

## Part 3

### Working With Crossover Functions

#### Chapter 8. Using CAD System

##### Background

The three main tasks performed by the Crossover section are:

1. Design and optimization of filters and crossovers.
2. Editing and modelling of the amplitude/impedance/time compensation circuitry.
3. Auditioning of the filtering capabilities using your sound card.

Frequency and time domain analysis is available on passive and active circuits. Fig 8.1 shows an example of a network created and analysed with this module's functionality.

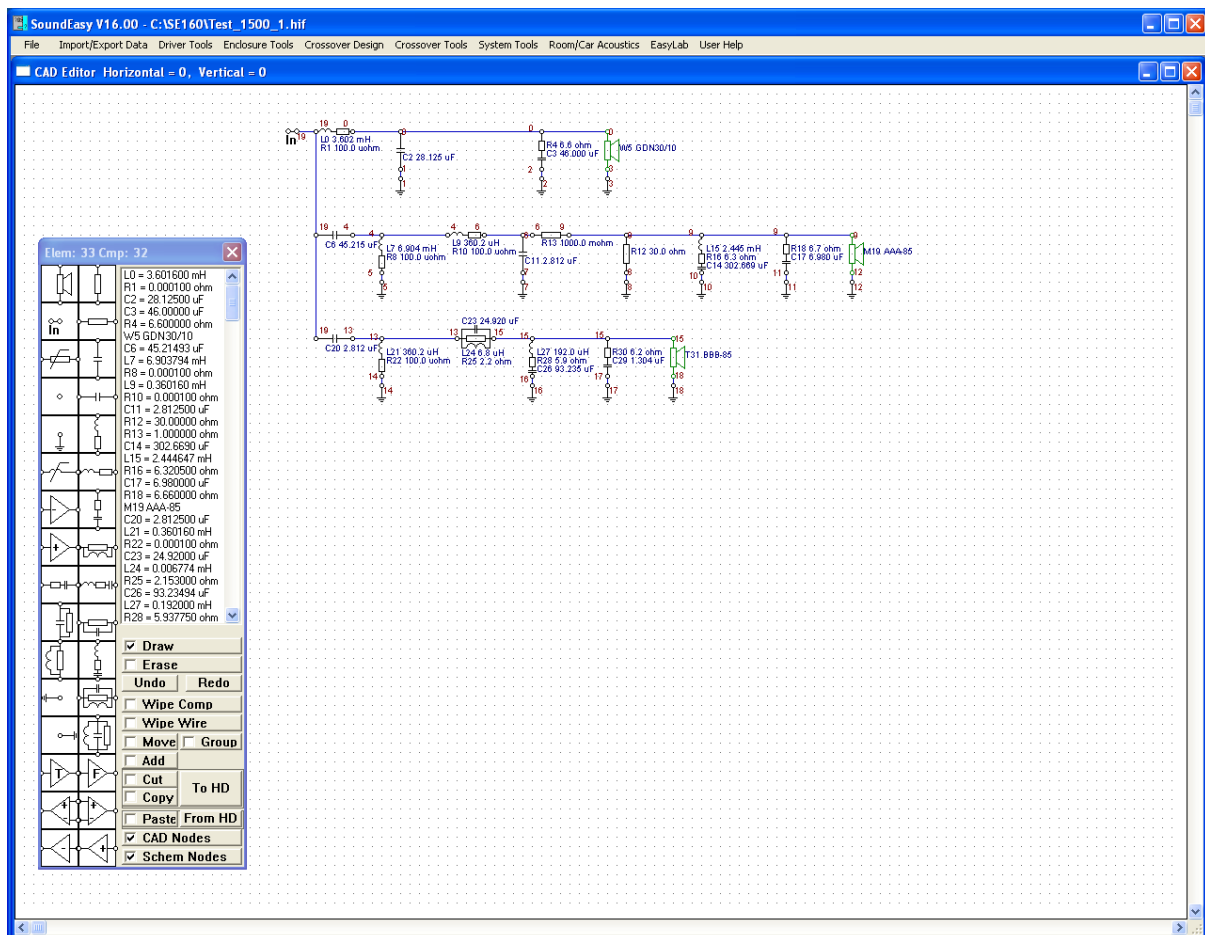


Figure 8.1. Example of fully edited schematic for a 3-way crossover

##### Pick And Place Component

All "simple" and "composite" component types, buttons for processing functions and list of used components are located on a floating selection tray. Schematic drawing is performed on separate screen. Components located on the floating tray can be "picked-and-placed" on the drawing area using the LEFT mouse button. There are 19 passive components to choose from. Some of the components consist of 3 elements, like LRC tank circuits. This approach helps speed up the calculation processes within the program. The list box rectangle on the floating tray will display automatically component values (in correct units). In order to pick a component from the provided list the user is required to:

1. Position the mouse pointer arrow above the required component .
2. **CLICK the LEFT mouse button.** The selected component will be highlighted in **blue**.
3. Drag the arrow pointer (and the component with it) to the chosen drawing area.
4. Once the component is displayed where it is wanted, **CLICK** the mouse button again. The component will be placed where you required.
5. Go to (1) and pick the next component.

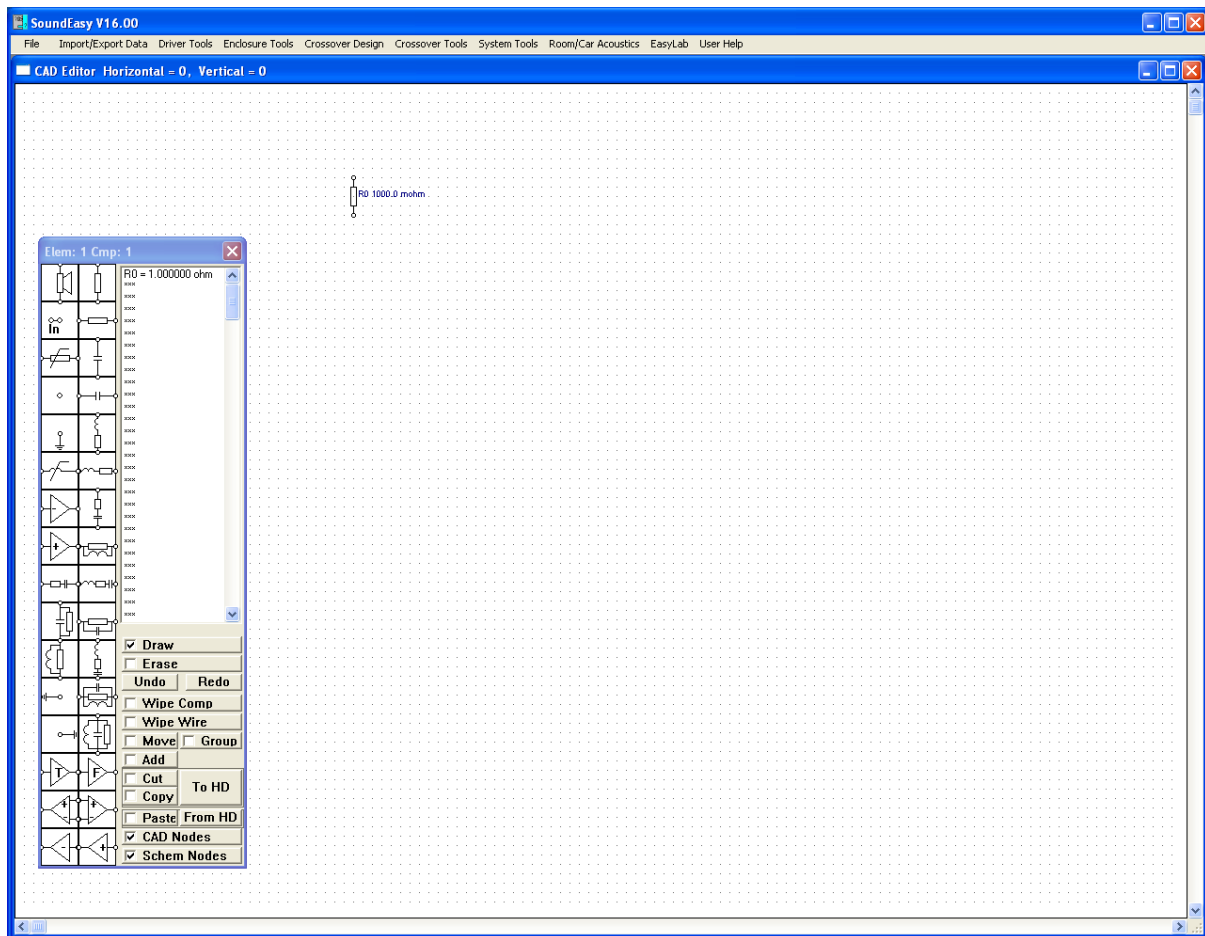


Fig 8.2 Resistor R0 placed on the CAD drawing area

To understand this simple process better still, refer to Fig 8.2 - 8.7. The process of building a small low pass filter is presented in a few steps. First, a load resistor R0 is placed on the drawing area (Fig 8.2). The resistor was "picked and placed" in the manner described above. Then, the C1 capacitor was positioned on the left hand side of the resistor. (It is assumed, the input to the network being built on the screen is on the left hand side of the screen and the output on the right hand side). Signal flow will be from the left to right as well.

You should now follow Fig 8.3 - Fig 8.6 and place all the remaining components.

### Connecting Components

Various connection types are shown on Fig 8.5. Listed below is a short description of each type.

1. In order to connect resistors R0-R1, position the pointer arrow on resistor R0 first, then press and keep pressing the **LEFT** mouse button. Next, drag the pointer across until it is placed above resistor R1, then release the mouse button. The connecting line drawn initially at an angle will disappear and a neatly drawn, right angle connection will be made for you. The important thing to remember is that R0 was the start and R1 was the end of this right angle line.

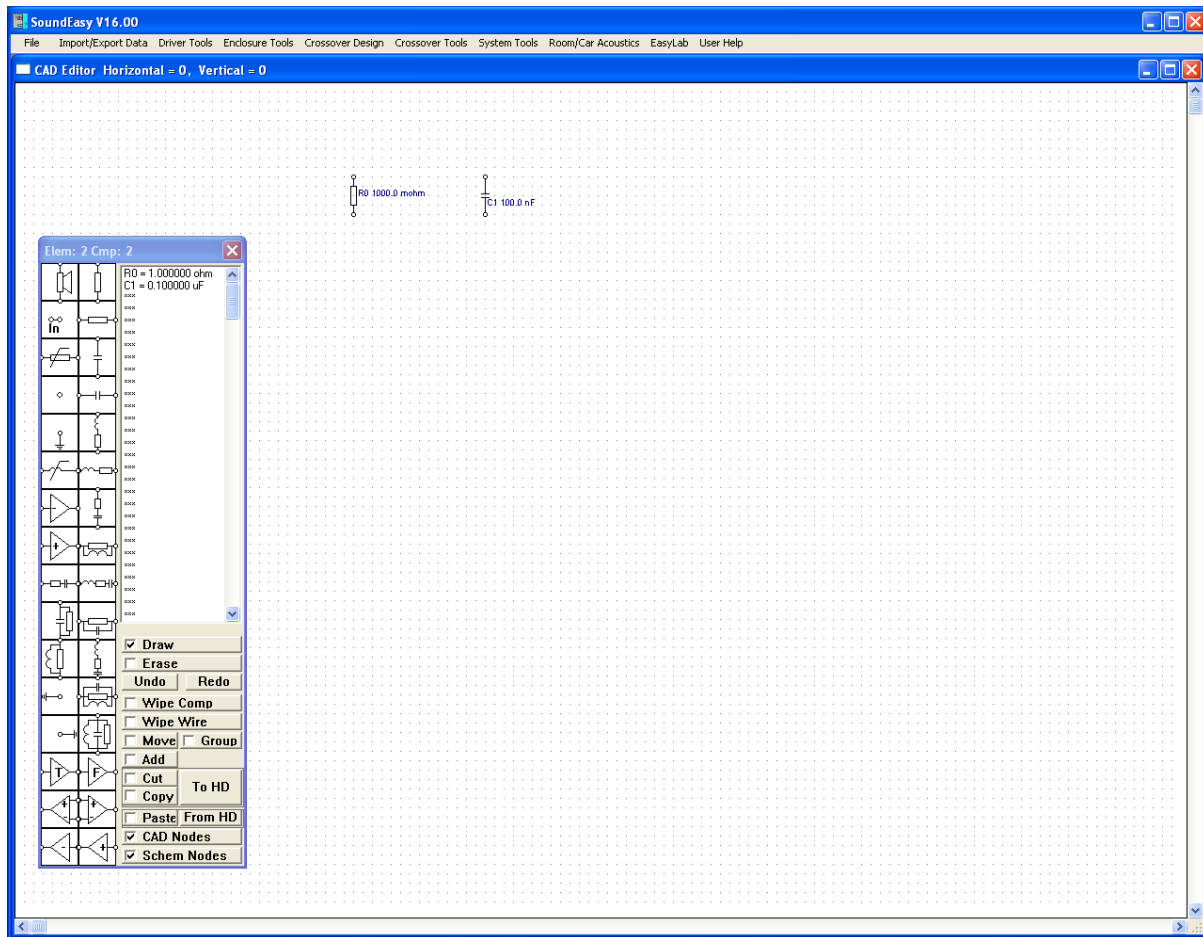


Fig 8.3 Capacitor C1 placed on the CAD drawing area

2. In order to connect resistors R2-R3, position the pointer arrow on resistor R2 first, then press and keep pressing the **LEFT** mouse button. Next, drag the pointer across until it is placed above resistor R3, and release the mouse button. The connecting line drawn initially at an angle will disappear and a neatly drawn, right angle connection will be made for you. The important thing to remember is that R2 was the start and R3 was the end of this right angle line.
3. In order to connect resistors R4-R7, position pointer arrow on resistor R7 first, then press and keep pressing the **LEFT** mouse button. Next, drag the pointer across, until it is placed above resistor R4, then release the mouse button. Again, the connecting line drawn initially at an angle will disappear and a neatly drawn, right angle connection will be made for you. Again it is important to remember that R7 was the start and R4 was the end of this right angle line.
4. In order to connect resistors R5-R6, position pointer arrow on resistor R6 first, then press and keep pressing the **LEFT** mouse button. Next, drag the pointer across until it is placed above resistor R5, release the mouse button. The line drawn initially will disappear and a neatly drawn, right angle connection will be made for you. It is important to remember is that R6 was the start and R5 was the end of this right angle line.
5. In order to connect resistors R8-R9, position pointer arrow on resistor R8, then press and keep pressing the **LEFT** mouse button. Next, drag the pointer to the right until it is placed above resistor R9, release the mouse button. The line drawn initially will be replaced by a neatly drawn, connection.
6. In order to connect resistors R10-R11 position pointer arrow on resistor R10, then press and keep pressing the **LEFT** mouse button. Next, drag the pointer down until it is placed above resistor R11, release the mouse button. The line drawn initially will be replaced by a neatly drawn, connection.

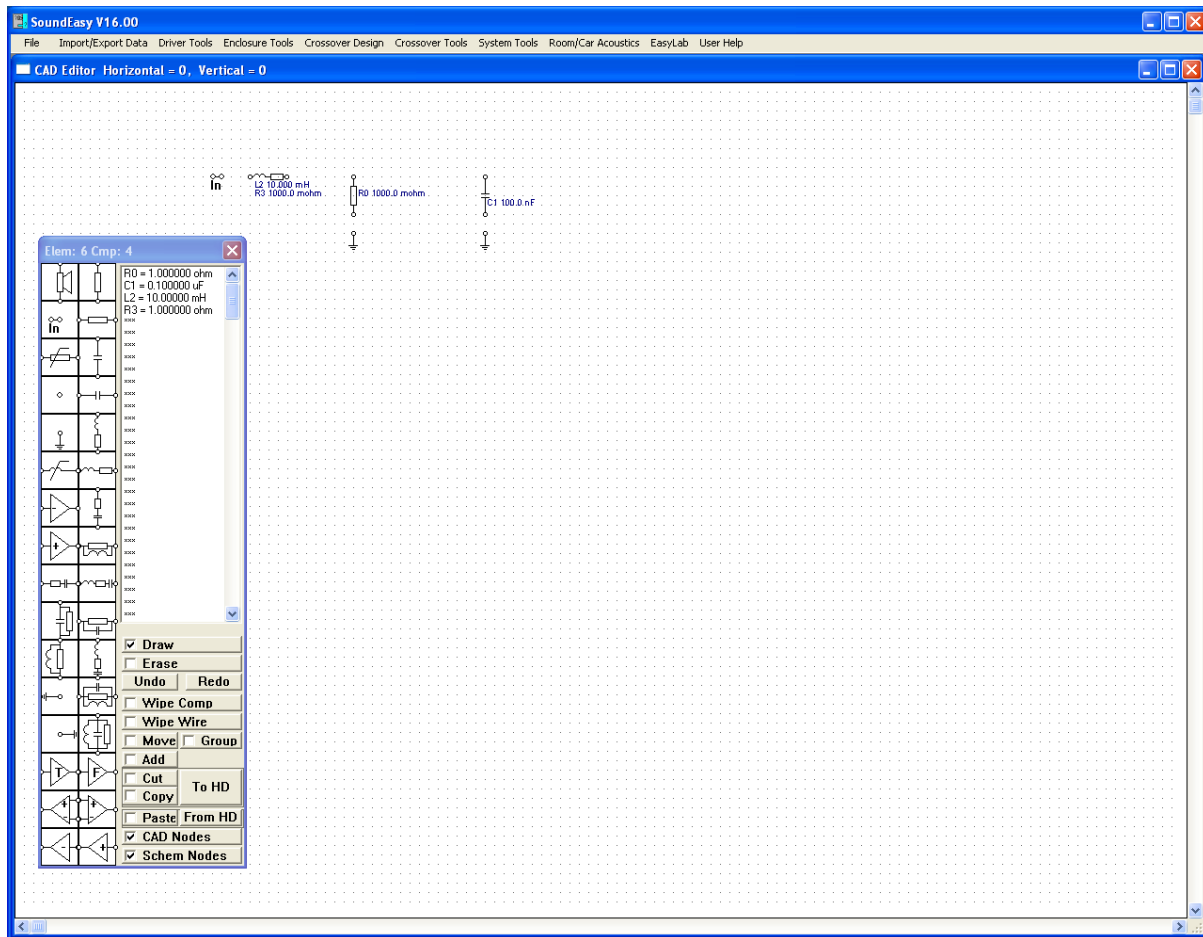


Fig 8.4 All LP filter components placed

7. In order to connect resistors R12-R13-R14, pick the "soldering point" node from the list displayed on the right hand side (the little circle) and place the node in the middle between R12 and R13 resistors. Next, position the pointer arrow on resistor R12, then press and keep pressing the **LEFT** mouse button. Drag the pointer to the right until it is placed above the node, release the mouse button. Place the pointer above the node, press and keep pressing the **LEFT** mouse button while moving the pointer right to the resistor R13. Release the mouse button. Place the pointer above the node, press and keep pressing the **LEFT** mouse button while moving the pointer down to the resistor R14. Release the mouse button. The line drawn initially will be replaced by a neatly drawn, connection. The node or "soldering point" must be placed before the connecting line was drawn. Having mastered the technique of connecting components, the user is now ready to continue with the example schematic in Fig 8.6. Place the arrow pointer on resistor R0 and press the **LEFT** mouse button, then drag the pointer to the node and release the mouse button. A left angle line is drawn connecting R0 and node - Node "0" was created. Continue with the rest of the connections following Figs 8.7 to 8.9. There is no need to exercise great accuracy when positioning the pointer arrow. The 'Crossover' tool cleans the drawing and makes all connections the correct way. Certain connections are not allowed. For example, R1-R8 cannot be connected or R6-R12 can not be connected without a node. If R0 was not there, one could place a node at the square 45 deg up and right from R6 and then connect R6 and R12 via the node. A fully developed schematic is shown in Fig 8.8. "Crossing the wires" is allowed and two nodes are assigned by the program - see Fig 8.5, R15,R16,R17,R18.

**Note 1:** Every network must have an Input node and some ground connections. Test signal is always injected between the Input node and the ground and measured or monitored at the nominated node and the ground (or two nominated nodes for floating loads ).

**Note 2:** Data file for each driver can hold up to 256 elements (components + solder joints + ground connections + input node) – DO NOT exceed this number.

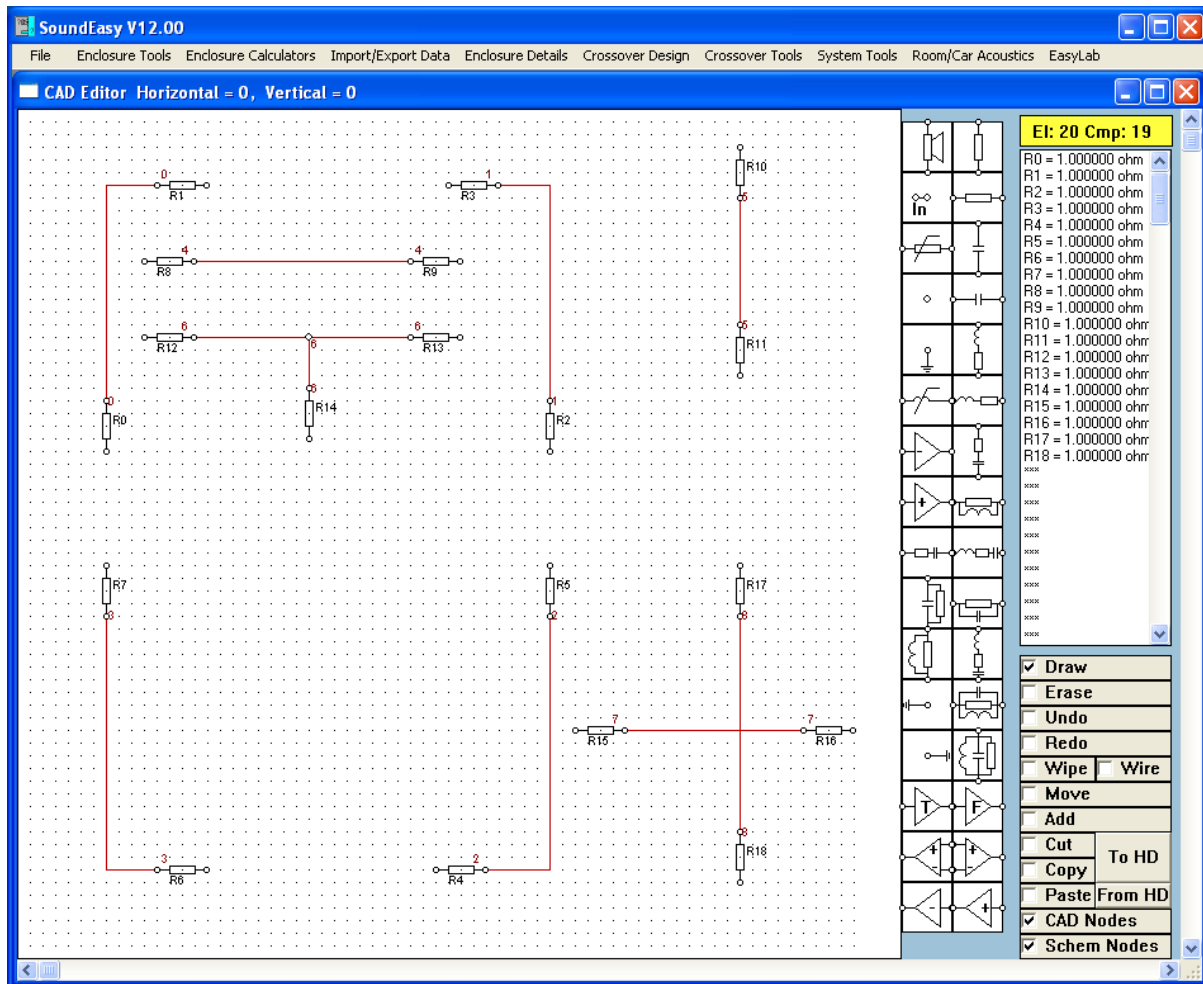
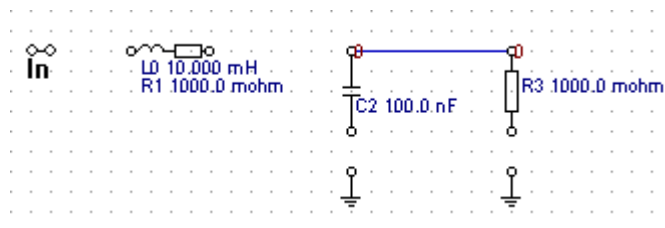


Fig 8.5 Various connection types



Please note, that link connects two nearest component terminals.

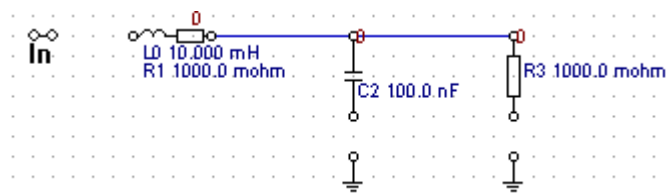


Fig 8.6 Making progressive connections between components

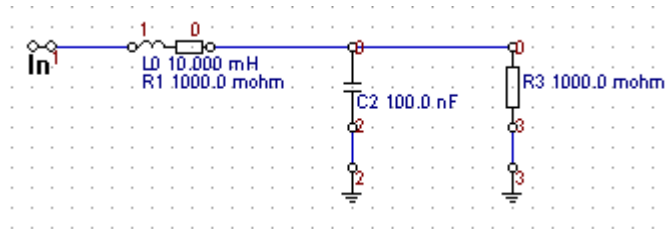


Fig 8.7 All connections for low-pass filter have been made.

### Entering Component Values

Please note, that starting from V12, component values are edited using the **RIGHT MOUSE BUTTON**.

The final step in creating the network is assigning values to the components. In order to assign a value to a component, position the arrow pointer above the required component and **CLICK** the **RIGHT** mouse button. As a result of this, a window will be opened with 1, 2 or 3 editable fields for entering the required value. The entered data must be a positive, real number. When finished, click on the "Done" button. Enter the value of the each component in the same way. The entered data is displayed under the "Parts List" heading as soon as the data entry window is closed. When all the values are assigned, the network is now ready for analysis (see Fig 8.7).

Driver CAD symbol must be associated with appropriate file data. For instance, woofer data must be associated with woofer CAD symbol - **CLICK** the **RIGHT** mouse button over the driver symbol.

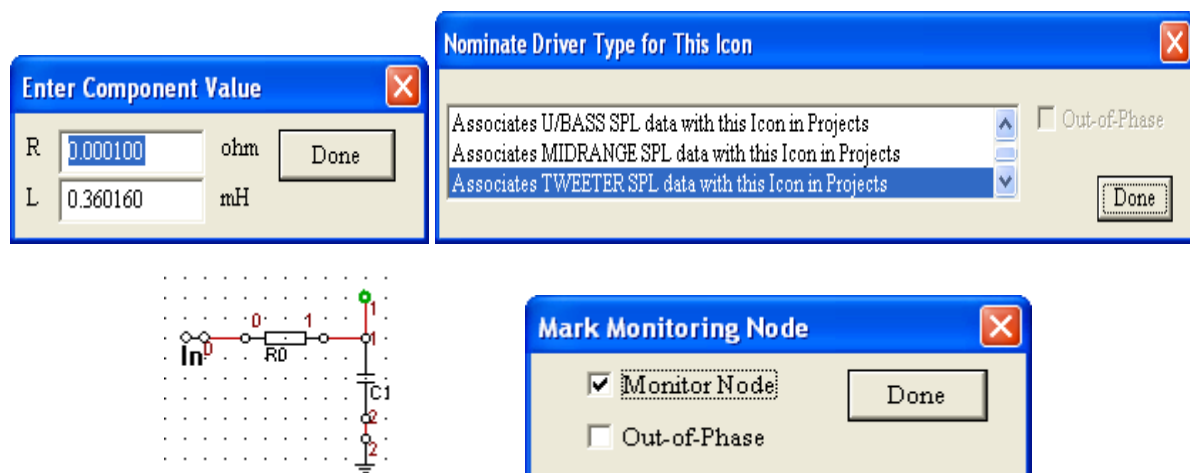


Fig 8.8 Typical Component Value Entry Box + Associating driver type with **driver icon** on the schematic. Also, an example of "Monitoring Node" on the schematic – only active in "Automatic Node Numbering" mode. Monitoring Node can be set "out-of-phase".

### Correcting Simple Error

The schematic capture technique which was implemented in the 'Crossover' tool allows the user to perform some simple corrections on the schematic. Most of the potential errors when connecting the components will be intercepted by the module before they are allowed to occur. If an unacceptable connection is made, the program erases the wrong line and flashes an error message window. However, the program will not detect a problem if the wrong type of component was placed on the drawing area. The program allows the user to replace one component with another one dragged and placed on the top of the first (wrong) one, if the new component is of the same orientation as the old one. It can be noticed, that some components listed on the right hand side of the "Edit" window lend themselves more easily to "series" connection with the signal flow (left and right terminals) and some components are of the "parallel" type, which is easier to connect as a shunt element to the ground (top and bottom terminals).

## Reversed OPAMPS in CAD system

Component list now includes the following elements:

1. Resistor – 2 orientations
2. Capacitor – 2 orientations
3. Serial L-R circuit – 2 orientations
4. Serial R-C circuit – 2 orientations
5. Parallel L-R circuit – 2 orientations
6. Parallel R-C circuit – 2 orientations
7. Serial R-L-C circuit – 2 orientations
8. Parallel R-L-C circuit – 2 orientations
9. Loudspeaker element – you can select woofer / upper bass / midrange / tweeter / upper tweeter driver files to be included in the analysis for SPL/ phase and time domain.
10. Input Node
11. Variable resistance element (for  $Z_{in}$  model)
12. Solder Joint
13. GND Node – 3 orientations.
14. Inverting OPAMP – 2 orientations
15. Non-Inverting OPAMP – 2 orientations
16. Differential OPAMP – 2 orientations
17. “T-Component” – Super Component. Assumes identity of a large number of filters, circuits and importer transfer functions.
18. “F-Component” – Implements filter transfer functions and Mimics 41 functions of Behringer 2496.

All components have been provided with two orientations. This allows you to see how the component will look in the schematic before it's placed.

Please note, that in addition to these components, you have over 100 built-in filters, crossovers and other circuits to choose from, when designing your schematic.

**Reversed OPAMPS →**

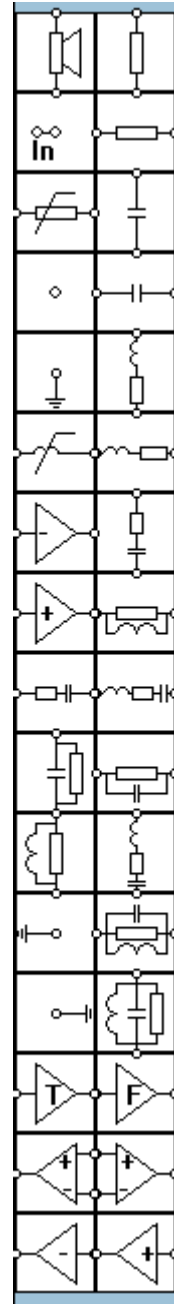


Fig 8.9 Menu “Pick” for CAD system.

## CAD Editing Functions

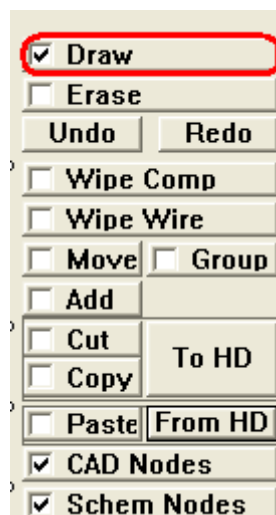
SoundEasy now includes 18 CAD processing related functions:

1. **Draw** – For “pick-and-place” components and drawing connections
2. **Erase** – For erasing single connection or component
3. **Overlap** – Changing components without disconnecting
4. **Undo / Redo** - For “undoing” the last 4 steps, and cancelling the last Undo operation
5. **Wipe** – For disconnecting and erasing the component
6. **Wipe Wire** – For erasing wires
7. **Move** – For moving a component around.
8. **Group** – for selecting several components at the same time.
9. **Add** – For adding a complete, pre-calculated filter or even a complete crossover to your schematic
10. **Cut** – For cutting a section of the schematic
11. **Cut To Hard Disk** – For cutting a section of the schematic and saving it to HD
12. **Copy** – For duplicating a section of the schematic
13. **Copy To Hard Disk** – For duplicating a section of the schematic and saving it to HD
14. **Paste** – For pasting the Cut section
15. **Paste From Hard Disk** – For pasting the Cut section from HD.
16. **Cad Nodes** – For switching ON/OFF dots (nodes) on CAD area.
17. **Schematic Nodes** - – For switching ON/OFF node numbering on CAD area.
18. In addition, there are over 100 **in-built filters and crossovers** to choose from. This approach allows you to use a calculator for deriving component values, so when the selected circuit is placed on the CAD screen, it will come fully functional. Possibly the simplest way of creating a filter is to load one of the in-built designs and modify it for your specific purpose. The process will be described in the next chapter.

When using functions like: loading in-built circuits, Add, Cut/Paste, Copy/Paste, Paste From HD, the selected circuit is placed in a buffer, so you can move the circuit around the CAD screen ( while pressing the left mouse button ) and place the circuit where you want it by releasing the left mouse button. It is likely, that you will find yourself using this functionality quite often.

### 1. Draw Function

The Draw function is the possible the most fundamental function of all, as it allows you to create your schematics. You can pick-and-place selected component and also link the components already placed on the CAD screen. The function is activated by placing the mouse pointer on the “Draw” check box and pressing the left mouse button. You should now see the “checkmark” as shown below. Now you can move the mouse pointer onto the component’s list item and press the left mouse button to select your choice. Keep the left mouse button pressed down, so you can pick the component and move it to where you want it on the CAD screen. Once the component is positioned, please release the left mouse button – the component you picked is now drawn on the schematic.

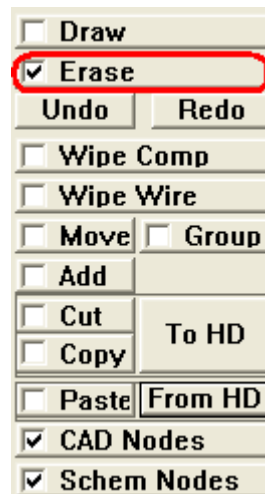




If you have two or more components placed on the CAD screen, you can connect them together with a link. The link is created by placing the mouse pointer tip on one of the component's nodes, and while the left mouse button is pressed down, you drag the mouse pointer to the other component's node to be connected. Then release the left mouse button.

## 2. Erase Function

The Erase function is your primary tool for removing components and links from the schematic. The function is activated by placing the mouse pointer on the "Erase" check box and pressing the left mouse button. You should now see the "checkmark" as shown below.



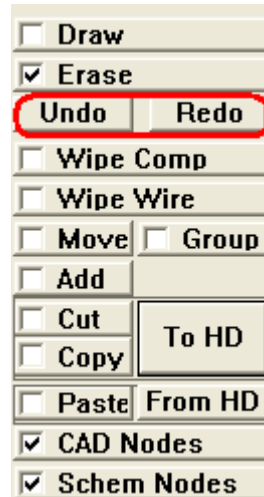
To erase a link between two components, perform the exact reverse of linking process. Place the mouse pointer tip on the start of the link, and while the left mouse button is pressed down, you drag the mouse pointer to the other component's node to be disconnected. Then release the left mouse button. Component can only be "erased" if all of it's links have been erased first. Then place the mouse pointer on the component to be erased and double-click the left mouse button. This will remove the component from the CAD screen.

## 3. Overlap Function

SoundEasy will accept one serial component placed on the top of another serial component and one parallel component placed on the top of another parallel component. Use the previously described "pick and place" technique to replace components. The parts list will be updated automatically. There is no button for the "Overlap Mode", just pick another component of compatible orientation and place it on the top the existing one.

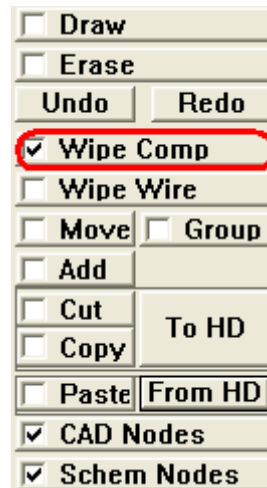
## 4. Undo / Redo Functions

The Undo function comes very handy if you decide to reverse the last four steps in creating your schematic. Single press on the "Undo" check box will take back one step. The program remembers the last four steps, so you can reverse all of them. If you happen to realise, that you have "undone" one change unnecessarily, you can use the Redo function to reverse the Undo action.



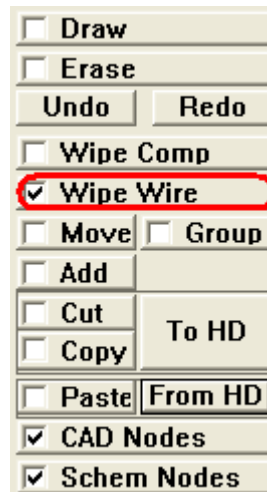
## 5. Wipe Function

Wipe function is a productivity-type function. In some circumstances, you may decide to remove a component from the schematic completely and permanently. You could use the Erase function to remove links one-by-one and then erase the component. Or, you can use the Wipe function to erase a component and all links connected to it. The function is activated by placing the mouse pointer on the “Wipe” check box and pressing the left mouse button. You should now see the “checkmark” as shown below. Then place the mouse pointer on the component to be removed and double-click the left mouse button. This will remove the component + all it’s links from the CAD screen.



## 6. Wipe Wire Function

Wipe Wire function is a productivity-type function as well. If you find yourself in a situation, where you need to erase many links quickly, the Wipe Wire function will set the program in “wire removing mode”, and then you can double-click on any wire to have it removed. The function is activated by placing the mouse pointer on the “Wire” check box and pressing the left mouse button. You should now see the “checkmark” as shown below. Then place the mouse pointer on the link to be removed and double-click the left mouse button. This will remove the link from the CAD screen.



## 7. Move Function

Components can be moved around the CAD screen freely. The component will stay connected and the links, that are affected by the move are highlighted in green. The function is activated by placing the mouse pointer on the “Move” check box and pressing the left mouse button. You should now see the “checkmark” as shown below. Then place the mouse pointer on the component to be removed and press the left mouse button. Keep the button pressed down as you move the component. Once you move the component where you want it, release the left mouse button. This will draw the component in the new location on the CAD screen.

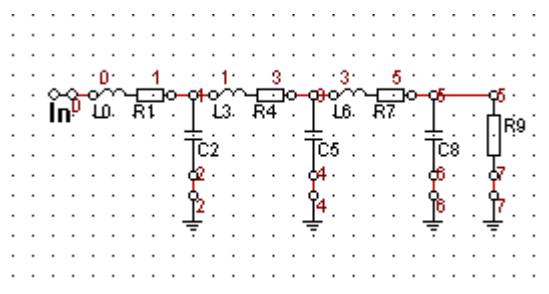
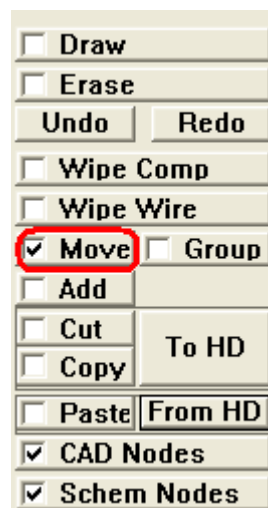


Figure 8.10 Original circuit

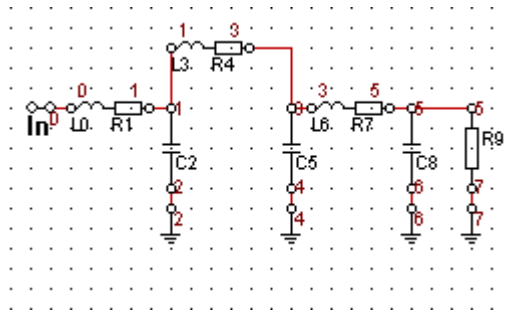


Figure 8.11 L3+R4 (lossy inductor) components have been moved.

You can also “Move” several components together. Select “Group” checkbox option and draw banding rectangle around components using left mouse button. Start drawing banding rectangle from top-left corner and finish with the bottom-right corner. Next, press left mouse button on one of the selected components and you will be able to move all selected components around.

## 8. Add Function

The “Add” function allows you to put the CAD system in “network add” mode. This is very powerful editing mode, as it allows you to add complete filter or crossover with all components pre-calculated.

Figure 8.12 shows the CAD screen with already uploaded 4-nd order Butterworth low-pass filter. Our goal is to create a crude band-pass filter by adding a 2-th order high-pass Linkwitz filter to it.

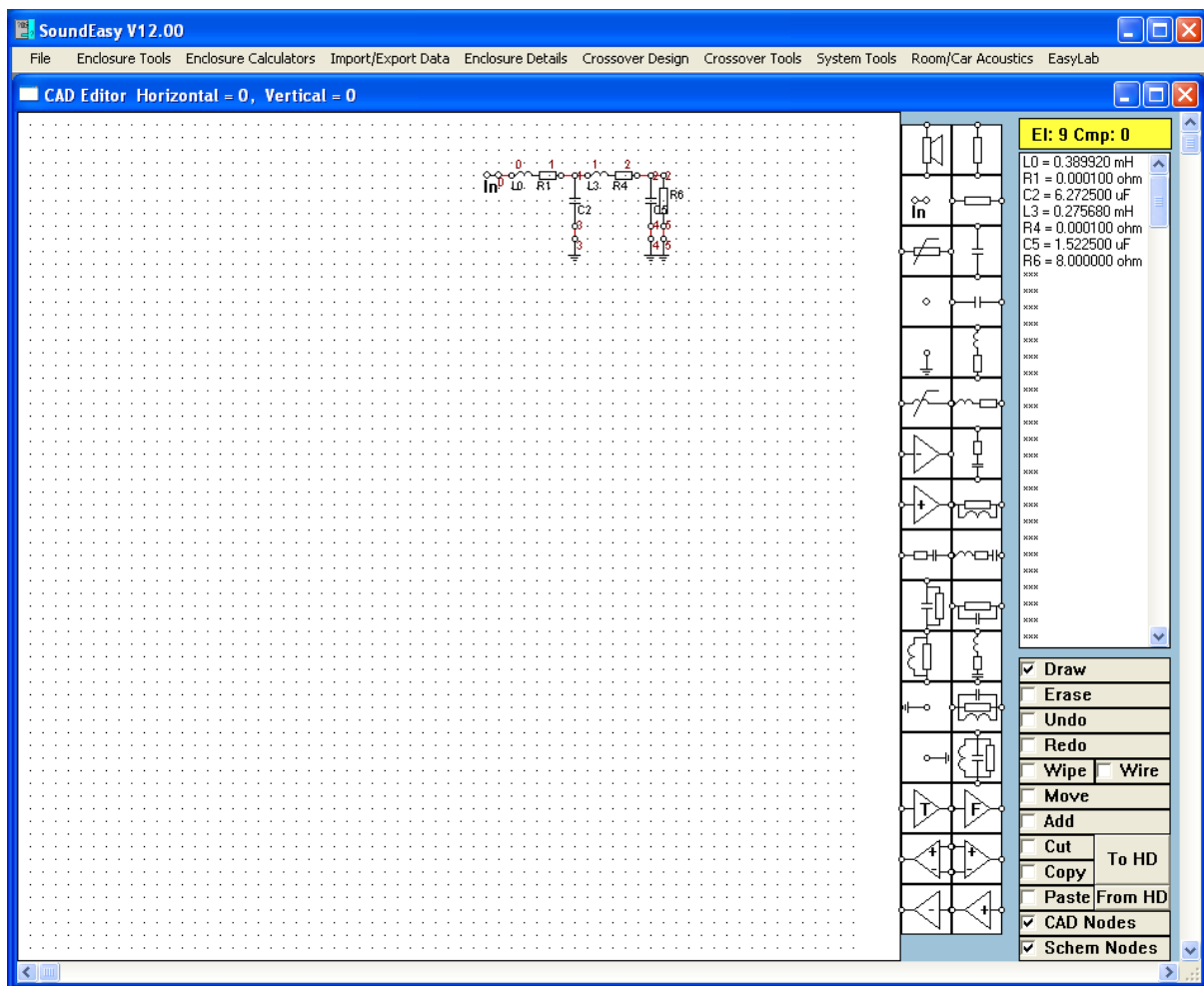


Fig 8.12 Low-pass filter placed on the CAD screen.

Next, place the CAD screen in the “Add Network” mode.

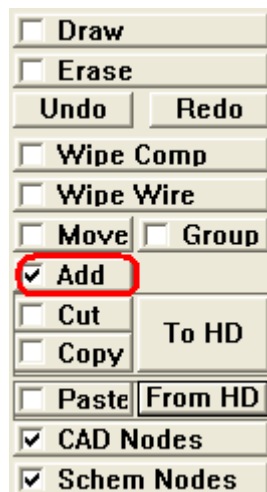


Fig. 8.13 CAD screen in “Add” mode.

Select the built-in filter you wish to use from the filters dialogue box as before. The filter will be placed at your specified location with all components calculated – see Figure 8.14

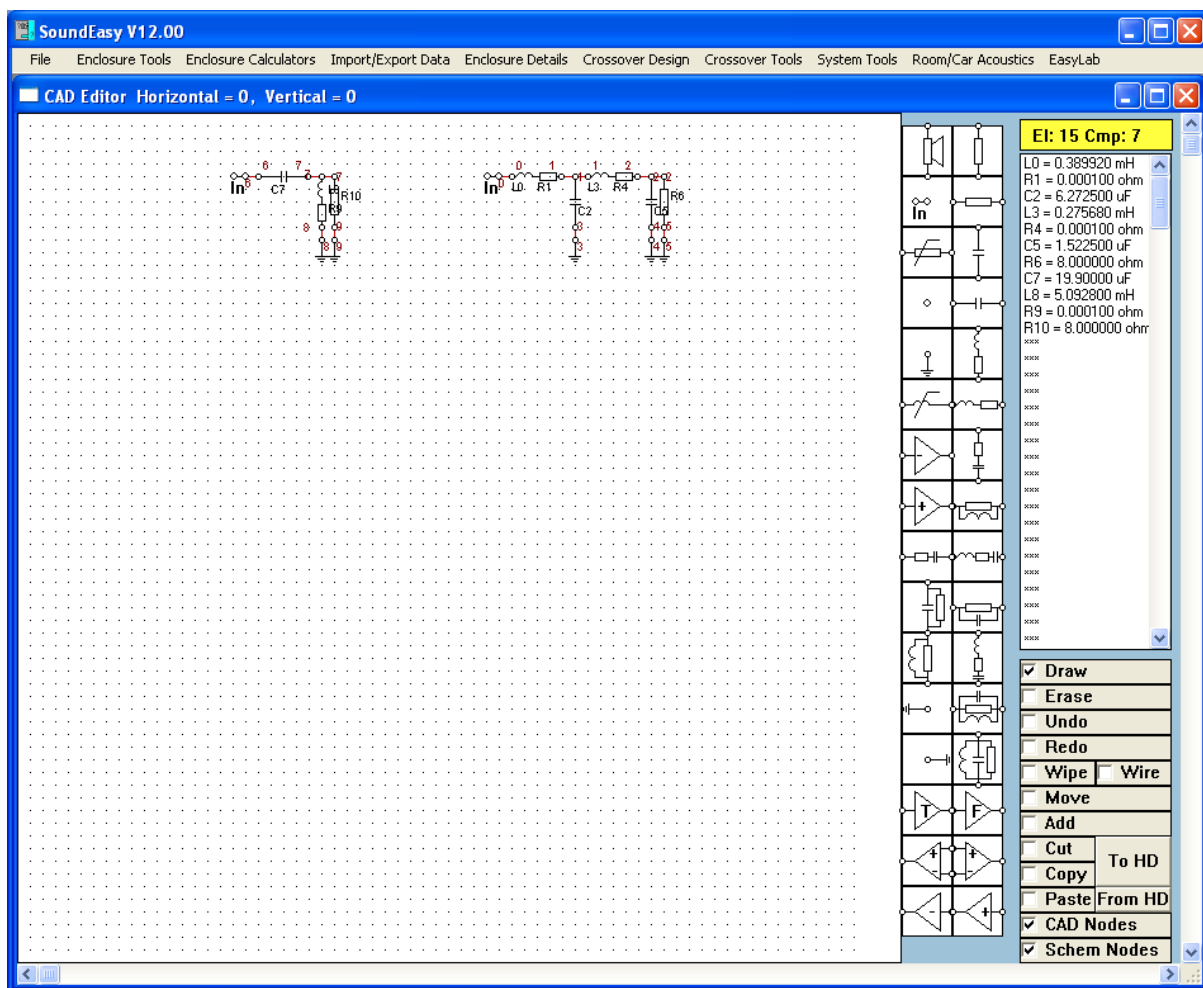


Figure 8.14 Low-pass and High-pass filters on the same screen

Please note, that you now have TWO inputs and all node numbering is duplicated – this is due to the fact, that you now have two independent circuits on one screen. Also, the first circuit has the load resistor connected to its output (R10) – and this component is now redundant.

Click on “Erase” button to erase the R10 (load for the first filter) and “IN0” (input to the second filter). Finally, click on “Draw” button and link the filters – as shown on Fig 8.15.

We have now created an asymmetric band-pass filter (uploaded 4-th order Butterworth low-pass and 2-th order high-pass Linkwitz filters). You can continue adding more and more circuits to create quite complex asymmetrical crossover very quickly. The “Add network” function allows you to freely mix-and-match filter + new filters and crossover + new filters. You can upload a crossover and then ADD to it more filters with all components calculated. Inevitably, some more circuit manipulation will be needed to produce your final filter or crossover.

SoundEasy performs basic checks for duplicate input nodes, wrongly anchored filters. Still, we recommend, that you are sensible about selecting schematics that are too large to fit on the available CAD screen.

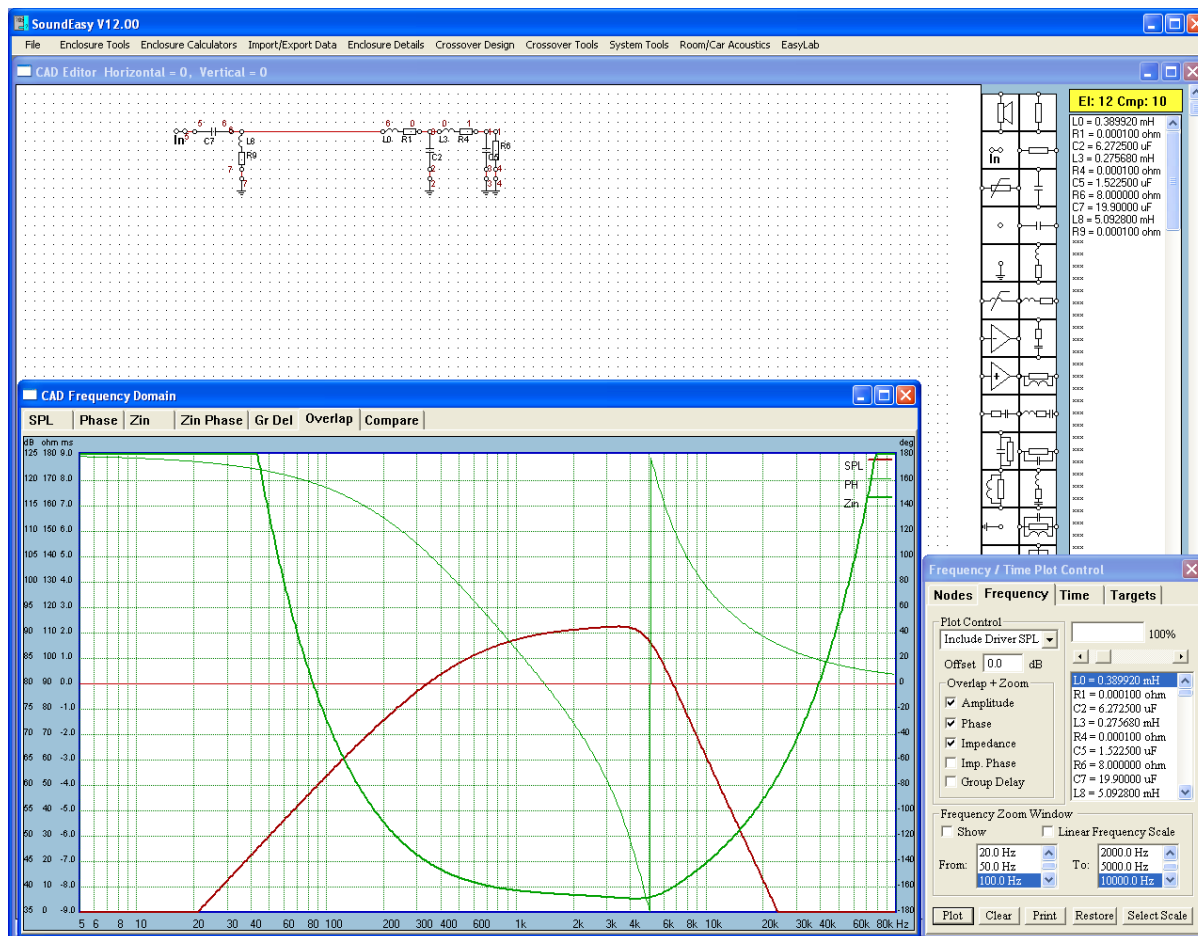


Fig 8.15 Two filters linked properly and their response as the band-pass filter

## 9. Cut Function

The “Cut” function allows you to quickly remove large portion of the schematic. The function is activated by placing the mouse pointer on the “Cut” check box and pressing the left mouse button. You should now see the “checkmark” as shown below. Please note, that the removed circuit is placed in a buffer, so it is still available for “Paste” operation.

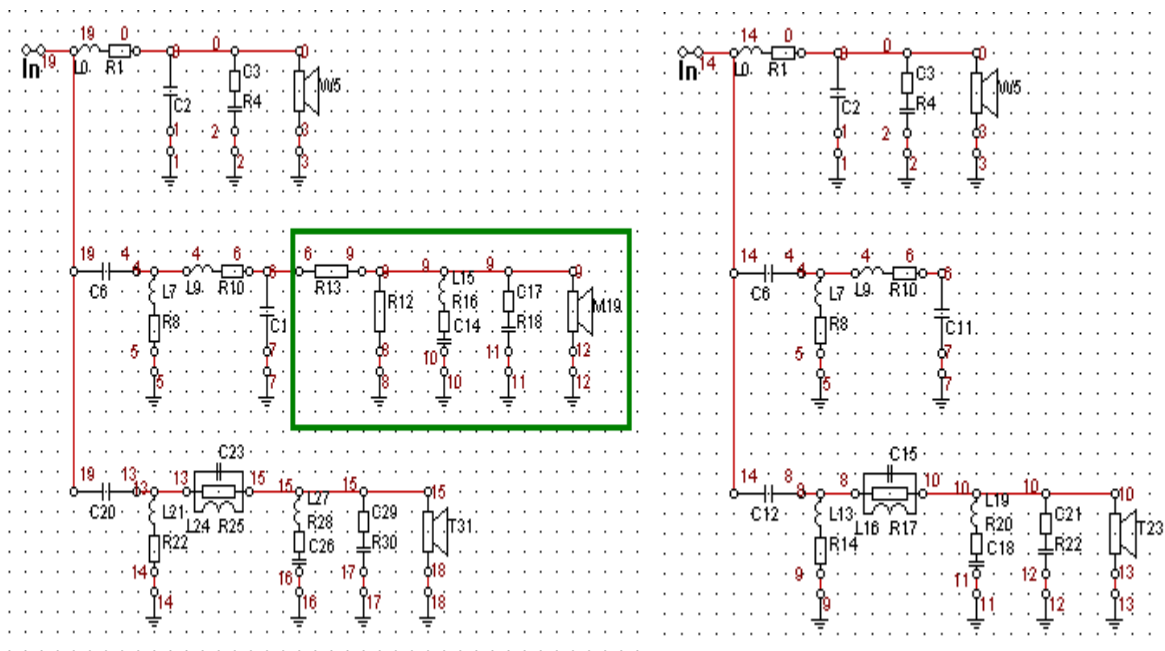
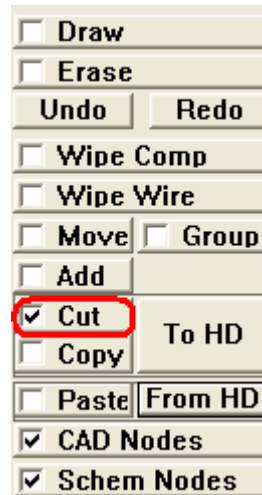


Figure 8.16. Simple schematic example “before” and “after” Cut operation

For the effect of complimentary “Paste” function see the following paragraphs.

Here is how it works:

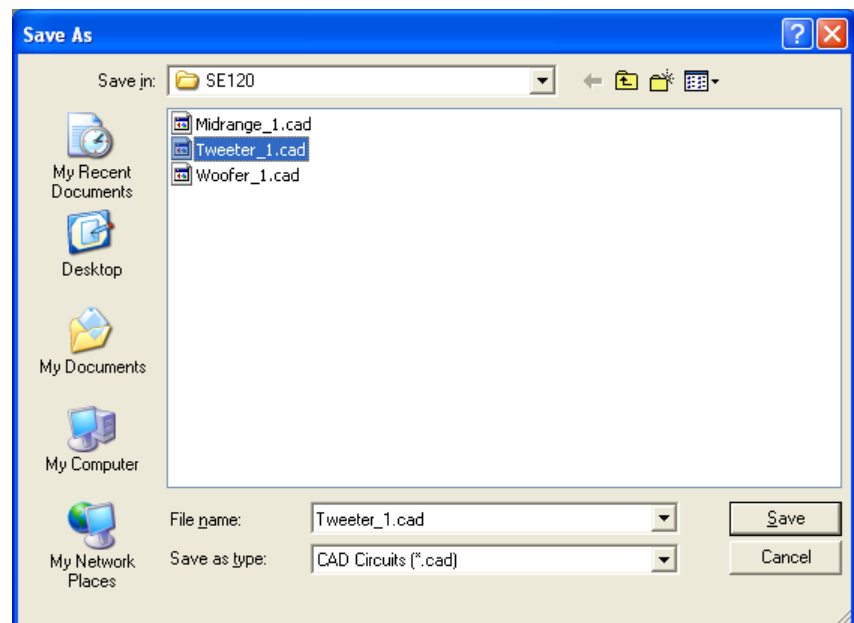
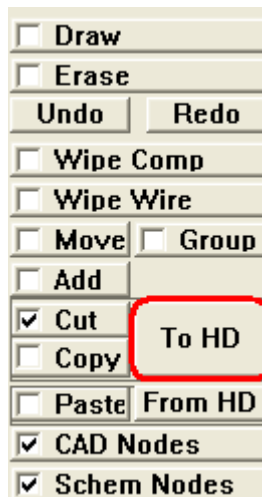
The sub network should be able to be bounded by a rectangle.

- Click on "Cut" check box.
- Place the mouse pointer on the "top-left" rectangle of the sub-network. Press the left mouse button and keep it pressed.
- Move the mouse pointer to the "bottom-right" rectangle of the sub-network. Release the left mouse button.

Move the mouse pointer where you want to place the sub-network. Click on the left mouse button (and keep it pressed), you can move the circuit around. This is your chance to make sure, that the sub-network is placed conveniently and will not overlap any other components. Release the left mouse button. You should be able to make **single "cut" and multiple "paste" operations**. All component values should be replicated.

## 10. Cut To Hard Disk Function

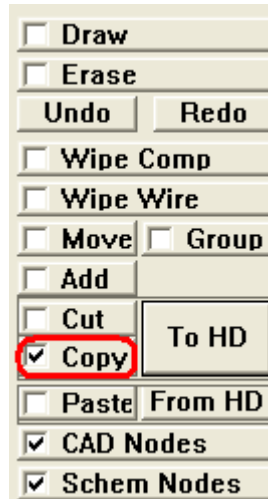
The "Cut To HD" function allows you to quickly remove large portion of the schematic and save it to your Hard Disk. The function is activated by pressing the "To HD" button. Please note, that you should "Cut" the circuit first, before selecting "To HD" button. Once the button is pressed, the standard "Save As" dialogue box will be invoked to allow you to enter a file name for the cut circuit. The circuits are saved with ".CAD" extension – see below.



## 11. Copy Function

The "Copy" function allows you to quickly duplicate large portion of the schematic. The function is activated by placing the mouse pointer on the "Copy" check box and pressing the left mouse button. You should now see the "checkmark" as shown below. Please note, that the removed circuit is placed in a buffer, so it is available for complimentary "Paste" operation.





Here is how it works:

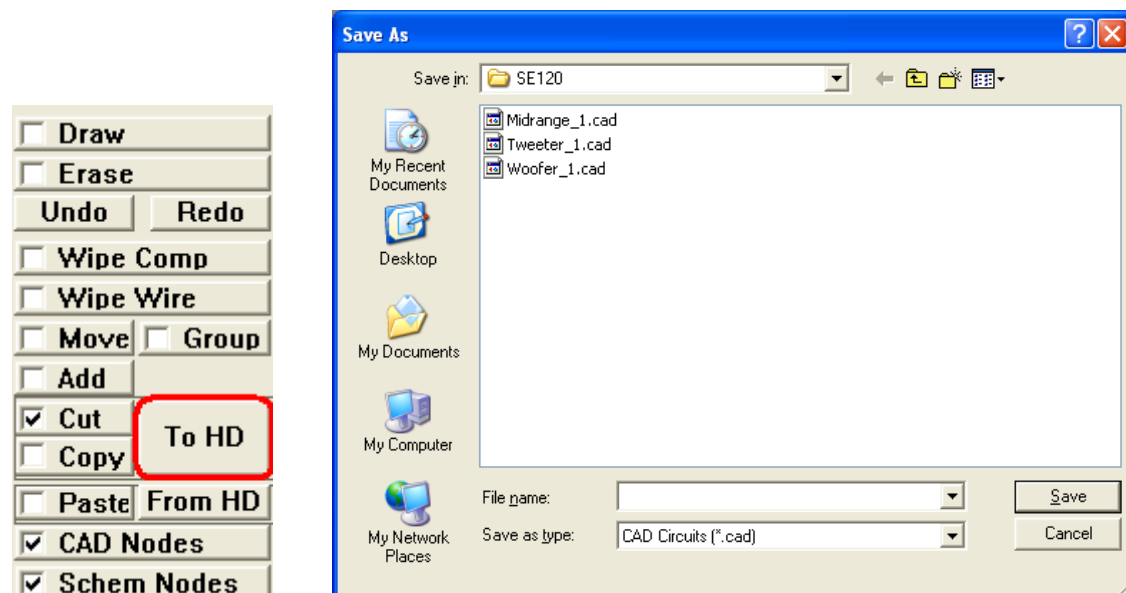
The sub network should be able to be bounded by a rectangle.

- Click on "Copy" check box.
- Place the mouse pointer on the "top-left" rectangle of the sub-network. Press the left mouse button and keep it pressed.
- Move the mouse pointer to the "bottom-right" rectangle of the sub-network. Release the left mouse button.

Move the mouse pointer where you want to place the sub-network. Click on the left mouse button (and keep it pressed), you can move the circuit around. This is your chance to make sure, that the sub-network is placed conveniently and will not overlap any other components. Release the left mouse button. You should be able to make **single "cut" and multiple "paste" operations**. All component values should be replicated.

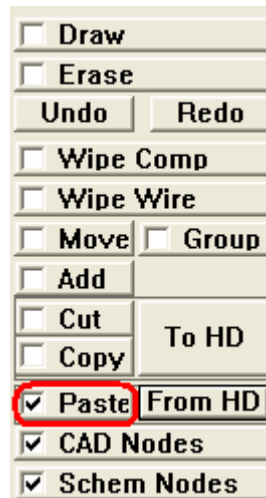
## 12. Copy to Hard Disk Function

The "Copy To HD" function allows you to quickly duplicate large portion of the schematic and save it to your Hard Disk. The function is activated by pressing the "To HD" button. Please note, that you should "Copy" the circuit first, before selecting "To HD" button. Once the button is pressed, the standard "Save As" dialogue box will be invoked to allow you to enter a file name for the cut circuit. The circuits are saved with ".CAD" extension – see below.



### 13. Paste Function

The “Paste” function allows you to transfer the circuit from a temporary buffer onto the schematic. Obviously, the circuit to be pasted, must have been placed into the buffer by Copy or Cut functions already. The function is activated by placing the mouse pointer on the “Paste” check box and pressing the left mouse button. You should now see the “checkmark” as shown below.



Move the mouse pointer where you want to place the sub-network. Click on the left mouse button (and keep it pressed), so you can move the circuit around. This is your chance to make sure, that the sub-network is placed conveniently and will not overlap any other components. Release the left mouse button. You should be able to make **single "cut" and multiple "paste" operations**. All component values should be replicated.

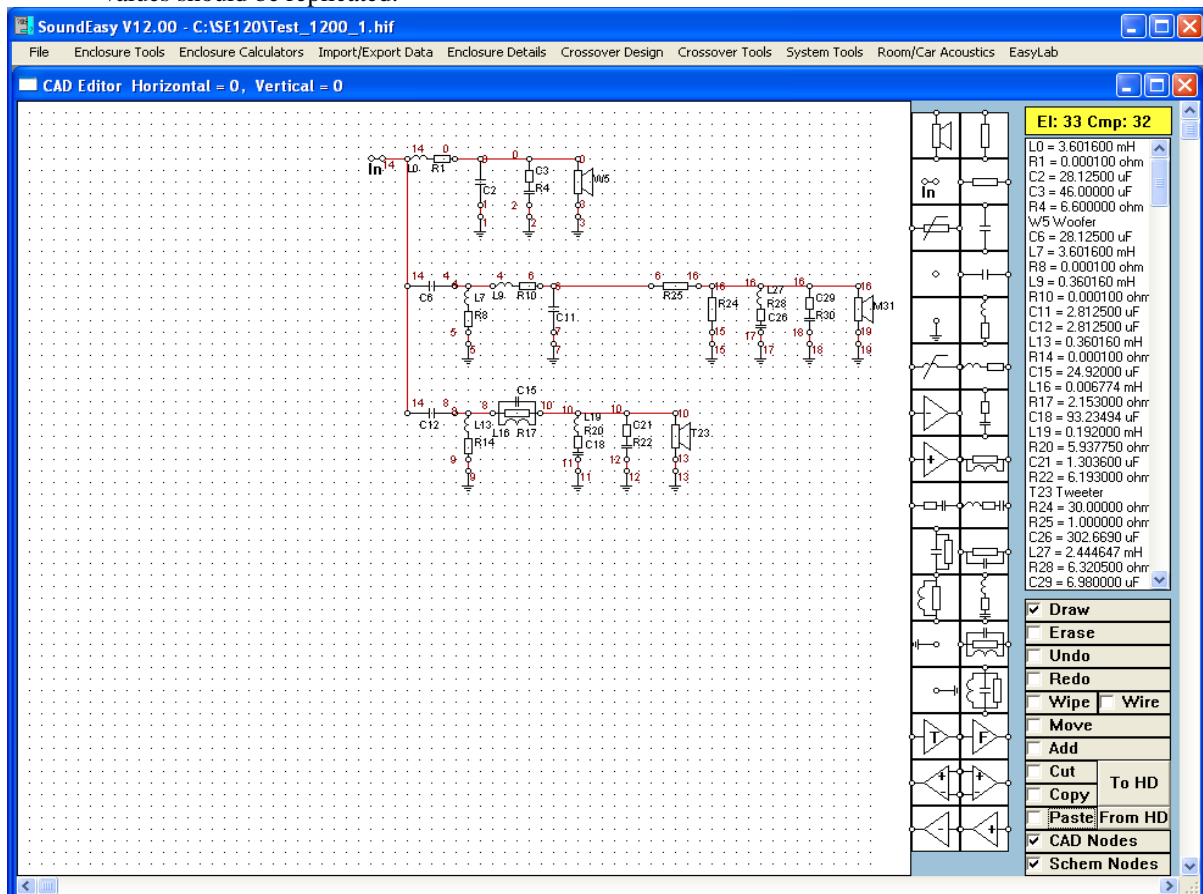
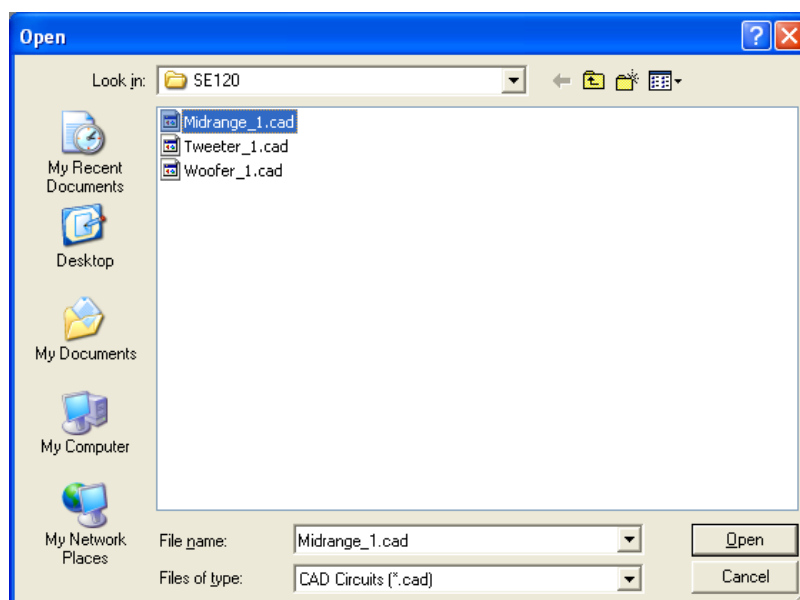
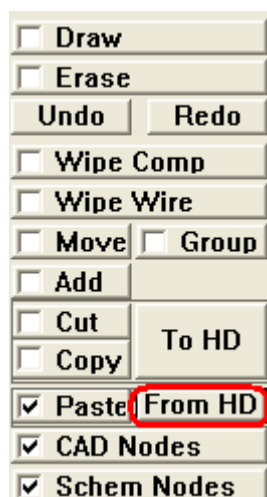


Figure 8.17. Simple schematic example “after” Cut-and-Paste.

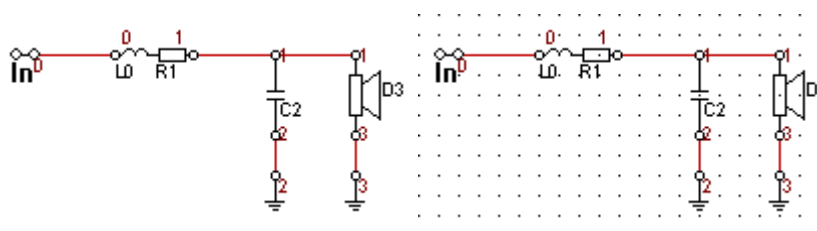
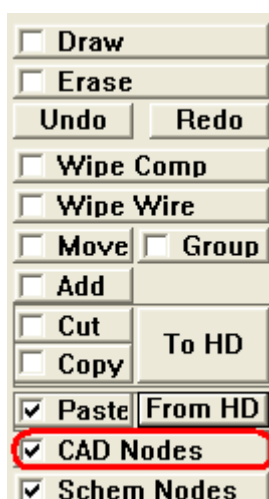
## 14. Paste From Hard Disk Function

The “Paste” function allows you to transfer the circuit from a temporary buffer onto the schematic. Obviously, the circuit to be pasted, must have been saved into the HD by Copy or Cut functions already. The function is activated by pressing the “From HD” button. Once the button is pressed, the standard “Open” dialogue box will be invoked to allow you to enter a file name for the cut circuit. The circuits are saved with “.CAD” extension – see below.



Once you select and load the circuit, move the mouse pointer where you want to place the sub-network. Click on the left mouse button (and keep it pressed), so you can move the circuit around. This is your chance to make sure, that the sub-network is placed conveniently and will not overlap any other components. Release the left mouse button. You should be able to make **single "cut" and multiple "paste" operations**. All component values should be replicated.

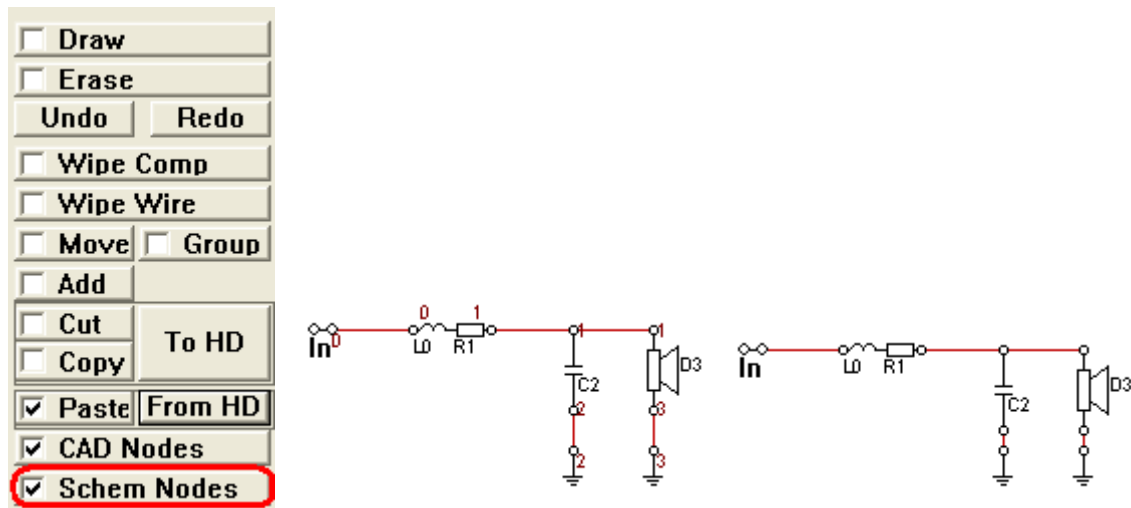
## 15. Cad Nodes Function



The “CAD Nodes” function is just a convenient switch to select/deselect location nodes on the CAD screen.

When the components are moved around the CAD screen, they are automatically placed on a “nodal raster”. This process manifests itself as slight “jumps” when the components are moved on the screen because the program attempts to place the components onto the nodes CLOSEST to the current position. Seeing the location of CAD nodes allows you to place the components more precisely. However, this is not strictly necessary, as you can move the components freely after they were originally placed.

## 16. Schematic Nodes Function



The “Schematic Nodes” function is just a convenient switch to select/deselect the display of electrical nodes on the CAD screen. You need the node numbers to tell the program where it should monitor the voltages for modelling network behaviour.

## Modelling Network Behaviour

The next step in the network design is the analysis of the network’s performance. This accomplished by plotting amplitude, phase group delay and input impedance of the network.

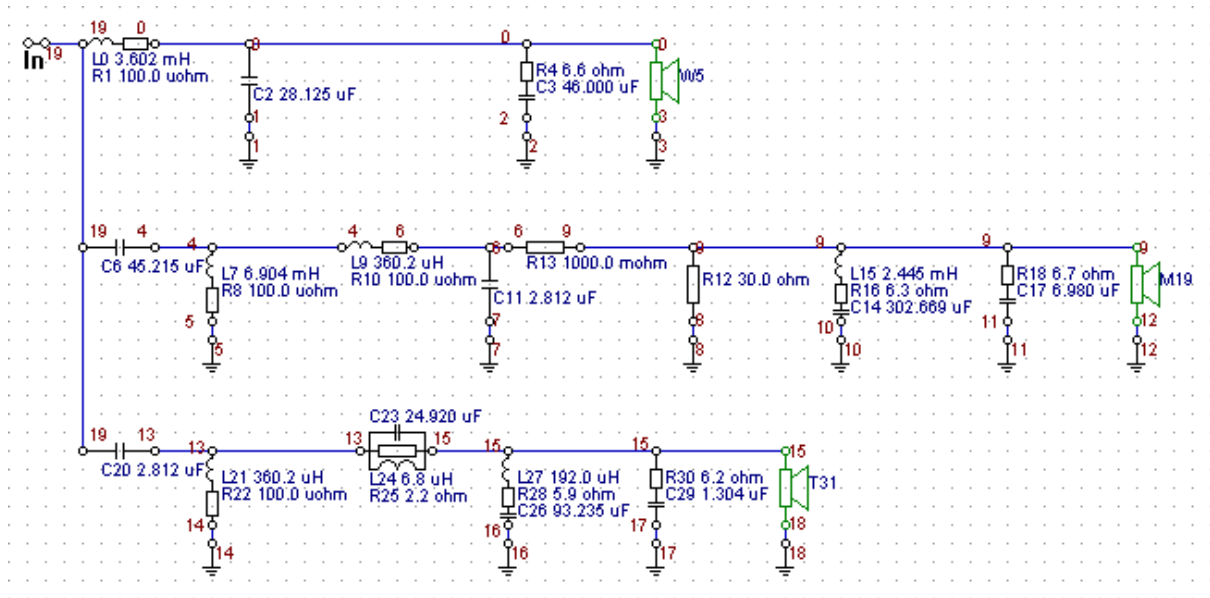
### Node Arithmetic

In order to plot transfer function ( like SPL ) of your circuit, you will need to determine which circuit nodes are involved. Input node is already determined for you, so the attention now need to focus on the output, or monitoring nodes. When you CAD schematic is fully edited, you will be able to read node numbers assigned to individual drivers.

Schematic below, shows the woofer connected between node “0” and ground “3”, midrange driver connected between node “9” and ground “12” and finally the tweeter driver connected between node “15” and ground “18”.

The transfer function is determined by calculating voltage difference between two nodes. Therefore, the second node number , typically the “To” nodes, would be preceded by a “-” sign to tell the program, that the voltage at this node must be subtracted from the “From” node. For instance, woofer transfer function on the schematic above would be calculated as:

$$W5 \text{ SPL} = \text{Node } 0 - \text{Node } 3$$



When your driver is connected “out-of-phase”, Both nodes: “From” and “To” must have inverted polarity. Using schematic above as an example, you can determine for midrange driver:

$$\mathbf{M19\ SPL = - (Node\ 9 - Node\ 12) = -Node\ 9 + Node\ 12}$$

Finally the tweeter’s transfer function, being connected “in-phase” would be determined from the following equation:

$$\mathbf{T19\ SPL = Node\ 15 - Node\ 18}$$

Following the schematic above, you would enter for woofer: “From” node “0” and “To” node “3”, for midrange driver: “From” node “-9” and “ To” node “12” and finally for the tweeter driver: “From” node “15” and “To” node “18”.

Electrically, voltage difference is always determined by measuring voltage between a Node X and the ground. You can easily observe, that Nodes 3, 12 and 18 are ground nodes. Therefore, voltage difference between ground and ground is always equal to zero. The program will detect this situation, and will respond with an error message.

For single-ended loads ( such as loudspeaker connected to ground on one end ), you must enter the grounded node as “\*”. This way, you tell the program to disregard this node. As a simple rule, you should assume, that all nodes entered in this TAB must be different from any ground node number. Given the above, your system SPL is determined from the following equation:

$$\mathbf{System\ SPL = Node\ 0 - Node\ 9 + Node\ 15}$$

Node arithmetic approach allows you to execute simple “add” and “subtract” voltage when calculating transfer functions. You can have up to 20 additions or 20 subtractions or any combination of the two operations not exceeding 20 operations. Series crossovers ( or floating loads ) have drivers with both nodes different from ground nodes. Therefore, you would enter both node numbers: “From” and “To”, into the TAB.

Fully edited nodes for the example above, are shown here:

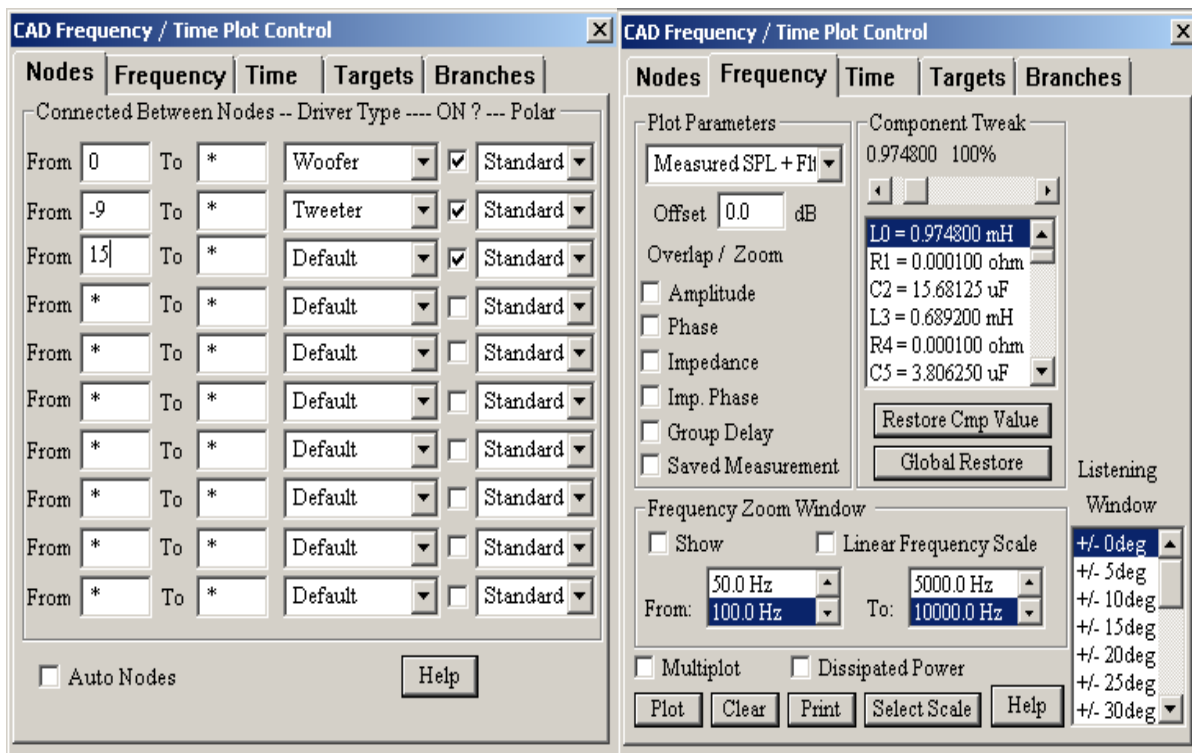


Fig 8.18 Tab for entering circuit nodes

Tab for selecting plots and component adjusting

### Automatic Node Numbering

This feature is turned ON from the Preferences screen. When activated, it will assign node numbers to your driver icons OR nominated nodes present on the schematic. **Your schematic must include driver icons OR marked nodes, for this feature to be operational.**

This automatic process incorporates simple rules and the nodal arithmetic described above. To make the best of the Automatic Node Numbering scheme:

1. Fully edit your CAD schematic.
2. Assign drivers (woofer, upper bass, midrange, tweeter or super tweeter) to each driver icon on the schematic - this is accomplished by right-click of the mouse button on the driver's icon to activate selection box.
3. Assign phasing to each driver – this is accomplished by right-click of the mouse button on the driver's icon to activate selection box. **Drivers “in - phase” are displayed in green colour, while drivers “out-of-phase” are displayed in blue colour.**
4. If you have a mixed-mode circuit, where you want the voltages to be displayed across the loudspeaker terminals AND at the nominated nodes, you **MUST** place node icons on the schematic (somewhere close to the circuit section where you want the voltage to be monitored), then link them to the nominated point, and edit the node parameters - nominate this node as “monitoring node” and assign arithmetic sign (or phasing) to it.

Once the above is completed, the Automatic Node Numbering algorithm will update your “Nodes” TAB data entry fields. You will need to turn drivers ON/OFF manually on the same TAB.

**There is one limitation associated with marked nodes. Please note, that using “marked nodes” approach you will not be able to link two marked nodes to obtain differential voltage across them. Marked nodes are always referenced to ground.**

## Nodes Tab

Schematic shown above, has the woofer connected between “From” node “0” and ground “3”, midrange driver connected between node “9” and ground “12” and finally the tweeter driver connected between node “15” and ground “18”. The “To” column of nodes would be used, if the driver was connected between two nodes AND none of them would be the “ground” node. For example, “series” crossovers have at least one driver connected to two floating nodes. When you assign component values to the schematic elements, you also need to assign driver type to each driver icon on the schematic. Simply press the right-mouse-button on the icon and you will be prompted to enter a number for identifying the driver. Driver type will be displayed in the “**Driver Type**” column. Finally, you can switch ON/OFF any driver from the analysis by checking/un-checking the boxes in the “**ON?**” column – see Figure 8.18. You must enter one (for single ended loads, second Nx enter as “\*”) or both node numbers (for floating loads) and **numbers must be different from any ground node number**. As explained before, in Manual Mode, driver polarity is reversed simply by **reversing the sign in front of the node number(s)** – see “-” in front of midrange driver on Fig 8.18.

The “**Polar**” column allows to select the measurement angle, at which the SPL was measured. The angles run in 5deg increments from -90deg to +90deg. “**Standard**” is the 0deg polar measurement. In order to use the off-axis SPL curves in modelling process, you must provide this data during the measurements for all angles.

## Frequency Tab

1. **Plot** – Button activates plotting for all selected functions.
2. **Clear** – Button clears the plotting window.
3. **Print** – Button opens printer dialogue box.
4. **Restore Cmp Value** – Button used to restore original component value (eg: after it’s value was changed by the slider).
5. **Global Restore** – Button used to restore ALL original component value (eg: after it’s value was changed by the slider). Restored values are the component values BEFORE you opened this dialogue box.
6. **Select Scale** – Button opens dialogue box for selecting vertical scale resolution.
7. **Offset** – Editable field for offsetting SPL plots in dBs.
8. **Slider and List-box** - Double-click on the list-box component to select it into the display field above. Then, you can use the slider to adjust component’s value.
9. **Plot Parameters** – Drop-down list box for selecting additional plotting parameters. See “**Additional Operations**” section.
10. **Help** – Brings up help information

Pressing **Plot** button the **Frequency TAB** will automatically plot all 5 transfer functions on individual TABs: SPL, Phase, Zin, Zin Phase and Group Delay.



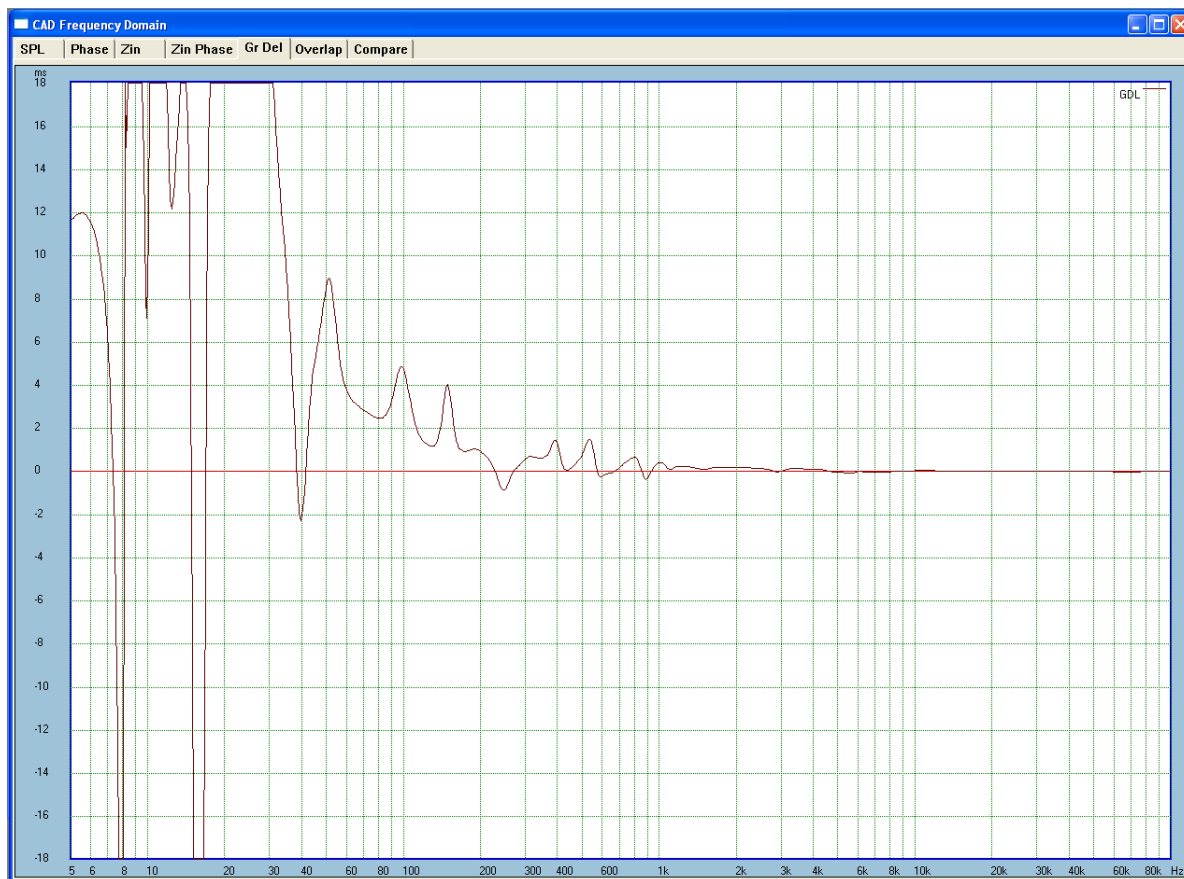
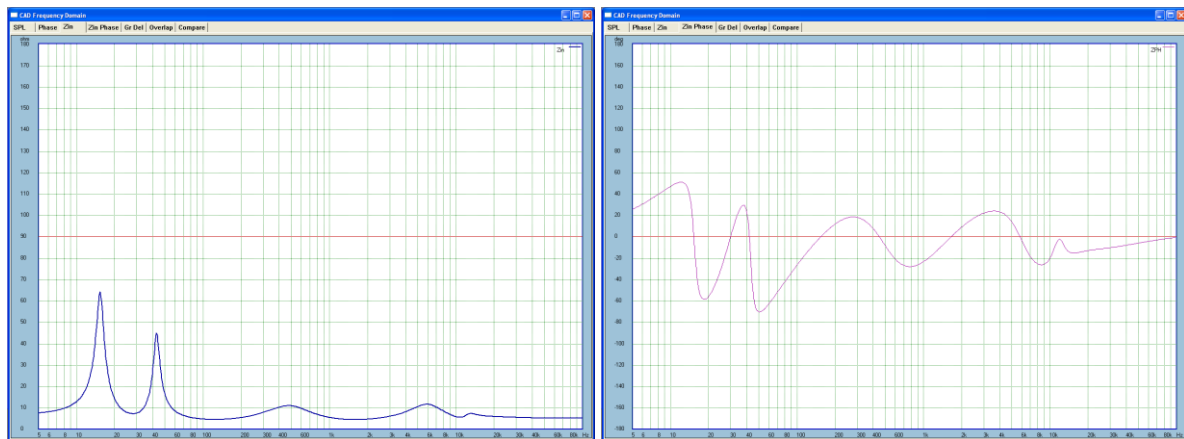


Figure 8.20 Example of 5 individual TABs plots + Overlay plot for our 3-way network.

Selecting any ( or all ) checkboxes on the “**Overlay + Zoom**” group **Frequency TAB** will automatically plot all 5 transfer functions on individual TABs: SPL, Phase, Zin, Zin Phase and Group Delay on the **Overlay TAB**.

Overlap + Zoom

☒ Amplitude

☒ Phase

☒ Impedance

☒ Imp. Phase

☒ Group Delay



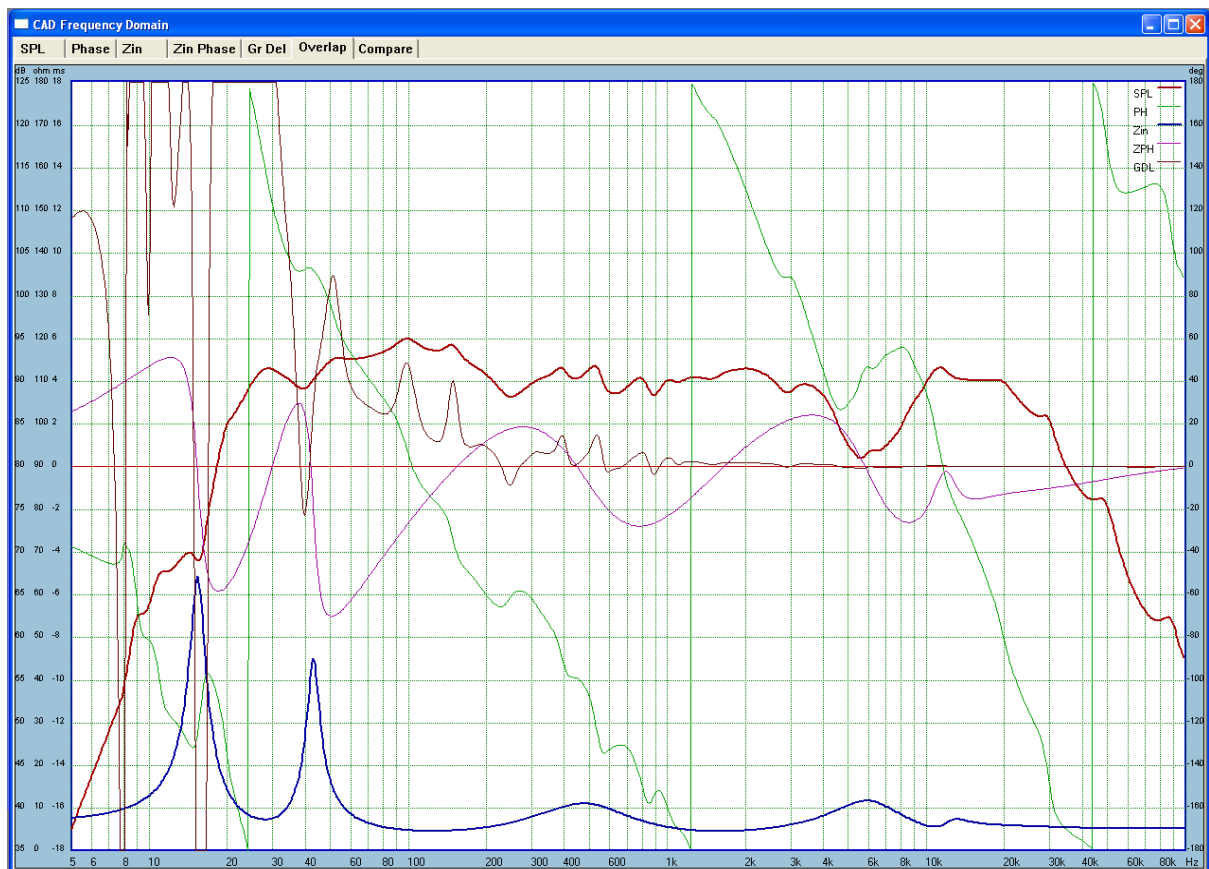
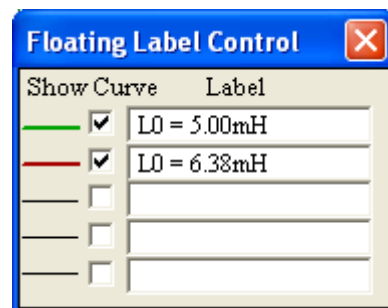
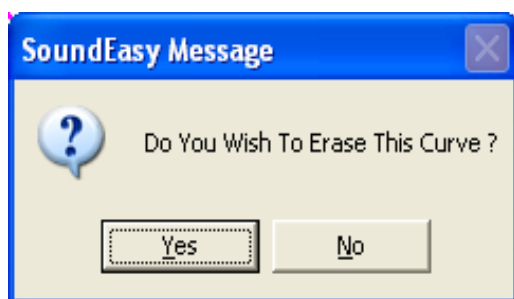


Figure 8.21 Example of 5 individual TABs plots on the Overlay TAB for our 3-way network.

### Curve Compare Function

The “Compare” function allows you to compare any curve from the 5 TAB’ed plots: SPL, Phase, Zin, Zin Phase and Group Delay on the “**Compare**” TAB for easy comparison the curves. You can select the desired curve by positioning the mouse pointer on the desired curve and pressing the left mouse button. The selected curve will be highlighted in pink.

When you release the button, the curve will be transferred to the “Compare” TAB. On the “Compare” TAB you will can also erase individual curves. Once again, position the mouse pointer over the desired curve and click left mouse button. The curve will be highlighted in pink, and a message dialog (shown below) will confirm, that you wish to delete this particular curve.



You can also **hide curves and add “Floating Labels”**. Press the right-mouse button on the plotting area, and a dialogue like the one above will pop-up. From there, you can select which curve you wish to view/hide. There are also a 25-character long labels for each curve. You can move the “Floating Labels” dialogue to the most convenient location. You can still invoke the “Vertical Scales” dialogue box by pressing right-mouse button on the non-active section of the “Floating Labels” dialogue box.

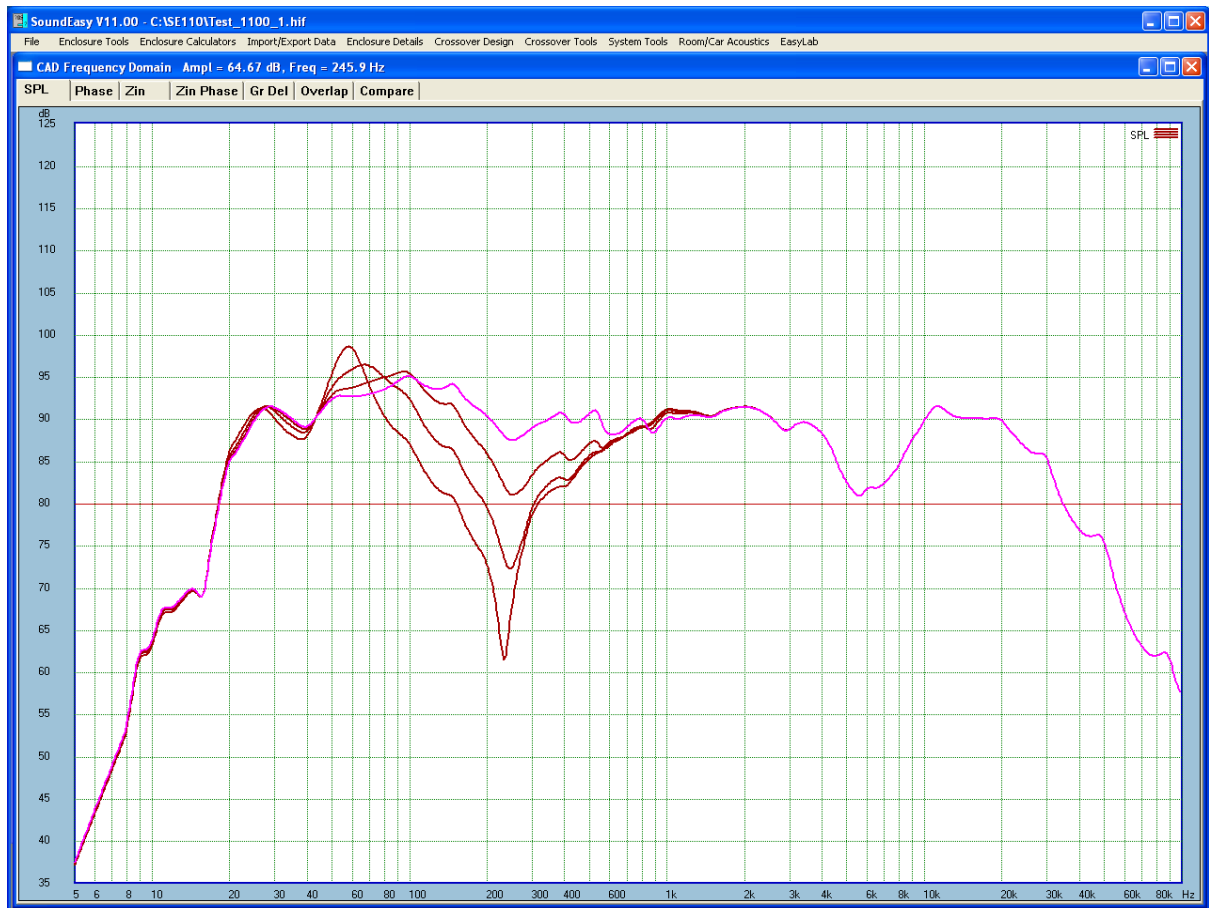


Figure 8.22 Example of selecting an SPL curve for transfer to “Compare” TAB.

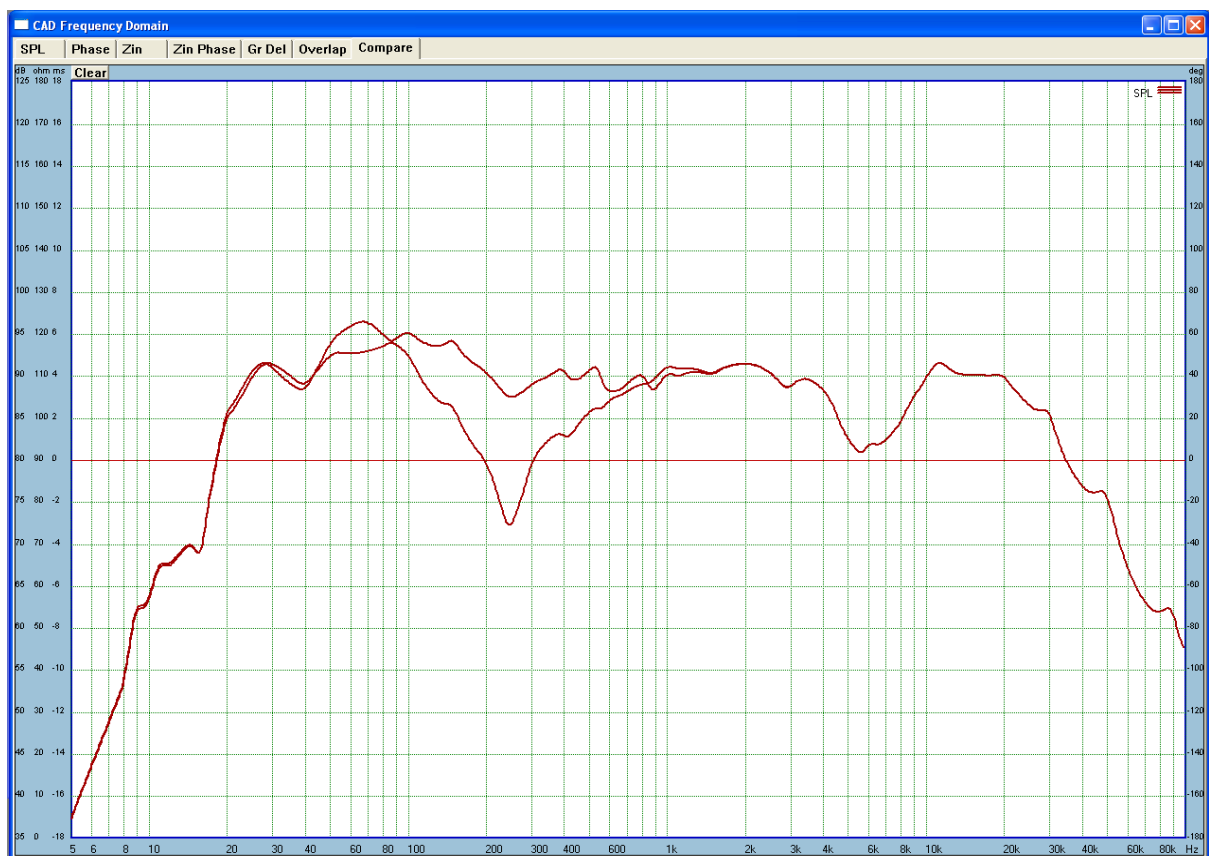


Fig 8.23 Example of “Compare” TAB with 2 SPL curves transferred form the SPL TAB.

The “Compare” TAB plotting window retains it’s plots until you exit the program. It’s primary function is to allow you to store curves, and compare it to other plots, as you progress through your “what-if” analysis. The “Clear” button simple clears the plots.

## Time Tab

### 1. FFT Method combo box –

**Phasing ?** check-box column – “In” phase indicates no phase reversal. “Out” of phase indicates phase reversed by 180 degrees.

### 2. Plot ? check-box column – “No” indicates do no plot this curve. “Yes” indicates inclusion of the curve in plots.

### 3. Sum FFT button – Plots summed response using FFT method

### 4. Nodal Method combo box –

**Phasing ?** check-box column – “In” phase indicates no phase reversal. “Out” of phase indicates phase reversed by 180 degrees.

### 5. Plot ? check-box column – “No” indicates do no plot this curve. “Yes” indicates inclusion of the curve in plots.

### 6. Sum Nodal button – Plots summed response using MNM method

### 7. FFT button - Plots summed response using FFT method. You **can include driver icon in the CAD** schematic when plotting time domain response using this method.

### 8. FFT Full button - Plots summed response using FFT method. You **must not include driver icon in the CAD** schematic when plotting time domain response using this method.

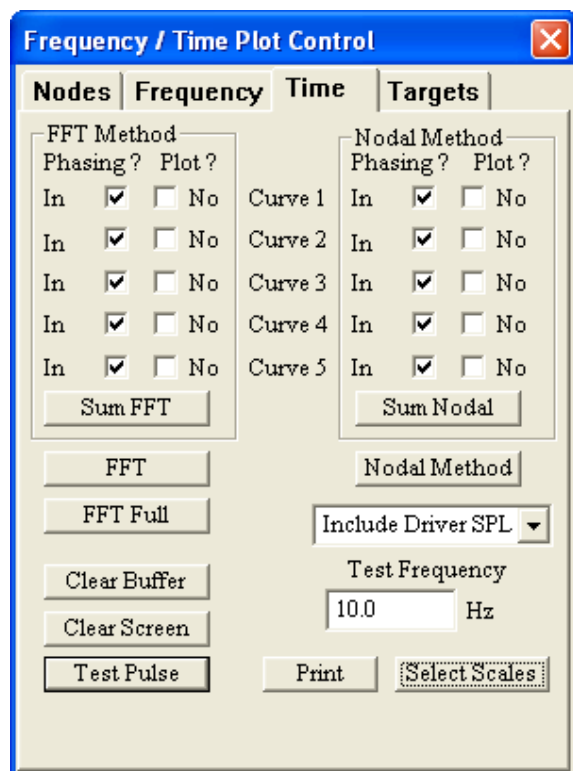
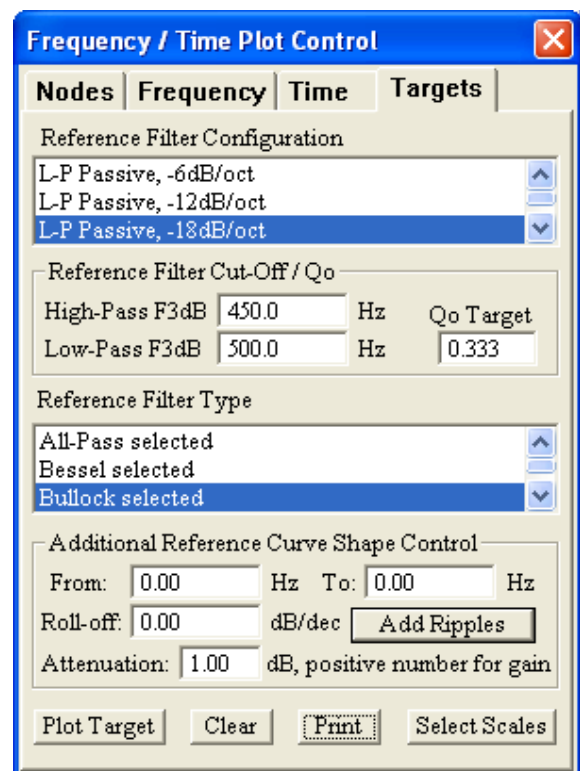


Fig 8.24. Tab for controlling Time-Domain plots



Tab for selecting target curves in CAD

### 11. Clear Buffer button – Clears memory storage allocated for 5 curves.

### 12. Clear Screen button – Simply clears the screen.

### 13. Test Pulse button – Plots the test pulse.

### 14. Test Frequency editable field – Enter your test frequency here.

### 15. Drop-down list box for selecting additional plotting parameters.

### 16. Print button – Invokes printer dialogue box.

### 17. Select Scales – Button opens dialogue box for selecting vertical scale resolution.

## Targets Tab

Allows you to plot target SPL/Phase curves in the frequency domain CAD plotting window.

1. **Reference Filter Configuration** – this list box allows you to select slope order and attenuation type.
2. **Reference Filter Type** – this list box allows you to select filter type.
3. **Reference Filter Cut-Off Frequencies/Qo** – three editable fields to accomplish just that. Enter frequencies in Hz.
4. **Add Ripples button** - Opens another box to enable you to add 3 ripples.
5. **Additional Reference Curve Shape Control** – group has four editable fields. The **“From”** field selects the frequency (in Hz) from which the response is to roll-off. . The **“To”** field selects the frequency (in Hz) at which the response is to flatten again The third field - **“Roll-off”**- is used to enter the required tilt in dB/decade. Usually, it is a small amount: 1-3dB/decade. The **“Attenuation”** parameter indicates, how much we would have to shift the 90dB reference line to match the system efficiency. For example: system efficiency is 86dB – enter **“Attenuation”** as **–4**.
6. **Plot Target button** - Activates plotting for all selected targets.
7. **Clear button** - Clears the plotting window.
8. **Print button** - Opens printer dialogue box.
9. **Select Scales button** - Opens dialogue box for selecting vertical scale resolution.

## What Is Compensation?

Measurement of the input impedance of a typical moving-coil, direct radiator loudspeaker, reveals that there are two main factors which affect the results: (1) the inductance of the voice coil and (2) the free air resonance of the vibrating system.

Voice coil inductance will result in a rising input impedance as the frequency of the signal is increased. At higher frequencies, the input impedance levels up because of the Eddy current shunt resistance across the voice coil inductor. At the lower end of the frequency range, a pronounced peak occurs. This is the free air resonance of the driver's vibrating system. For a typical driver, the DC resistance of the voice coil may vary from 3.5 - 14 ohms, but the factors mentioned can cause the modulus of the input impedance to reach 100-150 ohms. The input impedance of the driver is a complex function of frequency.

Whatever crossover network is designed, it is always assumed, that the crossover is loaded by a single resistor and this assumption is valid across the whole operating range of the filter. Now, the problem becomes more clear. In order to achieve the desired frequency response of the filter, a small and well designed passive network is inserted between the driver and crossover filter. The total impedance now presented to the crossover filter, consisting of the driver input impedance plus the compensation network impedance, is far less reactive. It might well be possible to cancel the reactive parts of the driver input impedance to almost negligible values. When the impedance of such a combination is examined, it should be an almost flat line. This is the designers ultimate goal when designing an impedance compensation network.

The second type of compensation is employed to improve the frequency response of the driver. Some small humps in the frequency response can be smoothed out or the excessive efficiency of the midrange or tweeter driver reduced to match the woofer efficiency. Once again, a small and properly designed network is inserted between the driver and crossover filter, but this time the amplitude of the driver and the compensation network is of primary interest. The third type of compensation involves time delay. In order to facilitate mounting of all drivers on a flat front panel, their acoustic centres need to be aligned. Usually, the higher frequency drivers are delayed to bring their acoustic centres in line with the woofer. SoundEasy offers a simple-to-use tool to design and evaluate many types of frequency/time compensation networks.

## Using The Calculators

The 'Crossover' tool has four handy built-in calculators for quick evaluation of four simple elements of the compensation networks shown in Fig 8.25 - 8.28.

### 1. L-Pad Calculator ( see Fig 8.25)

In order to reduce excessive sensitivity (efficiency) of the driver you need to insert a properly designed attenuator, consisting of two resistors. The L-Pad attenuator is best inserted between the driver (or driver and compensation network) and the crossover. The resistors must be able to dissipate the required power levels. To calculate the L-pad resistors, select "Calculator" option from the main menu and then highlight the "L-Pad" option. A small window will be opened with two editable data entry fields. The user is required to provide the drivers DC resistance for  $R_e$  and the required attenuation for Att then to click on the "Calculate" button. The resulting values for the parallel resistor  $R_p$  and series resistor  $R_s$  will be displayed below. The input resistance of the L-Pad network will be the same as  $R_e$ .

### 2. Zobel Network ( see Fig 8.26)

Zobel Network is used to cancel the effects of the voice coil inductance. In order to calculate the RC Zobel Network, select the "Calculator" option from the main menu and then highlight the "Zobel Network" option. A small window will be opened with three editable data entry fields. The user is required to provide all parameters and then click on the "Calculate" button. The resulting values for Zobel resistor  $R_z$  and Zobel capacitor  $C_z$  will be displayed below. Driver's amplitude response and total input impedance are plotted below. Please note the flat input impedance curve.

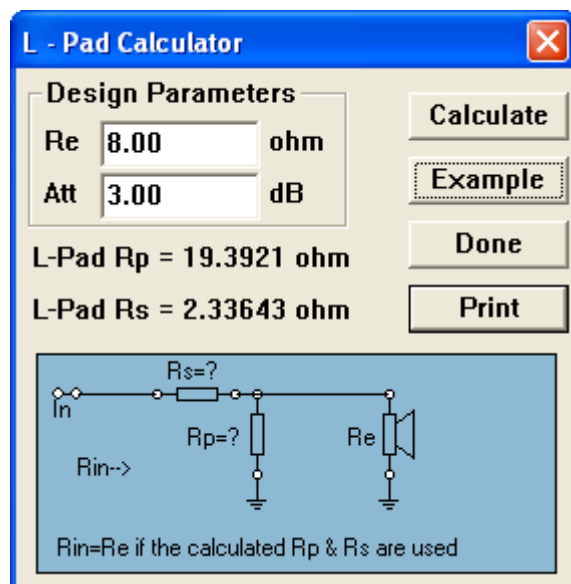


Figure 8.25 L-pad network.

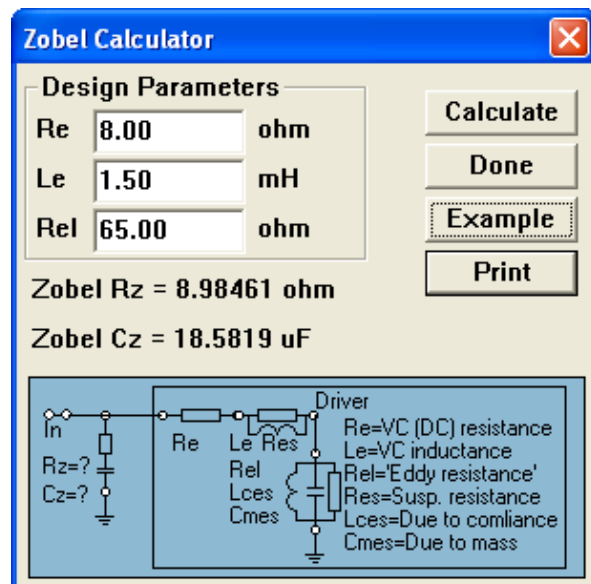


Figure 8.26. Zobel network.

### 3. Impedance Peak Equaliser ( see Fig 8.27 )

In order to calculate the components for the resonance circuit used to compensate for the impedance peak, select the "Calculator" option from the main menu and then highlight the "Parallel Notch" option. A small window will be opened with seven editable data entry fields. The user is required to provide all the data and then to click on the "Calculate" button. The resulting values for series components will be displayed below. The "before" and "after" impedance plots are shown below.

### 4. Amplitude Peak Equaliser ( see Fig 8.28)

In order to calculate the components for the parallel resonance circuit used to compensate for the amplitude peak, select the "Calculator" option from the main menu and then highlight the "Bump Compensation" option. A small window will be opened with five editable data entry fields. The user is required to provide driver's DC resistance for  $R_e$ , the amount of attenuation required in the centre of the hump for  $A_{0,h}$ , lower for  $F_l$  and higher for  $F_h$  frequencies at which the hump rises by  $A_{l,h}$  decibels. You would typically enter  $F_l$  and  $F_h$  as the frequencies at which  $A_{l,h}$  equals 1db and then click on "Calculate" button. The resulting values for parallel components will be displayed below.

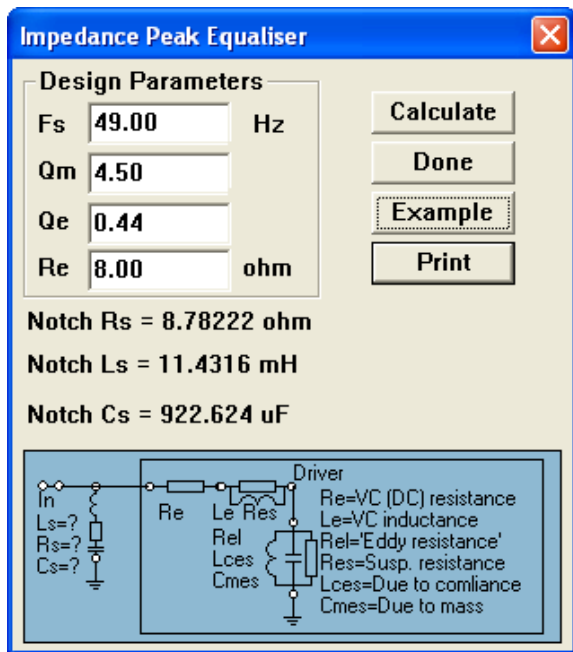


Figure 8.27. Impedance Peak Equalizer

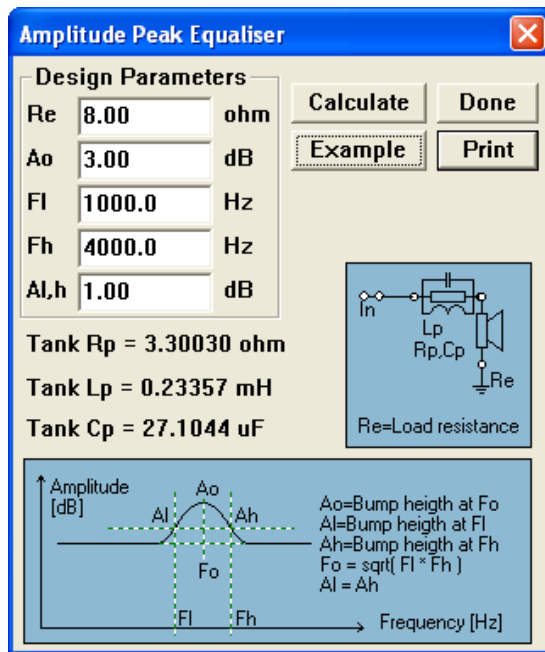


Figure 8.28 Amplitude Peak Equalizer.

## Lattice Networks

Compensation networks considered in this section have constant amplitude response and variable phase and group delay responses. They are commonly termed "all-pass" filters or Lattice Filters. One method of passively realizing all-pass function uses symmetrical lattices. A very useful representative of this family of filters is shown on Fig 8.29. It depicts a general topology of the symmetrical lattice filter. Please note, that both load terminals are floating - they are not connected to the ground terminal. A practical realization of the first order lattice network is given on Fig 8.30 and second order lattice network is shown on Fig 8.31.

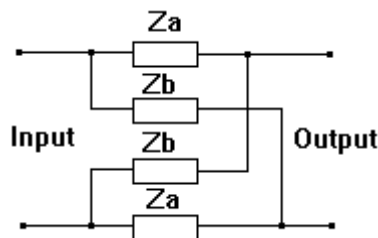


Fig 8.29 Lattice Network (Time Delay Filter).

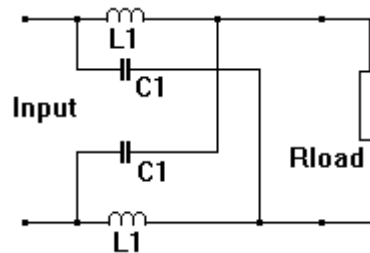


Fig 8.30 First order Lattice Network.

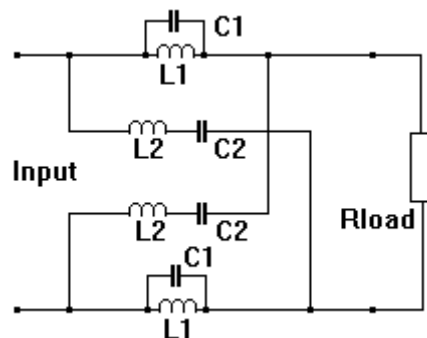


Fig 8.31 Second order Lattice Network

The larger number of elements in the symmetrical lattice arises because the network is not only symmetric with respect to the ports, but also balanced with respect to ground. For satisfactory performance, the impedance in two series arms must track at all frequencies, as must those in the two cross arms. These properties call for low-tolerance components, making the lattice network expensive to build and tune. We are mostly interested in the following characteristics of the filter:

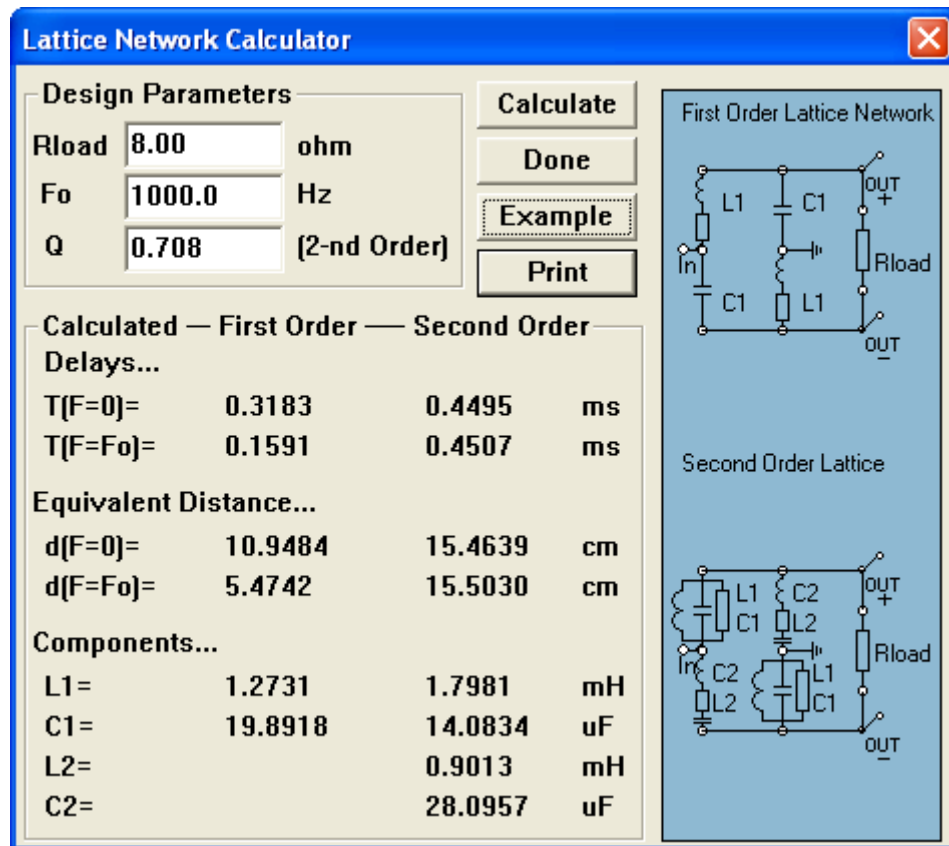


Figure 8.32. Lattice network calculator.

1. Ability to delay a signal in time without introducing frequency distortion.
2. Ability to reflect load resistance to the input terminals, the "constant resistance" property. This property will greatly assist in cascading the networks.

For the first order lattice, the above criteria will be met when:  $Z_a = -Z_b$  and  $|Z_a| = |Z_b| = R_{load}$

SoundEasy offers a built-in Lattice Network Calculator (LNC) - see Fig 8.32. This simple LNC allows the user to estimate attainable time delay offered by the first and second order lattices at a desired frequency ( $F_o$ ), for a given load resistance ( $R_{load}$ ). The LNC also calculates the L and C components of the first and second order lattice networks, such as the one shown on Fig 8.30 and Fig 8.31. The resulting components will be given for a constant resistance lattice. Larger delays can be obtained by cascading first or second order networks, so that their individual time delays can be summed. Time delay introduced by the equalizer (Lattice Network) will be added to the existing (if any) delay in the circuit. The first order all-pass network introduces maximum delay at  $F_o = 0.0$  Hz and the delay rolls-off with frequency. For the second order network, there are two parameters, that can be manipulated by the designer in order to obtain the required total delay: (1) Q-factor and (2)  $F_o$ . For Q-factors less than 0.6, the delay, introduced by the second order all-pass network tapers off with frequency. For Q-factors greater than 1.0, the second order lattice network allows the designer to create concentration of delay around a frequency, at which there is a 180 deg phase shift ( $F_o$ ). For Q-factors greater than 2, the delay exhibits a pronounced peak around the  $F_o$ , with slopes of -12dB/oct. The LNC is activated from the main menu by selecting (1) "Crossover Tools" and then (2) "Lattice Network" options. The first order lattice network requires load resistance ( $R_{load}$ ) and "cut-off" frequency to be entered on the calculator's data entry fields. The second order network requires additional Q-factor to be keyed-in.

**Caution !.** The LNC assumes that load resistance of the network ( $R_{load}$ ) is the DC resistance of the driver ( $R_e$ ). Remember, that lattice network should only introduce the required time delay, and reflect the load resistance to its input. Modifying  $R_{load}$  AND storing the data file will cause the value of  $R_e$  to be changed to  $R_{load}$ .

The total area under the delay curve is constant for lattice networks. As a consequence of this, higher Q-factors would cause concentration of the delay around the  $F_o$  at the expense of the delay at other frequencies.



## Additional Operations.

The introduction to the 'Crossover' tool was based on the woofer driver. The midrange and tweeter drivers have their own dedicated menus, but the operation of the menus is exactly the same as for the woofer driver. For detailed description on loading, saving and printing the user should consult Chapter 2. It is often beneficial to determine how much the "non-ideal" load disrupts the transfer functions of the designed network. It is easier to make this observation without the driver's acoustic response - in the "Exclude Driver" mode. The "Include Driver" and "Exclude Driver" options in the list box combine or exclude the previously edited driver's transfer function with the compensation network. The Impedance curve will not be modified by this option. The same option is also implemented in the 'Crossover' tool.

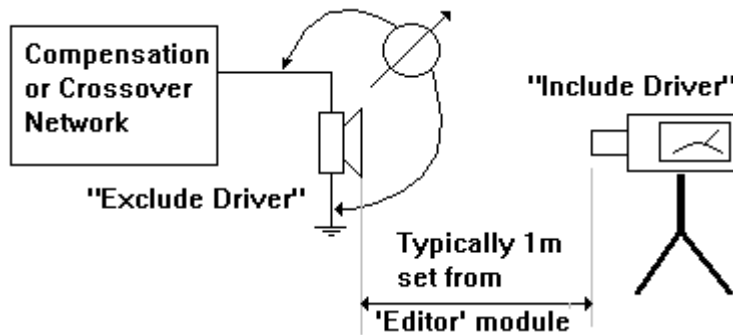


Fig 8.33 "Include Driver" and "Exclude Driver" options.

## "Crossover Tools" Menu Options

There are additional menu options available at the bottom of the "Crossover Tools" main menu bar. The bottom three are system-wide options and are active, while other (non-crossover CAD) screens are opened. You may for instance require to change a component value while working in system optimizer.

1. **"Use E-12 Series"** - this menu option allows you to approximate component values to the nearest E-12 series value.
2. **"Use E-24 Series"** - this menu option allows you to approximate component values to the nearest E-24 series value. These values are more accurate than E-12 series.

## "Preview" built-in circuit window

The program has a large number of built-in filter and crossover circuits to choose from. All these circuits can be loaded to the clean CAD screen with pre-calculated values. They can also be added to the existing circuit by using "Add" function. This way, crossover development is quite fast and efficient. All in-built circuits identify themselves to the user by their names, such as **"BP Active +/-18dB/oct"** circuit. This can be sometimes rather cryptic, so the circuit **"Preview"** window will come very handy in a situation, where you are uncertain, as to what the circuit name actually hides. Filter and crossover types and configurations are listed in two list boxes. Combination of user selections from both list boxes is needed for final selection of the circuit.

The **"Preview"** window is activated automatically when you open the built-in filter or crossover dialogue boxes and select any of the circuits from the provided list boxes. The **"Preview"** window will close automatically when you close the filter or crossover dialogue box. The built-in schematics are displayed automatically when you select any item from the list box, OR you can use cursor keys UP/DOWN to scroll through the schematics.

However, you may want to position and re-size the "Preview" window on the screen, so it will display full view of all built-in circuits. You will be able to keep the "Preview" window still opened, when you close the filter or crossover dialogue box using the "X" gadget in the top-right corner of the dialogue box. Now, the "Preview" window is active and can be adjusted as required.



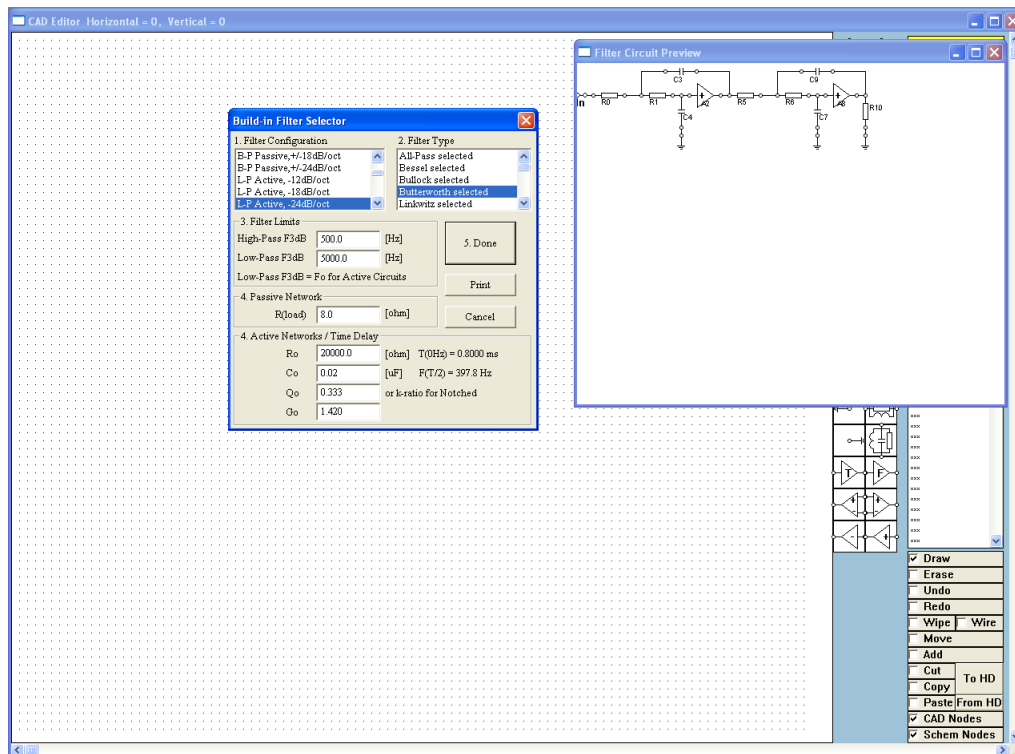


Figure 8.34 The “Preview“ window.

### Fast CAD Component Tweaking

In order to facilitate a shortcut to component value entry, we have implemented “Adjust” function. The function is accessed from a list box located on the “**Frequency/Time Domain Plot Control**” Tab – see Figure 8.35. The dialogue box allows you to manually adjust the value of the selected component **within 1% - 1000%** of its original value. All five curves: (1) Amplitude, (2) Phase, (3) Impedance, (4) Impedance Phase and (5) Group delay curves can be nominated to be displayed while the selected component is being tweaked. Finally, a “Clear” function is also provided to clear congested screen.

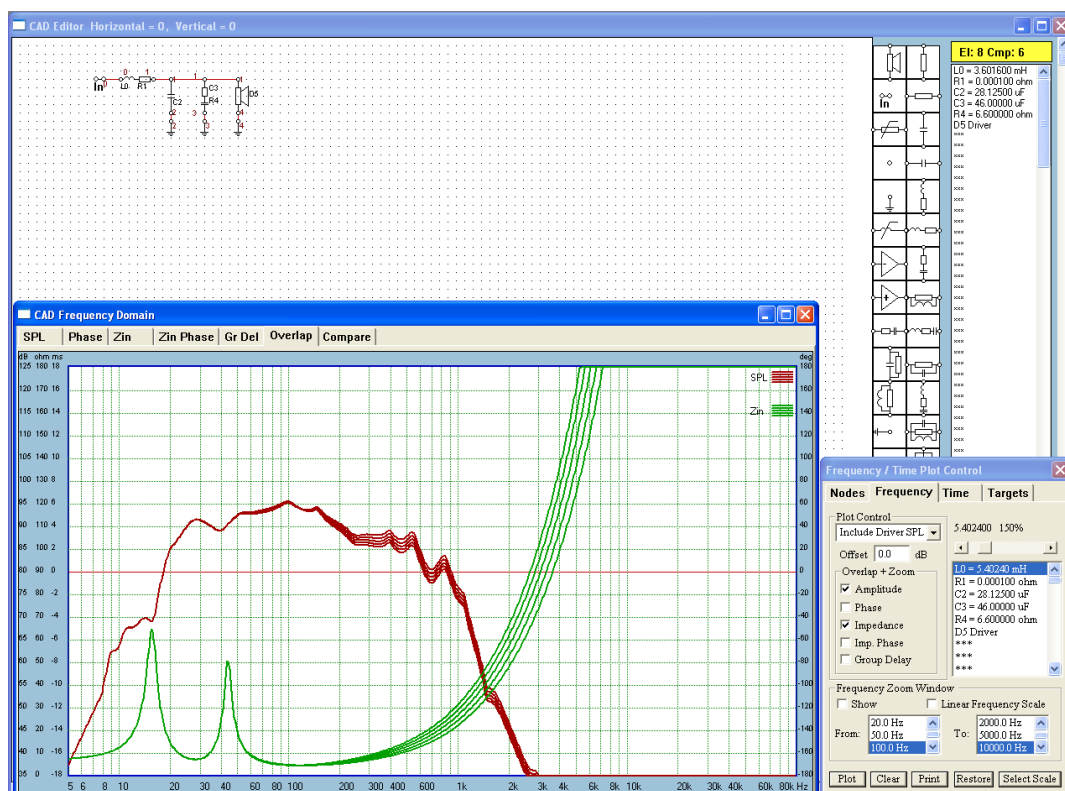


Figure 8.35. Example of “Tweaking” components using “Adjust” function.

## Frequency Domain ZOOM

Frequency Domain Zoom window allows you to zoom-in the frequency domain into the selected frequency range for detailed view of the plots in Crossover CAD and System plots. The Zoom window will be automatically opened if the “Show” box is “checked” – see Figure 8.36.

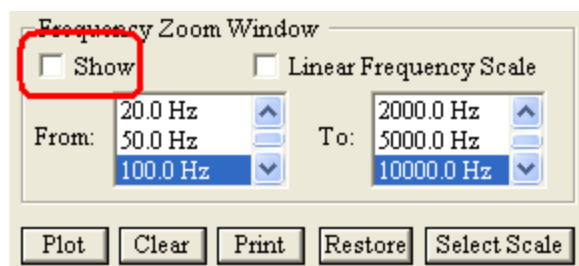


Figure 8.36 Zoom Window controls

The Zoom window has a separate settings for high/low ends of the frequency range of it's screen. These settings are fully independent of the main frequency domain screen in the CAD and System sections of the program. Vertical scale of the zoom window is linked to the main frequency domain window of the crossover section.

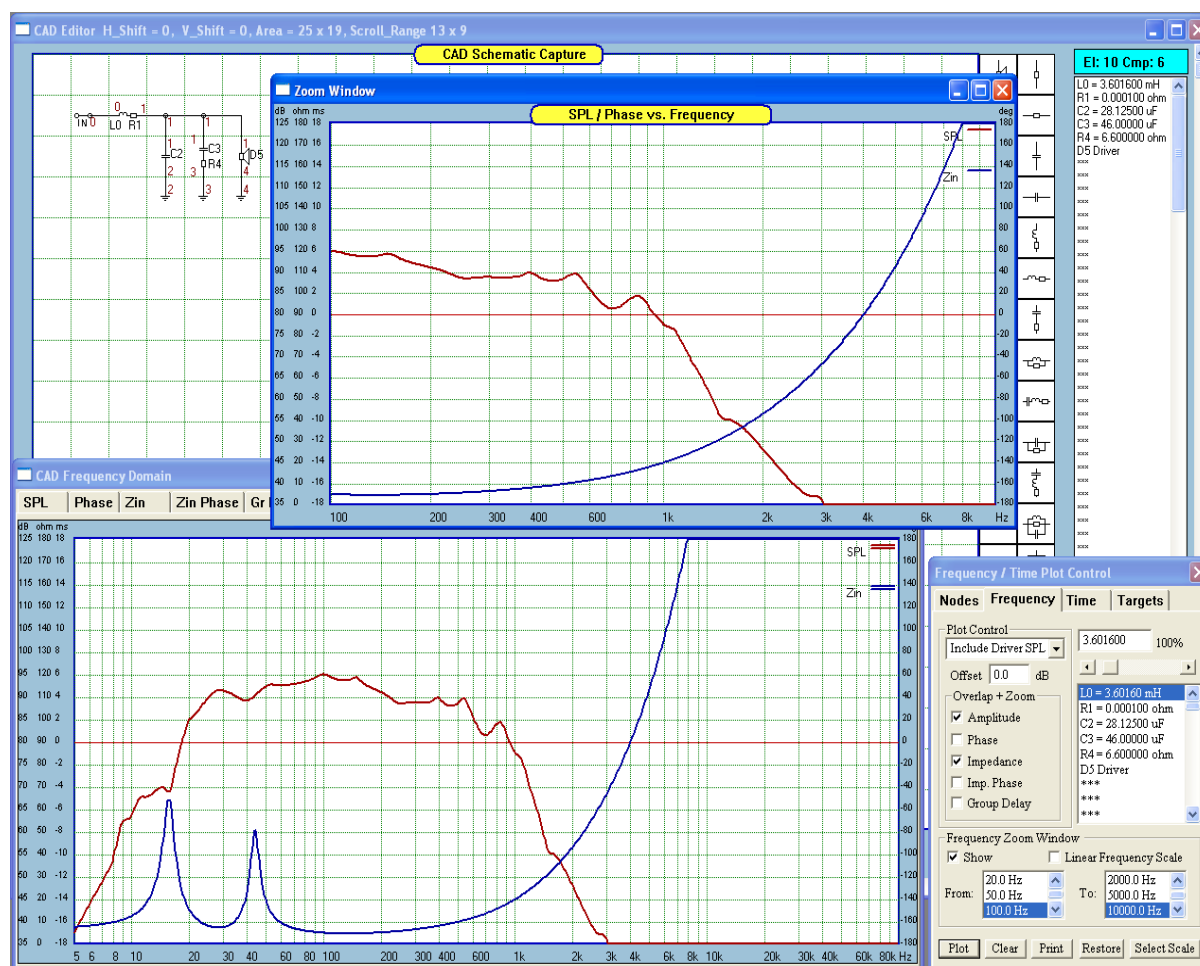


Figure 8.37 Zoom window used to focus on 100-10000Hz frequency range.

There are following practical issues are observable on the zoom window.

1. If you analyze a circuit, that has no driver-dependant data included in the analysis – like SPL or Zin from the driver file, than the circuit transfer functions are calculated directly for every frequency point of the screen. This is an excellent way of increasing screen resolution – a true zoom-in process.
2. If you analyze a circuit, that includes driver-dependant data in the analysis, then this data will be mapped onto the frequency scale of the zoom window.

## Linear Frequency Scale

Another very useful feature of the ZOOM window is the ability to switch to **linear frequency scale**. Frequently, you may need to determine if the circuit under analysis is actually a “liner-phase” type or how linear the phase really is. When operating the zoom window in linear frequency scale mode, this particular property of the circuit is clearly visible.

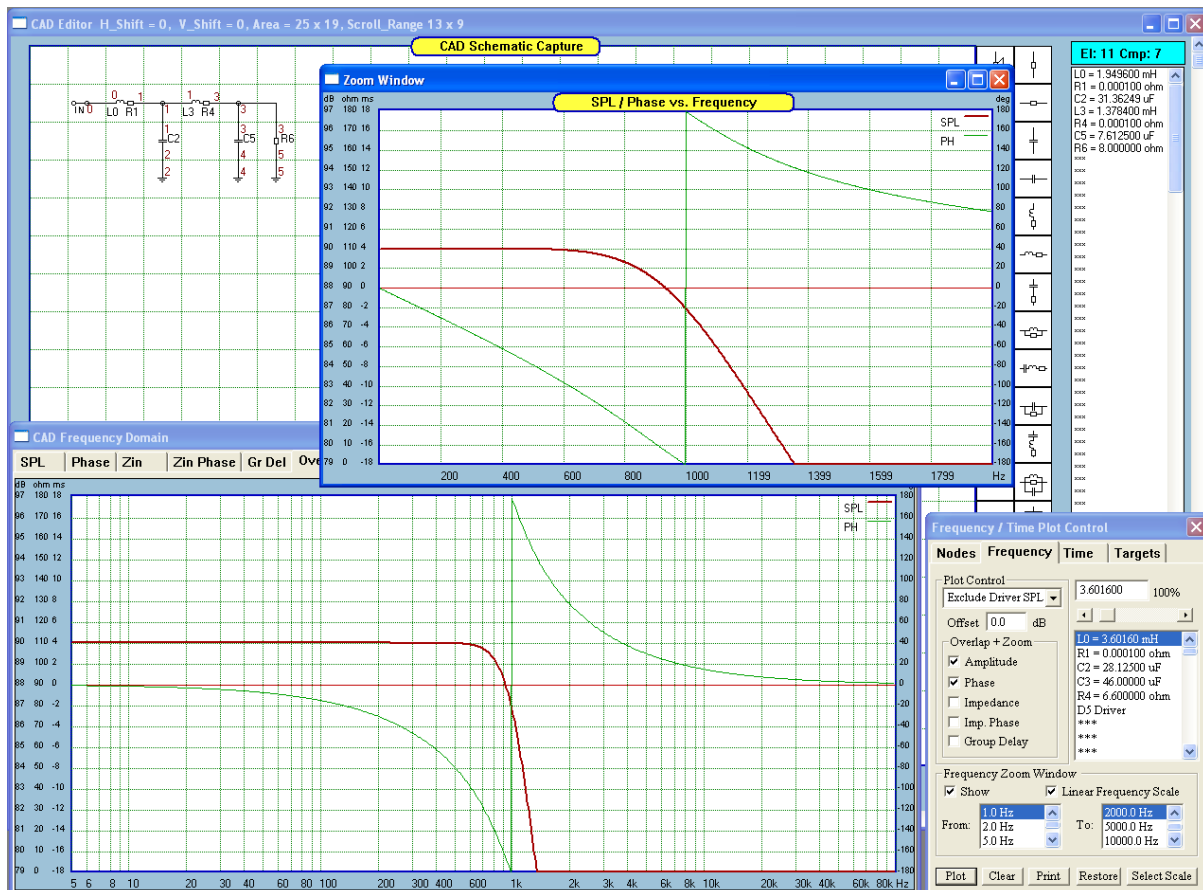


Figure 8.38. A 4-th order LP Butterworth filter shown in main window and ZOOM window with Linear frequency scale 1Hz – 2000Hz. Please note phase **non-linearity** between 400-2000Hz.

A filter will have linear phase if its impulse response is symmetric about its midpoint. As a result, systems with a linear phase response delay each frequency component of the input by the same amount, while nonlinear phase systems delay each frequency by a different amount, leading to a form of distortion known as phase distortion.

A good example of passive, linear phase filters are Bessel filters. These are characterized by good phase response within the pass-band – see figure below. The advantage of using linear frequency scale to evaluate phase response is highlighted on Figure 8.39, where the bottom (standard) window displays 4-th order LP Butterworth and Bessel filters using logarithmic frequency scale. Phase response of both filters is different, but one could not tell from this picture, which phase response is linear. The zoom window set to linear frequency scale makes it very easy.

Please also note the excellent zoom on the SPL and phase curves within 100Hz – 2000Hz shown on Figure 8.38 and Figure 8.39, while the standard screen runs in 5Hz – 100000Hz mode.

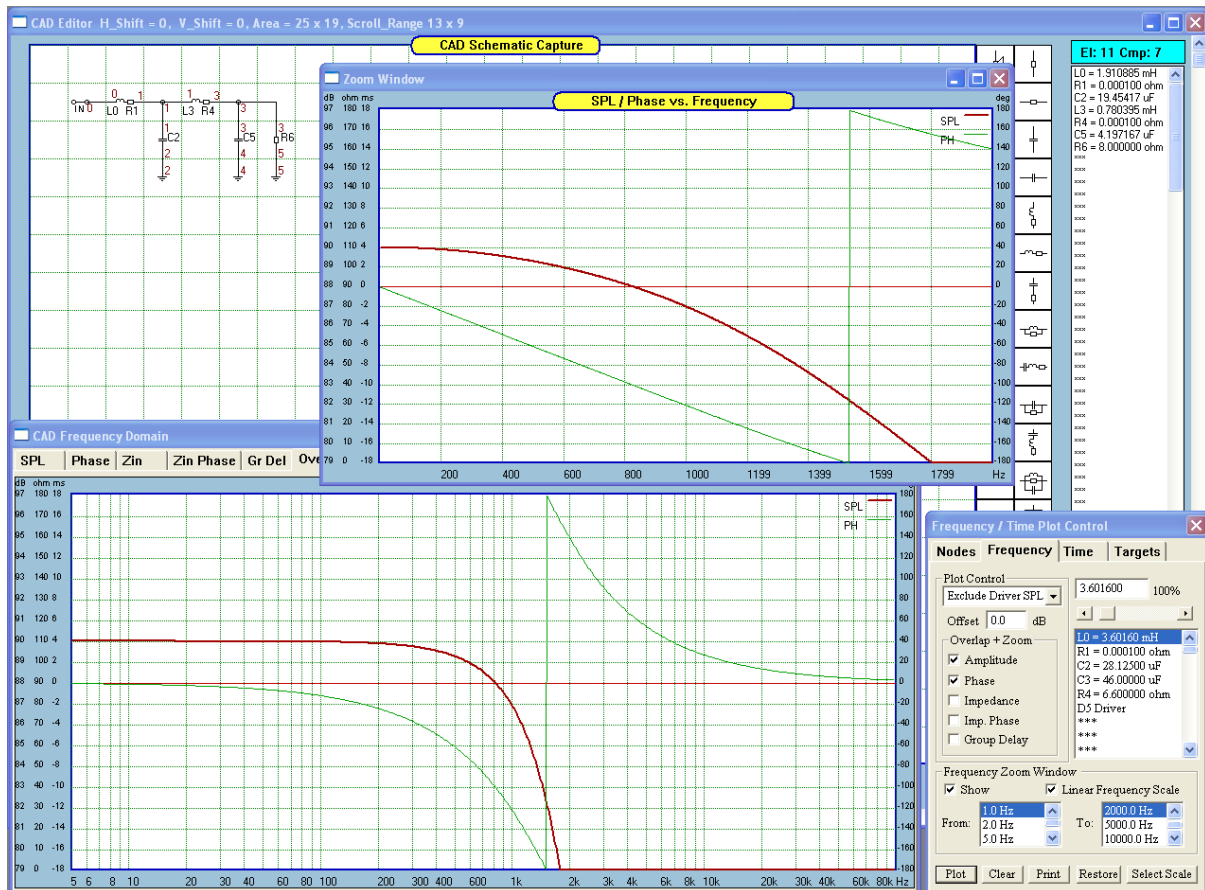


Figure 8.39 4-th order LP Bessel filter shown in main window and ZOOM window with Linear frequency scale 1Hz – 2000Hz. Please note phase **linearity** between 400-2000Hz.

The zoom window is highly automated. All requested curves will be re-plotted automatically when a new frequency range is selected. If the window is accidentally covered by another window, you can click on “Show” button twice to close and the re-open this window on the top of your screen.

## Multiplot

When plotting SPL of a multi-way systems, you may want to obtain SPL plots for ALL drivers in the system and finally the total system SPL. Traditionally, all this could be accomplished by switching drivers ON/OFF from the Nodes TAB, and replotting the SPL curves for each configuration scenario.

For time saving purposes, you may find the “Multiplot” handy and convenient. For this feature to work correctly, you will need to edit correctly the Nodes TAB.

1. Make sure, all drivers have their correct “Type” assigned.
2. Turn ALL drivers in the system ON.

The Multiplot function can be activated from the Preferences screen.

An example of properly set Node TAB is shown on the Figure below.

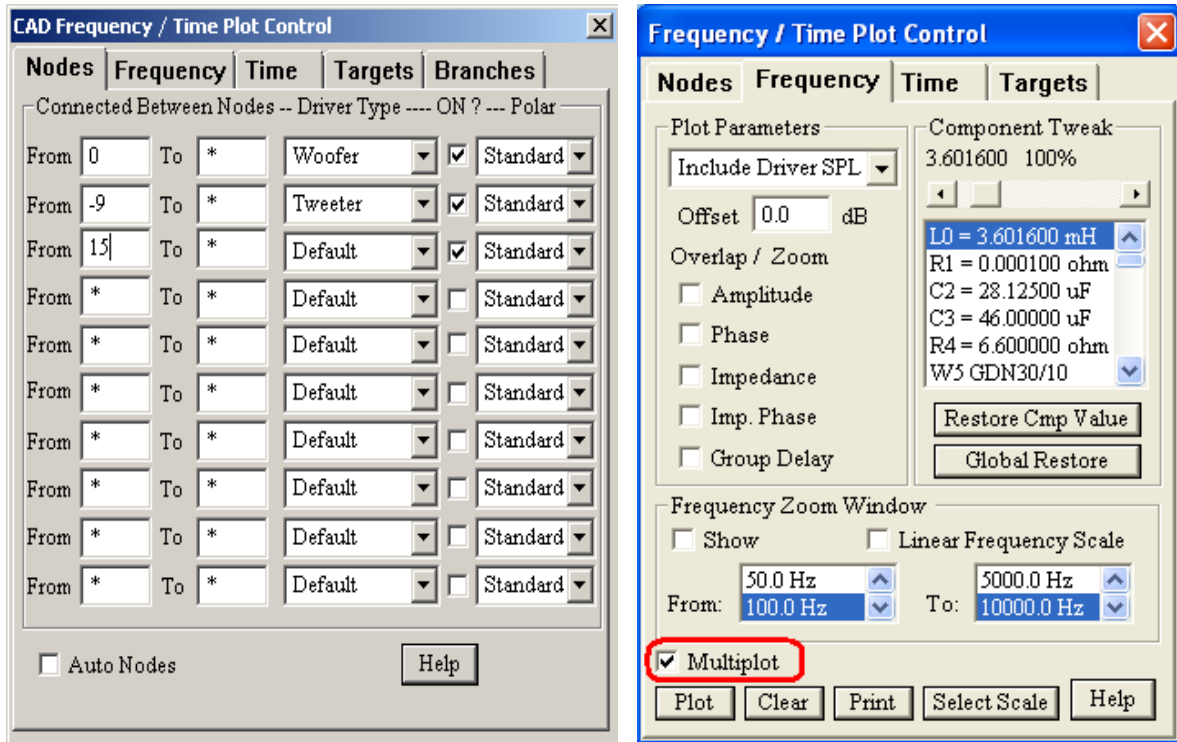


Figure 8.40. Editing Nodes TAB for Multiplot function

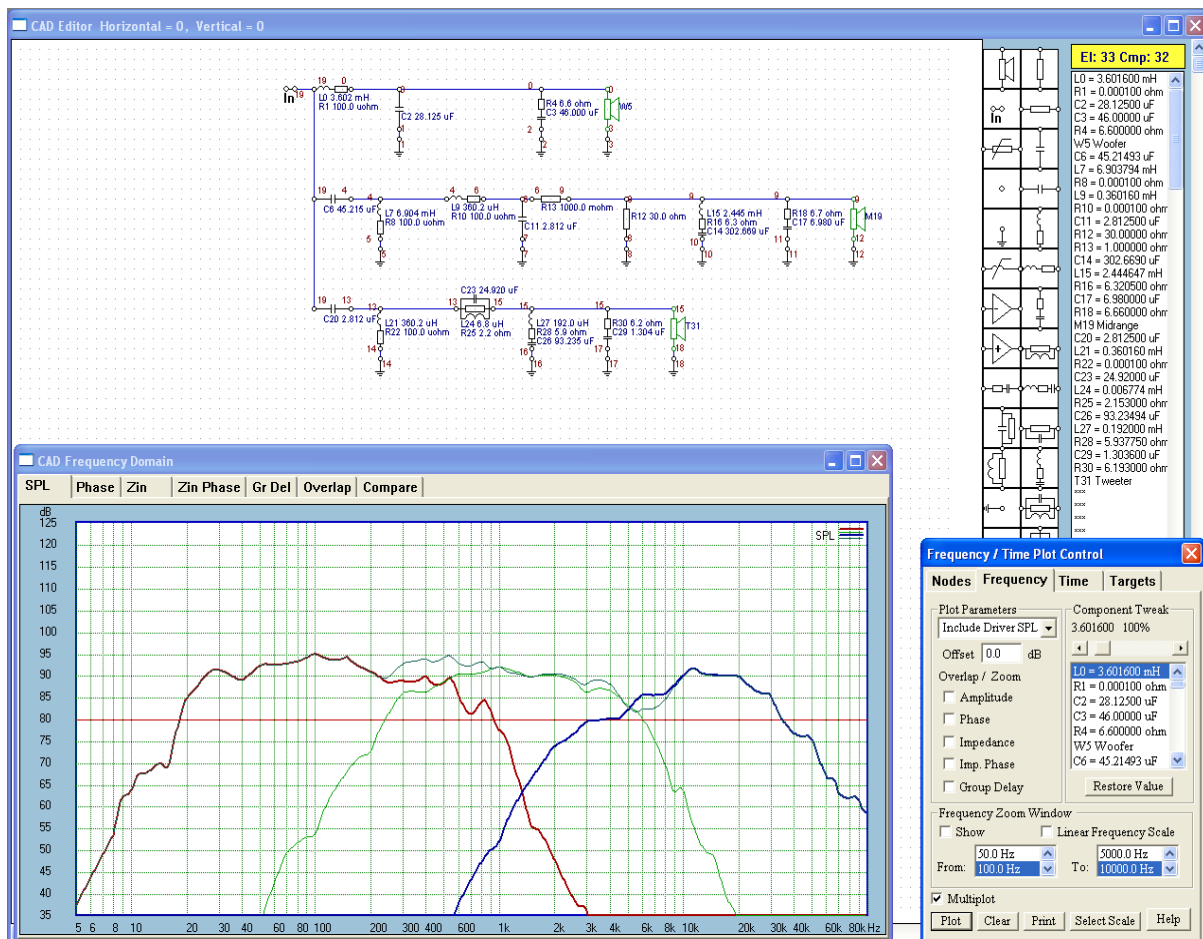


Figure 8.41. Example of Multiplot Mode – Please note different colours of the SPL curves.

## Save / Load Circuit

There is also an option of for Saving and Loading complete circuits. This feature will come handy when you create a library of your EQ circuits.

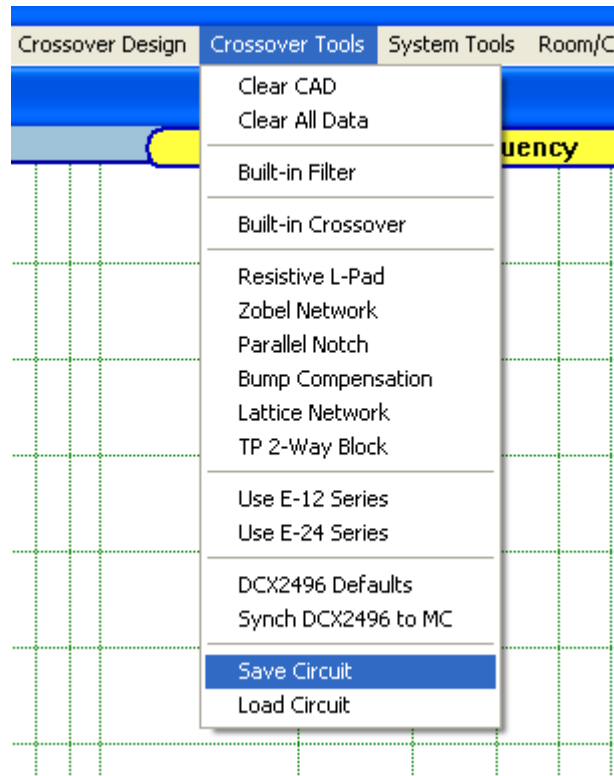


Figure 8.41. Selecting Save / Load circuit option.

## Tweaking Components – Four Methods

SoundEasy incorporates several options for tweaking components. Typically, tweaking involves making very small, incremental (step-by-step) changes in the component values and observe the effects of this change on the SPL plot. Practically, you would use the mouse, tap on the cursor keys, or use mouse wheel to scroll incrementally and see the SPL change immediately

### 1. List box + slider

This tweaking method has been designed to have all necessary controls located on a small area of the control TAB. You can select the component, tweak it using coarse, fine and rapid steps, and finally restore to the original value. All this is done using only mouse, and you do not have to take your eyes off the screen.

The method does not interfere with any screens and is consistent throughout the program. This method includes “auto-scroll” of the list box, so that a components is scrolled into list box’s view by clicking left mouse button on the desired component on the schematic.

SPL plotting screen may get behind CAD screen temporarily. The plotting screen gives you the essential feedback during tweaking, and it is absolutely necessary to see the screen during tweaking. However, by the same token, it is not necessary to see the screen during component selection process. Therefore, there should be no problem if the plotting screen temporarily hides behind the CAD schematic screen for a short moment. This way, the process is automated, and the user does not have to do anything at all to access components (even those potentially hidden behind plotting screen) on the CAD screen. The SPL plotting screen will be at the front, the moment you start tweaking.



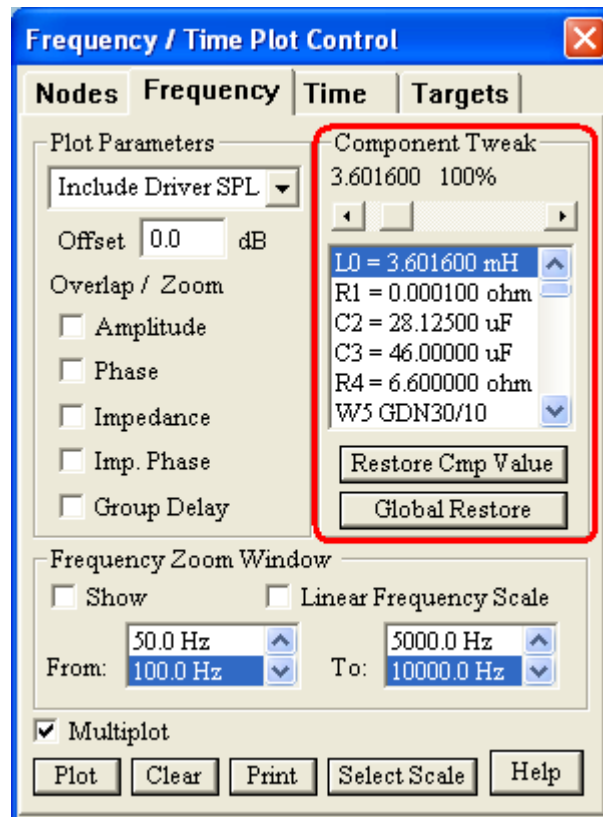


Figure 8.43. SPL plotting TAB with component tweaking controls.

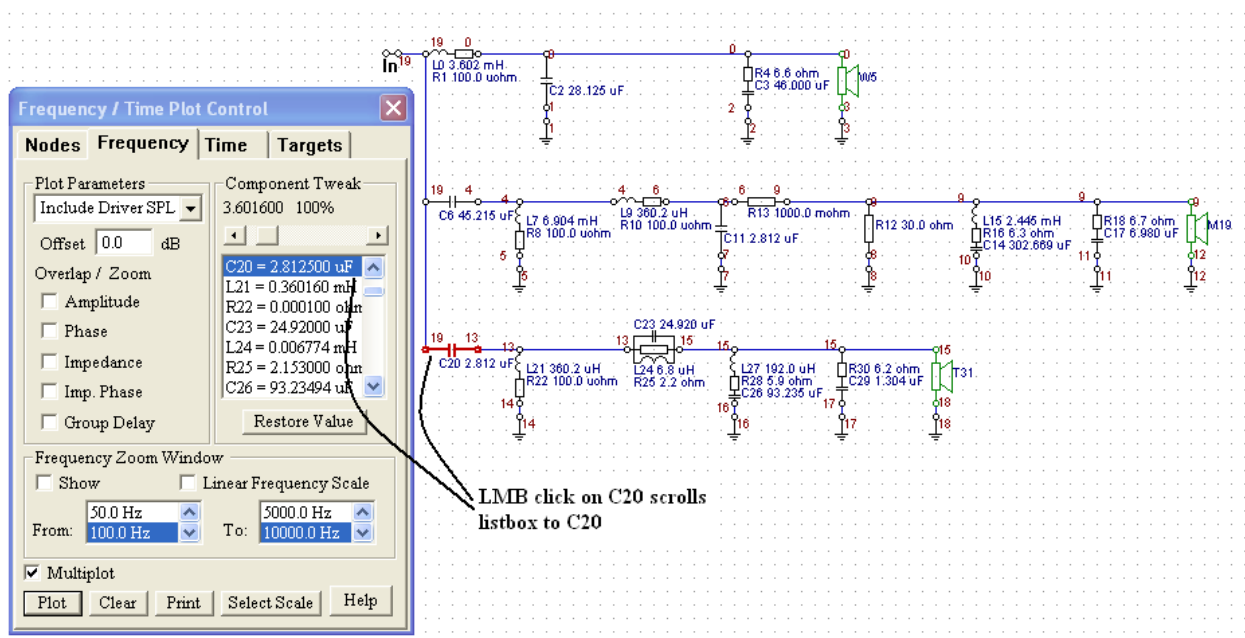


Figure 8.44. LMB clicked on a component C20, scrolls list box with C20 on the top.

In this method, you select components by double-clicking on the list, and then use slider to adjust component value. The slider offers you 3 options: (1) fine adjustment steps, (2) coarse adjustment steps, (3) rapid change to 0% or 1000%, or anywhere in between. This is a versatile adjustment element specifically designed for Windows.

New component values are automatically displayed in the list box and the display line above the list box.

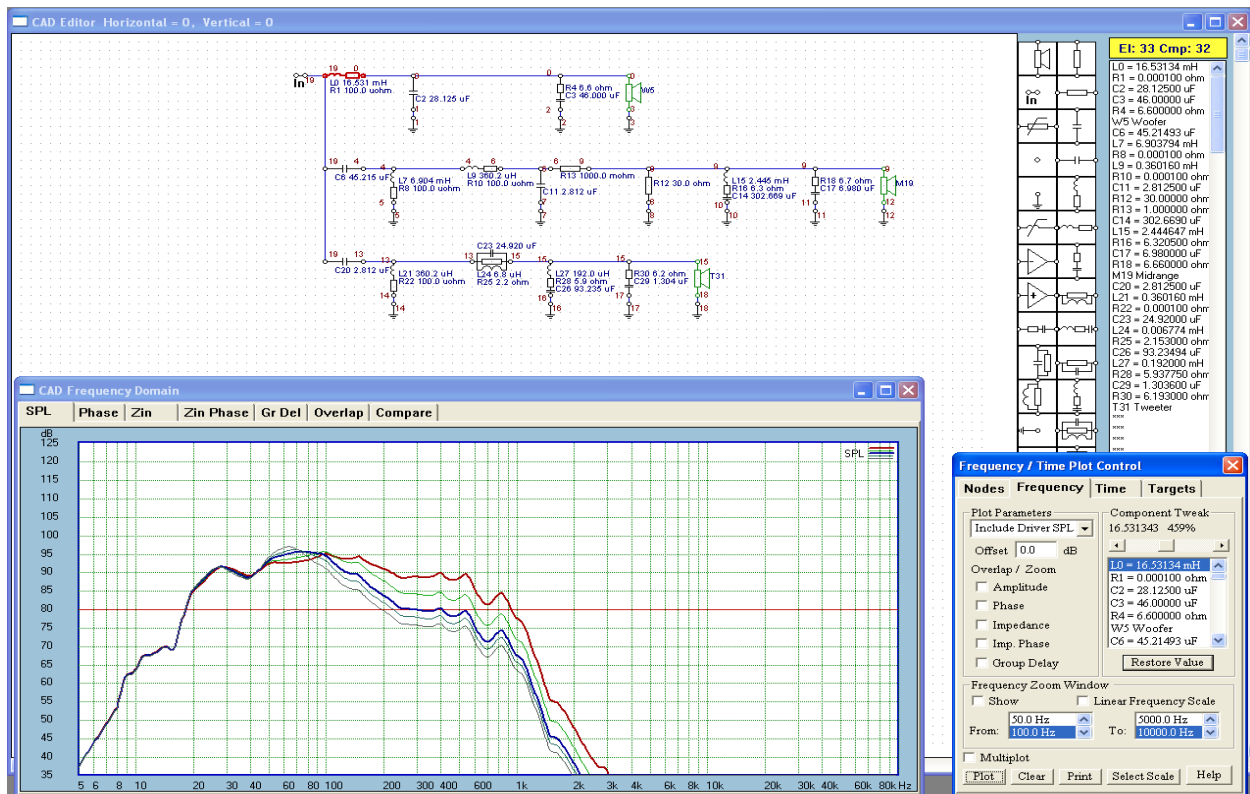


Figure 8.45. Example of tweaking L0.

## 2. List box + cursor keys

This tweaking method is similar to the one described above, and component selection is accomplished in exactly the same way. Once the component(s) is scrolled into view, you need to click the left mouse button on the desired component in the list box. This will activate the list box and the selected component. From this point, you can use arrow keys as follows:

UP arrow key – selects the next component above, in the list box

DOWN arrow key – selects the next component below, in the list box

LEFT arrow key – reduces the component value. Pressing CTRL key, will reduce step by 10x.

RIGHT arrow key – increases the component value. Pressing CTRL key, will reduce step by 10x.

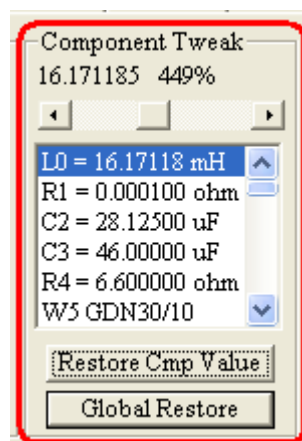


Figure 8.46. After selecting L0, use keyboard cursor keys to adjust component.



New component values are automatically displayed in the list box and the display line above the list box.

### 3. List box + cursor keys + mouse wheel

This tweaking method is also similar to the one described above, and component selection is accomplished in exactly the same way. Once the component(s) is scrolled into view, you need to click the left mouse button on the desired component in the list box. This will activate the list box and the selected component. From this point, you can use arrow keys as follows:

UP arrow key – selects the next component above, in the list box

DOWN arrow key – selects the next component below, in the list box

Scroll mouse wheel to INCREASE/REDUCE component value. Pressing CTRL key, will reduce step by 10x.

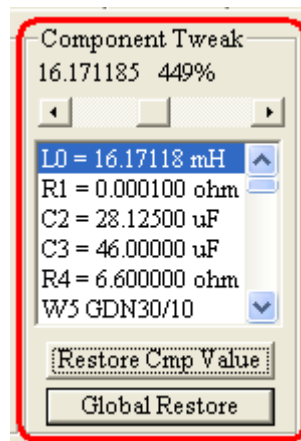


Figure 8.47. After selecting L0, use keyboard cursor keys & mouse scroll wheel to adjust component

New component values are automatically displayed in the list box and the display line above the list box.

### 4. Direct component selection from schematic

This option implements UP/DOWN cursor keys and mouse wheel to tweak, CTRL to adjust step, and TAB key to move between data fields.

Using this method, you select the component for tweaking directly from the CAD schematic. The component is selected the same way as for keying in/editing component value – by clicking right mouse button on this component.

When the dialogue box pops-up, the first data entry field should be selected (highlighted) already. You can use either UP/DOWN cursor keys to increase/decrease component value, or you can use mouse scroll wheel to adjust component value up or down. Pressing CTRL key, will reduce step by 10x.

This method does not interfere with any of the screens, and proves itself useful when you want to make quick, small adjustments on a couple of components. From practical experience, there will be larger tweaking operations, involving several components – then you would use Method 1-3. But there will be also situations, where you know, that components are wrong, and would just want to tweak one or two scattered component by small amount – then Option 4 with mouse scroll seems better suited.

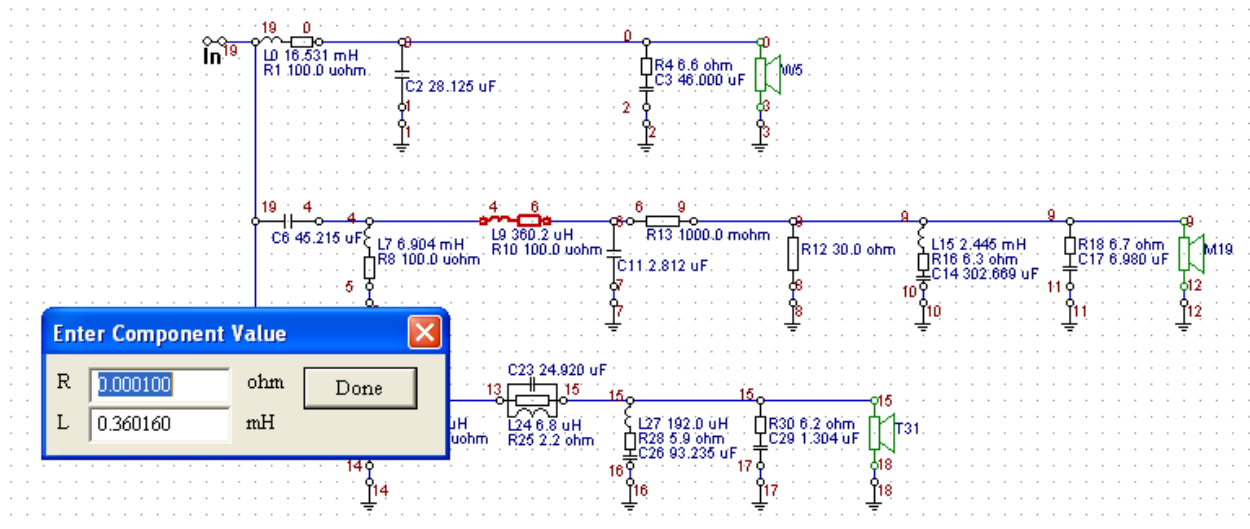
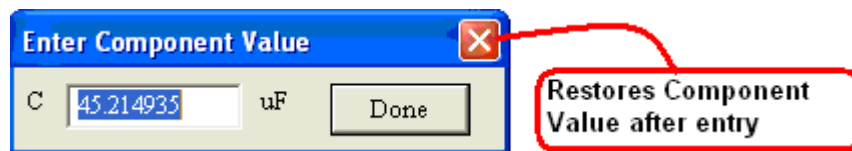


Figure 8.47. After clicking RMB on the component, use keyboard cursor keys OR mouse scroll wheel to adjust component

Finally, the “X” window button allows you to restore component value after you typed-in a wrong value. This function works for typed and scrolled values. Plotting screen is automatically refreshed upon exit.



### Grouping Passive CAD Components

In this chapter, we present a powerful, but rather advanced functionality of the program. It is suggested, that you would need at least an intermediate level of circuit design in order to fully exploit the power and flexibility of this function.

Consider a situation, where during a development of a 3-way crossover, a decision is made to change crossover frequency between midrange and tweeter. To make situation more dramatic, let's assume, that the crossover has already been equipped with other advanced features, like L-pads, impedance and amplitude compensation components. The dilemma can be resolved with some cut-and-paste exercise, or by starting from scratch, and developing a whole new complicated crossover – just to change one simple cut-off frequency.

In the scenario above, it is likely, that nothing else needs to change, except for some of the midrange and tweeter filter component values.

If the filters in your crossover have exactly the same topology as one of the built-in individual filters, you may be able to **re-calculate component values for the new cut-off frequencies, and drop the whole group of new component values directly into the crossover**. All built-in filters have input icon and load resistor icon included in their schematic. The input icon will be discarded during the substitution, but you can include the load resistor in your substitution, IF it was also included in your group of components, inside bounding rectangle. Due to the above requirement, it may be beneficial to develop crossovers with a bit of a forward thinking, as the Component Grouping function may one day come very handy and it will reduce otherwise necessary development efforts. There is however a cautionary note: SoundEasy contains over a hundred of built-in filters. There may be an odd filter, that has somewhat different topology, and will not match the filter in the crossover branch. This is rare, but please be aware of this possibility.

The whole process is shown below in a step-by-step fashion. Original midrange band-pass filter was designed to have the bandwidth of 500-5000Hz. The goal in the example below, is to move midrange band-pass filter's cut-off frequency from 500Hz down to 300Hz.

### Step 1.

Select “**Group**” checkbox from the floating tray. This will enable you to draw banding rectangle around a group of components you wish to change values for. In our example, these would be C4+L5+R6, which form a high-pass section of the band-pass midrange filter.

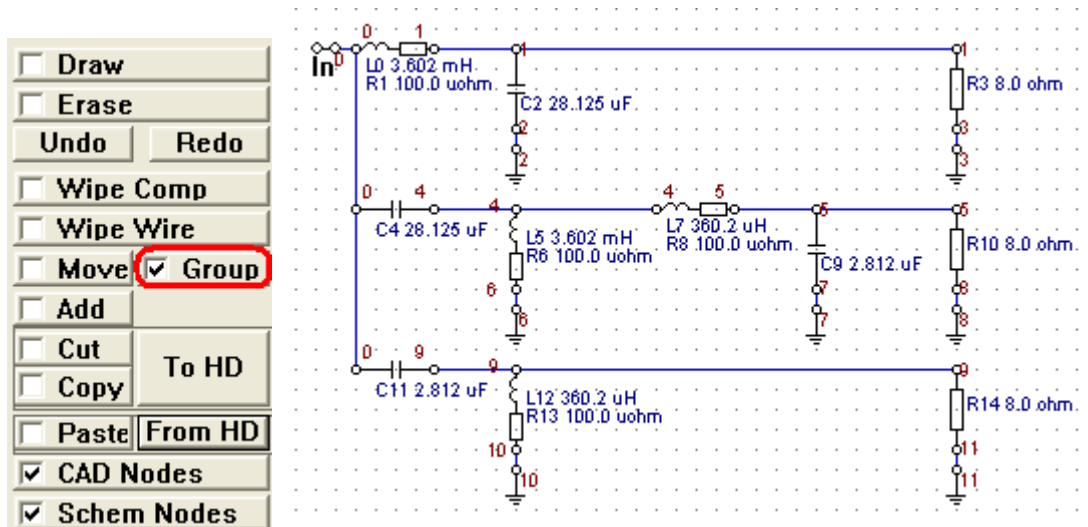


Figure 8.48. Selecting “Group” processing

### Step 2.

Draw the banding rectangle and release Left Mouse Button. You have now nominated components to be changed. Please remember, that you can only change a section of the schematic, that has the same topology as one of the built-in filters.

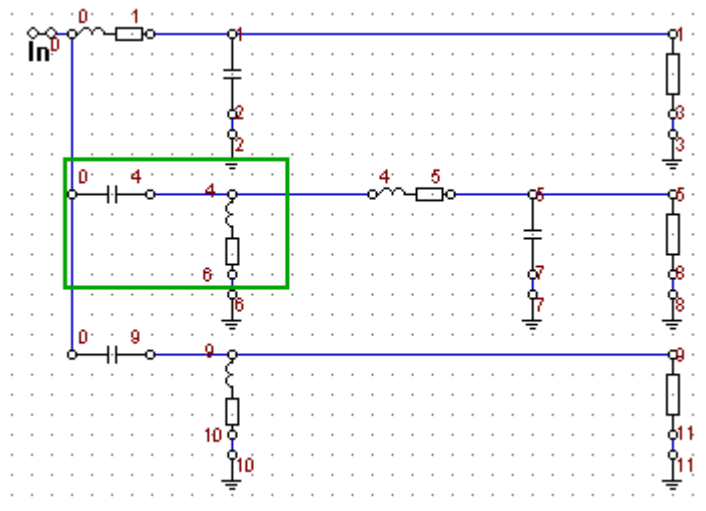


Figure 8.49 Components grouped and marked.

### Step 3.

Click on Right Mouse Button above one of the components in the group. This will open a dialogue box “**Group Component Editing**”, that will enable you to setup 3 sets of component values – each set for different cut-off frequency. Press “**Import Group1**” and this will open the familiar dialogue box, “**Built-in Filter Selector**”, used to import built-in filters.

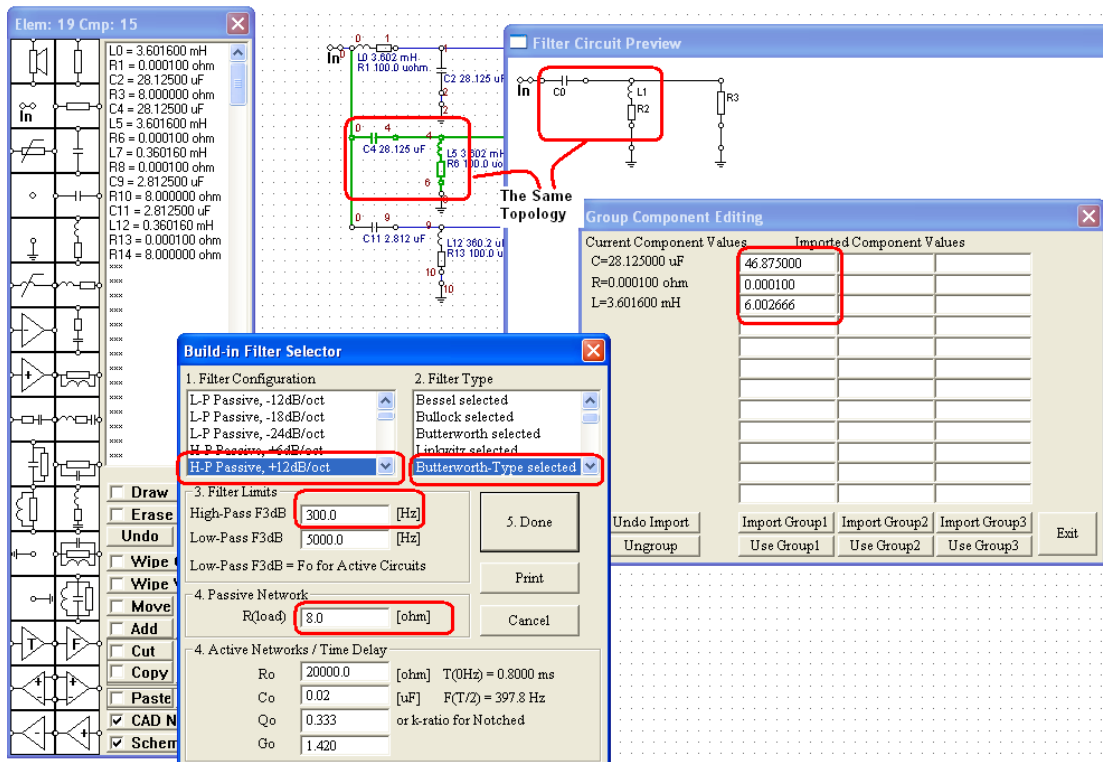


Figure 8.50. Selecting identical filter topology.

#### Step 4.

Select desired filter and all the necessary parameters. You will notice, that a display window, showing filter's topology will also open, so you can confirm that the topology of the CAD section you want to modify is the same as the filter you are using to pre-calculate new values. When finished, press **"Done"** button. This will close the filter selection dialogue and transfer new component values into the **"Import Group1"** set of data fields. Now, if you need to modify any of the data, you can do this now, as the data fields are editable. You can press **"Import Group X"** + Step 4 for all three sets of data.

#### Step 5.

Now, press **"Use Group1"** button to transfer the newly calculated component values into the original schematic. You will see these new values replacing the **"Current Component Values"** column.

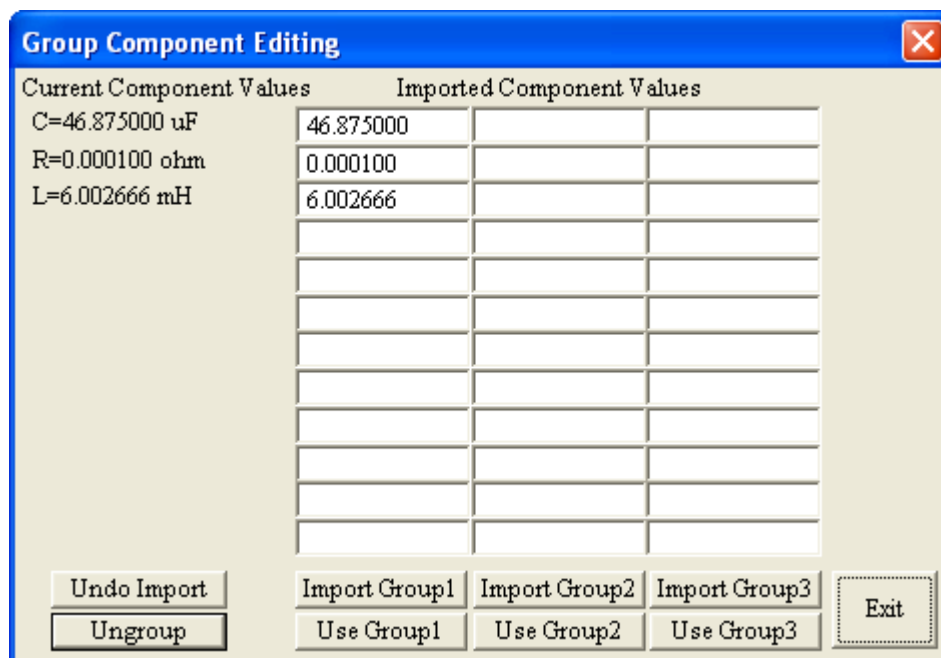


Figure 8.51. Group Component Editing dialogue box

## Step 6.

Press “Exit” button and this will close the “Group Component Editing” dialogue box and return the activities to the CAD screen.

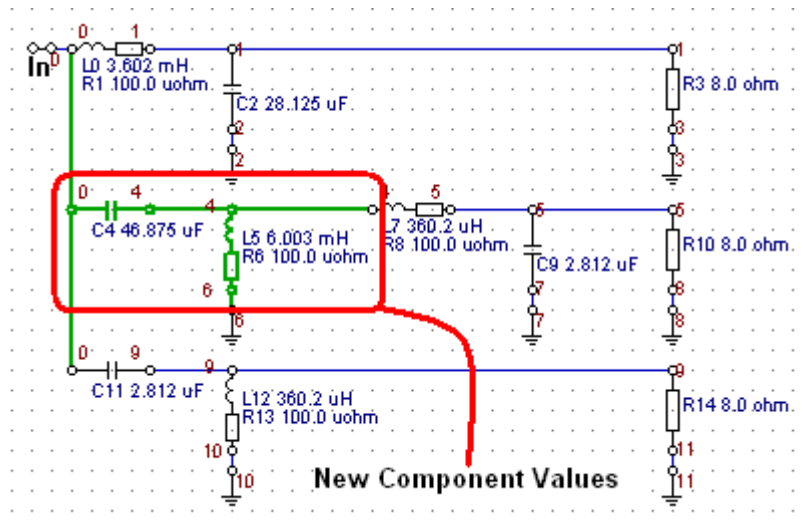


Figure 8.51. Grouped components with new values

Shown below are the frequency response changes “before” and “after” the substitution.

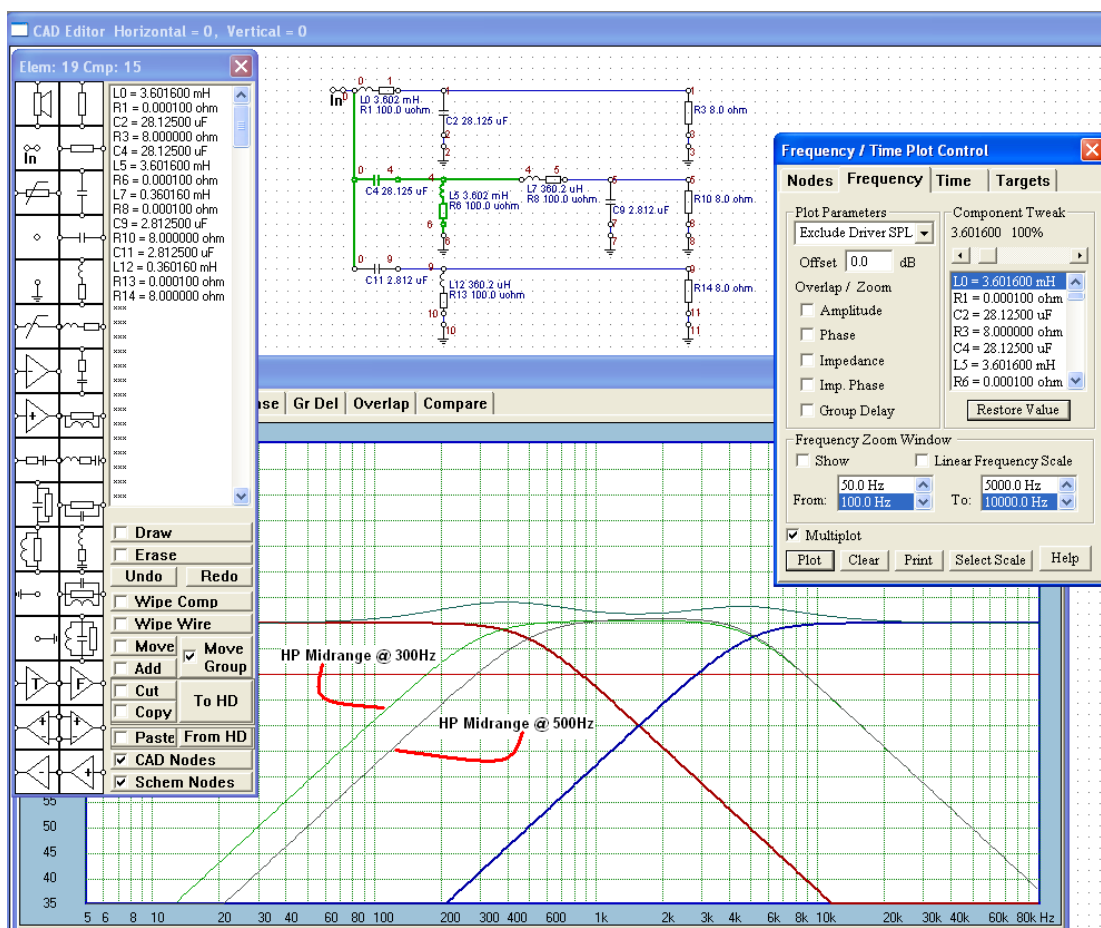


Figure 8.51. SPL plots of components with new values

When you become familiar with this method, you can preload the 3 sets of data fields with different components for different sections of the crossover.

When you become familiar with this method, you can preload the 3 sets of data fields with different components for different sections of the crossover. For instance, “**Use Group 1**” data column may contain alternative low-pass filter components, “**Use Group 2**” data column may contain alternative components for high-pass filter, and so on..... You will then be able to group different set of components with the banding rectangle and swap component values without resorting to the “**Built-in Filter Selector**” dialogue each time. There are 12 data fields available for new component values. Therefore, you will be able to deal with up to 4<sup>th</sup> order band-pass filters.

It is noticeable, that grouped components are highlighted on the schematic as long as the components are actually tagged as belonging to a group. (The group will hold the association until you select Move or Cut functions. These functions will automatically ungroup the components). However, this is handy, because now, you can click Right Mouse Button on any of the grouped components, and this will invoke the “**Group Component Editing**” box instantly. You can “**ungroup**” the components by pressing “**Ungroup**” button on the “**Group Component Editing**”. After that, each component can be edited again in a standard way, using the Right Mouse Button click over this component.

Finally, current software version allows the above features for passive crossovers only.

### Power Dissipation Screen

It is often very desirable to be able to understand power dissipation across your crossover network. Power can only be dissipated across resistive components, such as L-pad resistors, lossy inductors, that are represented on the CAD schematic as series coil resistors, and so on....There are up to 10 curves depicting power dissipated across nominated branches. Branches can be nominated from “Branches” TAB (see Figure 8.52) and can be turned ON/OFF using provided check boxes. To make the analysis more complete, you can also select input voltage to the crossover. This will be very useful to make sure, that the components can withstand full power from the amplifier, plus some safety margin.

Control buttons are self-explanatory. Branch Numbering of CAD schematic is automatic – necessary for Power Dissipation curves. Branch numbering can be toggled ON/OFF, via Automatic Node Numbers check box.

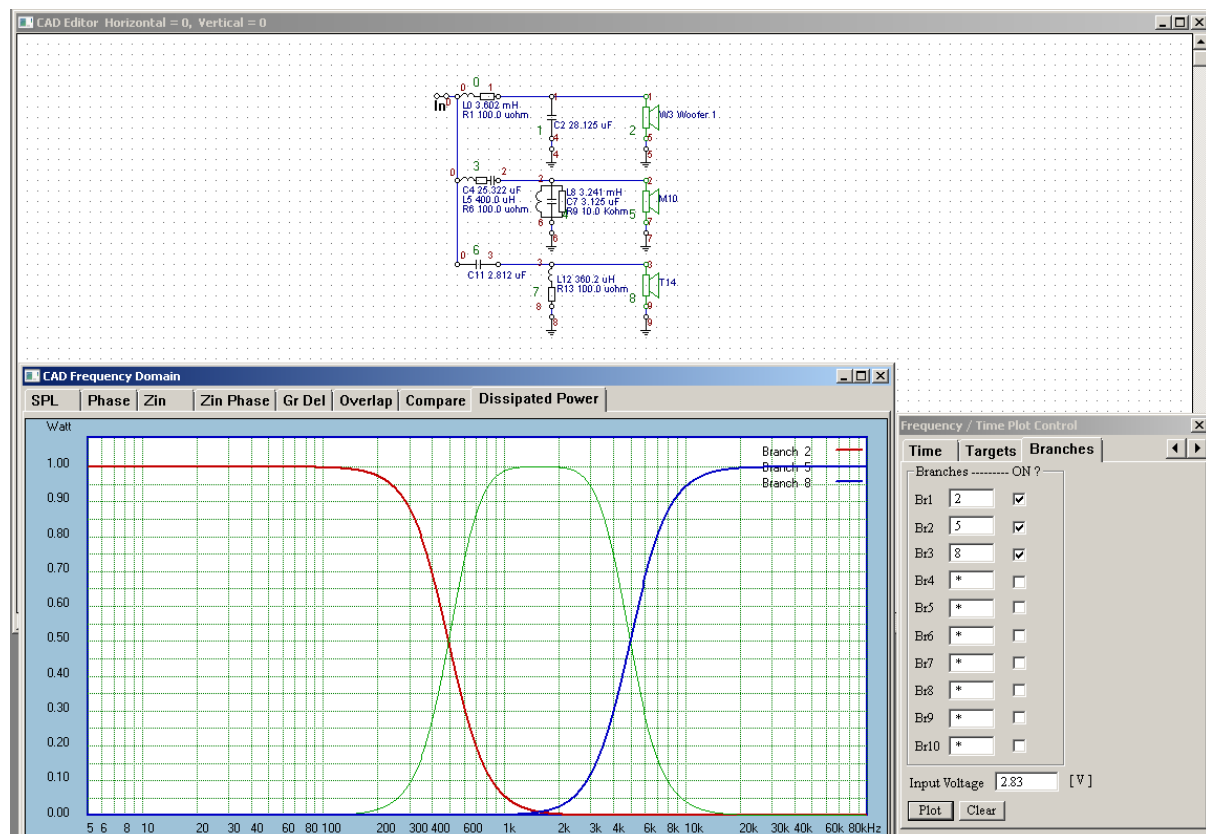


Figure 8.52. Power dissipation screen  
8.46

## Jumping Between Screens

Floating Pop-Up menu (appears at-your-fingertips) activated by Right Mouse Button allows you to jump instantly between 6 major sections of the program:

1. Editor control box,
2. CAD control box,
3. Enclosure Designer control box,
4. Polar Plots control box,
5. System Front Panel control box and
6. MLS Acquire Tab.

Example below shows RMB clicked upon a component section area ( this is display only area, therefore NOT an active Windows component area ). After the pop-up menu appeared, a jump to “Digital – MLS” section was selected.

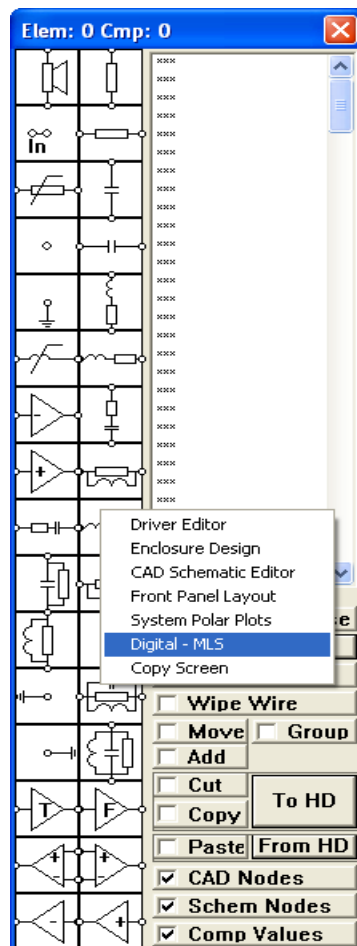


Figure 8.53. Jumping to MLS System.

Please note, that component values displayed on CAD schematic can be toggled ON/OFF using “Comp Values” checkbox.

## Recalling Direct Measurements To Compare With CAD simulation

At some point during the design process, you may wish to compare that last built prototype system with the current CAD simulations. The important measurements ( SPL/Phase, Group Delay, Zin and ZinPhase) have been saved in MLS system and these measurements can be re-called into CAD and System design screens to compare the measurements results with the current CAD simulations. You’ll need to select “**Saved Measurement**” checkbox option in the “**CAD Frequency / Time Plot Control**” dialogue box – see Figure 8.54 below.



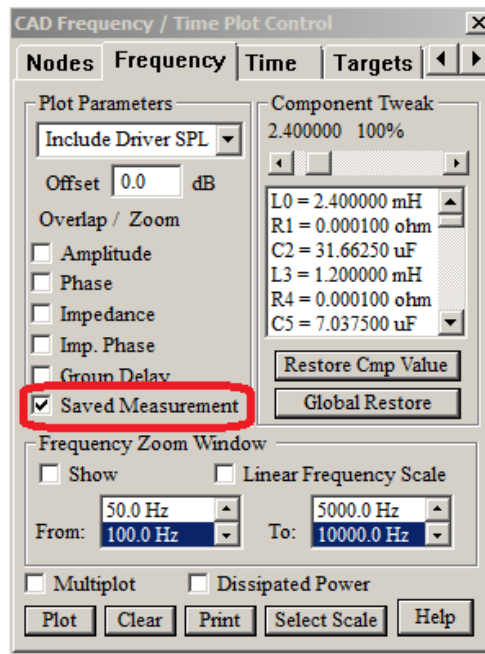


Figure 8.54 Selection of saved measurements in CAD System.

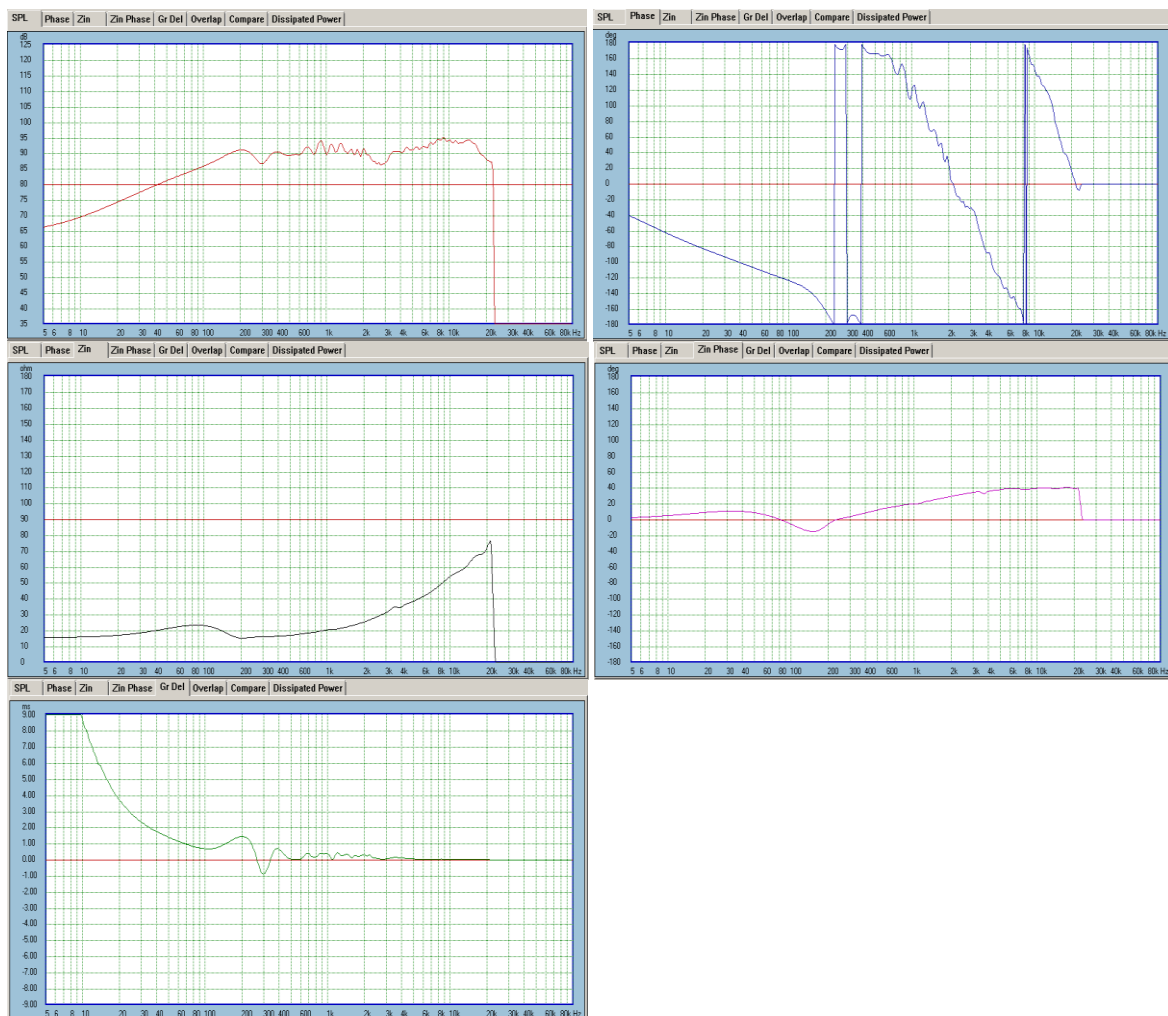


Figure 8.55. SPL / Phase / Zin / ZInPhase / GroupDelay measured curves displayed in CAD system.

Saved measured curves will appear in the corresponding TABs. The checkbox will flip ON/OFF the displayed curves.



## Mounting Coordinates In The System

Mounting coordinates on the front baffle are displayed in CAD screen – see Figure 8.56 below. This data is only available after the drivers were positioned on the front baffle – using system “Front Panel Layout” tools. The mounting coordinates are automatically saved into drivers’ files upon exiting from the “Front Panel Layout” dialogue box.

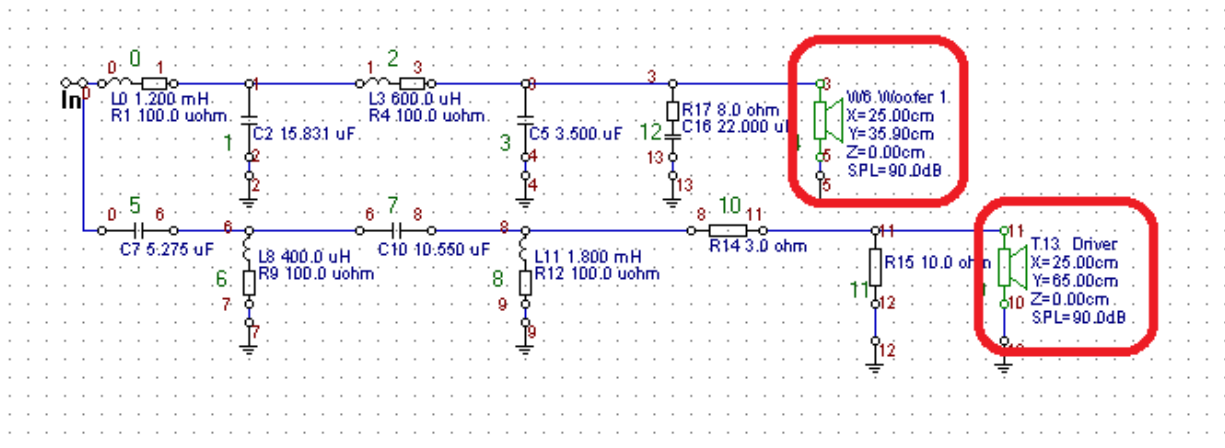


Figure 8.56. Mounting coordinates on the front baffle are displayed in CAD Characteristics screen.

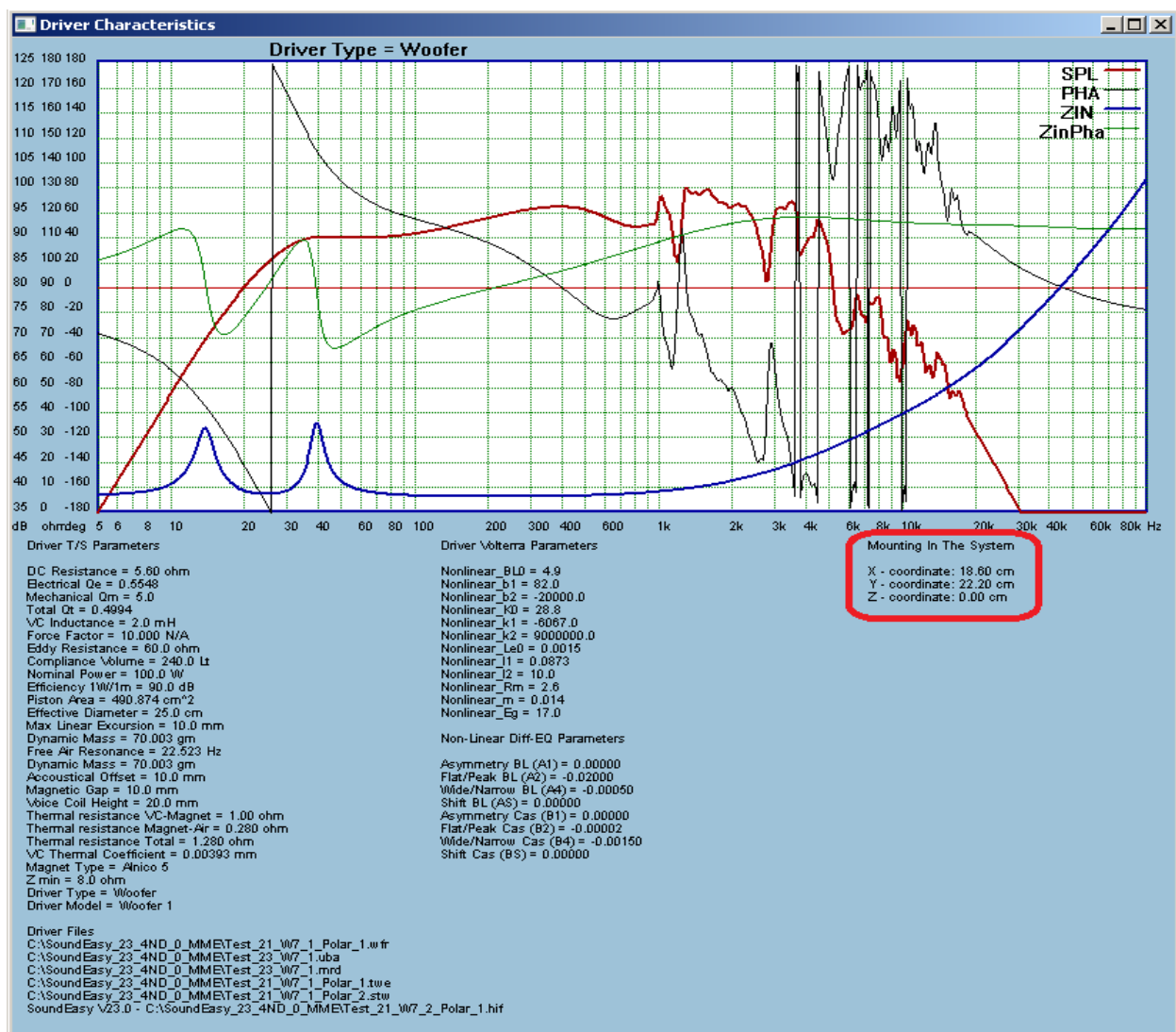


Figure 8.57. Mounting coordinates on the front baffle are displayed in Driver Characteristics screen.

## High-Order Filters

Implemented  $\pm 48\text{dB/oct}$  and  $\pm 96\text{dB/oct}$  HP/LP Butterworth filters and Targets – see below.

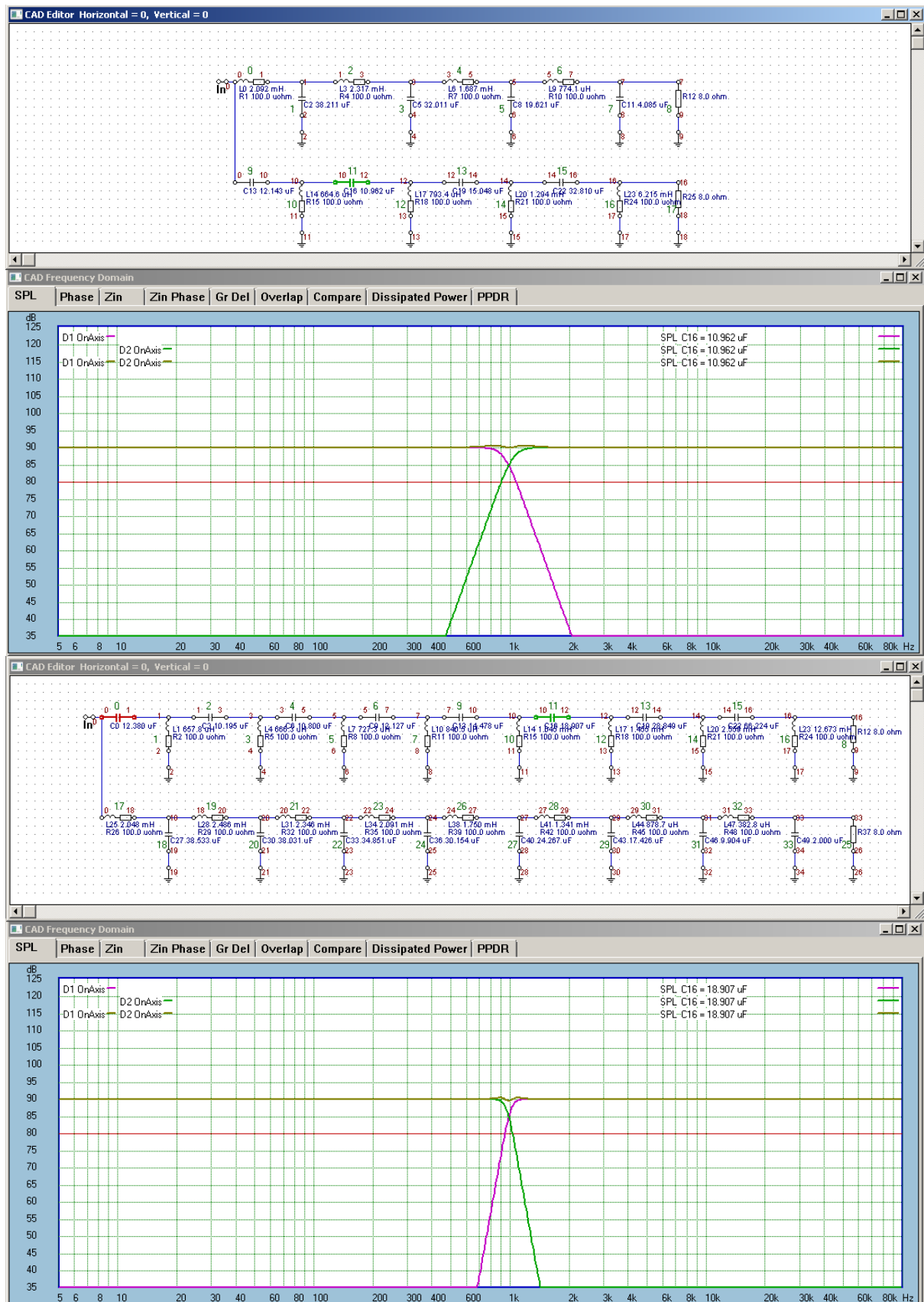


Fig 8.58.  $\pm 48\text{dB/oct}$  and  $\pm 96\text{dB/oct}$  HP/LP Butterworth filters.

The Butterworth crossovers shown above have very small ( $\pm 0.5\text{dB}$ ) ripple at the crossover frequency. It is unlikely, that this ripple would be audible. On the above Figure 58:

The 8<sup>th</sup> order crossover has shifted LP and HP frequencies to LP 3dB = 950Hz and HP 3dB = 1050Hz.  
The 16<sup>th</sup> order crossover has shifted LP and HP frequencies to LP 3dB = 975Hz and HP 3dB = 1025Hz.

For comparison, please examine a standard approach to a  $\pm 24\text{dB/oct}$  Butterworth crossover shown below. The is a  $+3\text{dB}$  hump at the crossover frequency, which may be audible. Linkwitz-Riley crossovers do not have this limitations.

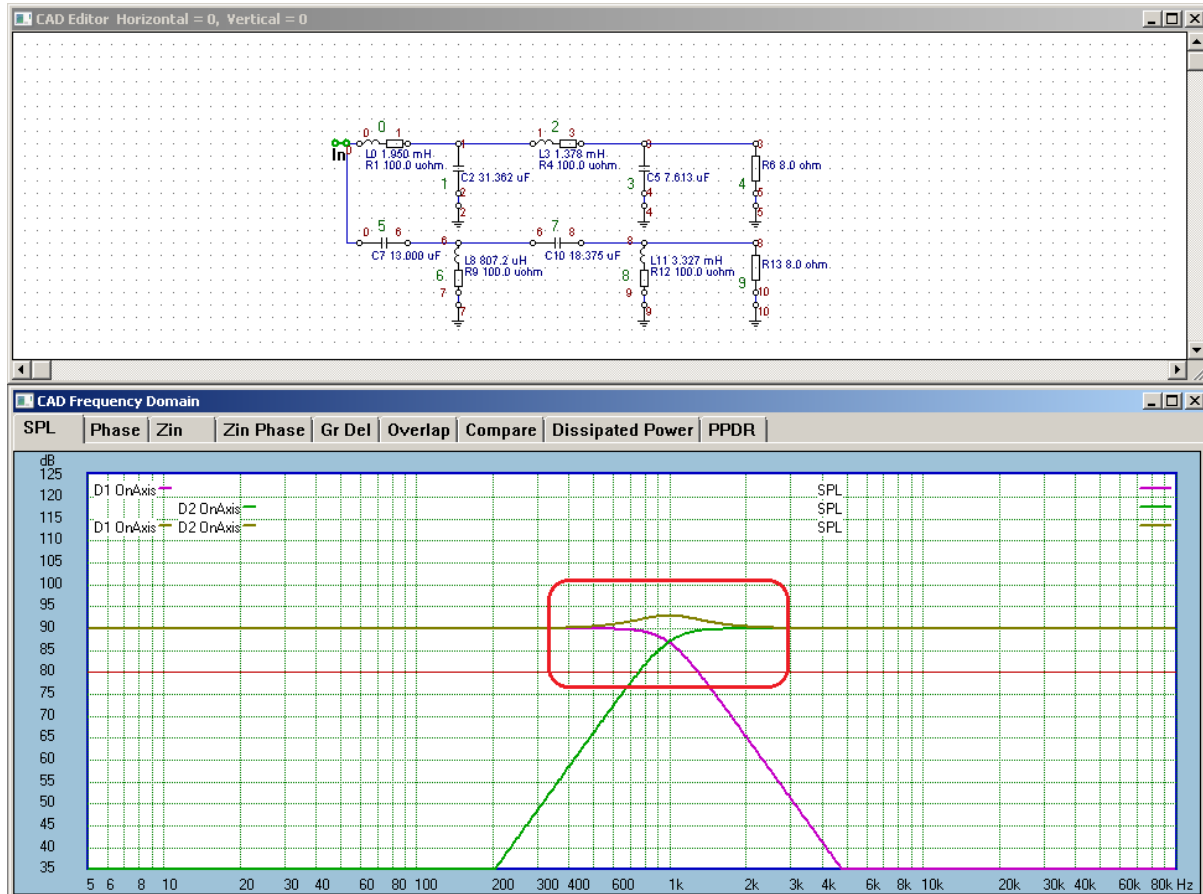


Fig 8.59.  $\pm 24\text{dB/oct}$  Butterworth filters.

### Shorted And Opened Component

Frequently used option in the crossover “What-if” analysis technique is the ability to remove a component from the crossover transfer function, without actually rewiring the schematic. A distinction has to be made whether the component is “in-series” with the signal flow or it “shunts” the signal to the ground. To remove the component from the filter function please apply the following rules:

1. Apply **Short** – if the component is “in-series” with the signal flow. Component will stay in **Blue colour**
2. Apply **Open** – if the component “shunts” the signal to the ground. Component will stay in **Pink colour**

These simple rules are demonstrated on the next example below. Component (R15) is shunting the signal to the ground causing a reduction in the power delivered to the load (tweeter). Therefore, to remove this component from the filter transfer function, it needs to be made “open circuit”. When this is done, we would expect the power delivered to the load to increase, as the shunting effect is no longer there. Indeed, when the “Open” option is applied to this component, the power delivered to the load (tweeter) is significantly increased.

Conversly, when the shunting component is shot-circuited, one may expect a dramatic reduction in signal flow. And this is also demonstrated on the Figure below. When the “Short” is applied to the shunting component (R15), virtually all power delivered to the tweeter is diverted to the ground – as indicated by the green curve.

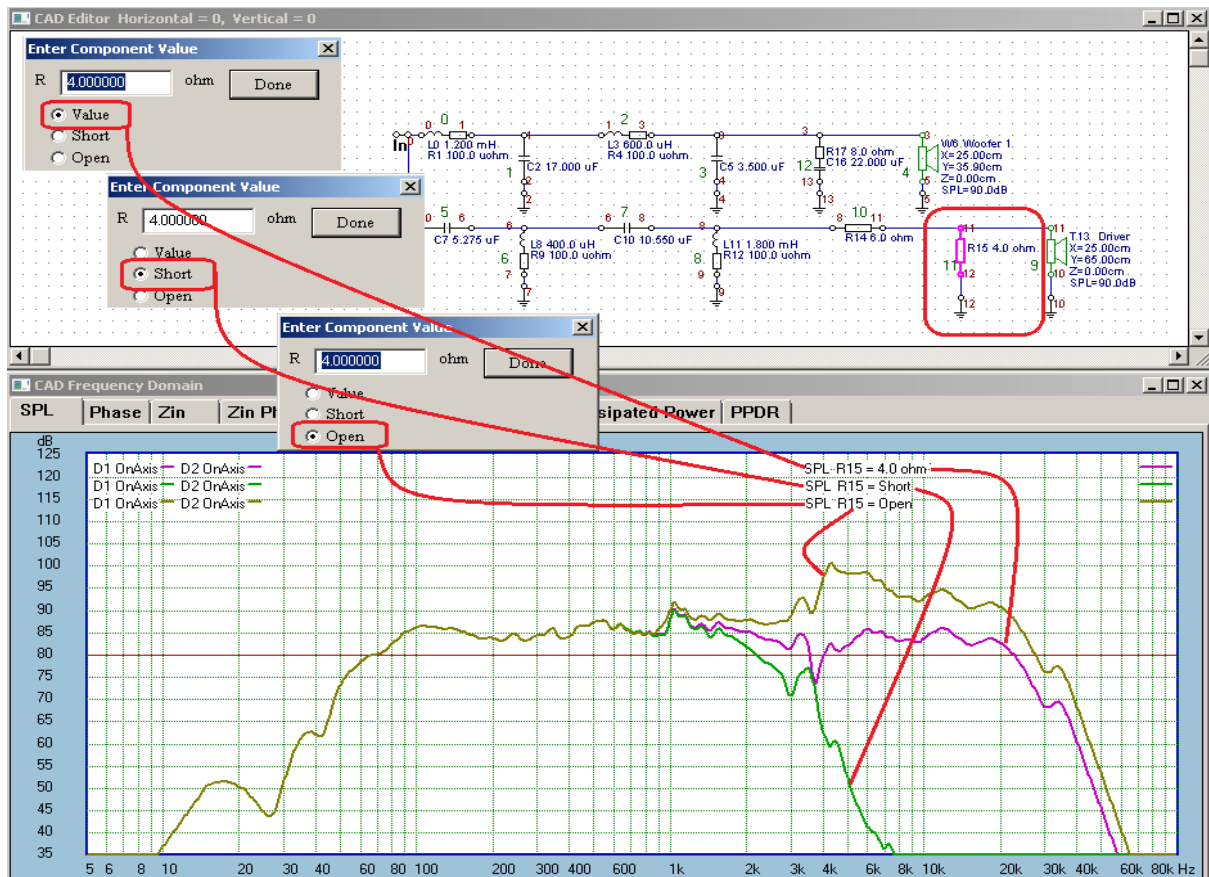


Fig. 8.60. Option to “Short” or “Open” passive component on the schematic without removing the component.

Finally, “Last Edited Component” stays highlighted on the schematic for easy localization. Plots shown below used mouse wheel.

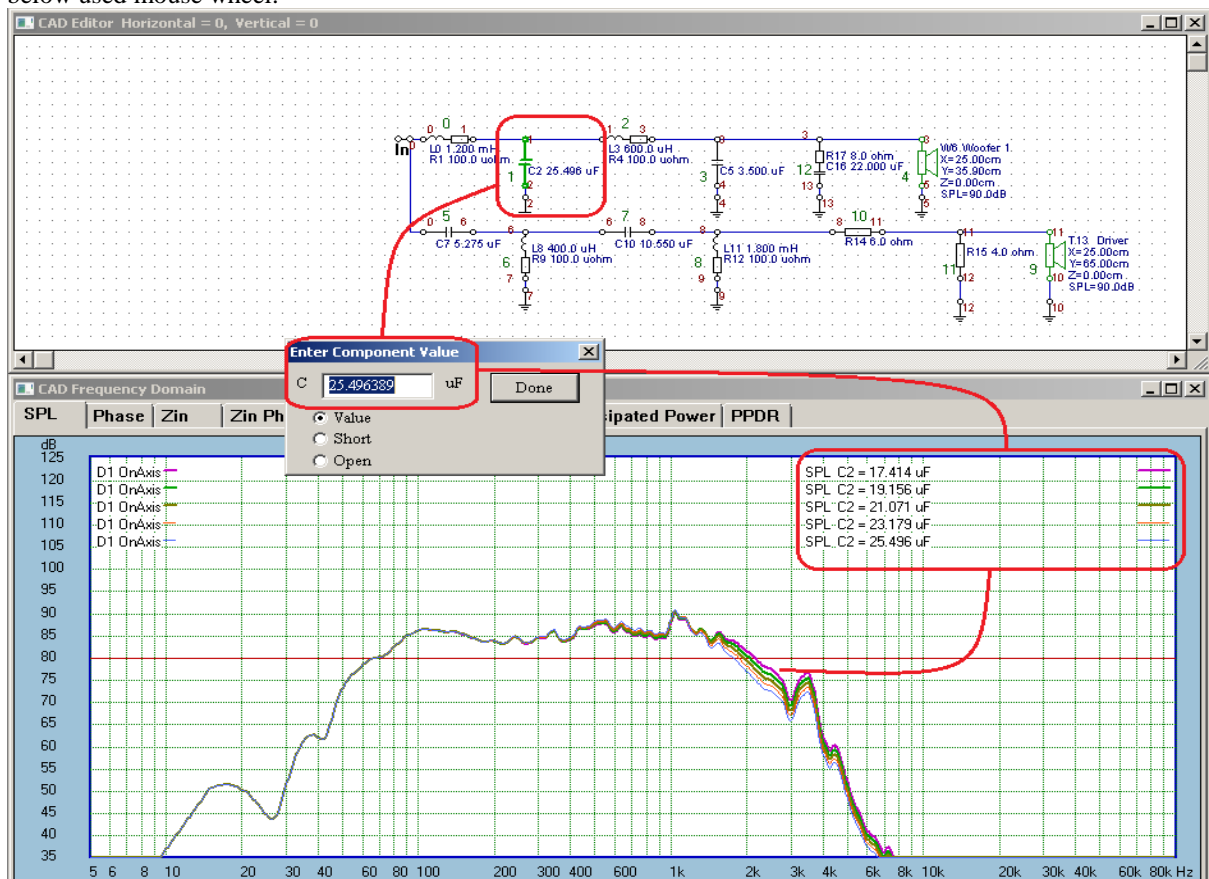


Fig 8.62. Labelling SPL curves for the last edited component in CAD Module

## CAD Plotting Options

There are 7 possible plotting options available from the CAD dialogue box shown below. Performance plots are designed to reflect two basic design options:

1. You rely solely on measured data and wish to use it for crossover/system performance review. In this scenario, driver's SPL performance was measured **with enclosure diffraction** over -90....+90deg angles and saved to driver's data file. All off-axis modelling will be based on measured curves. You should only select the following options available under "Plot Parameters": (1) Crossover ONLY, (2) Measured SPL+ Filt, (3) SPL of Driver Type Is and (4) Listen Window + Flt.
2. You rely on single, on-axis measurement **without diffraction**. In this scenario, driver's SPL performance was measured **without enclosure diffraction** and saved to driver's data file. All off-axis modelling will be based on program's modelling capabilities. You should only select the following options available under "Plot Parameters": (1) Off-Axis SPL + Diff, (2) Off-Axis SPL + Difr+Flt and (3) Off-Axis Diff ONLY.

All plotting options in Option 2 will respond to microphone pivot point coordinates and general microphone spherical location.

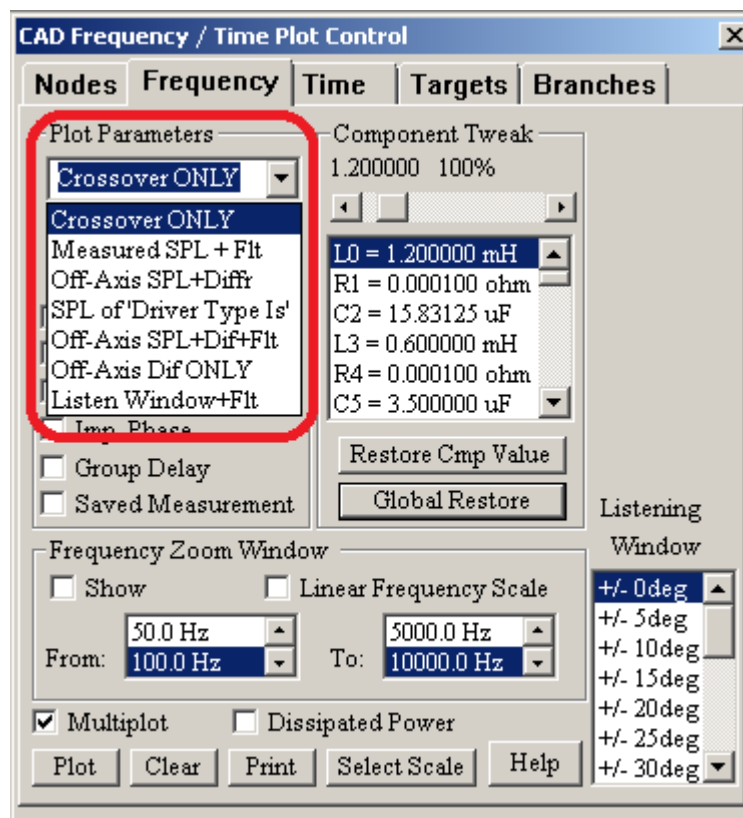


Fig 8.63. 7 plotting option available in CAD system.

Diffraction is calculated automatically (on-the-fly), provided the front baffle and microphone pivot location was correctly edited in the "Front Baffle Design" system.



## Frequency Response Analysis Options

Here is a detailed description of the plotting options.

**Meas(jw)** = measured driver's transfer function (or SPL/Phase).

**Flt(jw)** = filter transfer function.

**Dif(jw)** = enclosure diffraction.

1. **Crossover ONLY** – Use this option ONLY when a transfer function of a crossover or filter, Flt(jw), is required. This option DOES NOT include any driver SPL related parameters. Good for crossover study.
2. **Measured SPL + Flt** – Use this option to display curves of ON/OFF axis SPL, Meas(jw), measured **with diffraction, at each microphone position** and stored in driver file, and now being convolved with the filter Flt(jw). You can select different off-axis angles to display convolved Flt(jw)\*Meas(jw). This option DOES NOT respond to any microphone position selected from the System Screen. This option would be used when you have a full set of ON/OFF axis SPL measurements for all drivers in the system, perhaps performing final tweaks of the completed system. Since this option does not respond to microphone positioning on the System Screen. The measurements of each driver's SPL should be performed with the microphone positioned at the system measurement point – typically tweeter on-axis.
3. **Off-Axis SPL+Diff** – Designed primarily to work with **Generate H-Polar Plots** and **Generate V-Polar Plots** buttons. Therefore, use this option to generate complete ON/OFF axis SPL and Diffraction, Dif(jw), for current front baffle, and for angles -90deg.....+90deg in 5deg increments. **You must provide SPL measured on-axis in “half-space”, that is with box diffraction removed.** This option DOES respond to any microphone position selected from the System Screen. This option is convenient, when you do not have time to perform a full set of ON/OFF axis measurements. You would only provide single, on-axis measurement with diffraction removed for each driver, Meas(jw), and then use this option to calculate Meas(jw)\*Dif(jw) for all off-axis angles. In effect, you have created a full set of ON/OFF axis SPL measurements for all drivers in the system – this is needed for Option 2 above.
4. **SPL of 'Driver Type Is'** – Use this option to display original ON/OFF axis SPL data stored in driver's file. Data will be displayed without any additional processing. So, for instance, if you stored ON/OFF axis data without diffraction, it will be displayed as such. This option DOES NOT respond to any microphone position selected from the System Screen.
5. **Off-Axis SPL+Dif+Flt** - Designed primarily to work with **Generate H-Polar Plots** and **Generate V-Polar Plots** buttons. Use this option to generate complete ON/OFF axis SPL and Diffraction, Dif(jw), convolved with Flt(jw), for current front baffle, and for angles -90deg.....+90deg in 5deg increments. **You must provide SPL measured on-axis in “half-space”, that is with box diffraction removed.** This option DOES respond to any microphone position selected from the System Screen. Therefore, it can be used to simulate system ON/OFF axis performance, when only on-axis, “half-space” measurement for each driver is provided.
6. **Off-Axis Dif ONLY** - Designed primarily to work with **Generate H-Polar Plots** and **Generate V-Polar Plots** buttons. Use this option to display curves of ON/OFF axis diffraction curves, Dif(jw), ONLY. This option DOES respond to any microphone position selected from the System Screen.
7. **Listen Window+Flt** – Use this option to display SPL response AVERAGED over a selected “listening window” consisting of a range of horizontal or vertical angles. This option DOES NOT respond to any microphone position selected from the System Screen. This option would be used when you have a full set of ON/OFF axis SPL measurements for all drivers in the system, perhaps performing final tweaks of the completed system. Since this option does not respond to microphone positioning on the System Screen. The measurements of each driver's SPL should be performed with the microphone positioned at the system measurement point – typically tweeter on-axis

Please note, that “half-space” measurements are obtained by measuring a driver in the enclosure, and then subtracting diffraction calculated for the same enclosure and microphone location.

### Example

Front Baffle size and microphone location was edited (as per measurement system) in Front Baffle Design dialogue box. Driver was measured via MLS/ESS system **without diffraction and microphone placed at pivot position**. The resulting SPL curve is shown on Figure below.

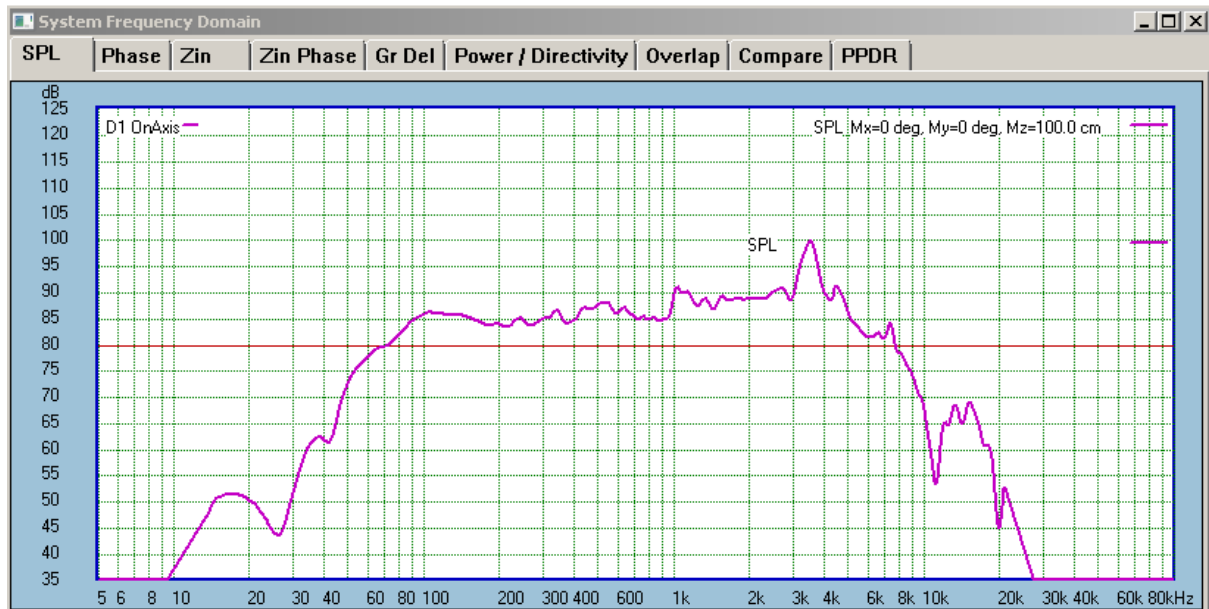


Fig 8.64. Measured SPL without diffraction.

If you continue to use “Standard” and “Measured SPL + Flt” settings, you will arrive at the display as shown below.

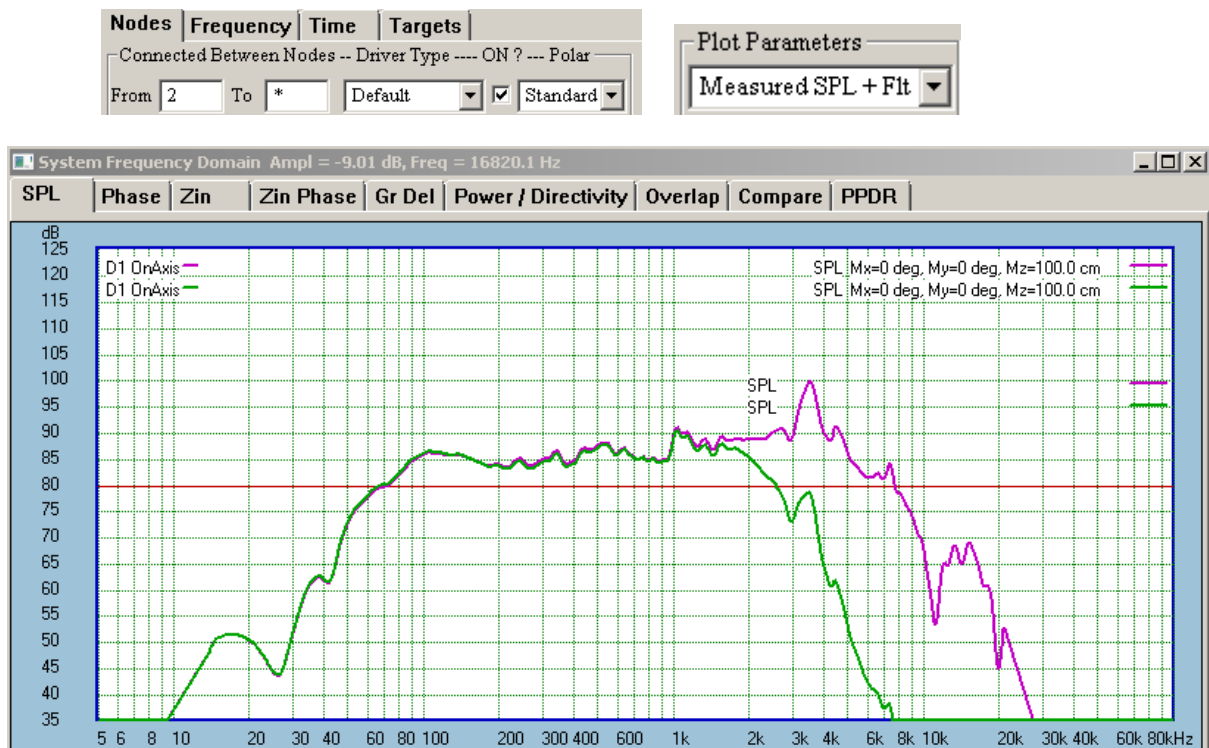


Fig 8.65. Pink curve is the on-axis SPL of the driver measured without the diffraction with and without the filter. Green curve is the on-axis SPL of the driver measured without the diffraction with and with the filter.

Please note, that you can measure the driver with diffraction, and use method described in Chapter 2, page 2.29 to easy subtract the diffraction.

We can now generate off-axis responses.

Off-axis responses were generated using **Generate H-Polar Plots** button in System Module. Microphone placement was the exact copy of the measurement placement, on-axis with the driver and Pivot Point was adjusted accordingly. Please switch to System section.

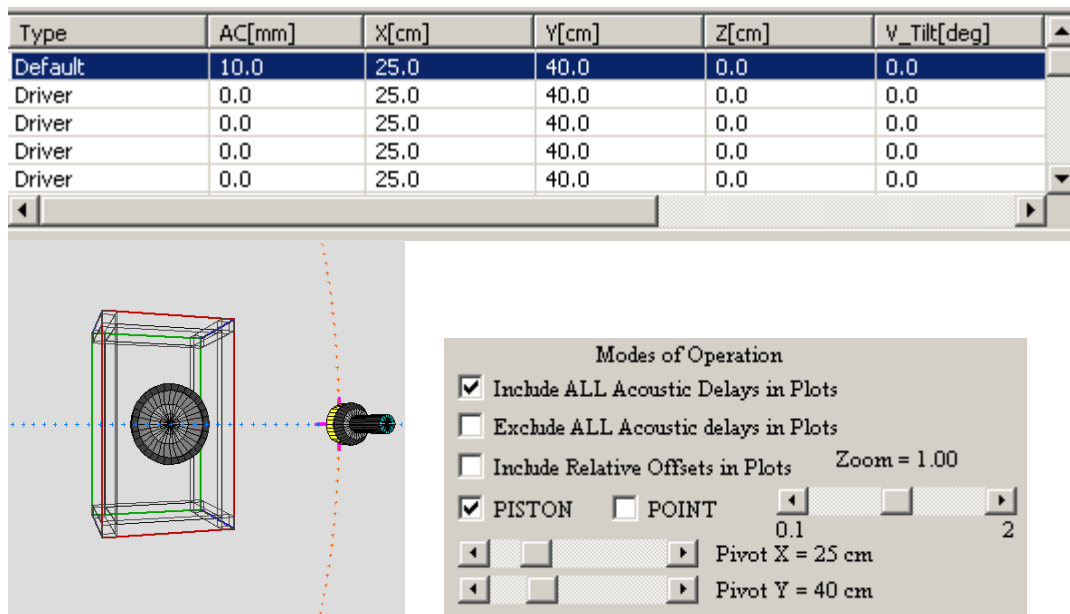


Fig 8.66. Setting pivot point to align it on driver's axis.

With the above settings at hand, we are ready to generate a complete set of off-axis SPL curves for this driver. Please select the following:

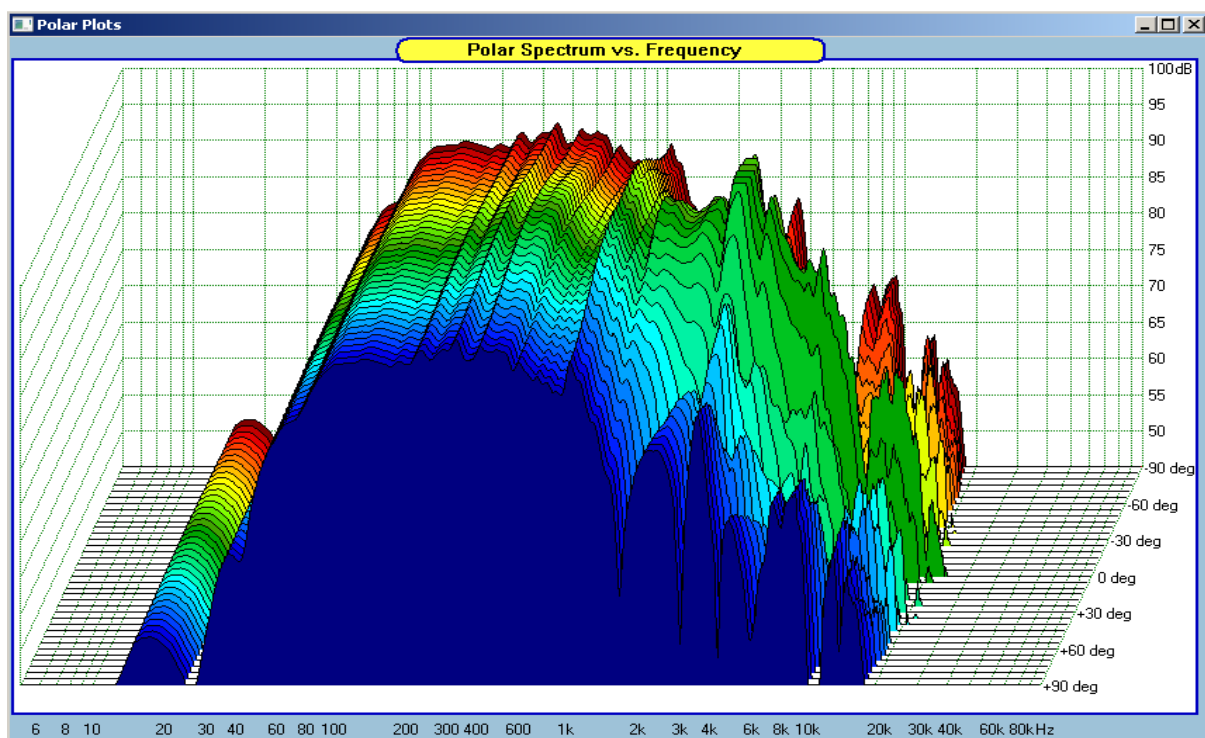


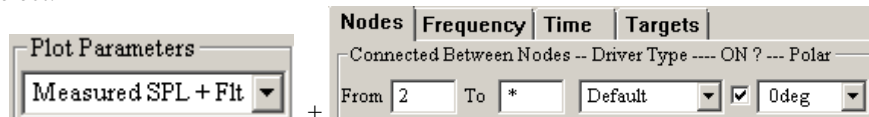
Fig 8.67. Polar Dispersion window pops up automatically.

Now, all the SPL plots from -90deg to +90deg have been generated and will include driver's off-axis pistonic model and off-axis diffraction calculated for corresponding angles, including the on-axis (or 0deg) angle. However, **the original on-axis SPL measurement performed without diffraction (stored in Polar location = Standard) has NOT been overriten**, and will remain as a "seed" in the driver file for future reference.

Please save the driver file.



This newly generated set of SPL plots has exactly the same characteristic as if the curves were measured, therefore, the data will be placed in the same data buffer as the measured curves. You can now access those curves via “Measured SPL + Flt” plotting option.  
Now, if you select:



The following curves are obtained: Pink = On-axis with diffraction, Green = On-axis + diffraction + Filter.

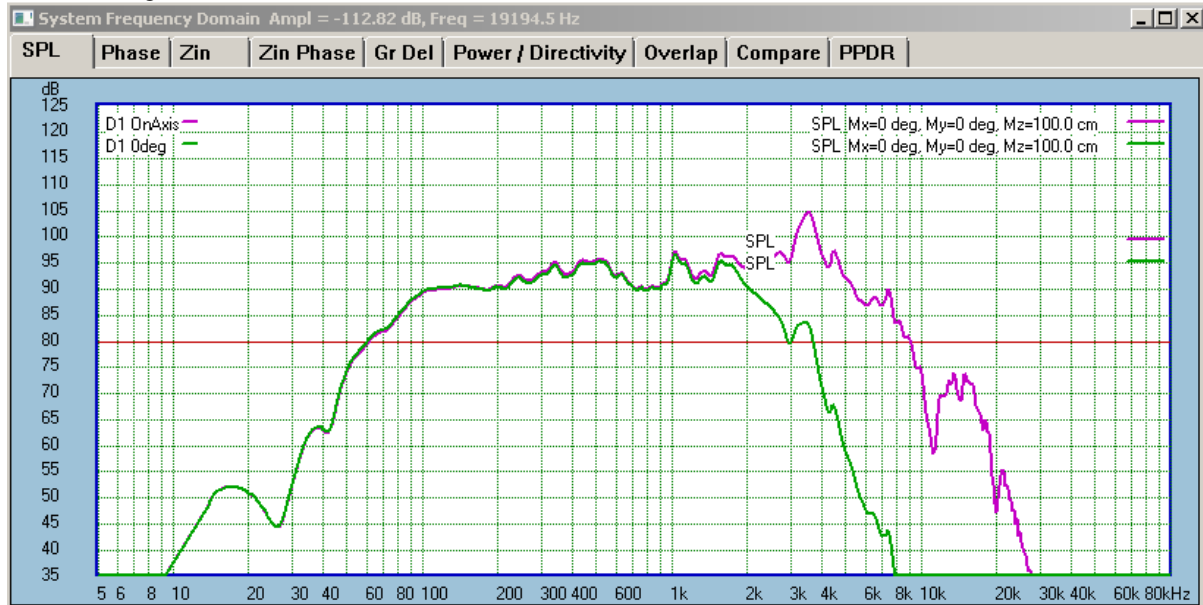
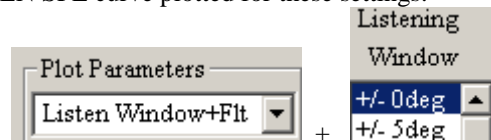


Fig 8.68. Calculated SPL + diffraction for 0deg angle.

It is observable, that the GREEN SPL curve plotted for these settings:



Should be the same as the PINK SPL for these settings: (Green was risen by 1dB to differentiate):

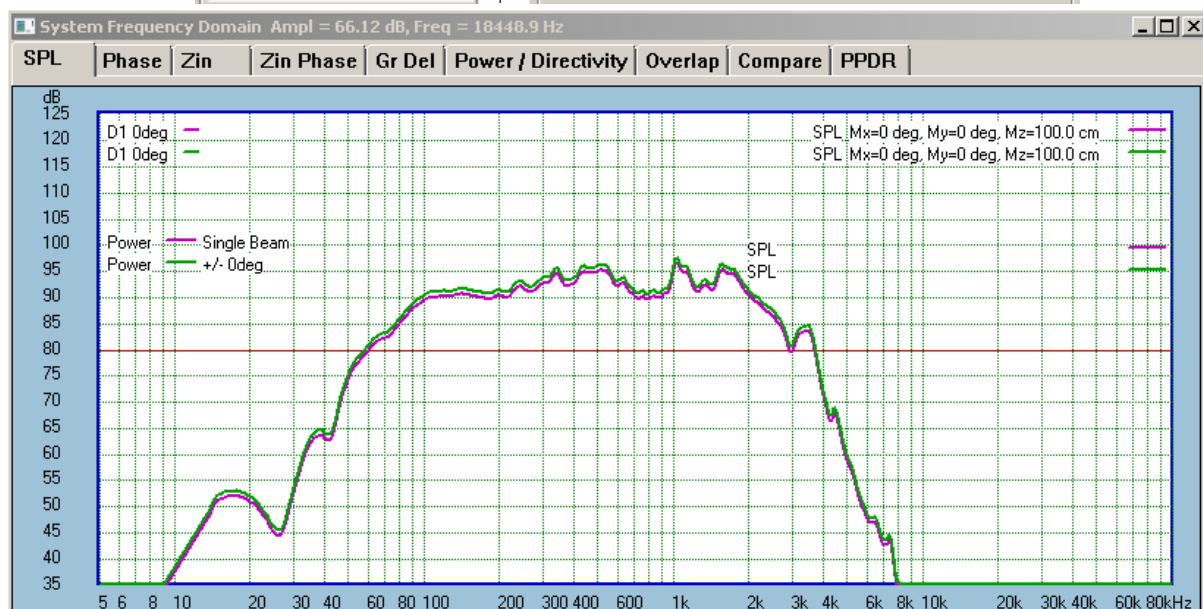
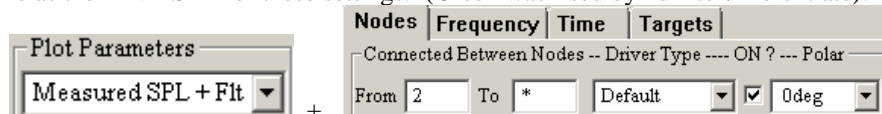


Fig 8.69. Equivalency of SPL curves.