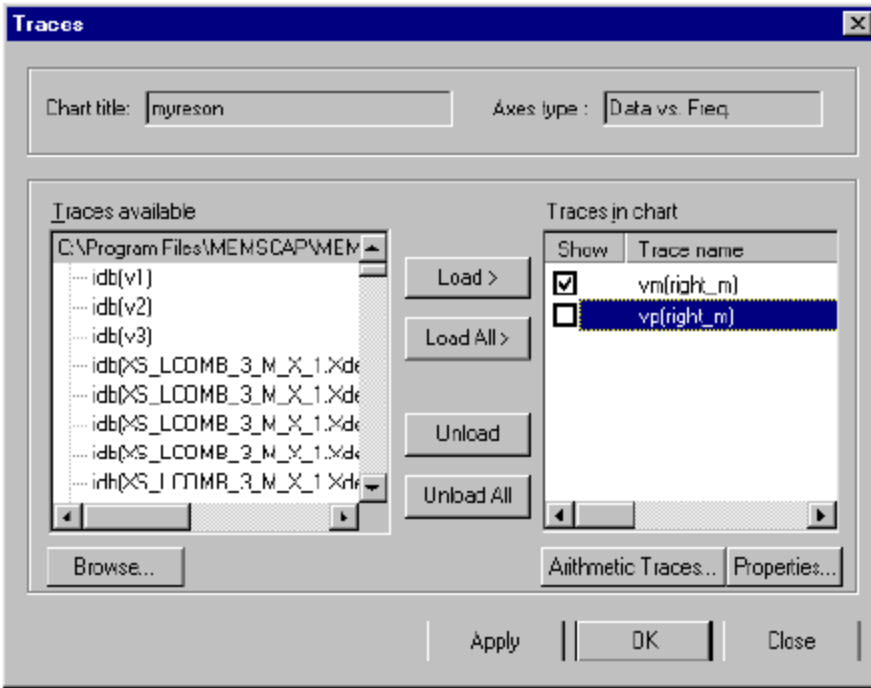


Figure 13: Traces dialog

- ✂ Click **Apply** and then **OK** to return to the chart.
- ✂ Click **Chart > Expand Chart** to better view the chart.

Notice that the displayed unit for **vm(right_m)** is **Volts**. The mechanical behavior of this system is modeled using electrical analogs of mechanical components; the mechanical displacement has been mapped to voltage. Therefore, **vm(right_m)** represents the displacement at the **right_m** node. Let's change the label of the dependent variable axis (Y-axis) from **Voltage Magnitude** to **Displacement**. The new Y-axis units will also be changed to **meters**.

- ✂ Choose **Chart > Options** to invoke the **Chart Options** dialog.
- ✂ Click the **Axes** tab.
- ✂ Set the **Chart title** to **Lateral Comb-drive Resonator**.
- ✂ In the **Y-axis** list, remove the information that appears and enter **Displacement(right_m)** under **Label**, and **m** under **Units** (Figure 14).

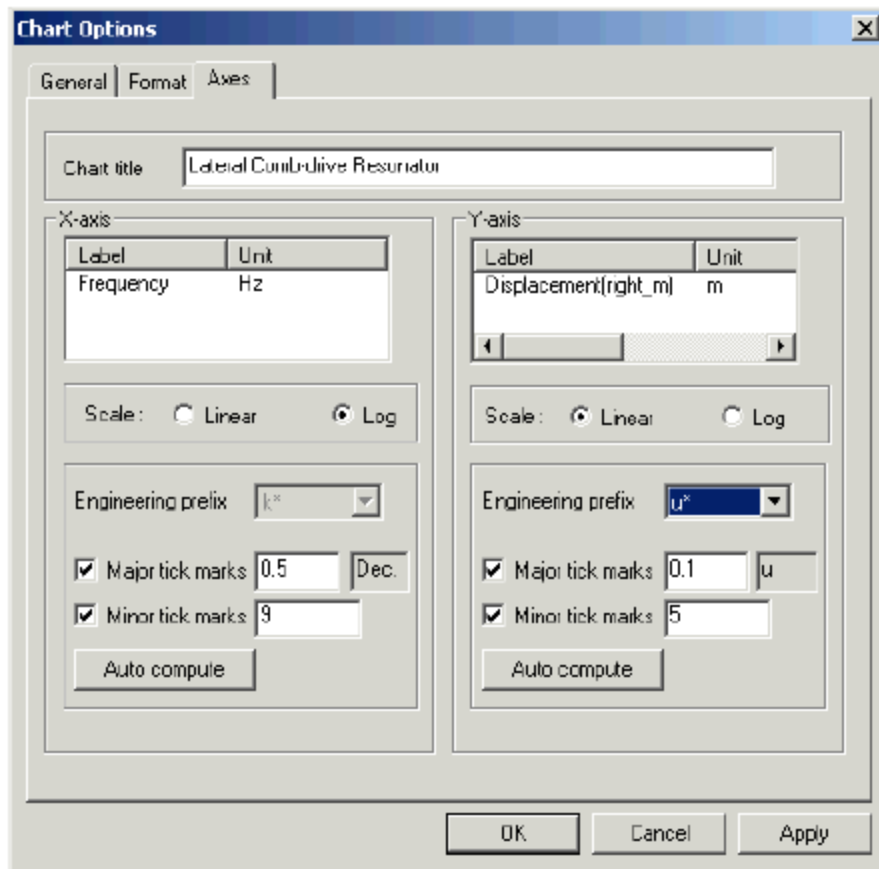


Figure 14: Chart Options dialog

✂ Click **OK**.

The chart is modified accordingly (Figure 15).

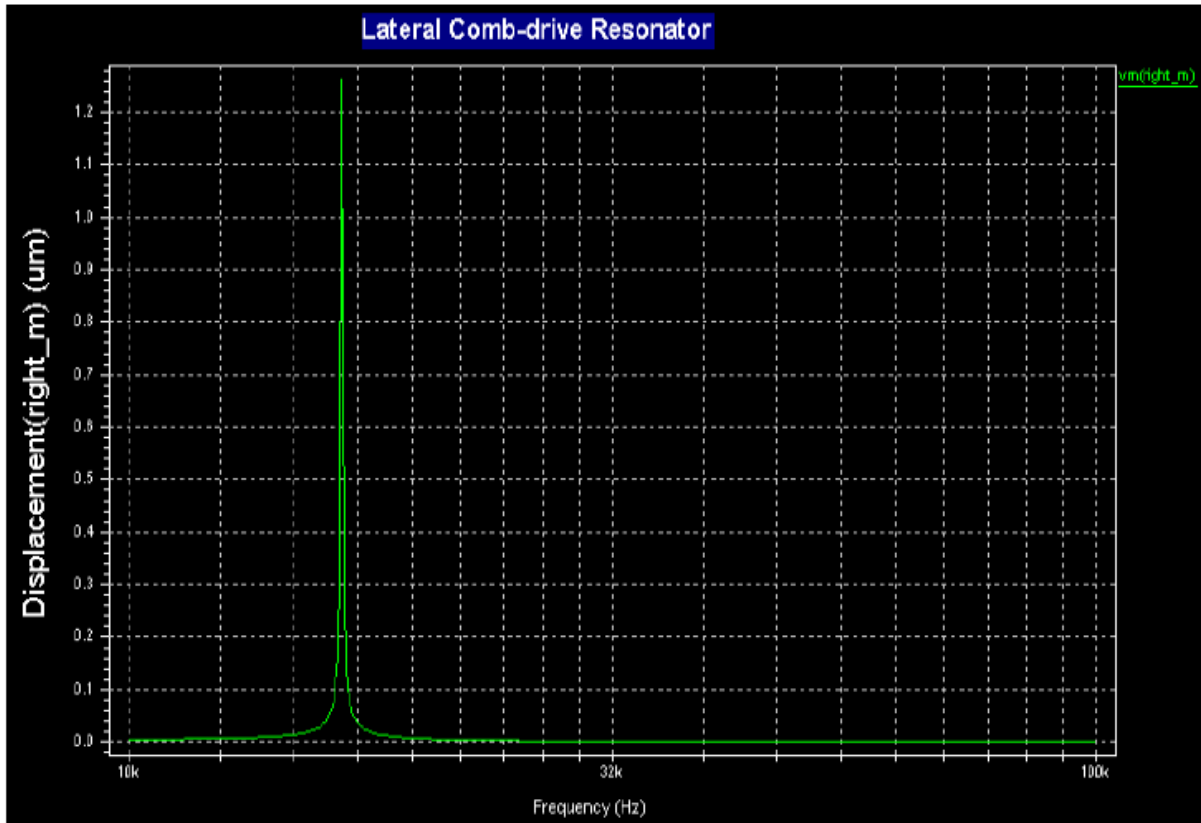


Figure 15: Customizing the amplitude waveform

You may want to measure specific values on the chart. Cursors are vertical lines, horizontal lines or + points that can be used to identify locations on the trace for measuring (reading). The vertical and horizontal line cursors extend to the full height or length of the chart. As the line cursors are moved along the chart, their coordinates appear at the margins of the chart window. We will demonstrate the use of **Vertical Bar** cursors below.

✂ Select **Chart > Cursors > Vertical Bars**.

Two vertical cursors should appear on the chart. They can be dragged with the left mouse button. The cursors' **X** axis locations (**x1** and **x2**) and the difference (**dx**) between the X axis locations of both cursors are displayed at the top left corner of the chart window. The **Y** axis location of the moving or last moved cursor is displayed under the trace name on the right side of the chart.

✂ Position the left bar so that it lines up with the tip of the trace peak. The frequency value is shown as **x1** and should be about 16.56 kHz. The displacement value can be found under the trace name- **vm(right_m)**.

x1, x2, and dx

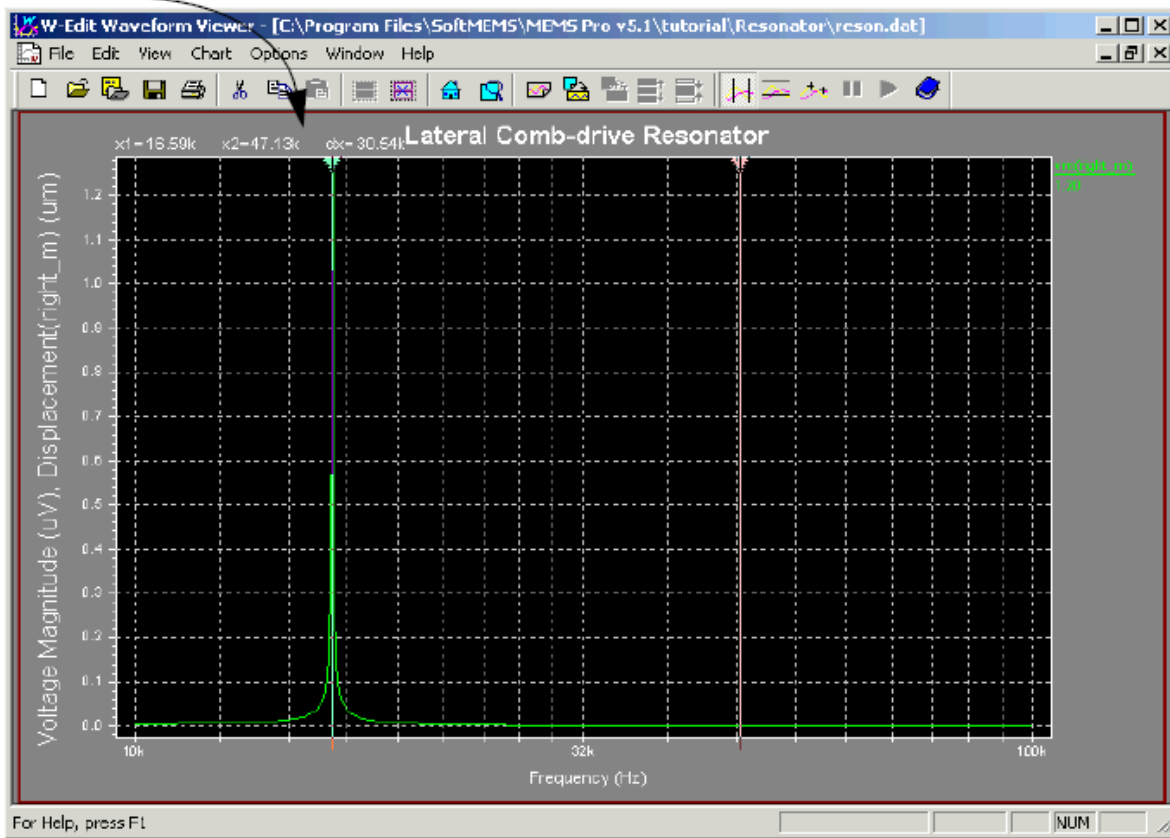


Figure 16: Using the Vertical Bars cursor

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

4. Generating a Layout

Launching L-Edit

✂ Launch L-Edit by clicking "Start > Programs > MEMS Pro v6.0 > L-Edit". 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.

✂ Select **File > Open** to open the **reson.tdb** file located under the **C:\Program Files\SoftMEMS\MEMS Pro v6.0\tutorial\Resonator** directory. The **Resonator** cell should appear as the active cell.

Each mask layer of the **Resonator** cell can be viewed by clicking the corresponding layer on the Palette bar (on the left) and then right click the mouse to "hide all" the rest layers.

✂ Familiar yourself with L-Edit and understand the layout of the **Resonator** cell.

✂ Select **File > New** and click **OK** in the open dialog box to create a new layout of a simple structure of your choice. For example, a cantilever beam, a bridge structure etc.

✂ Refer to MUMPs User Manual for design rules.

Save your layout as .tdb file in your folder.