Chapter 6. Examples

Frequency Domain

BoxCad program performs the following functions: (1) Schematic Capture, (2) Analysis in Frequency Domain and (3) Analysis in Time Domain. The program is a highly interactive package and the user is well advised to exercise some discipline when switching from one type of schematic to another.

As explained before, there are three types of schematics that may be created with BoxCad: (1) Acoustic Impedance, (2) Electrical Impedance and (3) Standard Network schematics. We begin our review of the program's tools in the frequency domain.

To start with, our task is to plot amplitude and phase responses of a vented system. To accomplish this, we will use in-build example data. Please follow these steps:

1. The user may recall, that calculators are used for entering data. From the main CAD window, select "Calculators" and then "Driver" under "ACOUSTICAL" menu option. A Driver Calculator - Acoustic Impedance Model box will be opened. Rather than entering driver's data, please click on "1.Example" button to make use of internally stored example data. Next, you must click on the "2. Expand" button to obtain mechanical parameters of the driver and finally, click on "3. Calculate" button to obtain acoustical impedance parameters of the driver. When using your own data, you need to perform the three steps described above. Finally, press "Cancel" button to close the box, or print the screen to obtain a hard copy.

2. Next, from the main menu, select "Calculators" and then "Rear Box + Port" under "ACOUSTICAL" menu option. A "Rear Box, Port Calculator" box will be opened. Once again, we will use internal data by pressing "1. Example" button and then "2. Calculate" button to obtain acoustical impedance model values of our enclosure and port. To close the box use "Cancel" button or print the screen to obtain a hard copy.

At this stage, we have calculated all necessary acoustical impedance parameters of the vented system. The user may now elect to create his/her own acoustic impedance schematic on the CAD screen using data just calculated. We will continue with the build-in schematics.

3. Please select "Models" menu the main CAD screen. This will open a pull-down menu with several options on it. Select "LoadModel" option and this will open a list box with a number of acoustical impedance (amplitude type of responses), electrical impedance schematics and three types of previously saved schematics. Please select "Acoustic Impedance - Vented" model and click on "Done" button. The program will now draw reasonably complete acoustic impedance schematic of the vented system on the main CAD screen. We are now ready to start plotting frequency responses of the circuit.

4. From the main menu, select “Tools” and then “Frequency” option. The frequency domain plotting window will now be opened. The user will notice “Plot” button in the right bottom corner of the window. Pressing this button will invoke a floating menu with several functions to choose from. Please remember, that the selected plotting function must match the type of schematic being display. Select “Amplitude” option under “Passive Only” section of the menu.

Next a dialogue box will be invoked to allow for selection of up to four branch currents (volume velocities). This arrangement allows the user to build a simple formula for adding/subtracting volume velocities. **Unused branches must have ‘*’ character in the data field and the only characters allowed are positive or negative integers.** If the built-in vented model was selected, the total system output is the difference between output from the cone and the vent (Branch 3- Branch 7). The user would enter 3 in the B1 field and -7 in the B2 field.

Acoustical pressure generator supplying input signal to our circuit has the strength of Pgen = (eg * Bl)/(Re*Sw), where: Sw = piston area of the driver, Bl = BL factor of the driver, eg = 2.83 volt generator output and Re = DC resistance of the voice coil. The Egen value is displayed in the dialogue box, allowing the user to modify it, if required. The Pgen value proposed by the program is calculated from the data entered in the Calculator dialogue box and causes the curve to be plotted around its free SPL level - in our example, it will be around 90dB. **Frequency response will be saved automatically in memory, but not to the disk**.

Other plots, that can be drawn at this point of time are: phase response, group delay and cone excursion. Input impedance and impedance phase can be only plotted after the electrical impedance model was loaded onto the CAD screen. Generally, the two steps described above are recommended for entering user supplied data.

Cone excursion may be plotted for different input power levels. The user may recall, that two main issues, described as power compression, have to be considered when applying higher power: (1) increased Re and (2) reduced BL factor. It is therefore necessary to determine the change in those two parameters and accommodate the results in the model for every new power level. The program provides a tool to accomplish just that. From the main please select “Calculators” and then “Thermal Analysis” option. The dialogue box, that will be opened, enables the user to calculate Re and BL at say, 100 watt input power. Press the “Example” button and record resulting BL, Qe and Re figures for 100 watt input power. These figures must be entered into the “Driver” calculator under the “ACOUSTICAL” option of the “Calculators” menu. Having keyed the new Re and BL, press “2. Expand” button to obtain mechanical parameters of the driver and finally, click on “3. Calculate” button to obtain new acoustical impedance parameters of the driver. It may be observed, that Rea parameter has dropped almost by half. The new Rea must be entered into the model and only now, the cone excursion plot can be drawn correctly. This entire process must be repeated for any power level selected from the “Thermal Analysis” box. Active networks can be evaluated by plotting their amplitude and phase frequency response. The only additional input the user has to provide is the node at which the output voltage is to be monitored. Two entries (nodes) can be selected with +/- sign to facilitate measurements across floating components. Combined networks (passive, whose frequency response was plotted before and currently displayed active CAD network) can also be evaluated. Good example here are powered subwoofers. Example of frequency domain plots are shown on Fig 6.1.

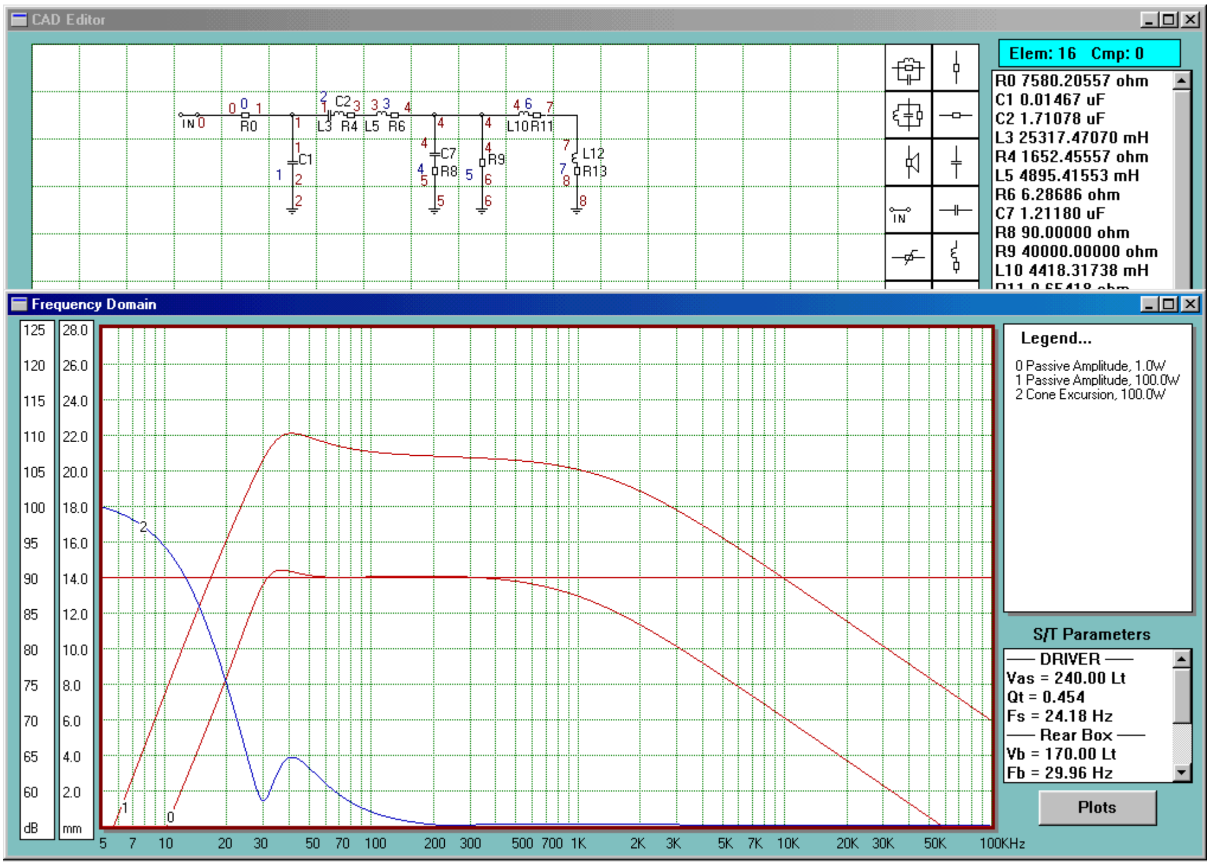

Fig 6.1 Vented system at 1W and 100W

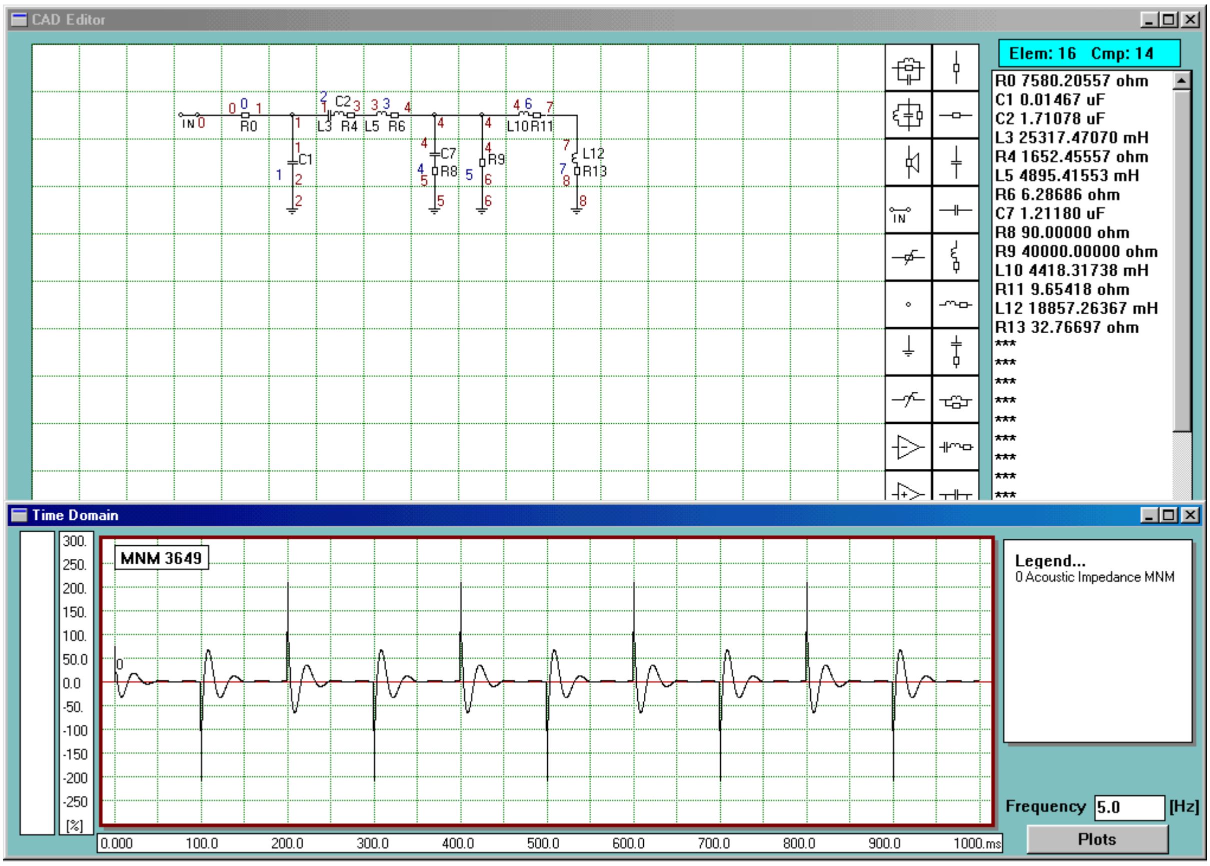

Fig 6.1 Vented system at 100W

Time Domain

In the time domain, the designer would be primarily concerned with circuit response to a bi-polar pulse or a train of pulses, our task would be to plot vented system response to a train of first 5 pulses of a 5Hz square wave – see Fig 6.2.

The program offers additional functionality in the time domain. The convolution (FFT&IFFT) method operates the same way as described in the main program, however, Modified Nodal Method (MNM) can display two type of responses: (1) a single pulse stretched over plotting screen and selected form 1...10 train of pulses, and (2) sequence of first X pulses, where X=1..10. Do not exceed 10 pulses. Please follow these steps:

1. As for the frequency domain above.
2. As for the frequency domain above.
3. As for the frequency domain above.

4. From the main menu, select "Tools" and then "Time" option. The time domain plotting window will now be opened. The user will notice "Plot" button in the right bottom corner of the window. Pressing this button will invoke a floating menu with several functions to choose from. Please remember, that the selected plotting function must match the type of schematic being display. Select "Acoustic Impedance" option under "Modified Nodal Method" section of the menu. Next a dialogue box will be invoked to allow for selection of up to four branch currents (volume velocities). This arrangement allows the user to build a simple formula for adding/subtracting volume velocities.

Unused branches must have '*' character in the data field and the only characters allowed are positive or negative integers. If the built-in vented model was selected, the total system output is the difference between output from the cone and the vent (Branch 3- Branch 7). The user would enter 3 in the B1 field and -7 in the B2 field.

When the MNM "Standard Network" option is selected from the "Plot" menu, the dialogue box will be opened to enable the selection of the desired plot. Networks may exhibit differences between the response to the first and consecutive pulses. Before the train of pulses is applied to the system, all voltages within the system are zero. The leading edge of the pulse is the transition from 0.0 to 1.0V, so the swing is +1volt. The first lagging edge swings down to -1volt, so the total negative swing is -2volt and from then onwards, the peak-to-peak voltage swing will be 2volt.

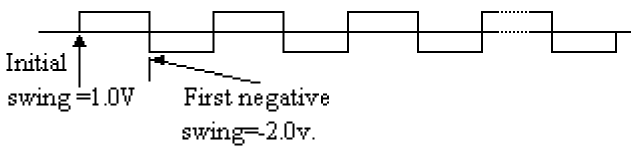

Fig 6.3

Example, shown on Fig 6.2 illustrates behaviour of a passive network. It can be seen, that it takes only 1 pulse for the circuit to settle into a stable pattern of pulses. As in the frequency domain analysis, the plotting function in time domain must also be matched with the type of CAD network displayed.

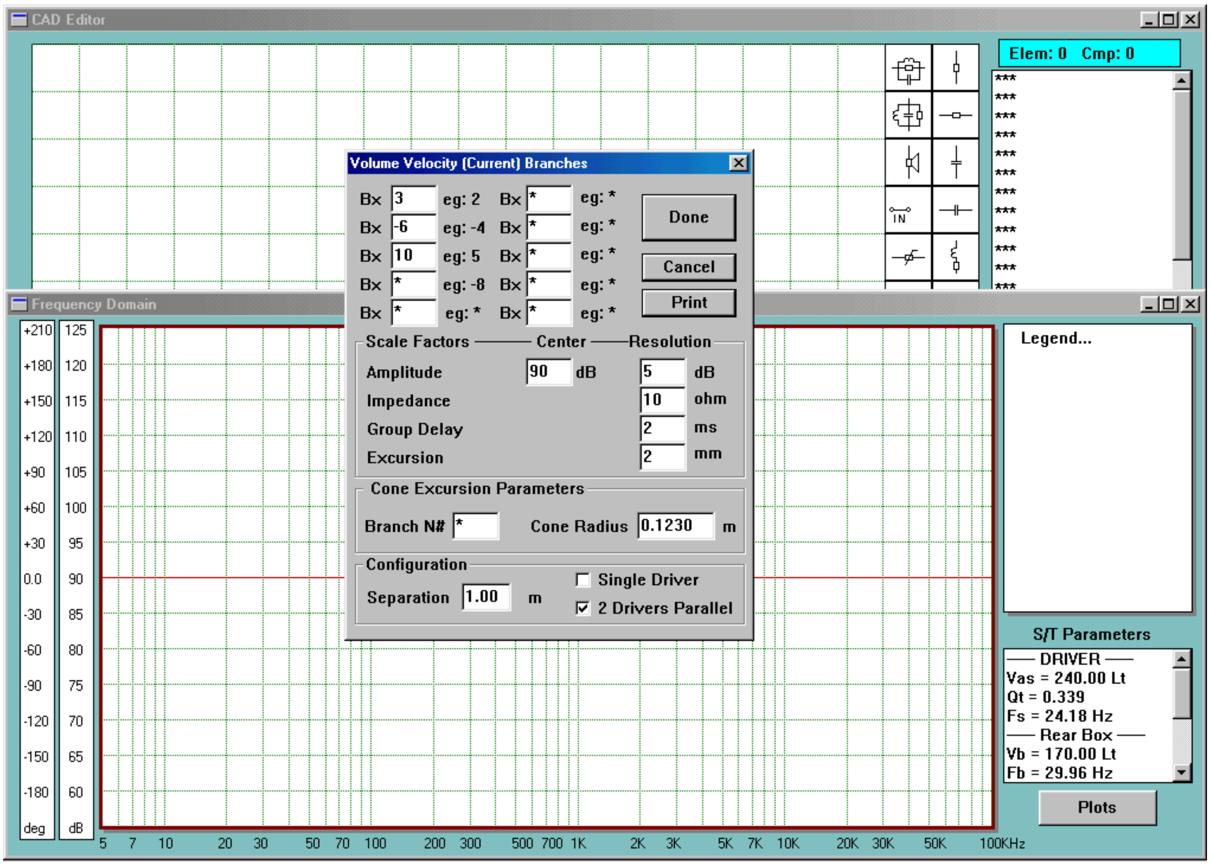

Fig 6.4 Acoustic impedance circuit control box

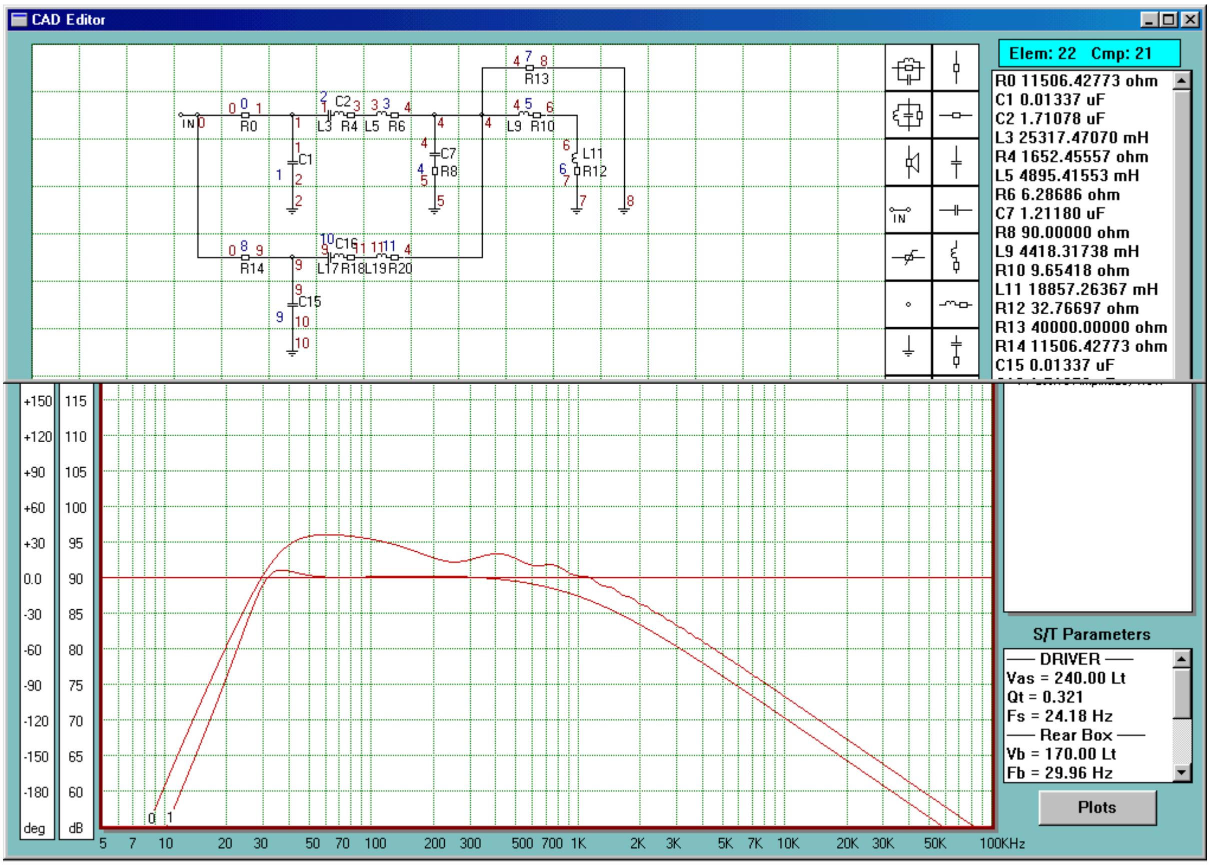

Fig 6.5 Frequency response of 2 drivers

Parallel Drivers

Two drivers mounted in the same enclosure will double the on-axis and inside the box pressure. This is modelled by adding a group of components (Fig 6.5 - R14, C15, C16, L17, R18, L19 and R20) simulating the second driver in parallel with the first driver and connecting branch 11 to node 4. Now, currents in branches 3 and 11 are flawing into node 4. Second factor affecting total system output is mutual radiation impedance. Mutual radiation impedance is modelled by an in-build power factor as described in SoundEasy manual - that is, if the “2 Drivers Parallel” box is checked in the plotting control dialogue box. Please do not forget about entering the required separation distance. Please review the following figures:

1. Fig 6.4 - shows correctly edited dialogue box.
2. Fig 6.5 - shows the “2 drivers parallel” CAD model. Frequency response “0” is the total system output and frequency response “1” is the same enclosure with just single driver.
3. Fig 6.6 - shows time domain response of the two drivers using MNM techniques.

BoxCad is a highly interactive and involving tool. This particular software design approach was deemed necessary to provide the designer with maximum processing power and flexibility. All error prone calculations can be eliminated from the design process by using the in-built calculators. Please take as much time as you need to get familiar with the package and its internal workings. We have equipped the software with a number of examples to guide you on the anticipated range of data, and because of that, BoxCad is ready to use even if you have no data at all. Please fell free to communicate to Bodzio Software what you like or dislike about this release.

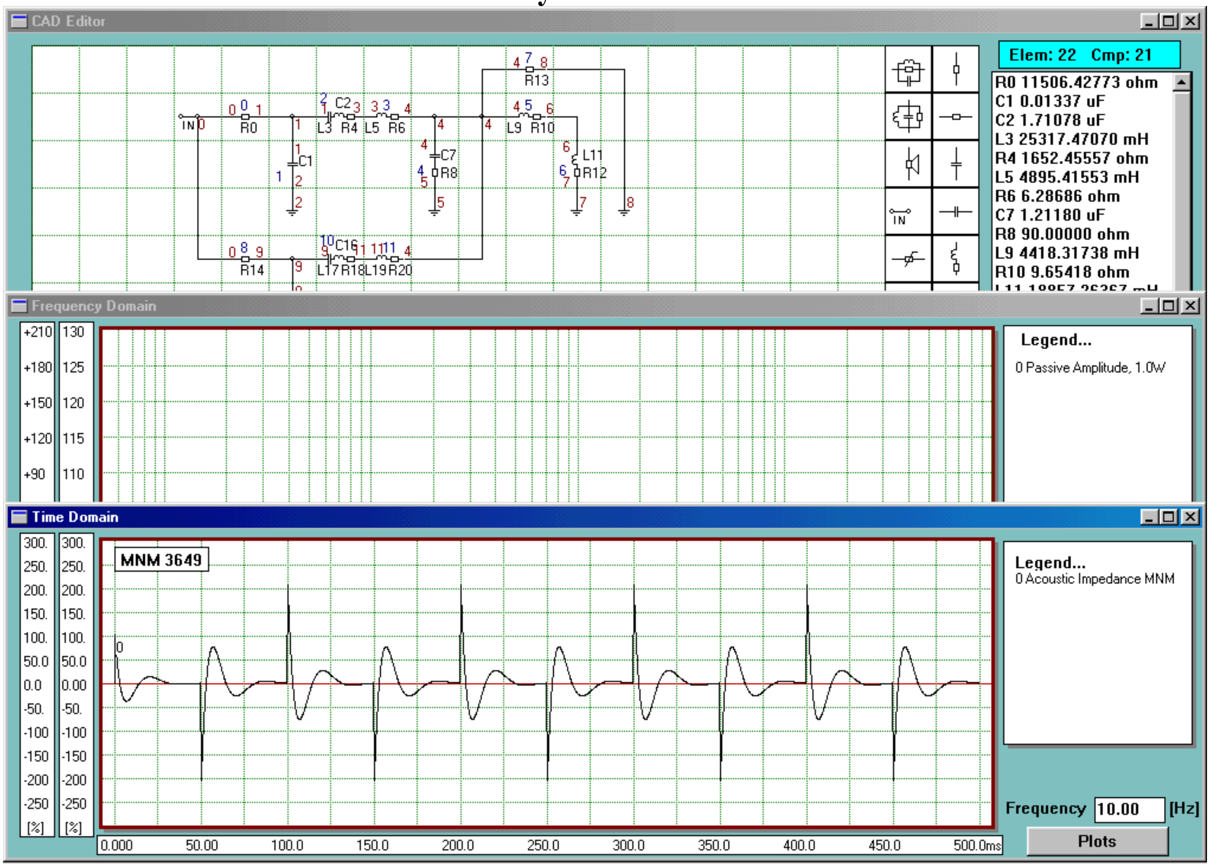

Fig 6.6 Time domain response of the two drivers using MNM techniques