Capsim® Tutorial

Linux/ MAC OS X Environment

Capsim ® Linux/MACOSX Version 6.0
(c) 1989-20016 Silicon DSP Corporation, All Rights Reserved

Table of Contents

Table of Contents ____ 2

List of Figures ____ 3

Introduction ____ 5

Running Capsim ____ 5

Placing Blocks on the Workspace ____ 6

Connecting Blocks ____ 8

Running a Simulation ____ 10

Errors During a Run or Editing ____ 17

Manipulating Plots ____ 17

Replacing Blocks ____ 18

Block Parameters ____ 19

Saving the Topology ____ 23

Creating an Hblock (Hierarchical Block) ____ 24

- HBlock Input Terminal ____ 25
- Manual Connections ____ 31
- Using a Hierarchical Block ____ 35
- Placing HBlocks on the Workspace ____ 35
- Moving Down an HBlock ____ 37

Probes ____ 37

HBlock Parameters: Arguments ____ 41

Global Parameters Through Arguments ____ 43

- Creating HBlock Parameters Through Arguments ____ 44
- Arguments as Global Parameters ____ 49

Actions on Multiple Blocks ____ 50

- Selecting Multiple Blocks ____ 50
- Removing Multiple Blocks ____ 51
- Moving Multiple Blocks ____ 52
- Converting Multiple Blocks to an HBlock ____ 52
- Replacing Multiple Blocks With an HBlock ____ 53

Multi-Rate Signal Processing, Eye Diagrams and Stacked Plots ____ 56

Multiplot Block and the Logic Analyzer Probe ____ 60

Complex Buffers and the Complex Plot Block ____ 64

Pole Zero Analysis ____ 66

Appendix A: Block Popup Menu Description ____ 72

Appendix B: Connection Popup Menu Description ____ 75

Appendix C: Command Line Graphic Settings ____ 77

List of Figures

Figure 1 Capsim Pulldown Menu Bar and Console 6
Figure 2 Capsim List of Blocks 7
Figure 3 Block layout for impulse, iirfil and spectrum 8
Figure 4 Block Popup Menu 9
Figure 5 Connected Blocks 10
Figure 6 Blocks Menu 11
Figure 7 Filter Impulse Response 11
Figure 8 Filter Frequency Response 12
Figure 9 Plot Popup Menu 13
Figure 10 Plot Style Dialog 14
Figure 11 Impulse Response with Bars to Highlight Discrete Samples 15
Figure 12 Create dB Plot Menu Item 16
Figure 13 Frequency Response in dB's 17
Figure 14 Replace Sub-Menu 18
Figure 15 Impulse Block Replaced by Sine Block 19
Figure 16 sine Block Parameters 20
Figure 17 Plot Menu 21
Figure 18 Group Plot Dialog 22
Figure 19 Comparison of Filter Response to 1000 Hz and 5000 Hz Sine Wave 23
Figure 20 HBlock Bock Layout 25
Figure 21 Connecting an Input Terminal (Select Workspace) 26
Figure 22 Input Terminal Connected 27
Figure 23 Connect add Block to node Block 28
Figure 24 Connect node to delay 29
Figure 25 Connect delay to add Block 30
Figure 26 Connect node to "Workspace" for Output Terminal 31
Figure 27 Redrawing a Connection 32
Figure 28 Manual Redraw 33
Figure 29 Manual Redraw Feedback 34
Figure 30 Delay Connections with Manual Drawing 34
Figure 31 integ HBlock with pulse and plot Blocks 35
Figure 32 pulse Block Parameters 36
Figure 33 Simulation Run with integ HBlock Showing Stair Case. 37
Figure 34 Topology for Probe Insertion 38
Figure 35 Connection Menu 39
Figure 36 Probe Dialog 40
Figure 37 Probe Simulation Results 41
Figure 38 HBlock Arguments 42
Figure 39 Argument Dialog Box 42
Figure 40 Argument Definition Dialog Box 43
Figure 41 Arg 0 44
Figure 42 Topology for Global Arguments 44
Figure 43 Arguments Dialog 45

Figure 44 Arg Dialog Box *45*
Figure 45 Updated Args: Number of Samples Arg *46*
Figure 46 Global Arguments *46*
Figure 47 sine Block Arguments with Connection to Arguments *47*
Figure 48 spectrum Block Parameters *47*
Figure 49 addnoise Parameters with Argument *47*
Figure 50 cubepoly Parameters with Arguments *48*
Figure 51 Result of Simulation *48*
Figure 52 Arguments as Global Parameters *49*
Figure 53 Edit Menu Showing Block Manipulation Items *50*
Figure 54 Blocks *50*
Figure 55 Start of Selecting Blocks *51*
Figure 56 Bounding Box *51*
Figure 57 Selected Blocks *51*
Figure 58 Selected Blocks Deleted *51*
Figure 59 Moving Selected Blocks *52*
Figure 60 Creating an HBlock and Replacing the Blocks *53*
Figure 61 Selecting Blocks *54*
Figure 62 Naming the HBlock *54*
Figure 63 Blocks Replaced with channel HBlock *55*
Figure 64 Drilling Down the HBlock *56*
Figure 65 Showing Block Names *57*
Figure 66 Eye Diagram *58*
Figure 67 Stacked Plots Showing Multirate Sampling *59*
Figure 68 Multiplot *61*
Figure 69 pulse Block Parameters *62*
Figure 70 wave Block Parameters *62*
Figure 71 Results of Simulation with Logic Analyzers *63*
Figure 72 Topology to Illustrate Complex Buffers *64*
Figure 73 cxplot Plots both the Real and Imaginary Parts of Complex Buffers *65*
Figure 74 IIR Filter Impulse Response *66*
Figure 75 iirfil Default Parameters *66*
Figure 76 tmp.pz File Contents *67*
Figure 77 Filter Design Pulldown Menu *68*
Figure 78 Pole-zero Layout for tmp.pz File. Also Shown is the Pole-zero Toolbar. *69*
Figure 79 Pole-Zero Analysis After Calculation *70*

Introduction

Welcome to Capsim. This is a tutorial that will expose you to many of Capsim's features. In the tutorial, we will examine the impulse response of a digital low pass filter. Next, we will observe the response of the filter to two sine waves at different frequencies. Finally, we will build a staircase generator Hblock (Hierarchical Block) from simple blocks. Other Capsim topics including logic analyzers, multi-plots, complex buffers, multi-rate signal processing and creating HBlocks from existing topologies will be presented.

Running Capsim

To begin Capsim, type:

```
$CAPSIM/BIN/capsim
```

Note: The environment variable CAPSIM must be set to the path where CAPSIM is installed. For example,

```
export CAPSIM=/usr/local/CAPSIM_V6_RELEASE
```

Important Note: When you run Capsim a window will appear for graphical block diagram development. The console will display messages and also is the TCL interpreter with the '%' prompt. Make sure you position the console so you can enter commands and view messages and simulation results in the console.

The screen will display a pull-down menu bar as shown in Figure 1. You will need to switch between the console and the Capsim workspace on some occasions. To display a new workspace in Capsim, select "New" from the File menu. An empty workspace will be displayed.

Before we proceed, exit Capsim using the File menu exit command or type exit in the console (in TCL mode). To work with this tutorial it is recommended that you create a directory and store all your work in that directory.

To make a directory, say, WORK type:

```
mkdir WORK
```

Then change directories to WORK (`cd WORK`). Type `pwd` to see the current working directory.

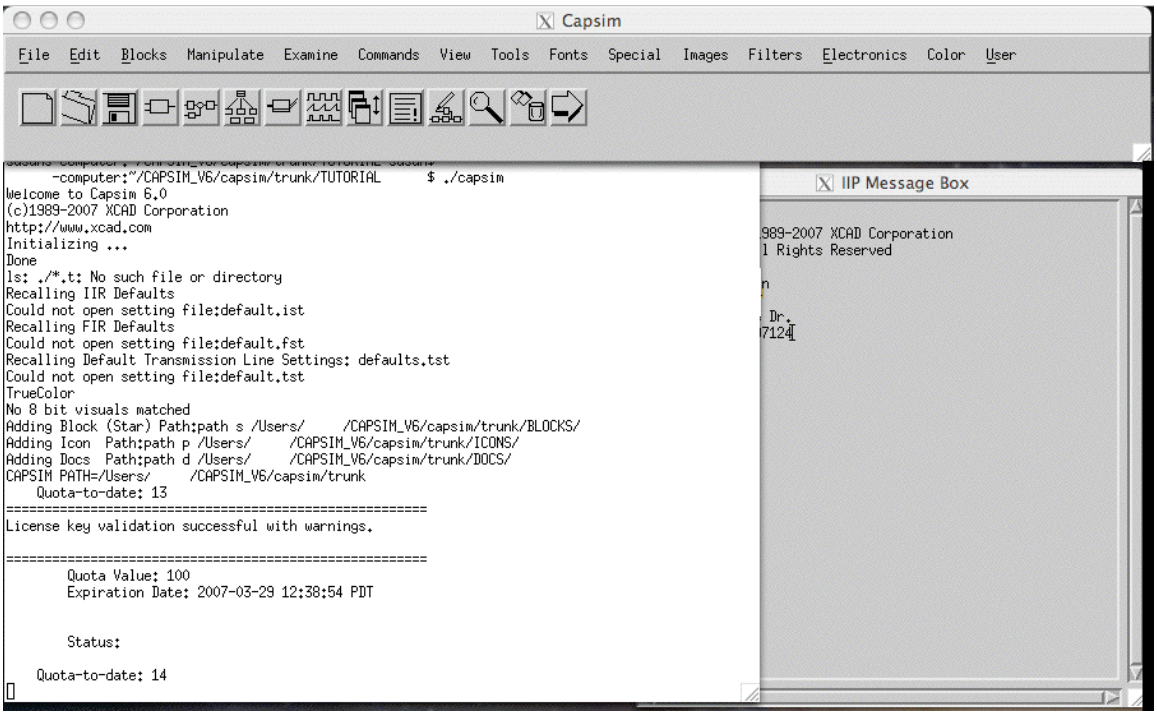


Figure 1 Capsim Pulldown Menu Bar and Console

In the first part of this tutorial, we will place the following blocks on the workspace: an *impulse* block, to generate an impulse; an *iirfil* block, to simulate an IIR digital filter; and a *spectrum* block to plot the impulse response and frequency response of the filter. Later, we will replace the impulse block with a sine wave generator block in order to observe the low pass IIR filter's response to two tones at difference frequencies.

Placing Blocks on the Workspace

To begin, move the cursor to the Blocks menu in the pull-down menu bar. Select Blocks … (See Figure 6). The Capsim block selection box (Block List window) will appear as shown in Figure 2.

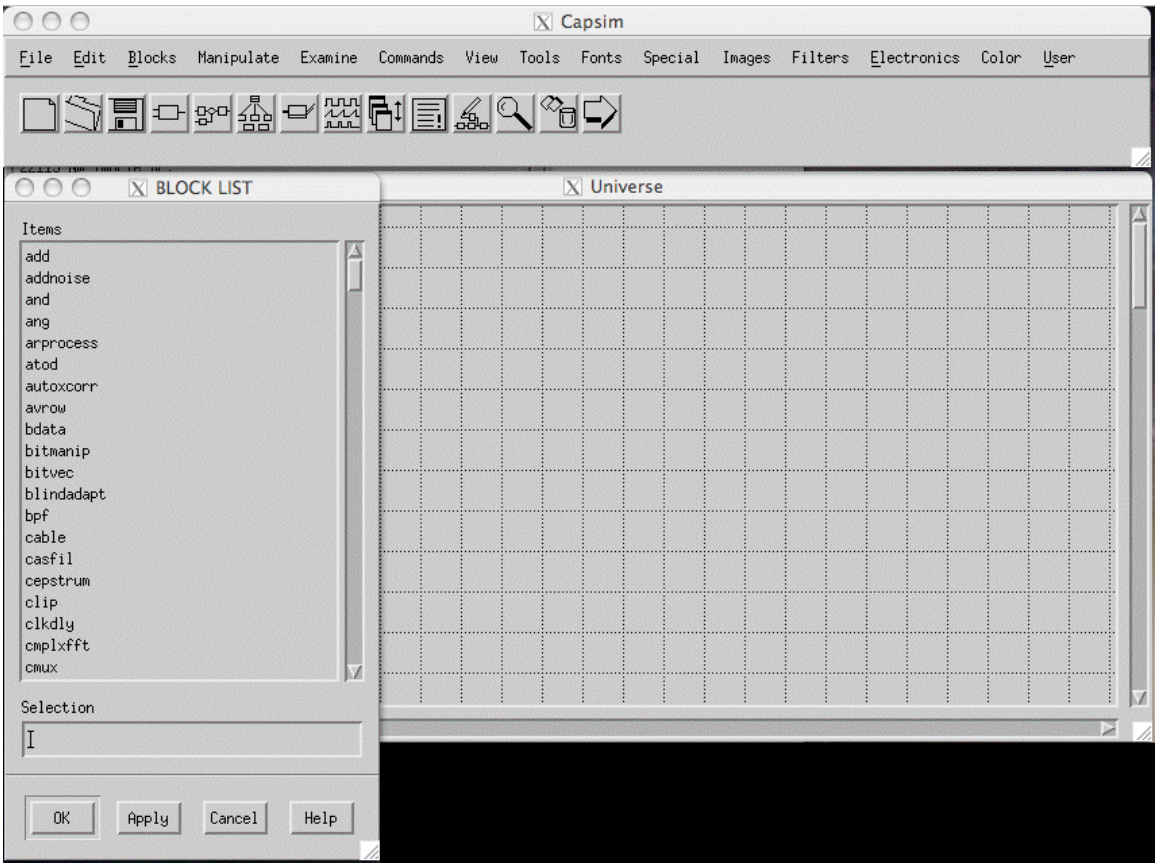


Figure 2 Capsim List of Blocks

Move the Block List window to a convenient location on the screen so that most of the workspace is visible. (You can do this by first placing the mouse on the top bar in the Block List window and then dragging it to the new location with the left mouse button down.) You can also activate the block list by clicking on the Blocks icon in the workspace tool bar . If you wish you may resize the workspace to any size you desire. To place a block on the workspace, find the block in the block list, click on its name and then click on the *Place* button (you can also select it by typing its name in the text field). You can scroll through the block list using the scroll bar. To begin, find the *impulse* block. Click the mouse button to highlight it. (The Selection box should display impulse). Now, click on the Place button. Move the cursor to the workspace. Notice that the cursor has now changed to a hand cursor. Observe the layout in Figure 3. Move the hand (place) cursor to the left of the workspace as shown. Click the mouse button. The impulse block will appear on the workspace. If your paths are setup you will see an impulse icon. Otherwise a block will appear with impulse written in it. Now, get the *iirfil* and *spectrum* blocks from the block list and place them on the workspace as shown in Figure 3. After you have placed the blocks, move the cursor to the Block List window's

cancel button and press the left button to get rid of this window since we don't need it for now.

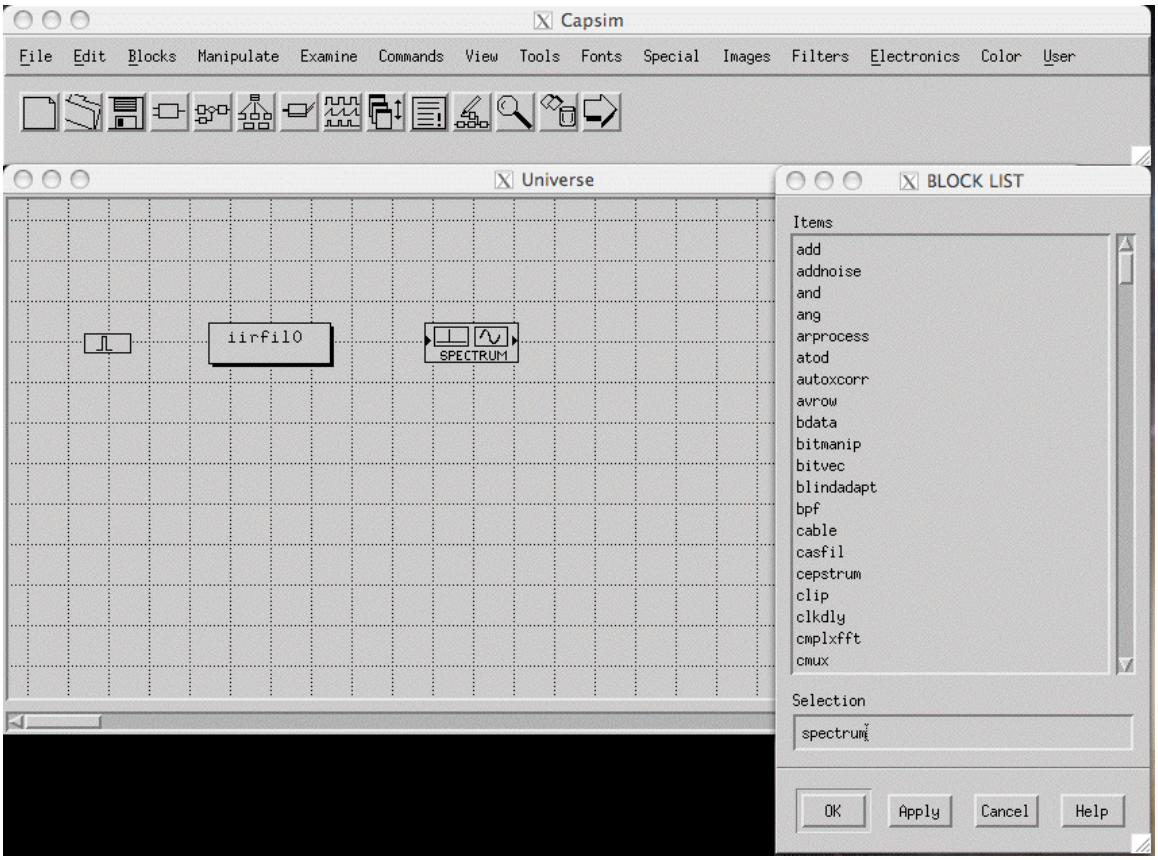


Figure 3 Block layout for impulse, iirfil and spectrum

Connecting Blocks

There are several ways to connect blocks. We will first try the auto connect mode with no fast connect. We will explain fast connect later. To connect the *impulse* block to the *iirfil* block, move the mouse over to the impulse block so that it is highlighted (a box appears around the icon). Now click the left hand mouse button. A popup menu appears as shown in Figure 4. This popup menu allows you to do many things with the block. For now, just move the cursor so that the *Connect an Output* option is highlighted. To select this option, click the right hand mouse button. The popup menu will disappear and the cursor will change to a question mark. The reason for the question mark is that Capsim needs to know which block you want to connect to. Move the cursor over to the *iirfil* block so that it is highlighted. Now click on the left hand mouse button. The impulse block is automatically connected to the *iirfil* block. Repeat the same procedure to connect the *iirfil* block to the *spectrum* block. Your topology should be as shown in Figure 5. We are now ready to run a simulation.

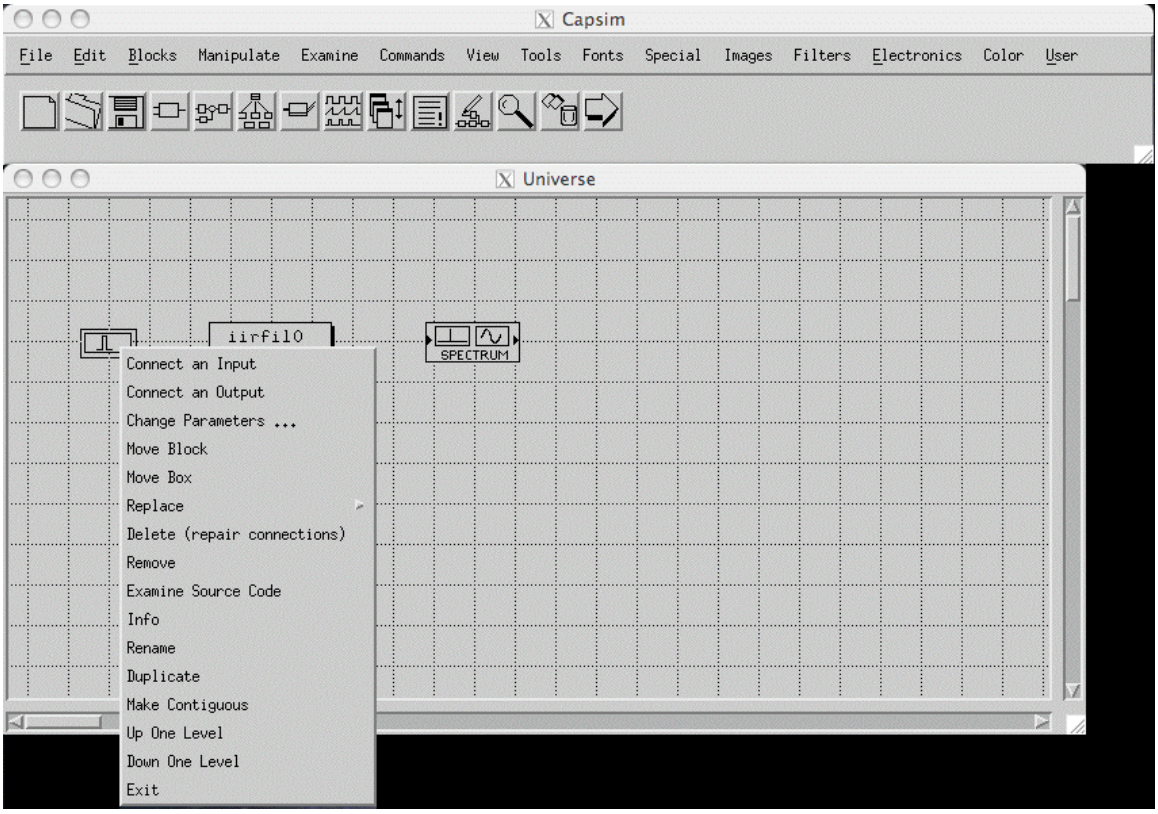


Figure 4 Block Popup Menu

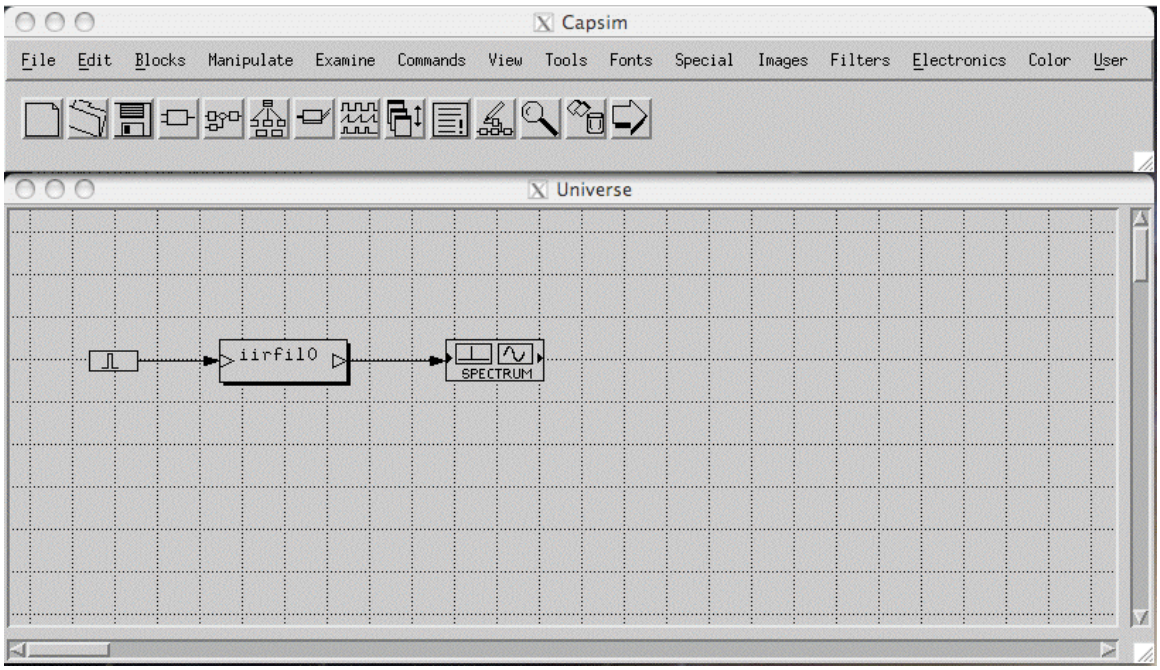


Figure 5 Connected Blocks

Note, when you click on a Block and the block popup menu appears, make sure that you select Exit Menu to exit the menu if you do not make a selection. Otherwise, you may need to click the refresh icon, to be able to select the menu again.

Running a Simulation

Move the cursor to the pull-down menu bar and select Blocks. Place the cursor on the Run option and press the left hand mouse button. See Figure 6. Or you can click on the Run icon in the workspace toolbar . The impulse response in Figure 7 and the frequency response in Figure 8 will appear. The two plots are independent windows that are stacked on top of each other. Move them around for a better view. You can also use the plot shuffle tool to view the plots.

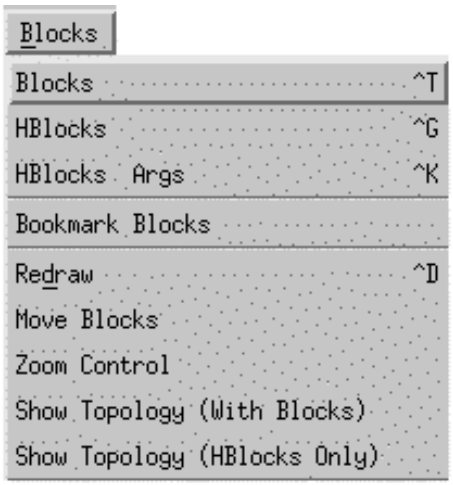


Figure 6 Blocks Menu

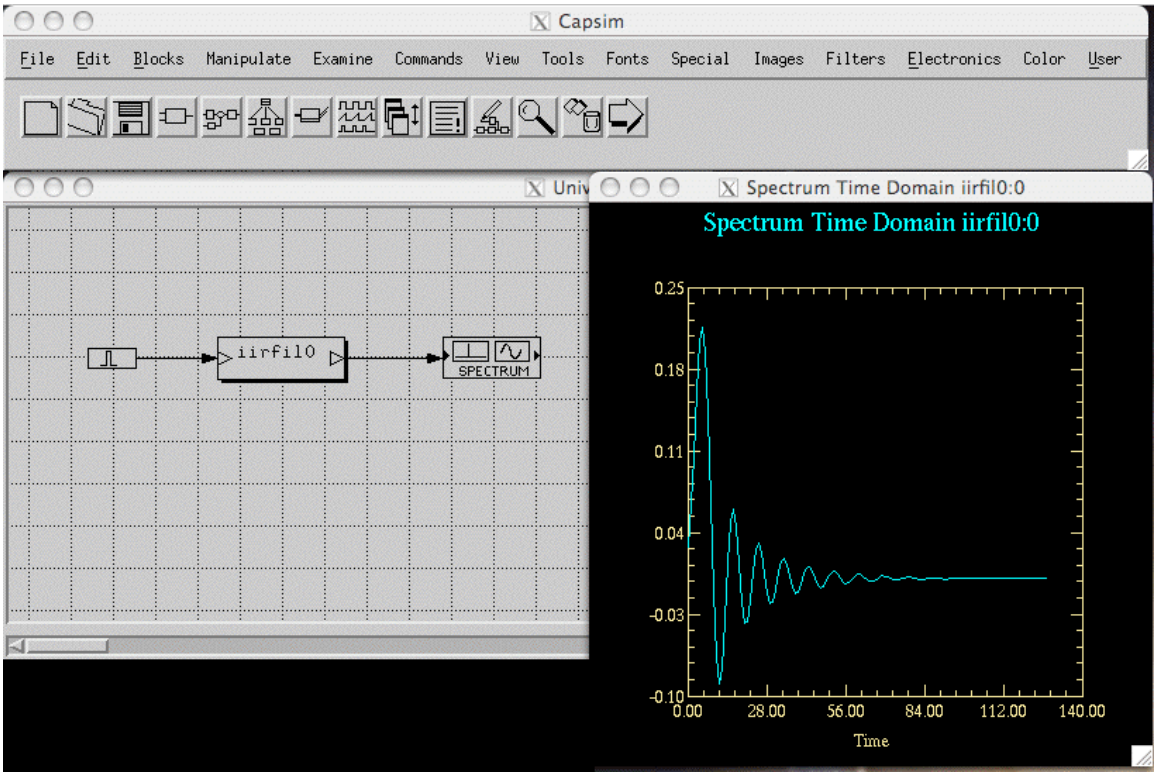


Figure 7 Filter Impulse Response

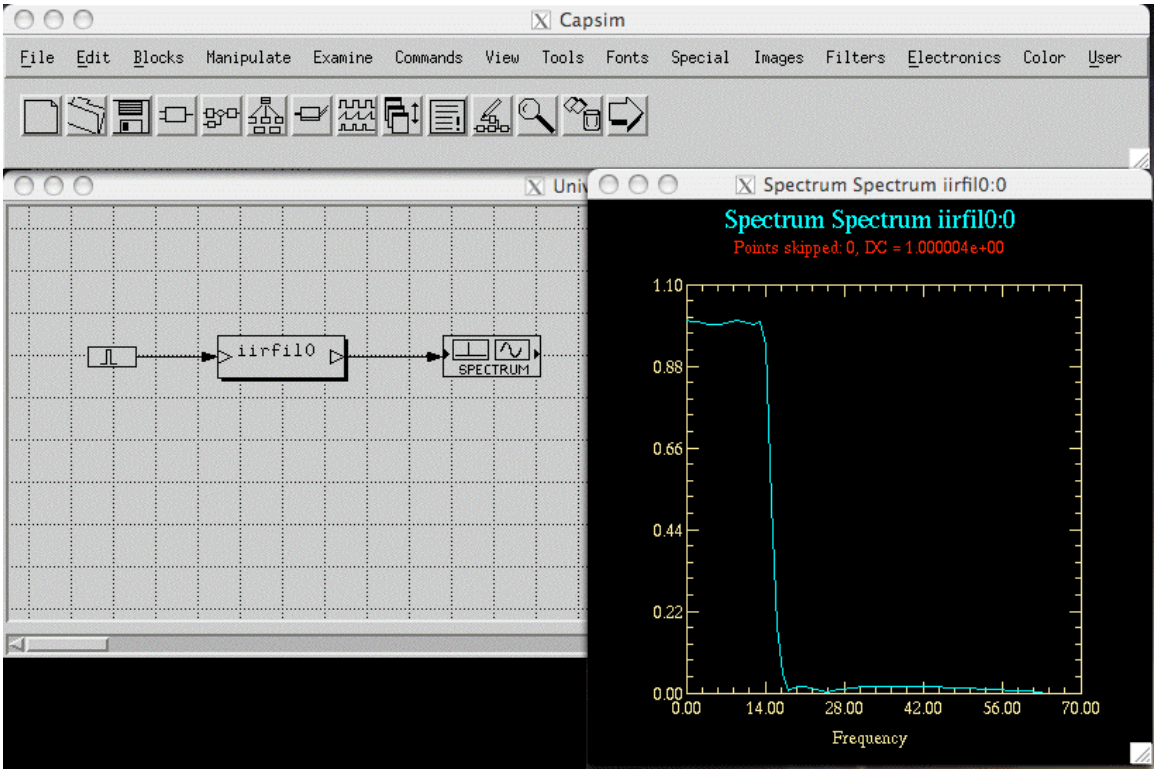


Figure 8 Filter Frequency Response

Place the cursor in the impulse response plot. Click the left mouse button to select the plot. In many DSP applications it is appropriate to display the plots such that the discrete nature of the plot is highlighted. To do this, click on the plot to display the plot popup menu as shown in Figure 9. Select *Curve Style*. The dialog box in Figure 10 appears. Select Bar from the Curve Style radio button group and click Okay. The impulse response will change to a discrete sample plot (bar chart) as shown in Figure 11. Note that you can set all curves to draw bar charts by default. More on this later (Use save plot layout in File pulldown menu and load it again or store it as default).

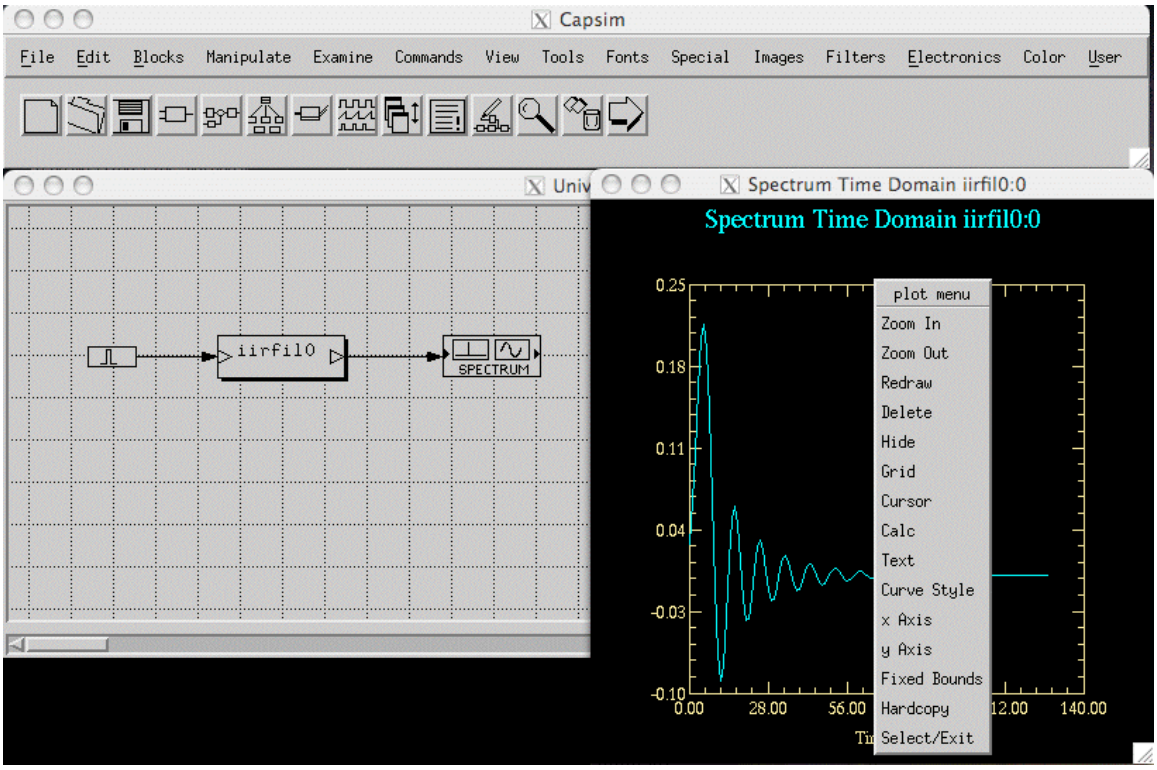


Figure 9 Plot Popup Menu

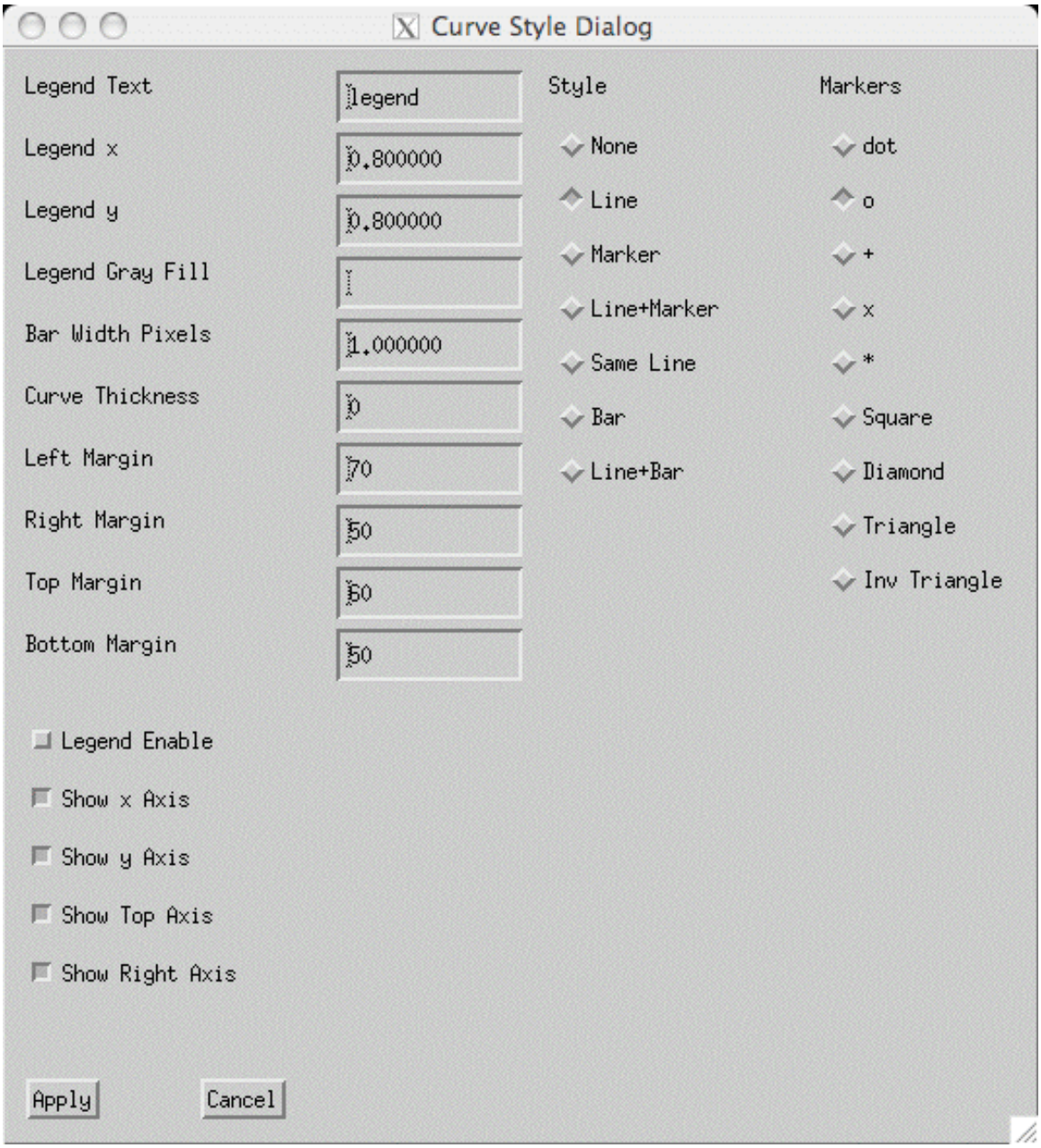


Figure 10 Plot Style Dialog

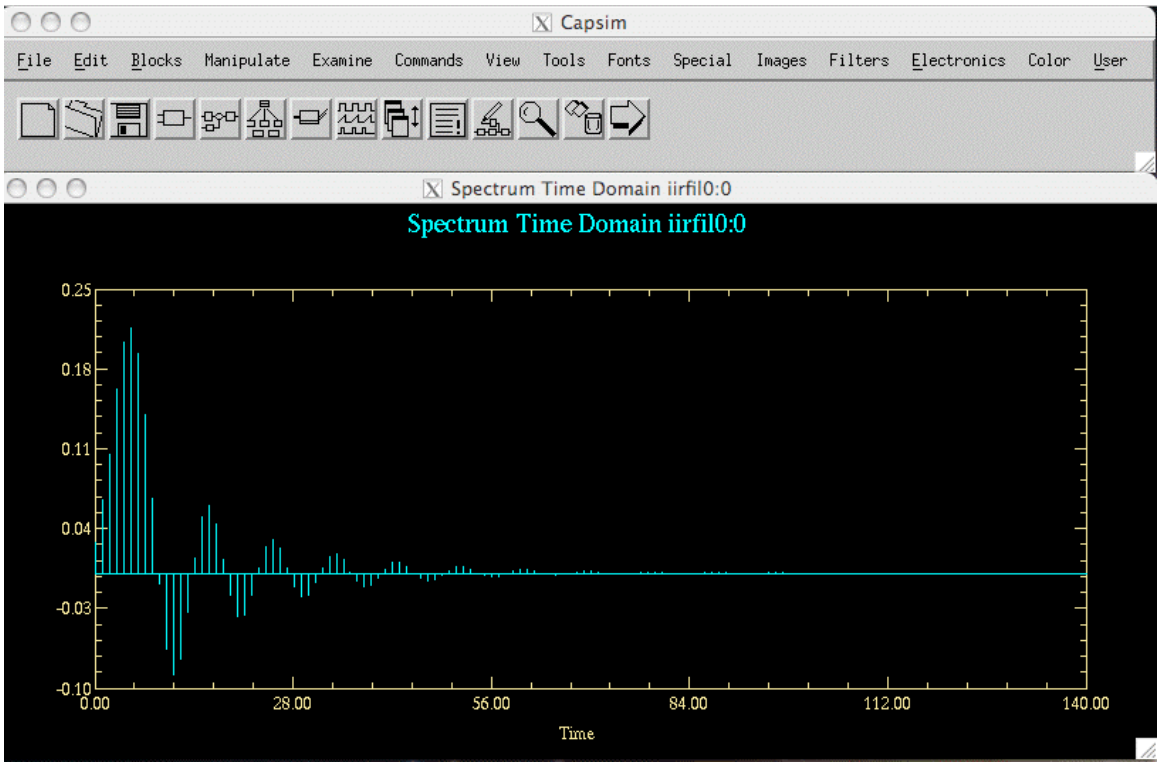


Figure 11 Impulse Response with Bars to Highlight Discrete Samples

Another option is to display the frequency response in dB's. To do this select the frequency domain plot by clicking in the plot area and select *Select/Exit* (or *Redraw*) to make the plot the current selected plot. Next select "Create DB New Curve" from the Plot menu. See Figure 12. A new plot with the magnitude in DB will appear as shown in Figure 13. (You can also set the *spectrum* block to produce magnitude in dB).

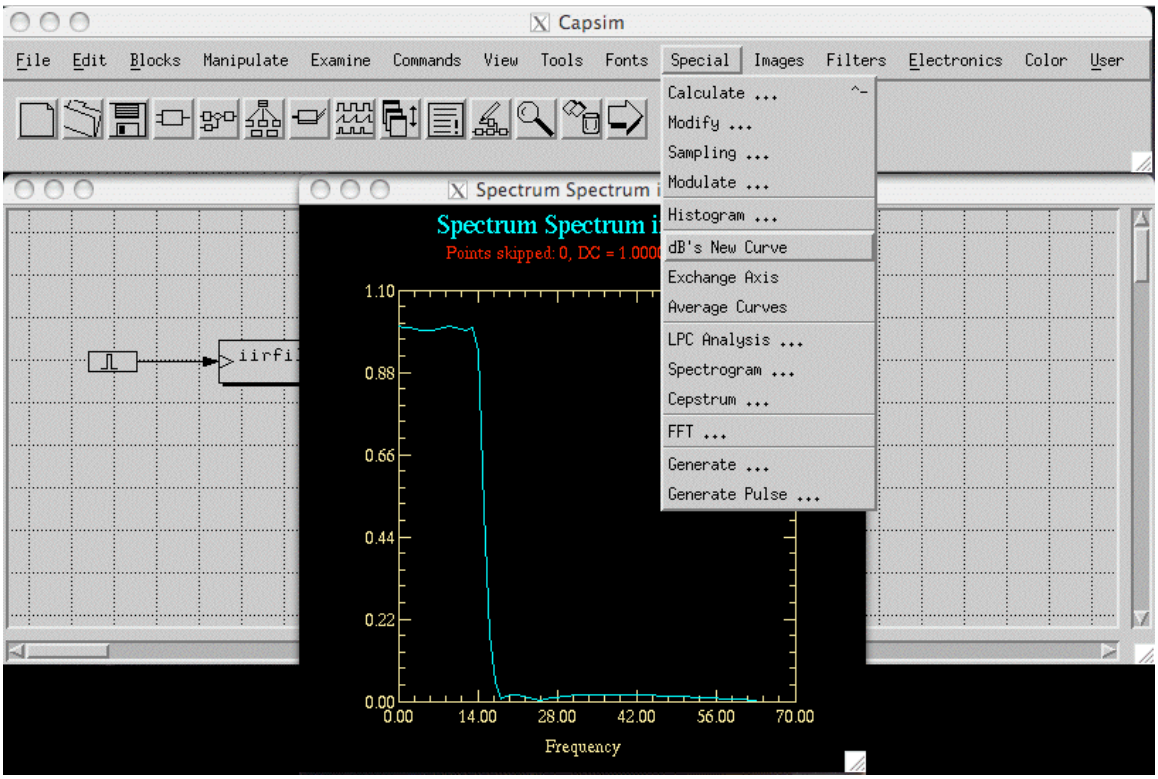


Figure 12 Create dB Plot Menu Item

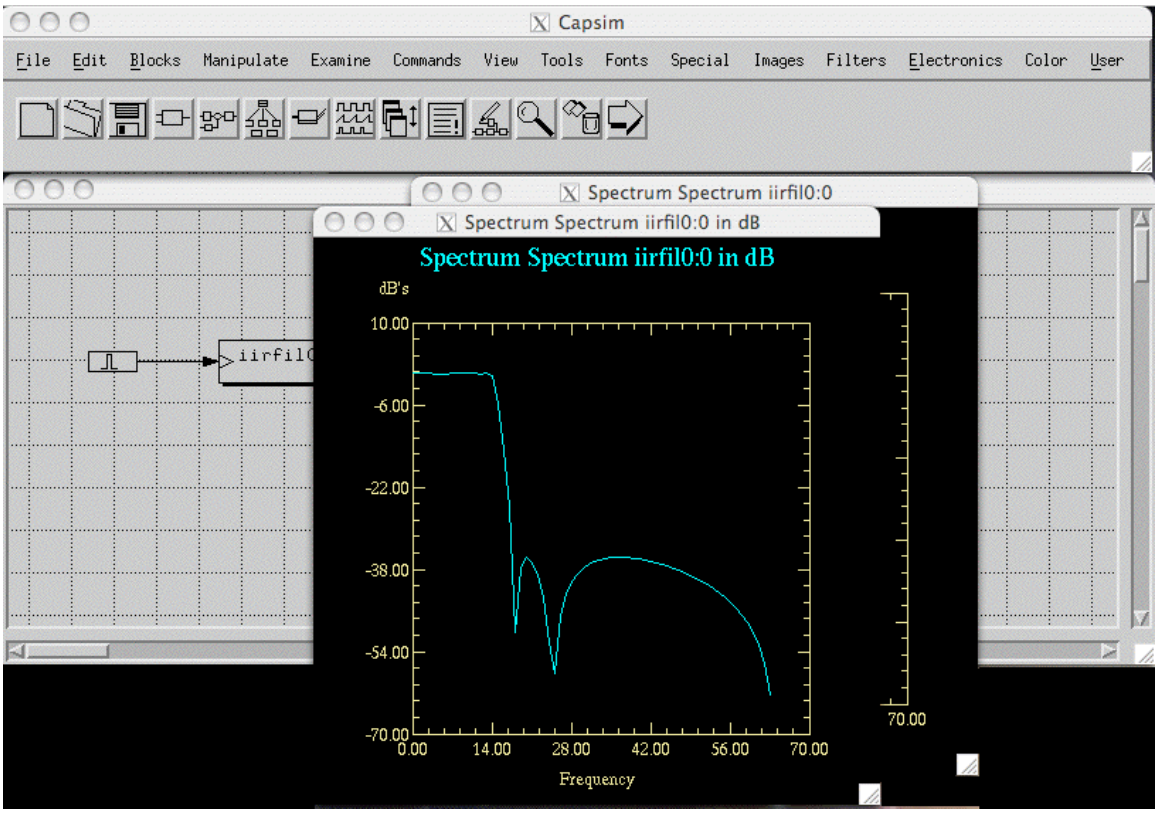


Figure 13 Frequency Response in dB's

Errors During a Run or Editing

Capsim will inform you of errors through the console window. Read the message and correct the error. At this stage of the game it is probably better to start all over again.

Manipulating Plots

You can do many things to the plots such as zoom in and zoom out, change the axis and title labels, change the plotting style to bar charts, etc. Check out the options available for each plot. For now, we are interested in deleting the two plots we just created. Select the edit menu from the pull-down menu. Select "Delete All Plots". The two plots will disappear. Note that there is no warning. In Capsim, you will usually perform a lot of simulations comparing results, changing parameters and running simulations again and again. You will appreciate the no warning! When deleting ,multiple plots a warning is given. You can use the delete plots tool from the tool bar.

Replacing Blocks

In the following sequence, we will replace the *impulse* block with the *sine* block so that the response of the low pass filter to two tones at different frequencies can be simulated. One tone will be in the filter pass band and the other will be in the filter stop band. The responses will be compared using the group plot facility.

To replace the *impulse* block with a *sine* block, move the cursor to the impulse *block* so that it is highlighted. Click the left mouse button. The block popup menu will appear as before. Move the cursor to the Replace option. Place it over the arrow (->). A popup menu appears as shown in Figure 14. Move the cursor horizontally so that it highlights the Blocks option. Now press the mouse button. The Block List window will appear again. Move it over to a convenient location. Now, scroll through the block list and find the *sine* block. Click on it. Then click the Place button. The *impulse* block is replaced with the *sine* block as shown in Figure 15. After the block has been replaced cancel the Block List window by clicking on the Cancel button.

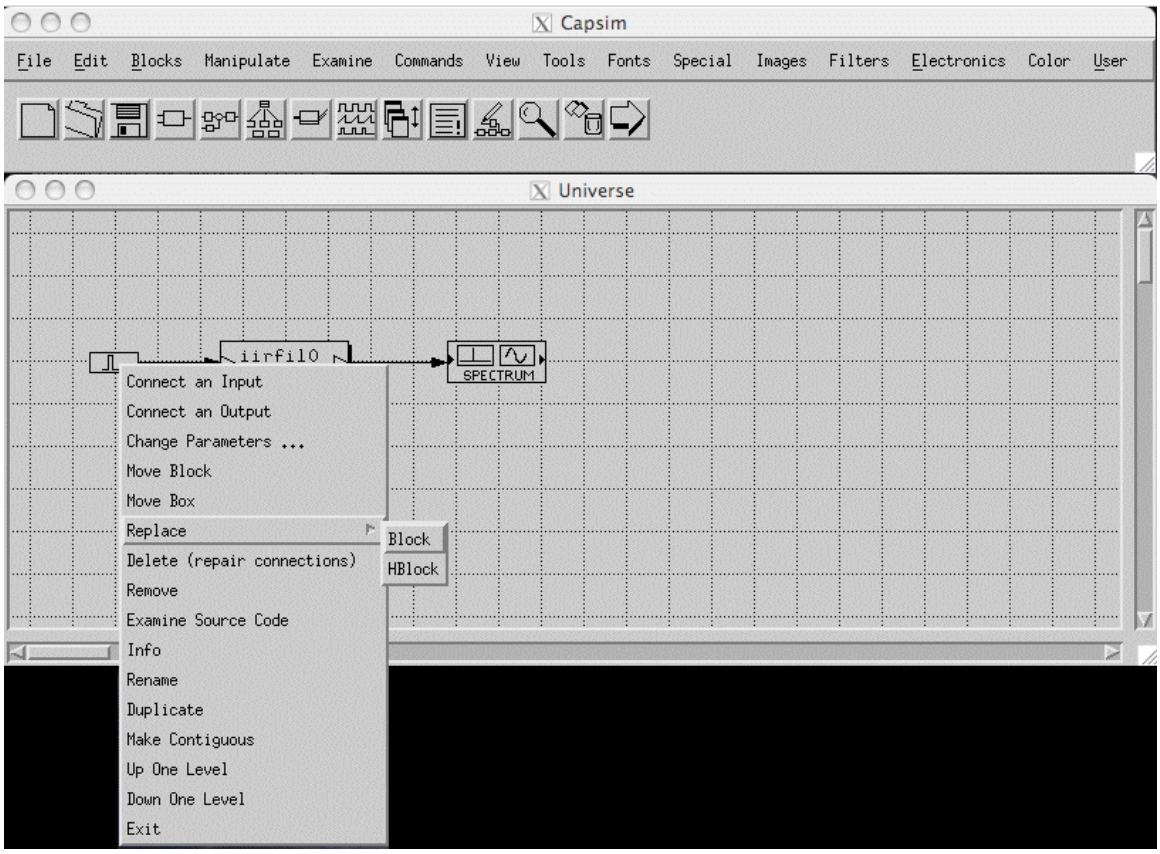


Figure 14 Replace Sub-Menu

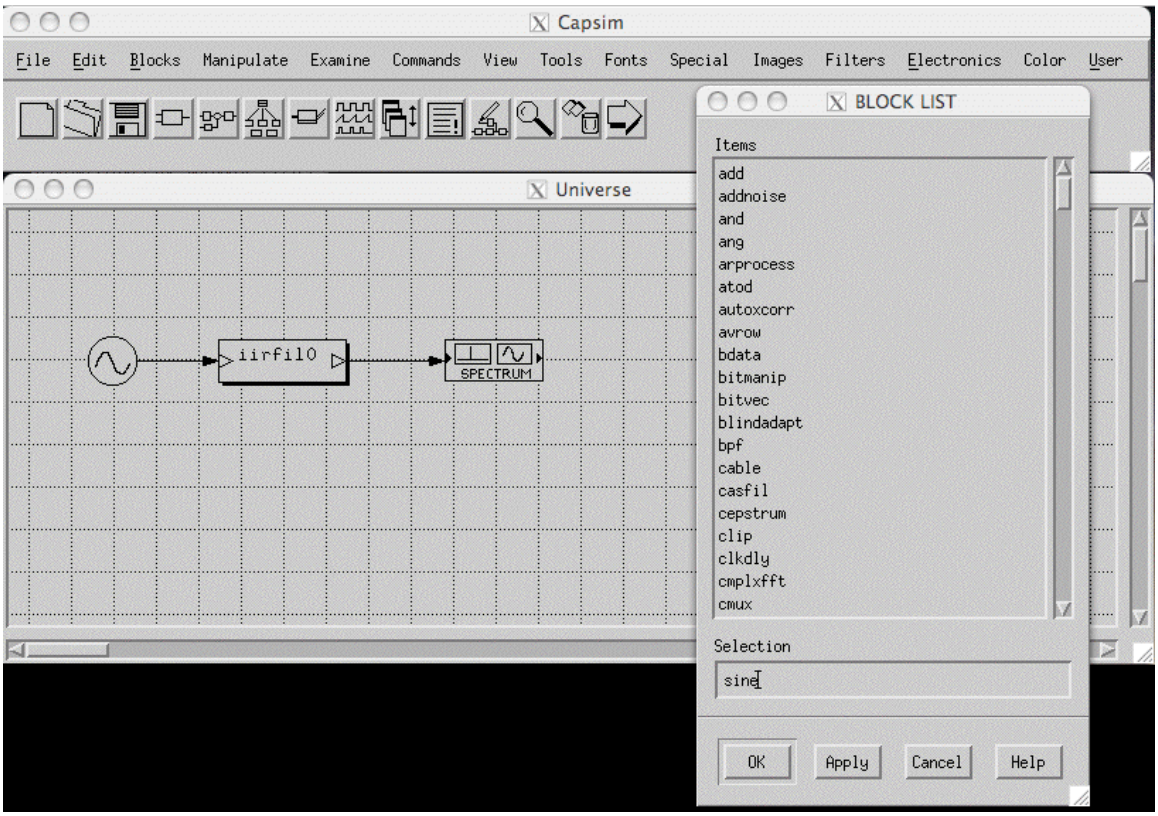


Figure 15 Impulse Block Replaced by Sine Block

Run a simulation by the procedure described before. Two plots should appear. In one plot, the time domain response appears. In the second plot the frequency domain plot appears. The IIR filter implements a low pass filter with 35 dB attenuation at 4.4 kHz. In the next section we will investigate the response of the filter to two sine waves at 1000 and 5000 Hz. Note that the sampling rate is 32000 Hz for both the *sine* and *iirfil* blocks. You may also change the sampling rate parameter in the spectrum block to 32000 Hz so that the x axis displays Hz.

Block Parameters

The frequency of the *sine* block was 1000 Hz. The number of points generated was 128. To see what the parameters of the *sine* block are, move the cursor to the *sine* block and click the mouse. The popup menu will appear. Move the cursor to the *Change Parameters* menu item and click the left mouse button. The *sine* block parameter box

will popup. The definitions and the values of the sine block parameters are displayed. This dialog box is useful when multiple simulations are run where a parameter is changed between simulations.

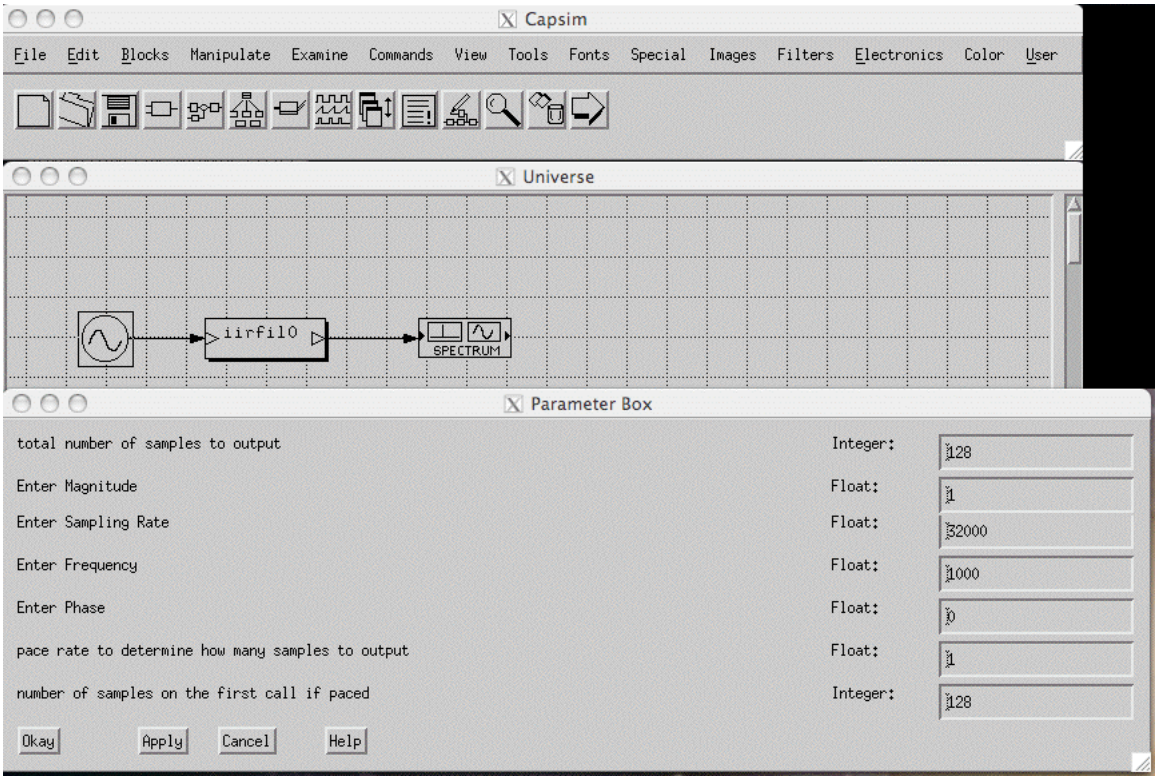


Figure 16 *sine* Block Parameters

Set the *total number of samples to output* to 1050. Click on OK. Run a simulation. Now delete the frequency response curve. Use the delete from the plot popup menu. The idea is to keep the time domain plot. Now display the parameters of the *sine* wave again as in Figure 16. Change the *Enter Frequency* parameter to 5000 Hz. Click on OK. Run the simulation again. Delete the frequency domain plot. The idea is to compare the filter response to a 1000 Hz tone to a 5000 Hz tone in the time domain. To do this, select *Group Plots* from the *View* menu in the menu bar. See Figure 17. The Group Plot Dialog box appreas. Select Time Domain Plots. Click on the Group Plots check box. Click on Okay. A new plot is generated in which you can compare the response of the filter to the two tones. See Figure 19.

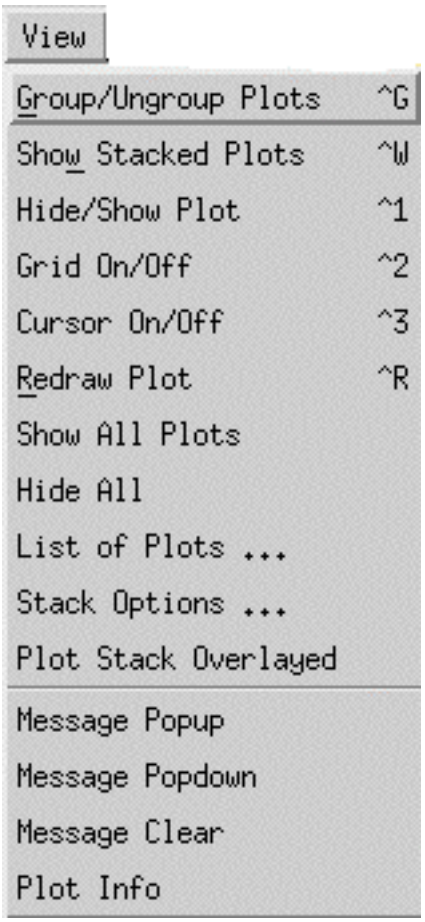


Figure 17 Plot Menu

Figure 18 Group Plot Dialog

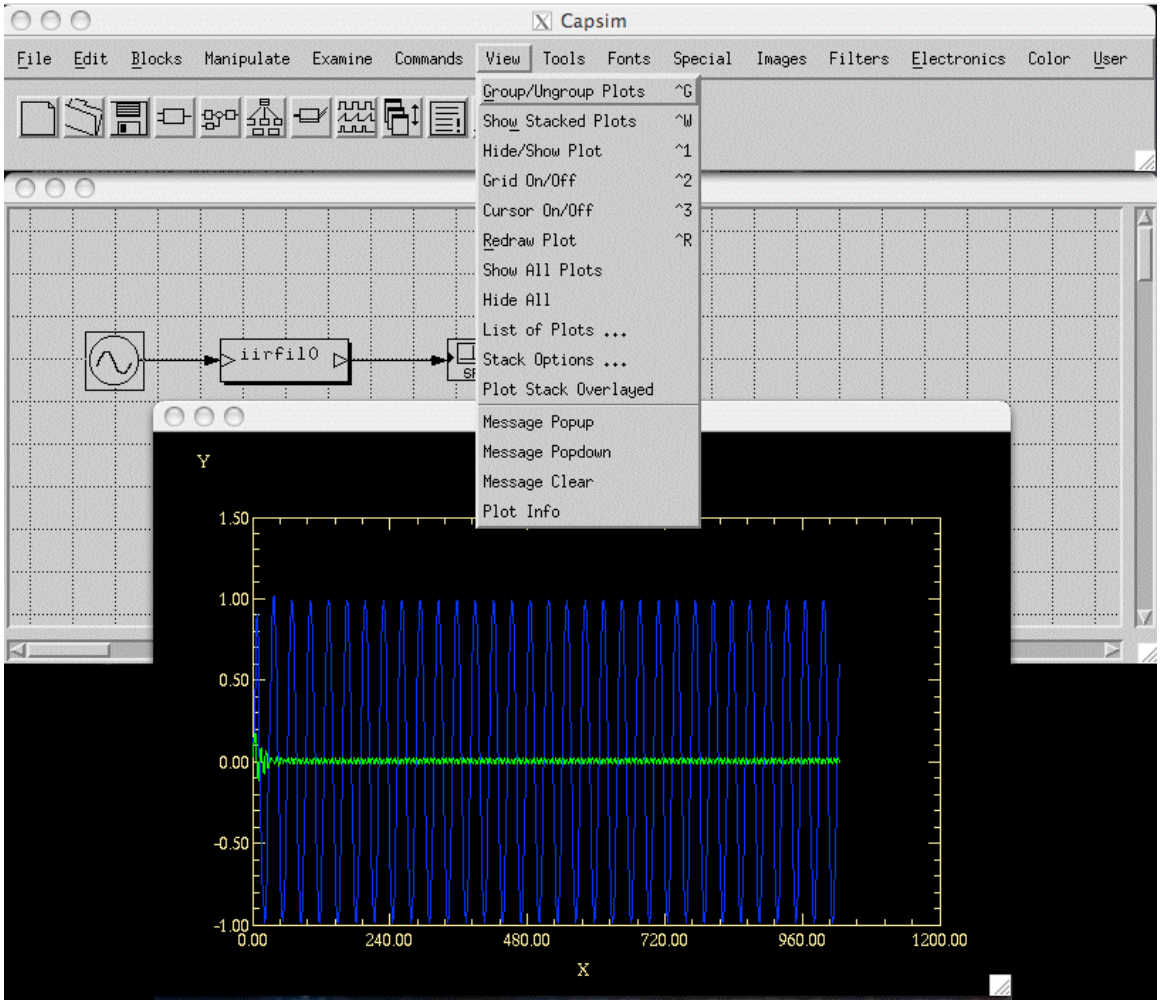


Figure 19 Comparison of Filter Response to 1000 Hz and 5000 Hz Sine Wave

Saving the Topology

It is time to save your work. Select the File option from the Capsim pulldown menu. The File pulldown menu should appear. Select the "Save Topology As ... " option. A dialog box should popup requesting the name of the topology. Type in *sinetest* and then click on the OK button (you can name it whatever you like). The topology has now been saved. If you make any more modifications and wish to save the changes, just use the Save

Topology option in the File pulldown menu, or use the save icon in the topology toolbar. Make sure to save it in the current directory (WORK).

To exit Capsim, select the Exit menu item from the Capsim File pulldown menu. NOTE: THERE IS NO WARNING WHEN YOU EXIT CAPSIM. The reason for this is that in the Capsim model for design and simulation you enter and exit Capsim often. Changing code, rebuilding etc. After a while the nagging warning dialogs become a pain. So save your work often and be a power user!

Creating an Hblock (Hierarchical Block)

Capsim is a hierarchical block diagram simulator. Complex blocks can be made of existing blocks. These blocks are called HBlocks. HBlocks can contain other HBlocks as well as blocks. In this tutorial, we will construct a staircase generator HBlock from three existing blocks. We will implement a pure integrator and drive it with periodic impulses.

Startup Capsim again from scratch (exit from the current session and start Capsim from the command line). Select New from the File menu. Place the *add*, *delay* and *node* blocks on the workspace as shown in Figure 20. The *delay* block is indicated by the z^{-N} icon and the *node* block by the small square. The *delay* block delays its input samples by an amount specified by its parameter. The default is 1 sample delay which is what is needed in the staircase HBlock. Thus, we don't need to change its parameter. The *node* block is useful when more than one buffer needs to be connected to a block's output. That is, the *node* block takes the samples from its input buffer and forks them out to as many output buffers that are connected.

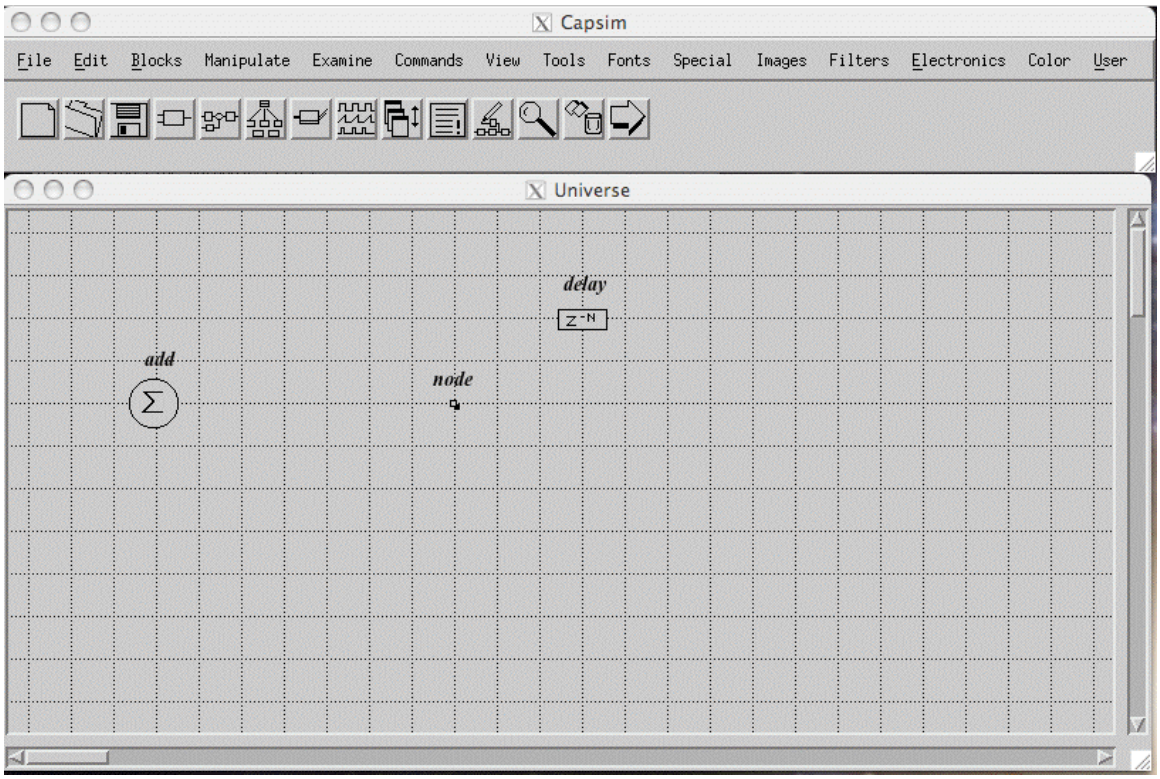


Figure 20 HBlock Bock Layout

HBlock Input Terminal

An HBlock needs to receive and send samples to the blocks that are connected to it. The mechanism through which samples are input and outputted from an HBlock are the input/output terminals.

We need to connect an input terminal to the *add* block as shown in Figure 22. To do this, click on the *add* block. Select “Connect an Input” from the popup menu (Figure 21). When the cursor changes to "?", move it to the **left** of the *add* block and click on an empty space in the workspace. The terminal connection in Figure 22 results.

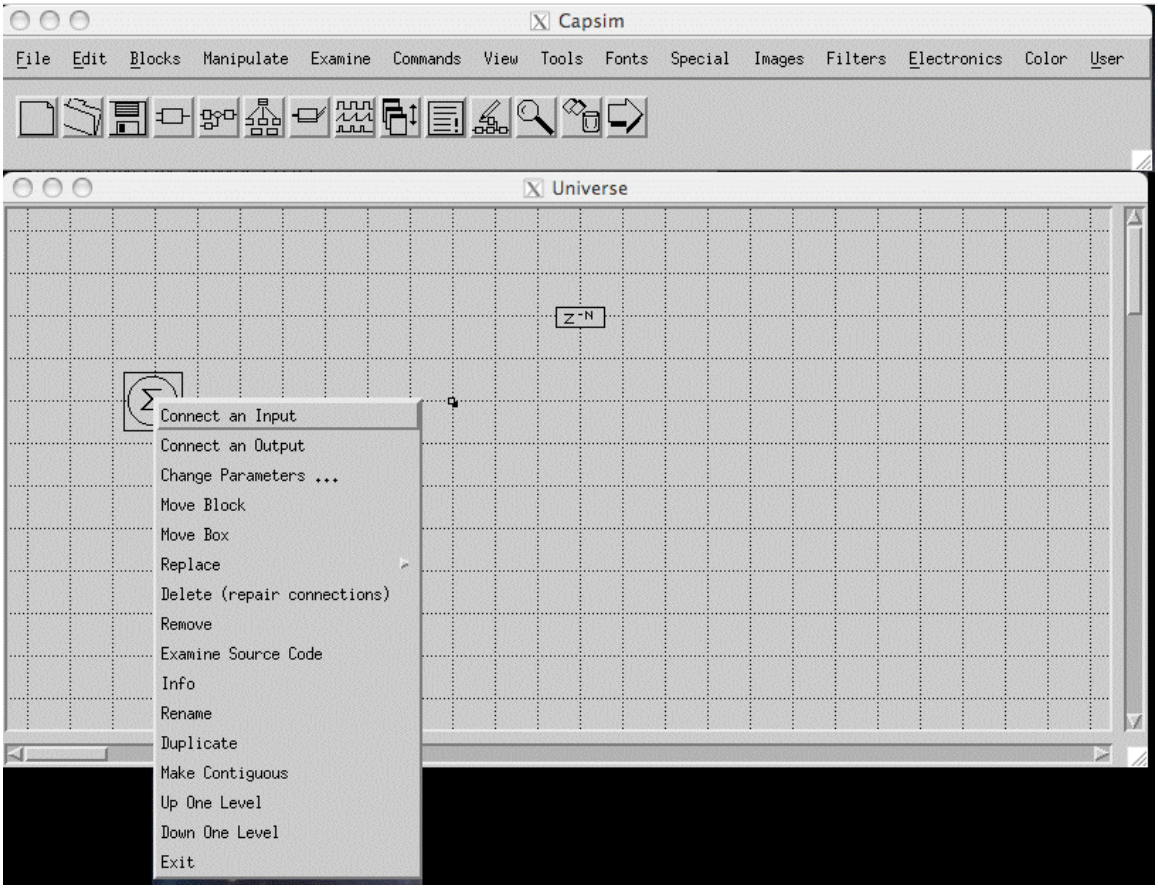


Figure 21 Connecting an Input Terminal (Select Workspace)

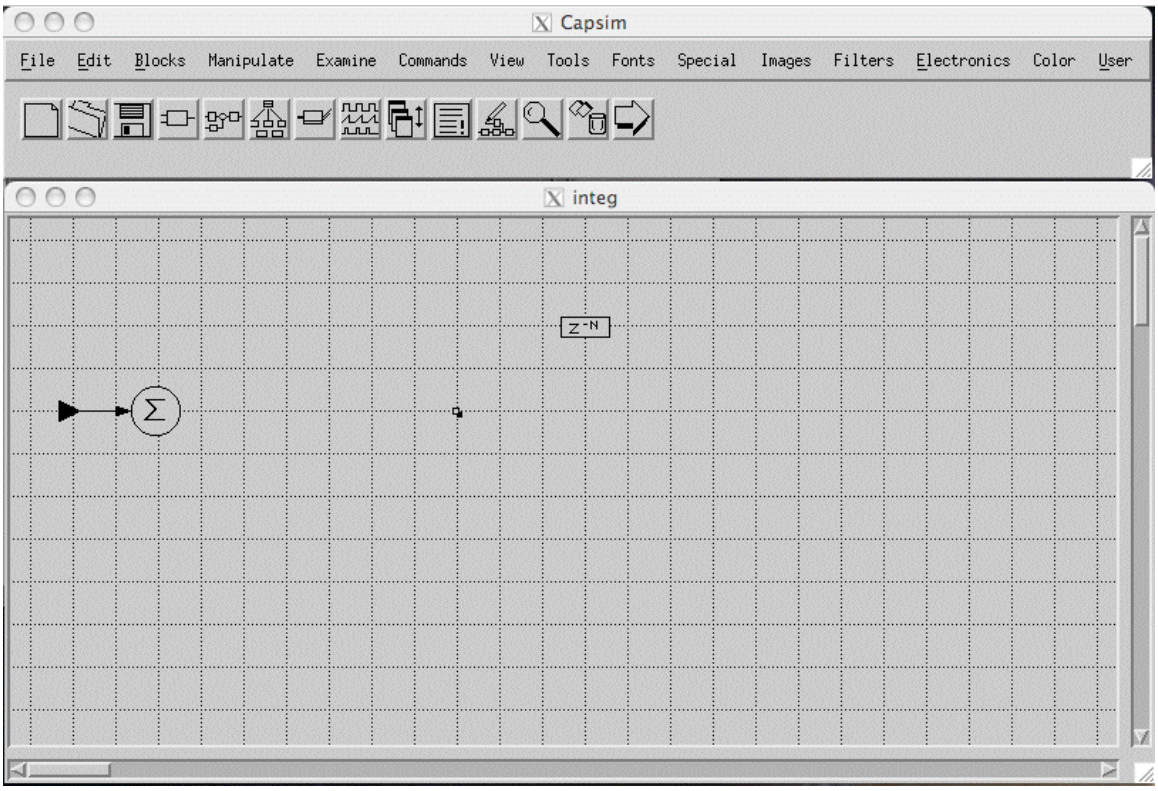


Figure 22 Input Terminal Connected

Next connect the *add* block to the *node* block. See Figure 23. Connect the *node* block to the *delay* block as shown in Figure 24. Connect the *delay* block to the *add* block as shown in Figure 25. Finally connect an output terminal to the *node* block. To do this, click on the *node* block. Then select "Connect an Output". Next click on an empty point in the workspace to the right of the node block. An output terminal will appear as shown in Figure 26.

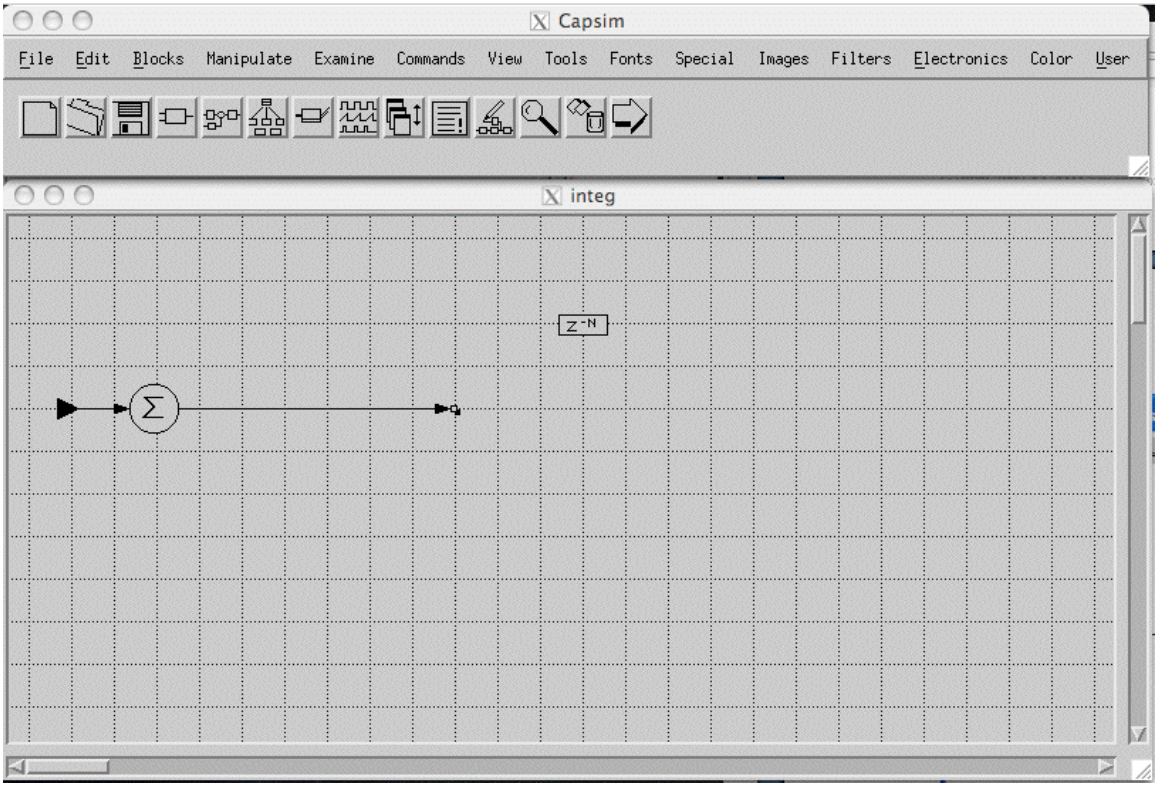


Figure 23 Connect add Block to node Block

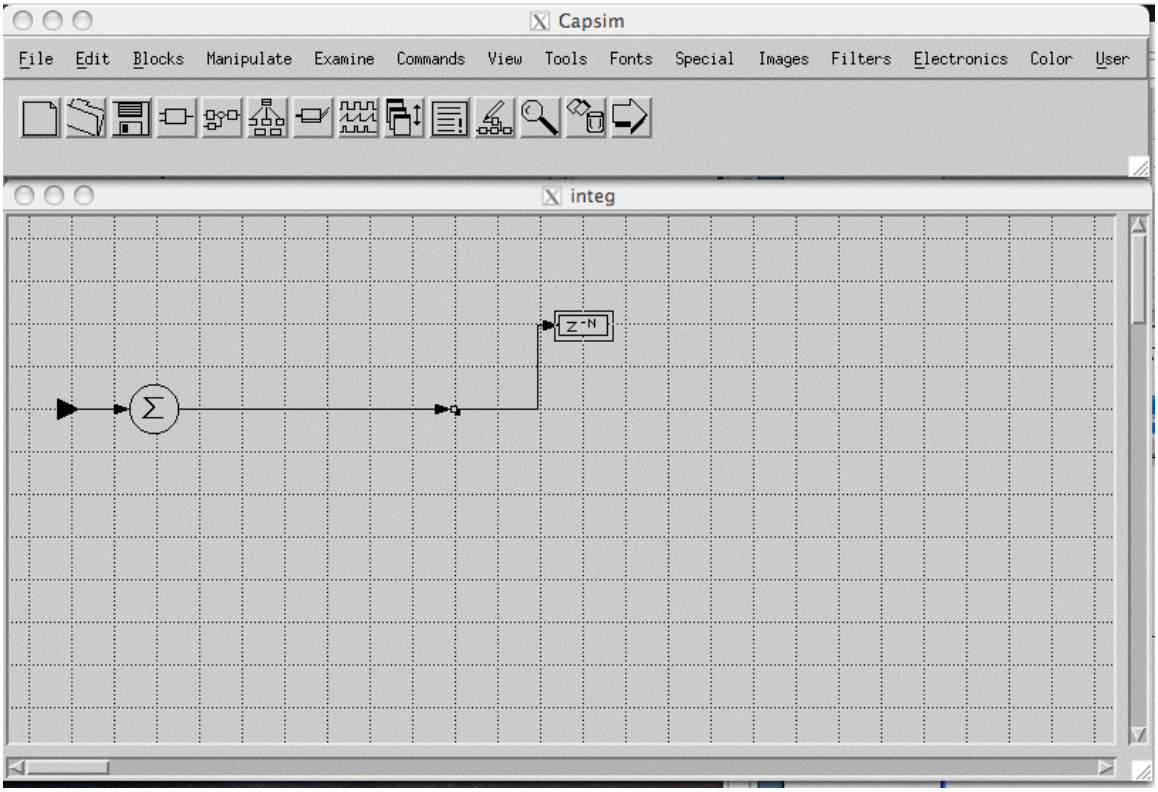


Figure 24 Connect node to delay

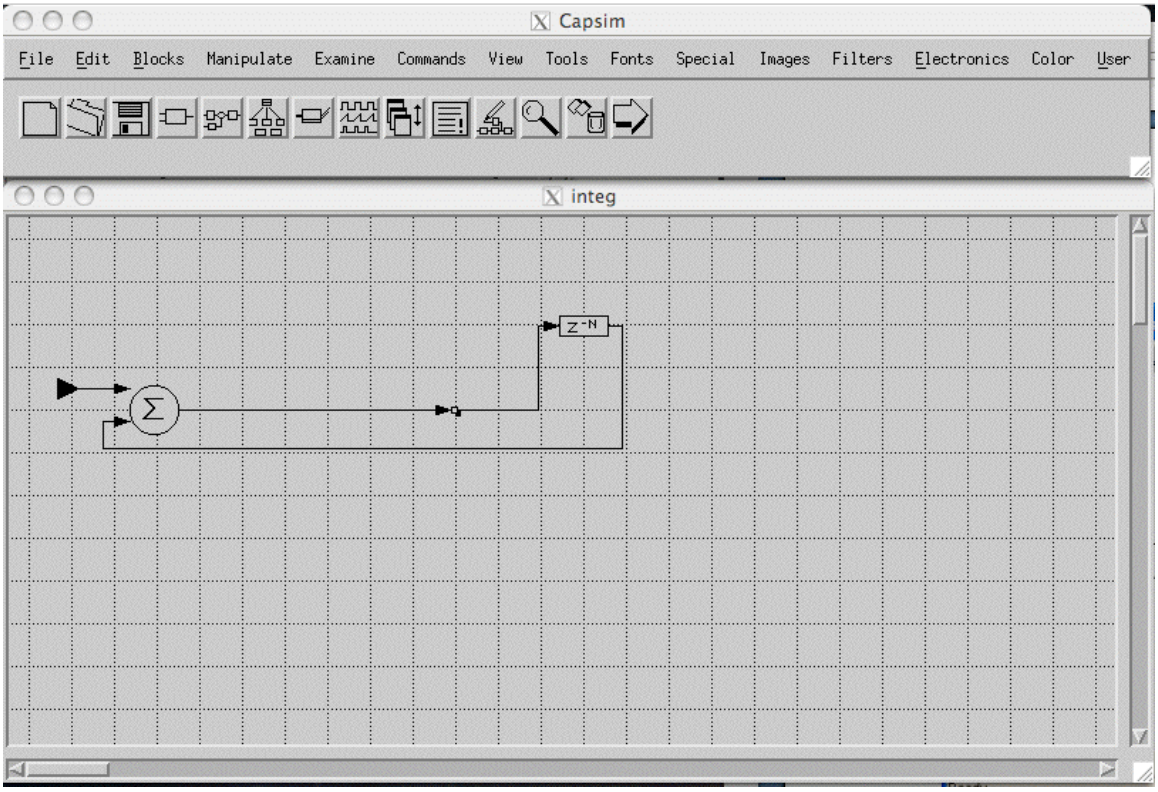


Figure 25 Connect delay to add Block

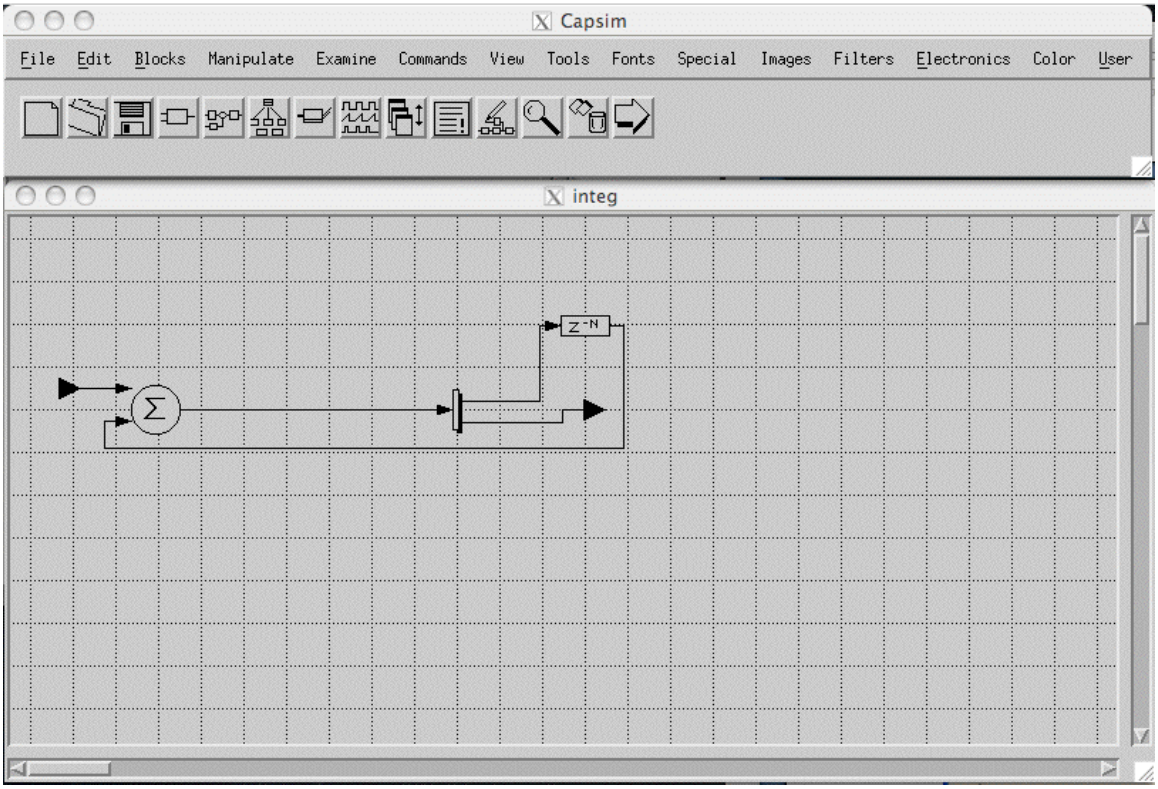


Figure 26 Connect node to "Workspace" for Output Terminal

Now that you have created the HBlock, you must save it. Select the “Save Topology As ... “ from the Capsim File pulldown menu. When prompted for a topology name type *integ* in the text field. Click on OK.

Manual Connections

This section is optional. On most occasions Capsim’s automatic connections are the best choice for connecting blocks. However, especially when feed back is involved, manual connections are necessary. The procedure is to let Capsim make the connection. Then click on the connection and select *Redraw (same numbers)* from the popup menu. See Figure 27.

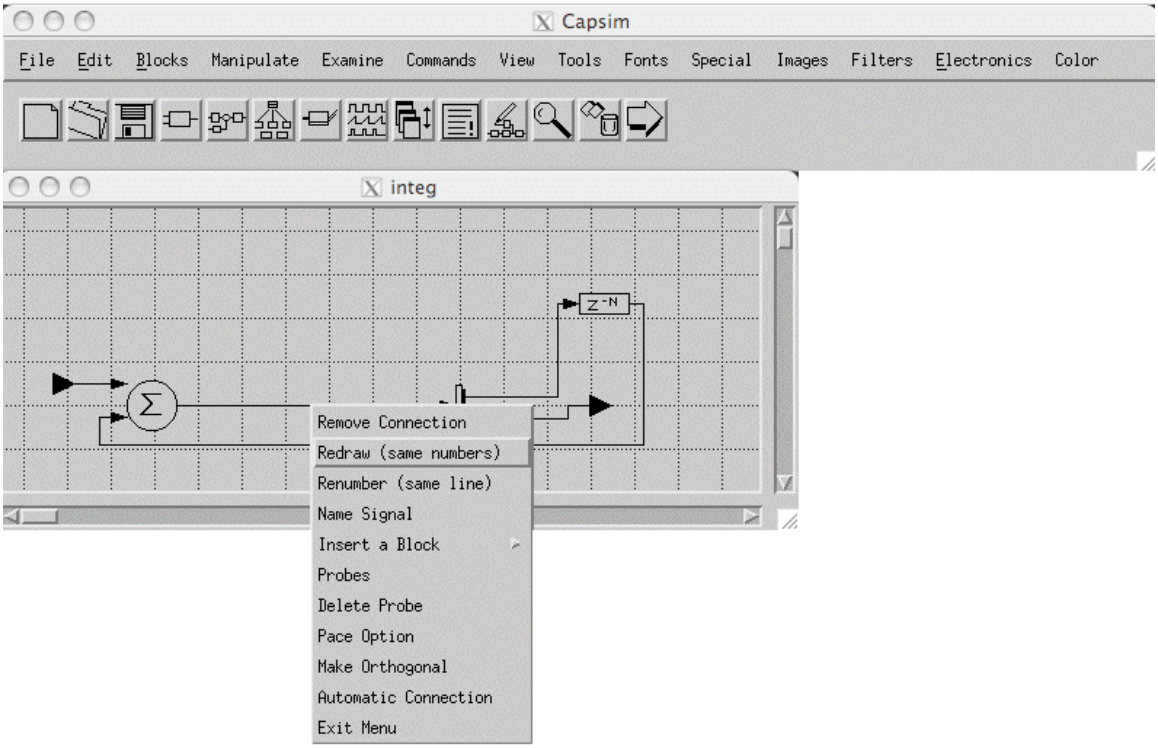
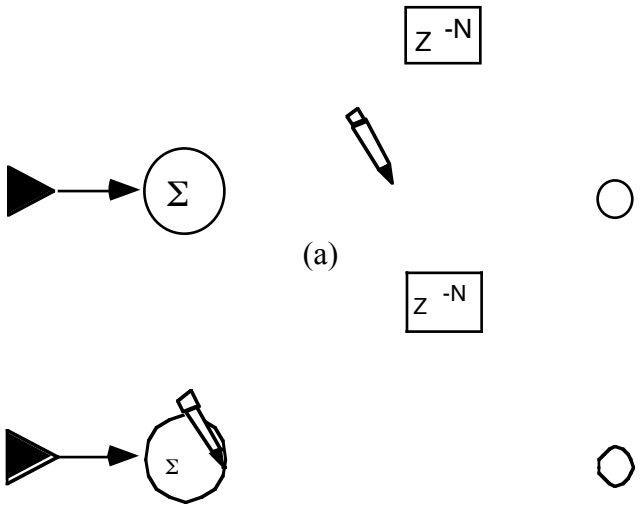


Figure 27 Redrawing a Connection

The pencil cursor indicates that you are in manual connect mode. Move the pencil cursor to the *add* block as shown in Figure 28(b). Click the mouse button once (do not hold down the mouse button). Now move the pencil cursor. A rubber band will follow the cursor. See Figure 28(c). Move the pencil to the *node* block as shown in Figure 28(d). At this point, double click the mouse button. A line will be drawn with an arrow indicating that a connection has been made. If there are any residuals on the screen, click on Redraw in the option box.



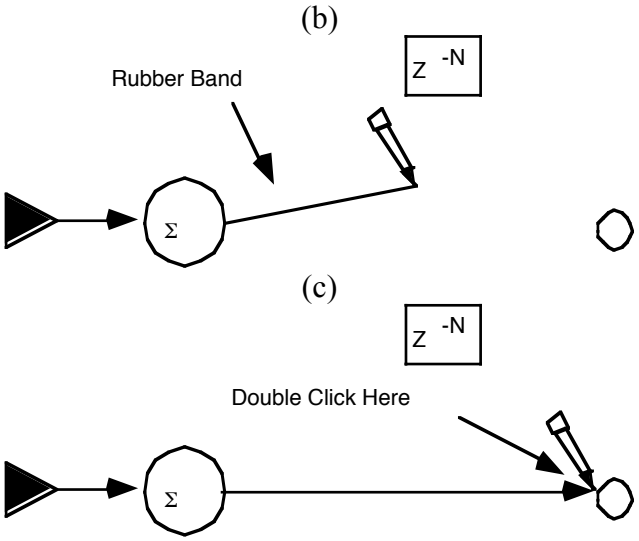
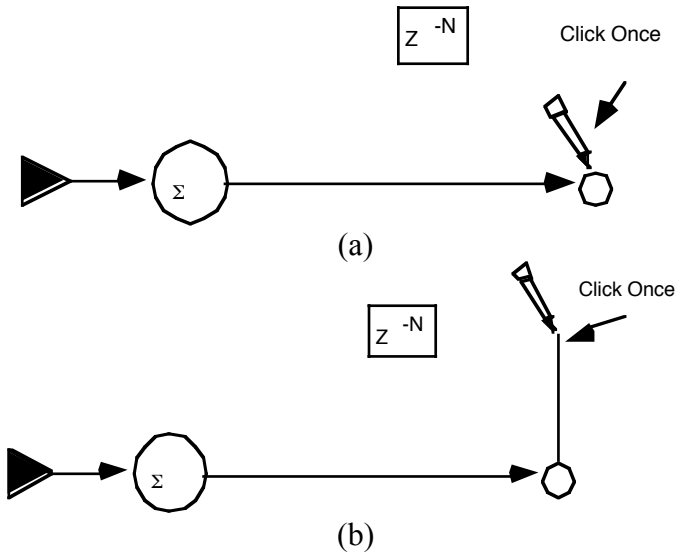
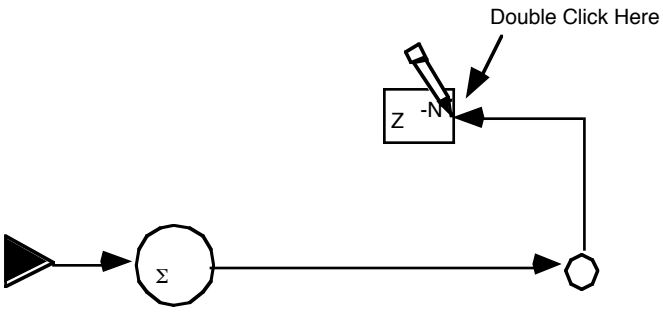


Figure 28 Manual Redraw

To make the connection from the *node* to the *delay* block, first move the delay block to the right as shown. Then click on the connection between the *node* block and *delay* block and select *Redraw*. The cursor will change to a pencil indicating that we are ready to make a manual connection. Place the pencil at the *node* block as shown in Figure 29(a). Click once. The rubber band should appear. Now move the pencil to the location shown in Figure 29(b). Click once. Move the pencil to the delay block as shown in Figure 29(c) and double click. A connection will be made.





(c)

Figure 29 Manual Redraw Feedback

Follow the same procedure to make the connection from the *delay* block to the *add* block. The final block diagram is shown in Figure 30.

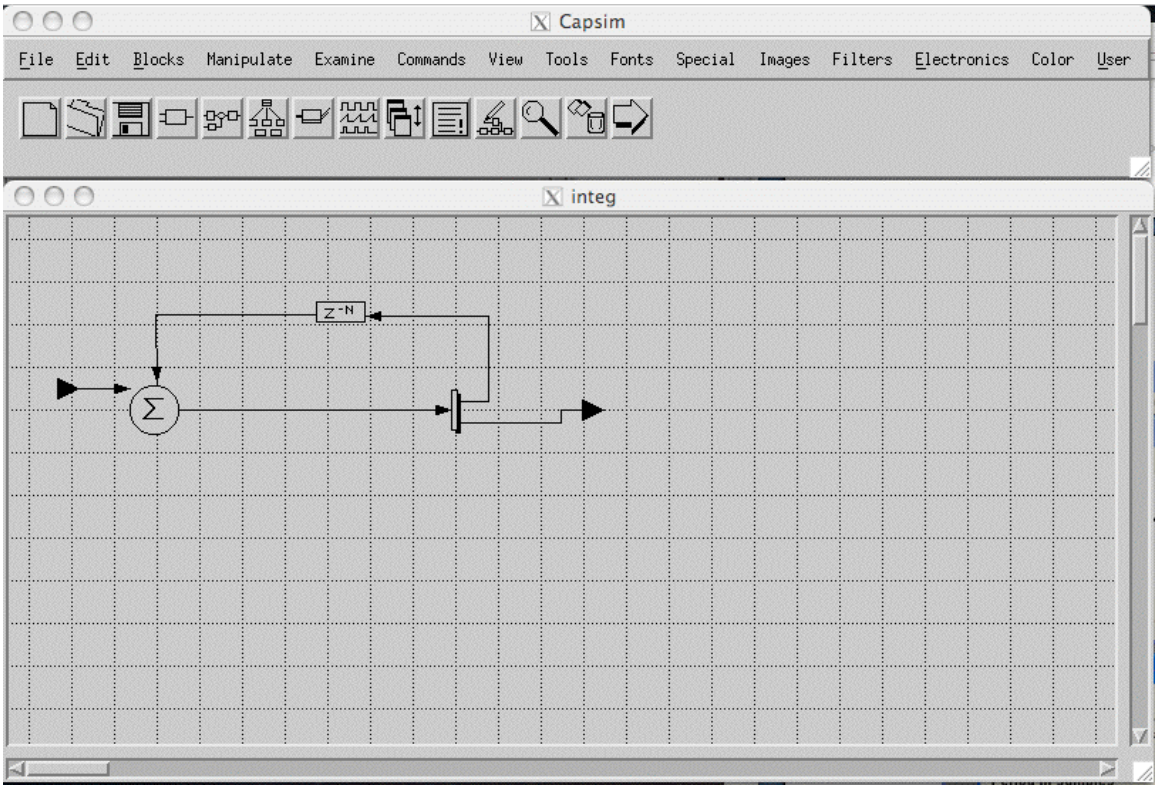


Figure 30 Delay Connections with Manual Drawing

Using a Hierarchical Block

At this point, select Clear All from the File pulldown menu. Click OK on the warning dialog. The workspace will disappear. You are ready to create the universe.

Select *New* from the File pulldown menu. A new workspace will be created. We are going to create a topology as shown in Figure 31. The topology is formed from the *pulse* block, the *integ* HBlock, and the *plot* block. Layout the *pulse* and *plot* blocks as you did before by selecting Blocks from the Block pulldown menu and selecting the blocks from the Block List.

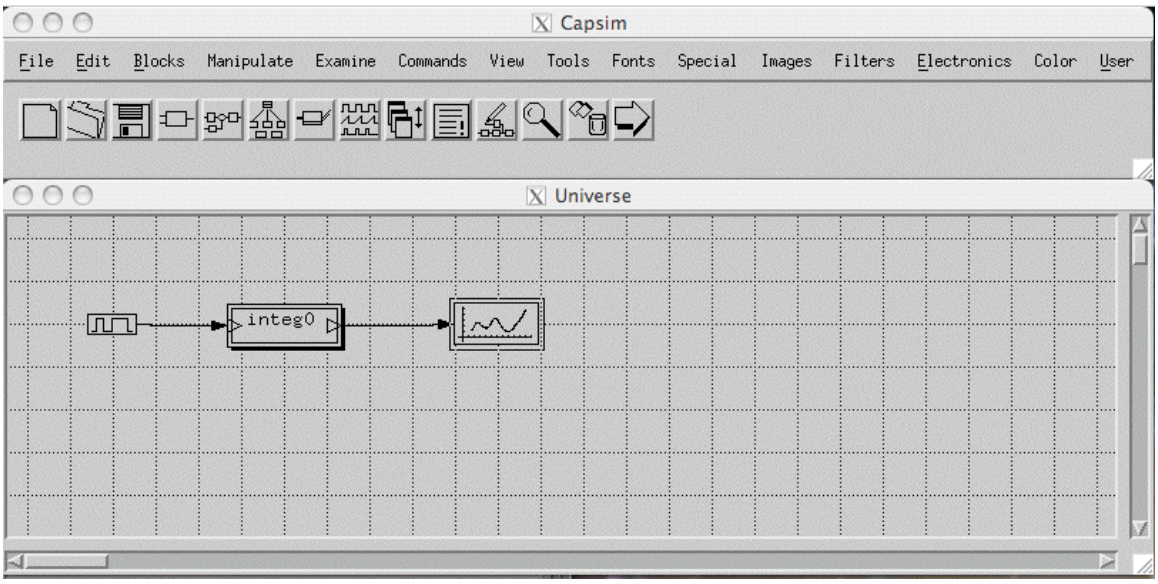


Figure 31 *integ* HBlock with *pulse* and *plot* Blocks

Placing HBlocks on the Workspace

To place the *integ* HBlock on the workspace, select the HBlock option form the Block pull down menu. Or, click on the HBlock icon in the tool bar . An HBlock List window will appear just as in the case for placing blocks. You should find the *integ* HBlock in the list. Move the window to a location so that you can place the *integ* HBlock onto the workspace. Select the *integ* HBlock from the list by clicking on it and then clicking on the Place button. Move the cursor to the workspace. It should turn into a hand

place tool. Move it between the *pulse* and *plot* block. Click the mouse button. The *integ* HBlock appears. HBlocks appear with a double box. Make the connections shown in Figure 31 using the procedure described before. Before we run some simulations we need to change the *pulse* block's parameters. Click on the *pulse* block (if in fast connect, click again). The *pulse* block popup menu will appear. Move to the parameters item and select it. The pulse parameter dialog box should appear. Change the samples per period to 10 and the pulse width to 1. See Figure 32. Click on the OK button. You are ready to run a simulation. It is a good idea to save your work first, however, before any simulation just in case you crash. So save the universe as *integtest* for example. Run the simulation. You should get a plot of a staircase as shown in Figure 33.

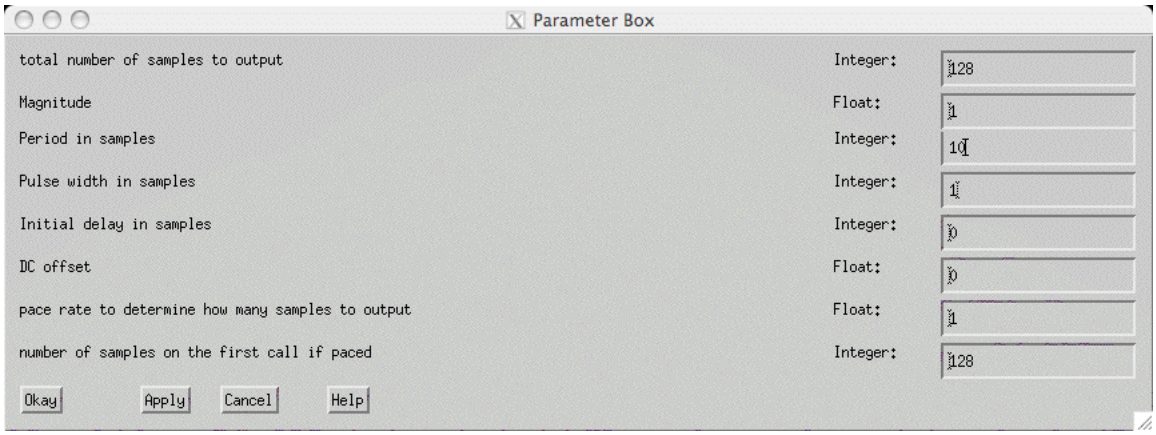


Figure 32 *pulse* Block Parameters

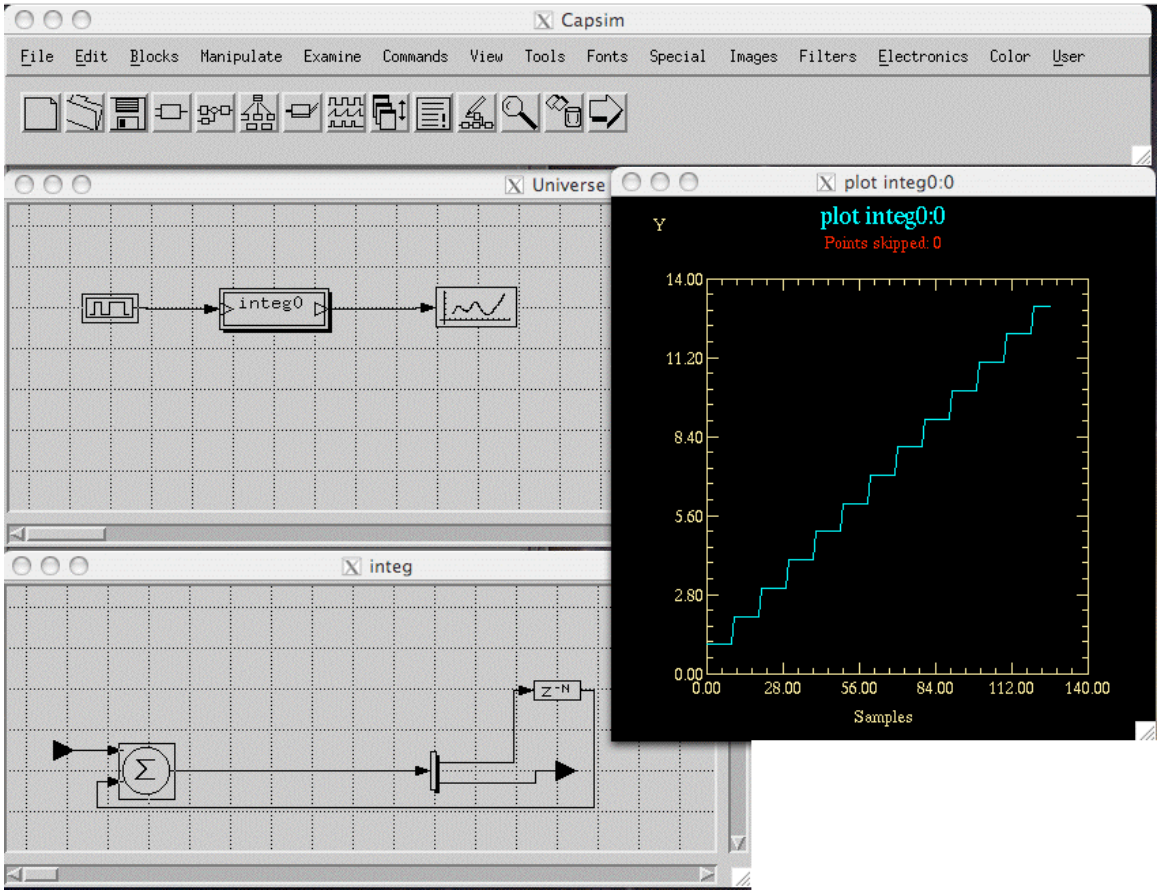


Figure 33 Simulation Run with integ HBlock Showing Stair Case.

Moving Down an HBlock

The topology of the HBlock is also shown in Figure 33. In Capsim you can work on a number of HBlocks at once. To bring up a workspace with the *integ* HBlock, move the mouse to the *integ* HBlock. Click on the left button to popup the block menu. Select "Down One Level". A workspace with the *integ* HBlock will appear. You can resize it if you wish. To make a workspace current when multiple workspaces are on the screen, move the mouse to an empty area on the workspace you want to make current. Click the left mouse button. You can now edit the workspace.

Probes

Probes are attached to connections to monitor the samples. They include a *spectrum*, *plot*, *prfile* to print samples to a file, *hist* a histogram generator and *stat* a statistics probe among others. To see how probes work, create a new workspace and layout the blocks shown in Figure 34. The blocks are, from left to right, the *sine* block, the *addnoise* block, and the *prfile* block. Connect the blocks as shown. Note that the *addnoise* block simply

adds "white" Gaussian noise to its input samples and the *prfile* block by default prints the samples to the console. Hence, when you run a simulation, nothing seems to happen unless you monitor the console. The samples are printed to the console. To observe what is going on, let us attach a probe to the connection between the *addnoise* block and the *prfile* block. Move the cursor to this connection and click the left button on the mouse (the connection must be highlighted by the appearance of two squares at its endpoints). A connection popup menu appears as shown in Figure 35.

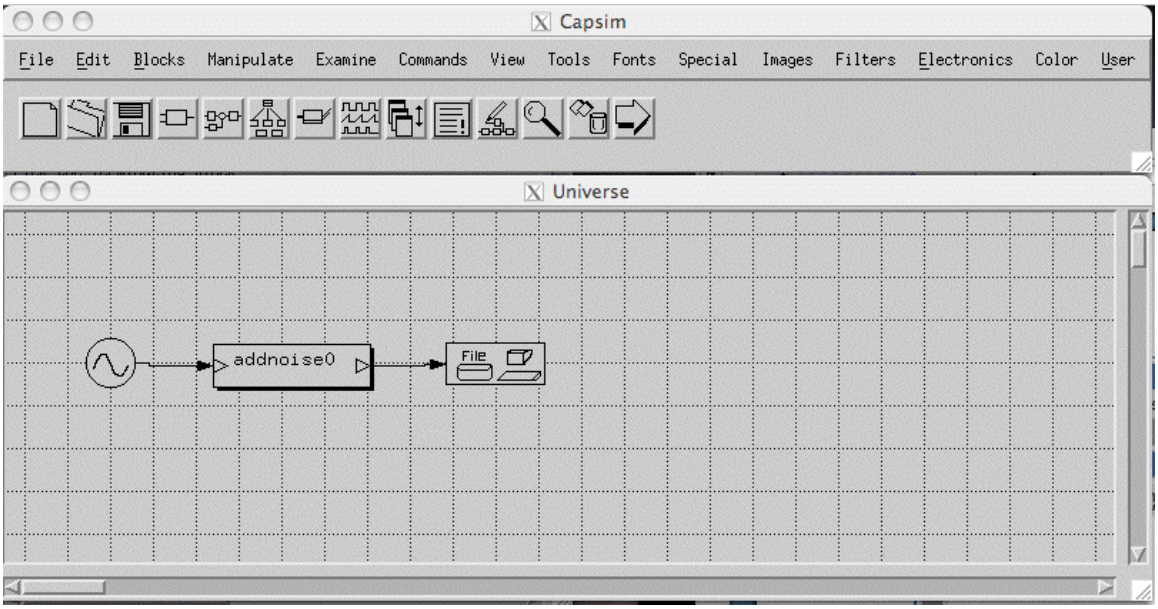


Figure 34 Topology for Probe Insertion

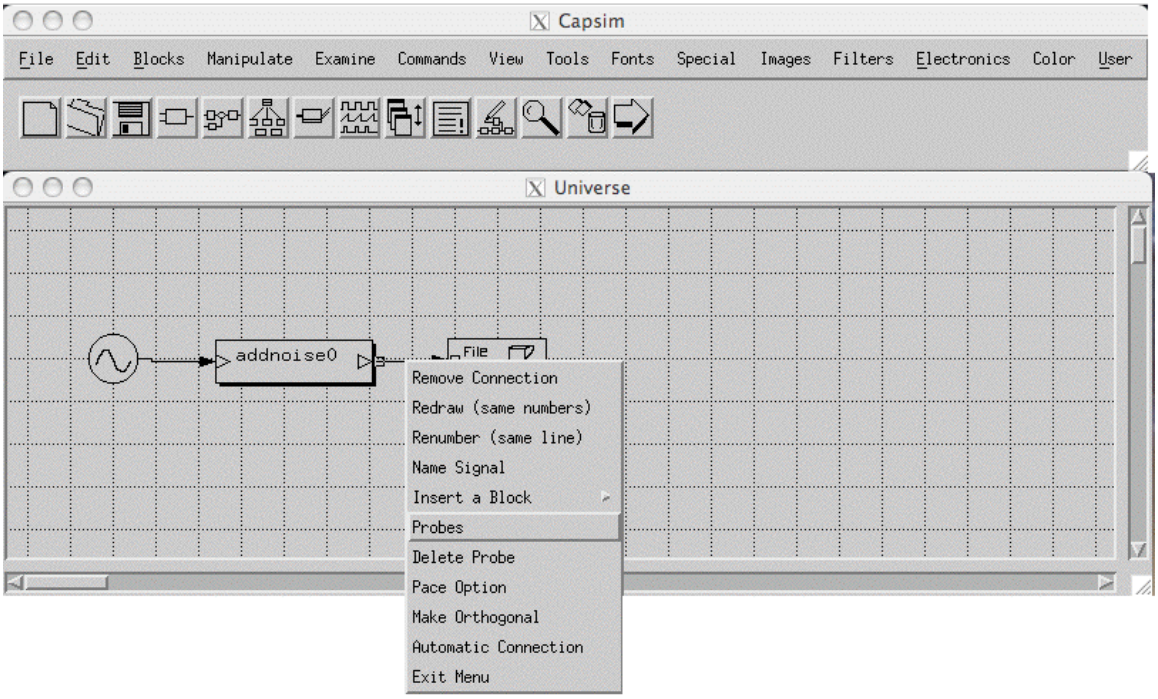


Figure 35 Connection Menu

Move the cursor to the Probes item and click the right mouse button. A Probe Dialog Box appears. See Figure 36. To enable a probe, click on the toggle box by its name. So click on the *spectrum* toggle to enable it. A probe will appear on the connection as shown in Figure 36. Before you run a simulation, set the total number of samples parameter in the *sine* block to 1050. Now, run a simulation. Observe the time domain and frequency domain plots. Disable the spectrum probe and enable the *hist* probe. Run a simulation. You should get a histogram. Later, change the *sine* block amplitude parameters to zero. The histogram should appear as a bell shaped curve since the samples are Gaussian distributed.

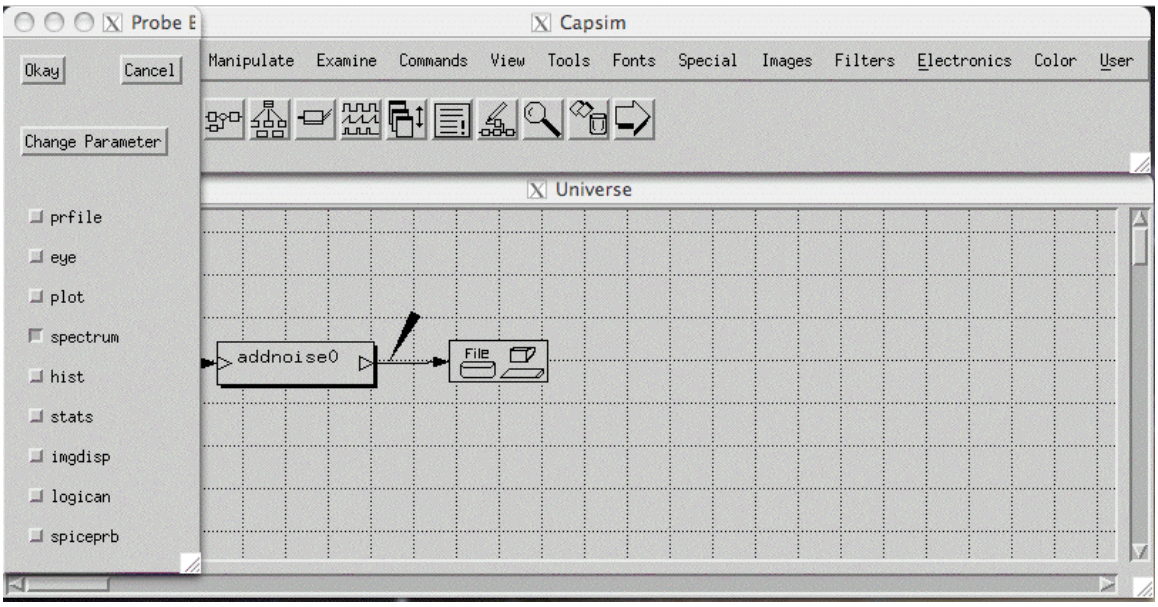


Figure 36 Probe Dialog

Probe parameters may be changed by clicking on the Change Parameters button. A popup menu will appear with a list of probes. Select the probe you want to change the parameters for (it must be enabled), and click the left mouse button. A parameter dialog box will appear for that probe. Change the parameters as described earlier. If you no longer need the probe dialog box click on OK or Cancel. You can delete all probes using the connection popup menu.

Save your work!

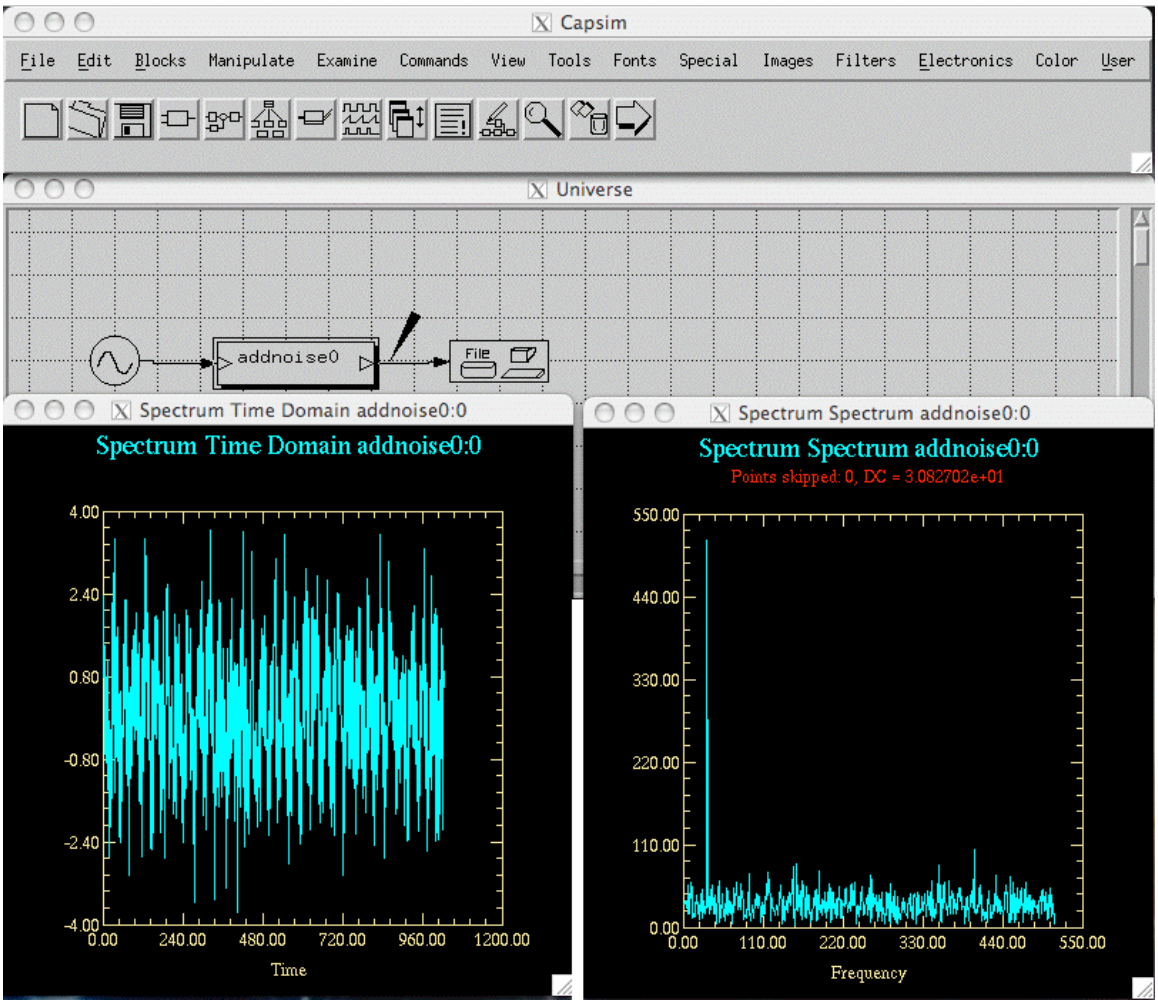


Figure 37 Probe Simulation Results

HBlock Parameters: Arguments

The argument command establishes a model for a HBlock parameters. The basic idea is illustrated in Figure 38. An HBlock is composed of interconnected blocks. These blocks may have parameters which are local. That is, the user of the HBlock does not need to change these parameters. On the other hand, some of the block parameters need to be accessible to the user of an HBlock. These parameters are made accessible through the arguments mechanism. Thus, each parameter that is "exported" from the HBlock is specified by *arg n* instead of a value. The block will then get the parameter value from argument number n. See Figure 38.

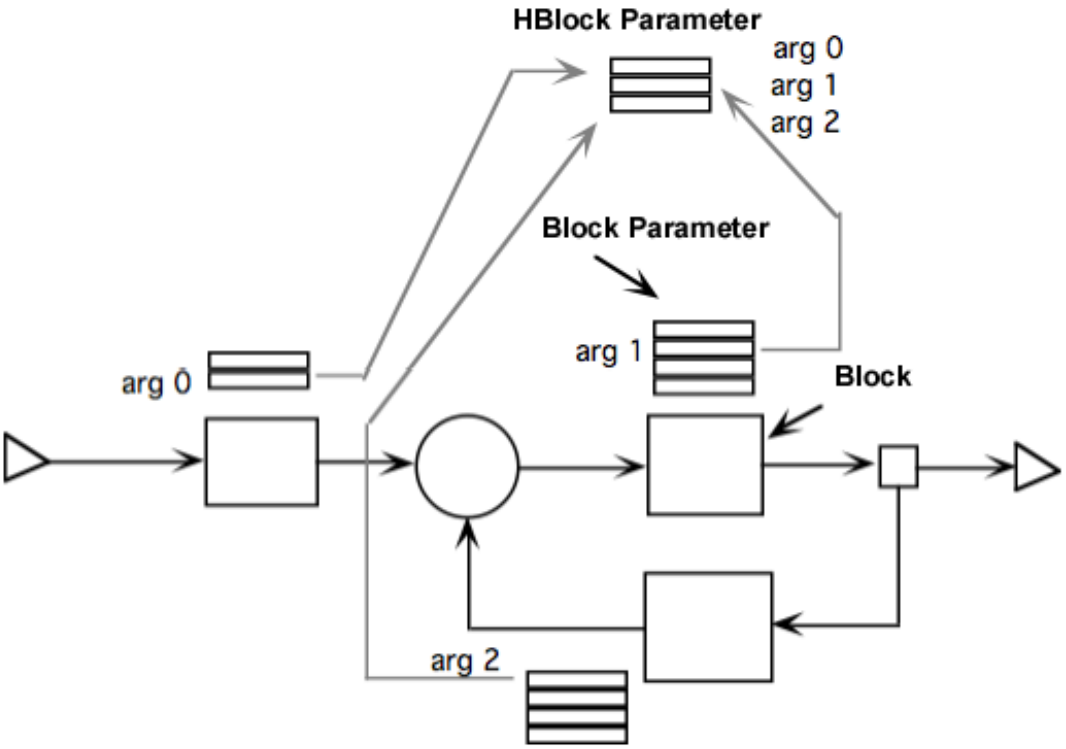


Figure 38 HBlock Arguments

It is clear that argument *n* must be defined prior to usage by the block. So the first step in using arguments, to specify HBlock parameters, is to define them. To do this, the Arguments item in the Blocks pulldown menu must be selected. Or you can click on the arguments icon from the tool bar . The dialog box shown in Figure 39 will appear.

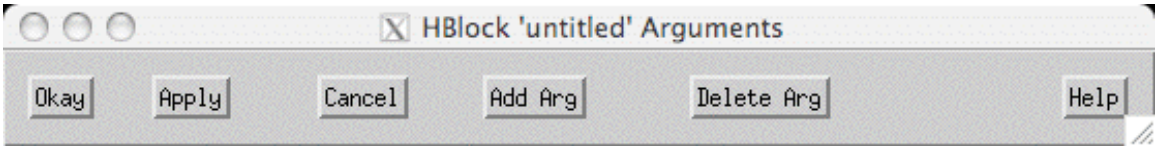


Figure 39 Argument Dialog Box

Initially, there are no arguments. So you must click on the add argument button. This will cause the argument definition dialog box shown in Figure 40 to appear. Using this dialog box, you select whether the parameter is an *int*, *float*, *file*, or *array*. You also define a

prompt and an initial value for the parameter. This new argument will then appear in the argument dialog box when you press Okay. The argument can always be modified at any time. The first argument created is arg 0 and so on. To access this argument in a block, you type *arg0* for the parameter value.

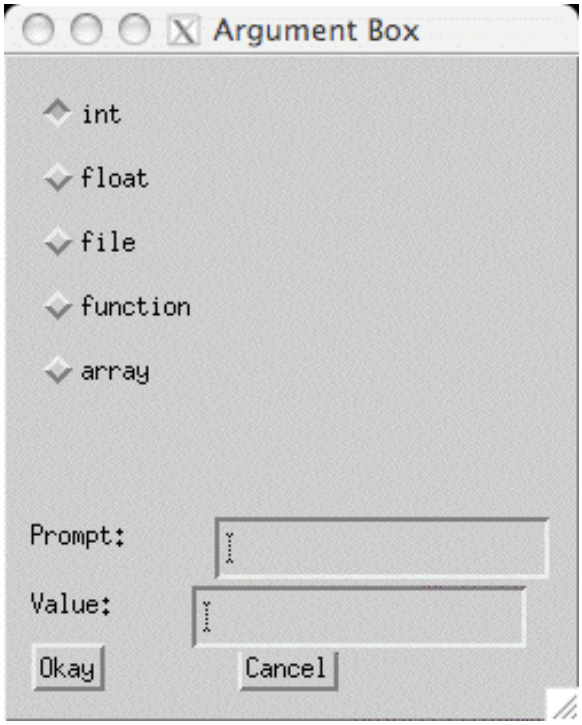


Figure 40 Argument Definition Dialog Box

Global Parameters Through Arguments

The use of arguments is not limited to HBlocks. A universe (the top level HBlock) can also have arguments. In fact arguments can be used to set a global parameter for the top level universe. For example, the number of samples to generate and consume is the same for several source blocks. Instead of specifying the same number of samples for each block, the parameter arg 0 is specified. Using the argument command the value and prompt for arg 0 is set. Then if the number of samples was changed, only arg 0 is modified. Otherwise, the parameter for each block must be changed individually. This concept is illustrated in Figure 38.

In the next tutorial we will illustrate the use of HBlock arguments and global universe arguments.

Figure 41 Arg 0

Creating HBlock Parameters Through Arguments

In this section we will analyze a channel with nonlinear distortion and additive Gaussian noise. The purpose of the tutorial is to show how parameters are created for a universe/HBlock. When the HBlock is used as a block, these parameters will be available to the user just as block parameters. We will further show how arguments can be used in a universe for global parameters.

First we create the topology shown in Figure 42.

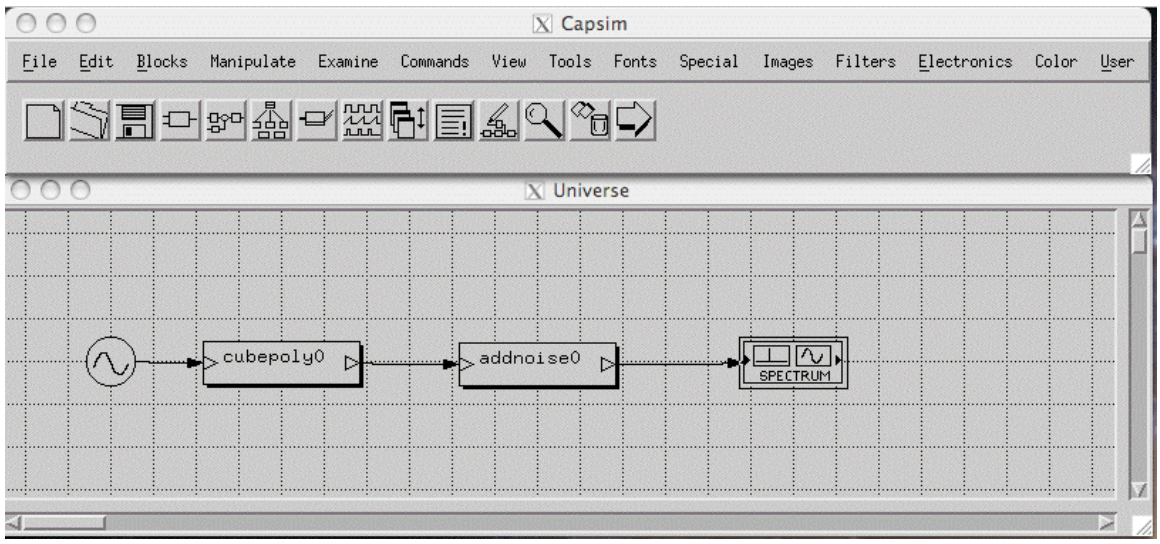


Figure 42 Topology for Global Arguments

Once the topology is created, save it and name it *distortion*. We are ready to define arguments for the topology. The *cubepoly* block will introduce nonlinear distortion. The *addnoise* block will add noise to the signal. There is a parameter which is common to the *sine* and *spectrum* block, that is, the sampling rate. Instead of changing it for each block, we will define a global argument parameter called *Sampling Rate*. We will then connect the global parameter to each block's *Sampling Rate* parameter. But first, we will create an argument for the number of samples to generate. We will also create other arguments that will be connected to block parameters.

To create the “Number of Samples” argument, click on the arguments icon in the tool bar or select arguments from the Blocks pull down menu. The argument dialog will appear shown in Figure 43.

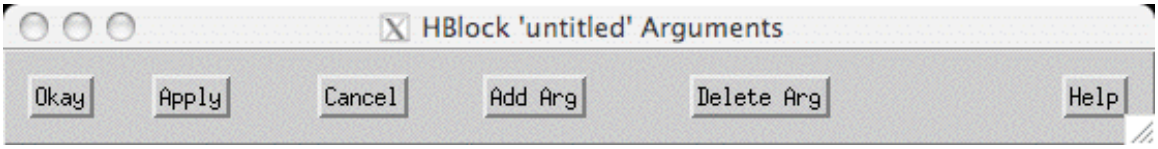


Figure 43 Arguments Dialog

Click on the *AddArg* button. The argument definition dialog box shown in Figure 44 appears. Fill in the prompt and value as shown. Also, very important, select the *int* radio button for the type.

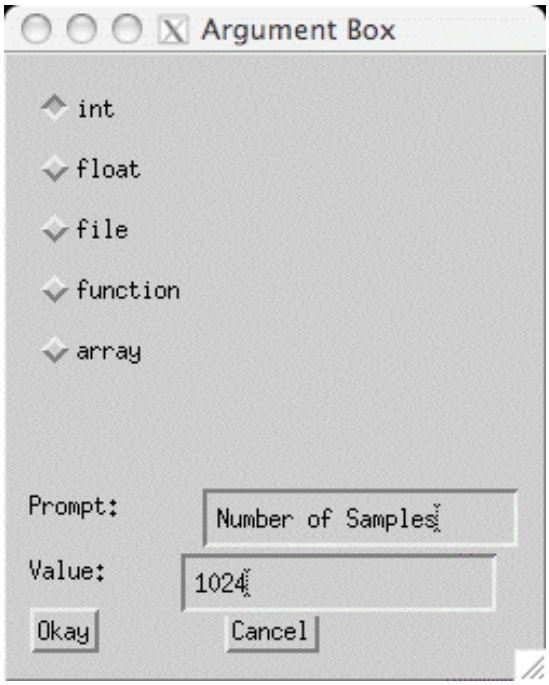


Figure 44 Arg Dialog Box

When you click on OK, the dialog disappears and the HBlock Arguments dialog is updated as shown in Figure 45.

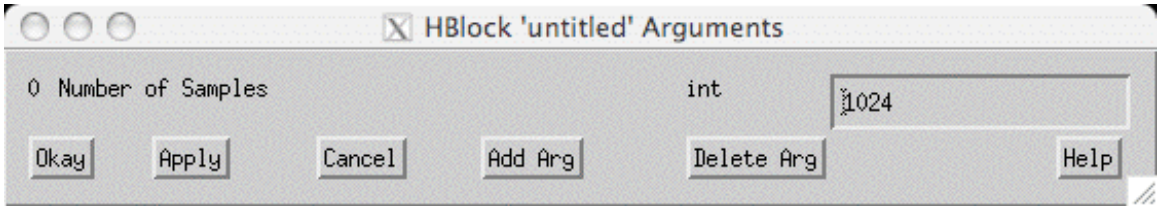


Figure 45 Updated Args: Number of Samples Arg

By following the same procedure, add the Sampling Rate, Cubic Factor and Noise global arguments. See Figure 46. Note that arguments are identified by numbers starting from zero.

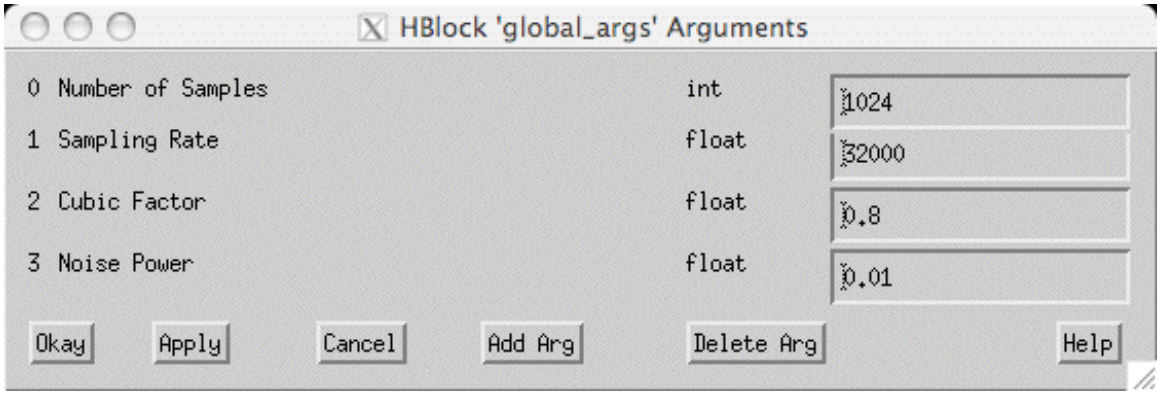


Figure 46 Global Arguments

The next step is to connect the global arguments to the block arguments. At this point make sure you save your work. Click on the *sine* block and select "Change Parameters" from the Block popup menu. The *sine* parameter dialog box appears. Change the "total number of samples to output" argument to *arg0*. That is type *arg0* as the parameter value. Also set the sampling rate to *arg1*. Observe that *arg1* is the sampling rate in Figure 46. Click on OK. Open the *sine* block parameter box again and notice that the parameters now reflect the global argument values. Figure 48 shows that the Sampling Rate for the *spectrum* block also points to *arg1*. Set the parameters for the *addnoise* and *cubepoly* blocks as shown in Figures 49 and 50. Save your work.

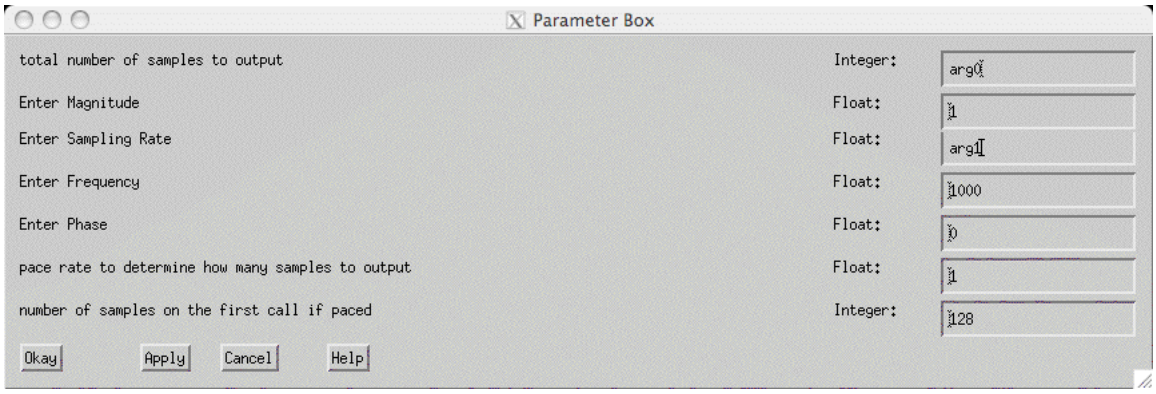


Figure 47 *sine* Block Arguments with Connection to Arguments

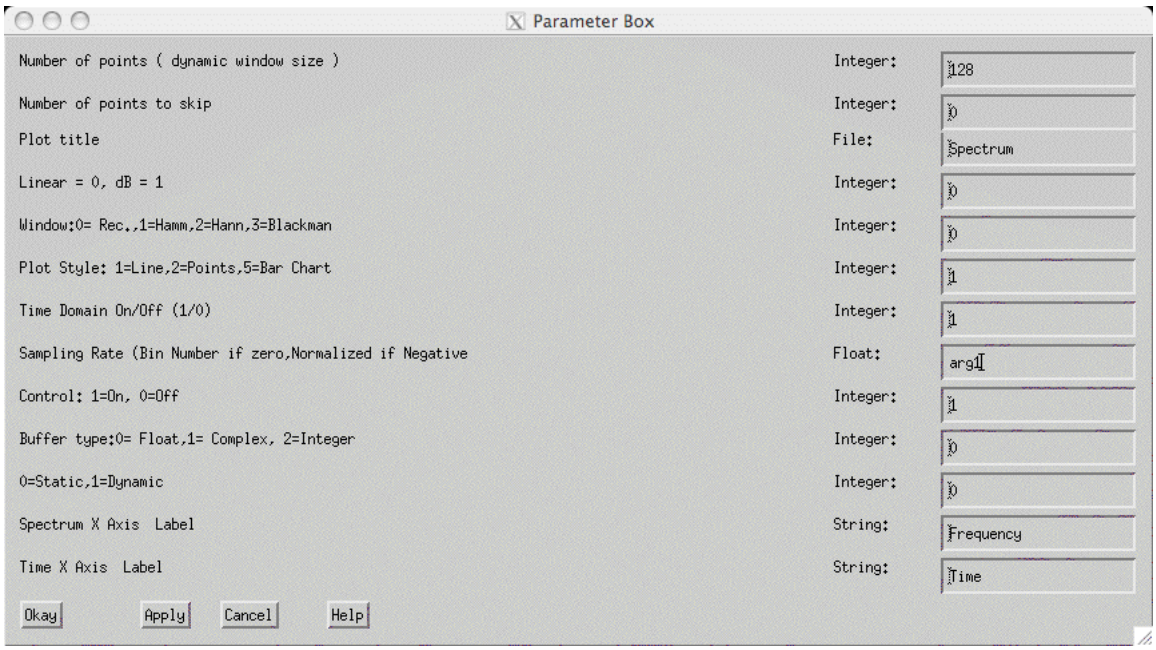


Figure 48 *spectrum* Block Parameters

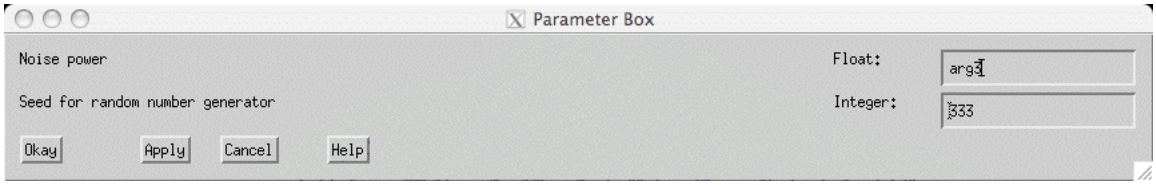


Figure 49 *addnoise* Parameters with Argument

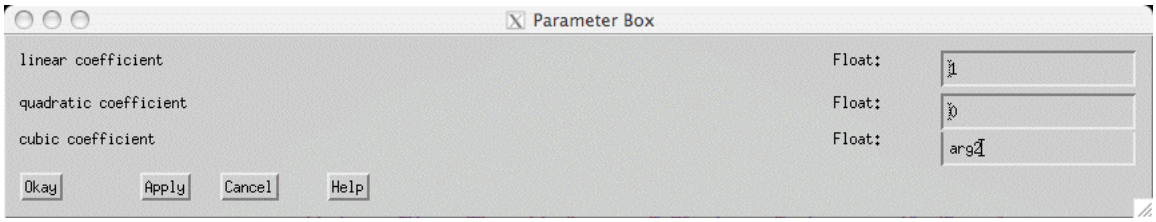


Figure 50 *cubepoly* Parameters with Arguments

When you save your work, the file *distortion.t* is updated. This file is a text file that describes the topology including the arguments. You may want to review this file to see how the arguments are stored. The result of running the simulation is shown in Figure 51. Notice that the *spectrum* has a maximum frequency of 16000 Hz which is half the 32000 Hz sampling rate. The *spectrum* block received this value from the global argument.

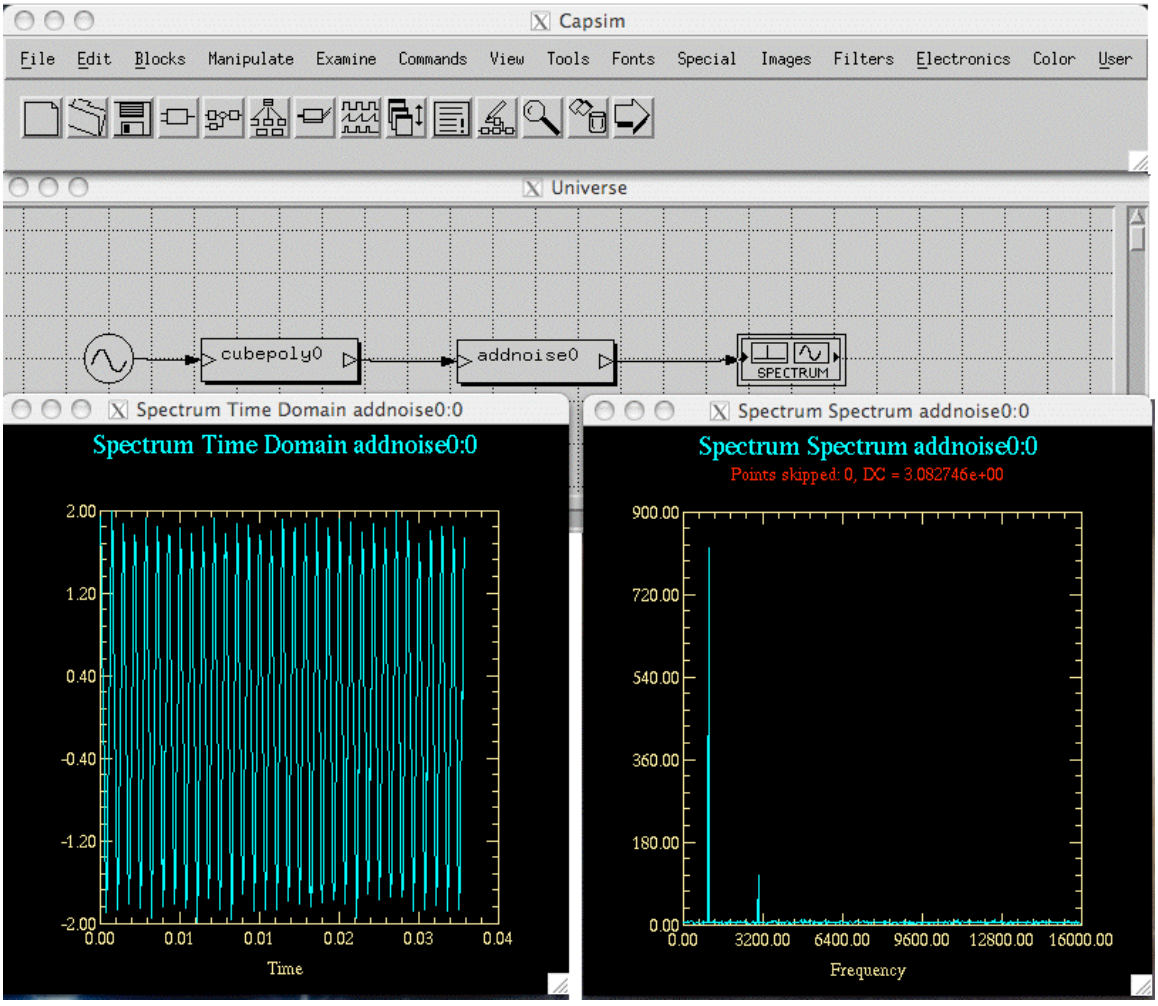


Figure 51 Result of Simulation

Try increasing the non-linear distortion by modifying the global argument arg 2. Set it to say 2. Run the simulation. Compare the result with your previous simulation. Note that the third harmonic increased. You can see this by expanding $\sin^3(x)$. If you want to create an overlay plot, select stack and overlay options from the Plot menu. Select frequency domain. Then select overlay plots to compare the plots in the frequency domain.

Arguments as Global Parameters

As was shown in the previous example, arguments can be used to implement global arguments. An example of a global argument is the number of samples in multiple source blocks. Figure 53 shows the idea behind using arguments as global parameters.

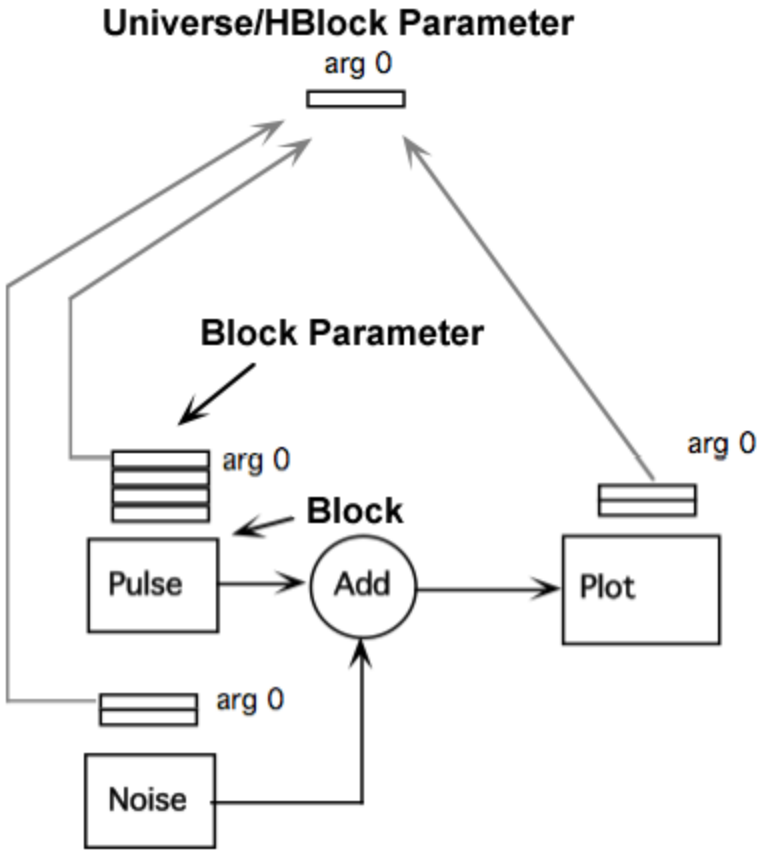


Figure 52 Arguments as Global Parameters

Actions on Multiple Blocks

On many occasions you need to manipulate a group of blocks. Examples are when you want to move a number of blocks, delete them, or convert them into an HBlock. The Edit pulldown, shown in Figure 54, provides the means for multiple block editing.

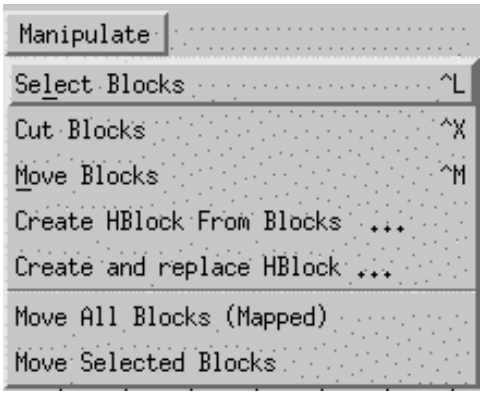


Figure 53 Edit Menu Showing Block Manipulation Items

Selecting Multiple Blocks

The procedure for selecting a group of blocks is shown in Figure 55 through Figure 58. We will work with the example topology shown in Figure 62. First click on the Select menu in the Edit pulldown menu, Figure 54. We want to enclose the blocks to be selected in a bounding box. After you click on Select, move the cursor to the workspace. It should turn into a hand. Move the hand to the lower left corner of the bounding box as in Figure 56 and click the mouse (make sure to release the mouse button). Next move the hand cursor to the upper right corner and click the mouse as in Figure 57. The selected blocks are indicated by the check mark as shown in Figure 58.

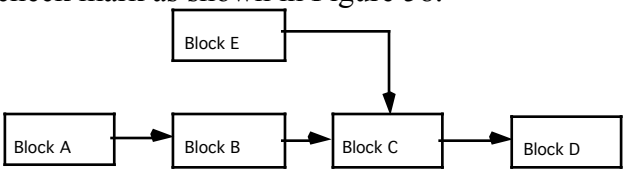


Figure 54 Blocks

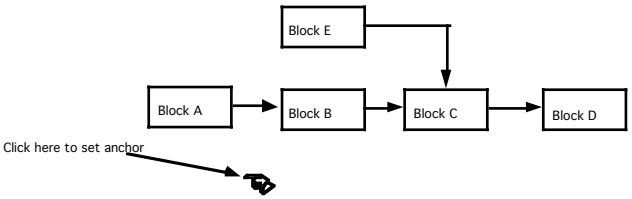


Figure 55 Start of Selecting Blocks

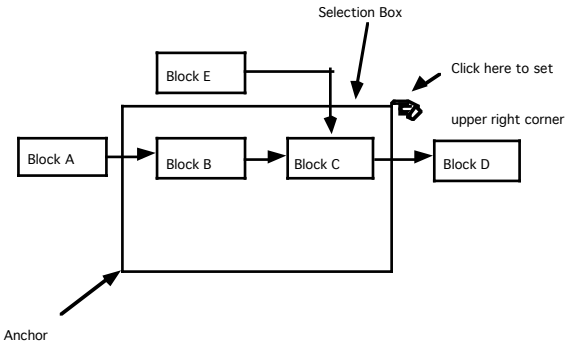


Figure 56 Bounding Box

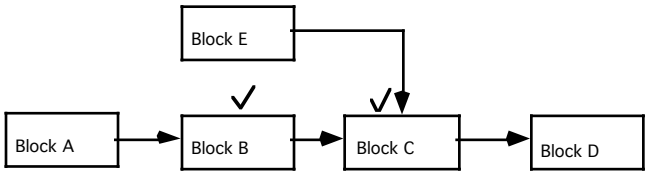


Figure 57 Selected Blocks

Removing Multiple Blocks

To remove selected blocks, click on the Remove pulldown menu item in the Edit menu. The blocks will be removed along with all connections to and from the blocks. Note that if all the blocks in the topology are selected, one block will still remain. You can delete this block with the block popup menu remove menu button.

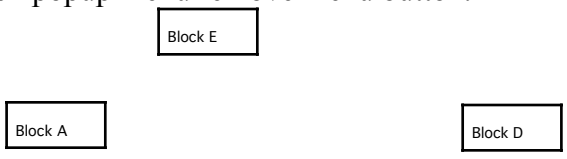


Figure 58 Selected Blocks Deleted

Moving Multiple Blocks

To move multiple blocks, select them as described above. Next select the Move menu from the Edit pulldown menu. Move the cursor to the workspace. A rubber band will appear anchored at the center of the bounding box, as in Figure 60. Move the tip of the rubber band to the desired new center of the bounding box. Click the mouse. The blocks and the connections will be moved. See Figure 60.

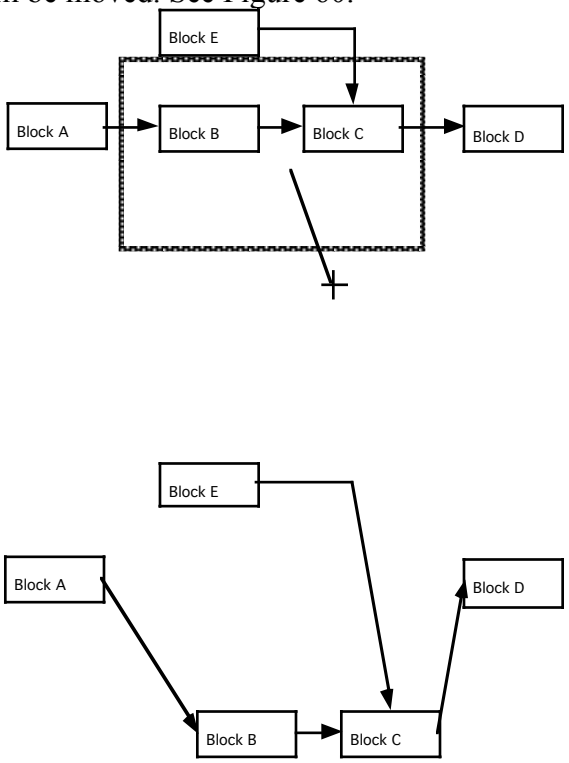


Figure 59 Moving Selected Blocks

Converting Multiple Blocks to an HBlock

You can create HBlocks from blocks within a universe or HBlock. This is a very useful option. To create an HBlock first select the blocks using the procedures described above. Next select Create a HBlock from the Edit pulldown menu. You will be prompted for a HBlock name. Note that after you create the HBlock you can use it immediately. However, it will be at the bottom of the HBlock list. For an example see Figure 61. Note that all connections from selected blocks to other blocks become output terminals. All connections from blocks to the selected blocks become input terminals.

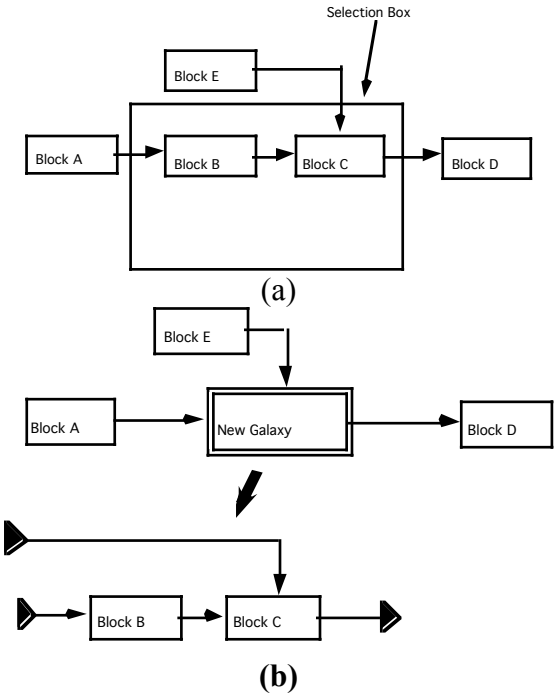


Figure 60 Creating an HBlock and Replacing the Blocks

Replacing Multiple Blocks With an HBlock

This is a powerful option since the HBlock is created and replaces the selected blocks. You can test the HBlock within the universe to check functionality. The procedure is the same as in converting multiple blocks to a HBlock, except that after you specify the HBlock name, the blocks are replaced by the HBlock. As an example, Figure 62 shows the *distortion* topology that was previously created.

First we select the *cubepoly* and *addnoise* blocks. Next we select the "Create HBlock and Replace" menu item from the *Manipulate* pulldown menu. We type in the name of the HBlock as *channel*. The blocks will be replaced with the *channel* HBlock as shown in Figure 63. In Figure 64 we see that the *cubepoly* and *addnoise* blocks have been replaced by the *channel* HBlock. Figure 65 shows the *channel* HBlock using the "Down One Level" from the block popup menu. Note the output of the *addnoise* block is now an output terminal.

NOTE: Remove all probes before creating HBlocks using the Manipulate menu.

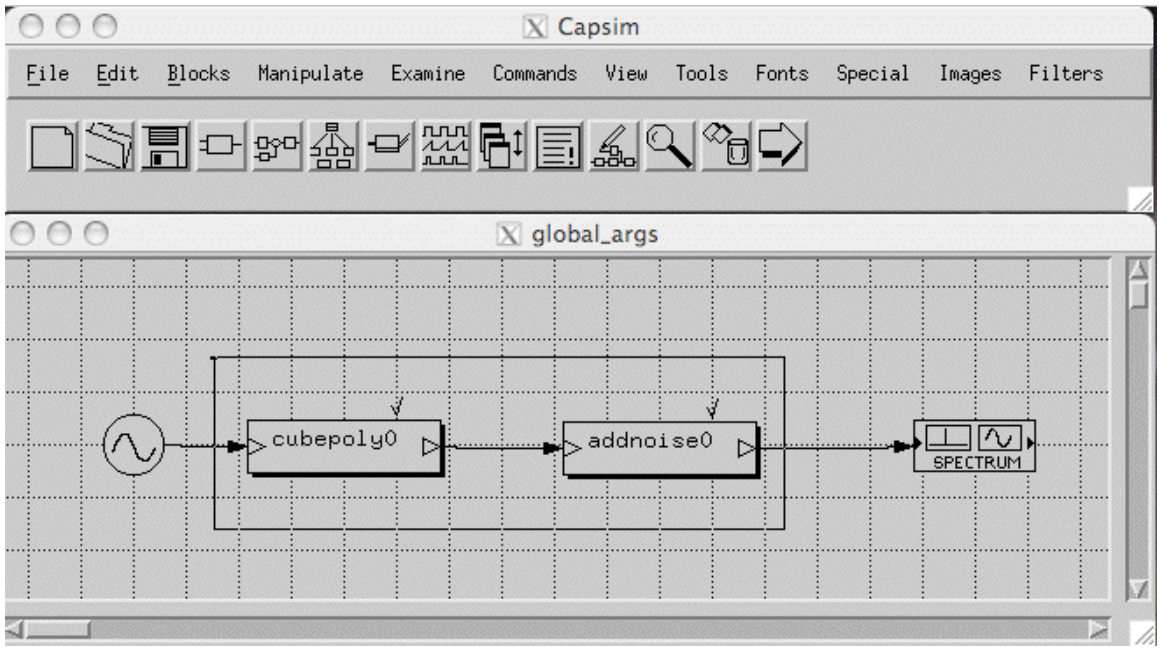


Figure 61 Selecting Blocks

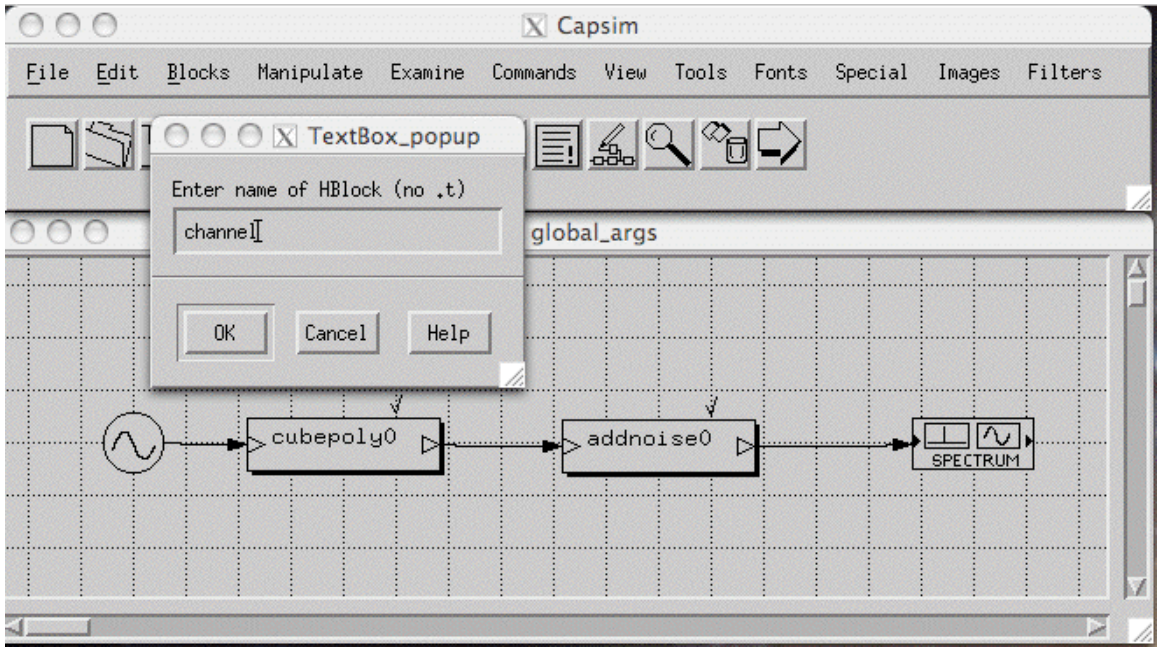


Figure 62 Naming the HBlock

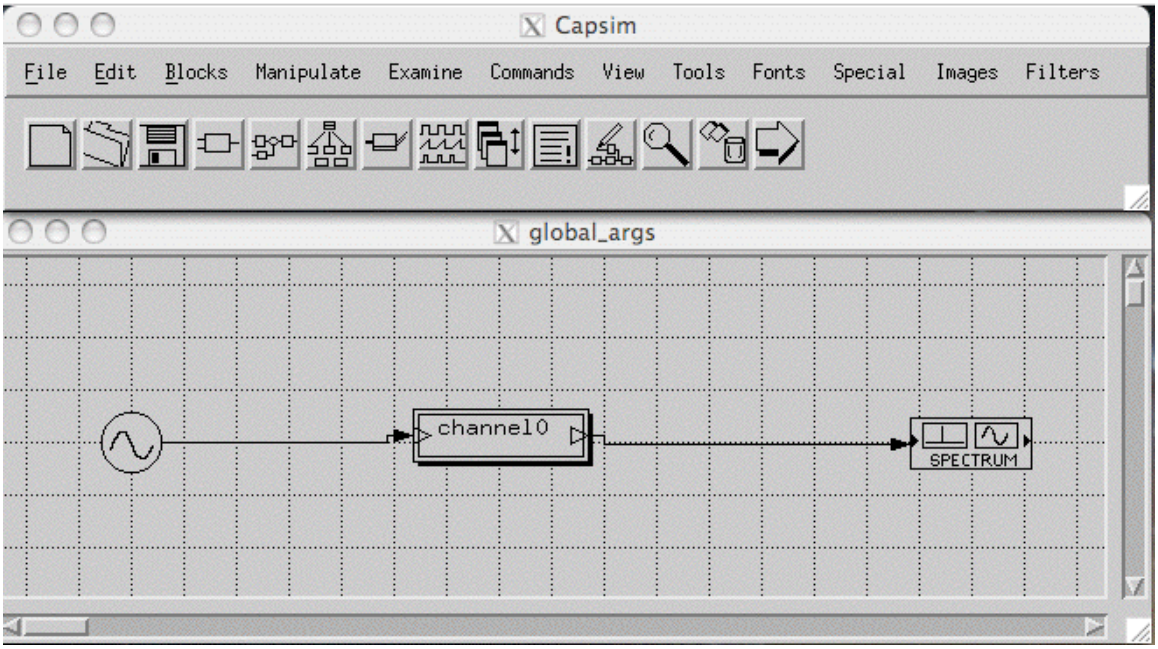


Figure 63 Blocks Replaced with channel HBlock

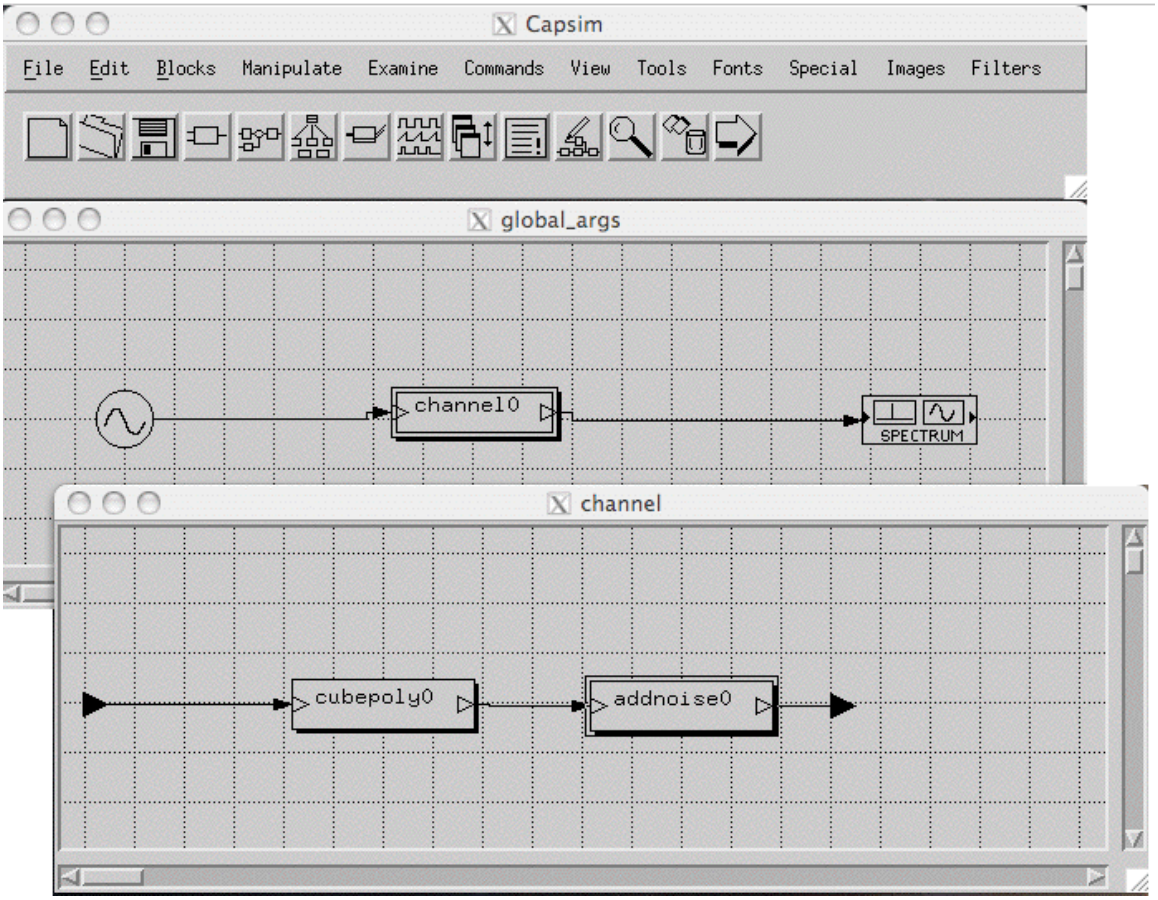


Figure 64 Drilling Down the HBlock

Multi-Rate Signal Processing, Eye Diagrams and Stacked Plots

In this tutorial we will demonstrate multi-rate signal processing, eye diagrams, and the logic analyzer probe for generating stacked plots.

Create a new universe called *baseband* using the blocks indicated in Figure 66. The blocks are from left to right, *bdata*,*linecode*,*sqrtnyq*,*addnoise*,*sqrtnyq* and *eye*. Save the universes as *baseband*. Note that using the option in the Command Menu in the menu bar we have displayed the options and turned on “Show Block Names”.

The *bdata* block generates pseudo random bits, one bit at a time at the bit rate. We can arbitrarily say that the bit rate is 1 Mega Bits per Second (Mbps). The *linecoder* block encodes the bits into bipolar format (-1 for a "0" and +1 for a "1"). The line coder also increases the sampling rate. In this simulation we will oversample by 8. Thus each baud interval will be 8 samples. Each bit that is generated by the *bdata* block will result in 8 samples generated by the *linecode* block. The *linecode* block generates the encoded data as impulses. Each impulse is filtered by the *sqrtnyq* block for pulse shaping. Pulse shaping is necessary for zero inter-symbol interference. It will also bandlimit the signal.

The channel is modeled as an additive noise channel. The pulse shaping is split between the transmitter and receiver for an overall Raised Cosine pulse shape. Thus the sampling rate at the output of *linecoder* is 8*1Mbps=8 MHz.

Run the simulation. You should get the eye diagram shown in Figure 67 where we are using 100% excess bandwidth.

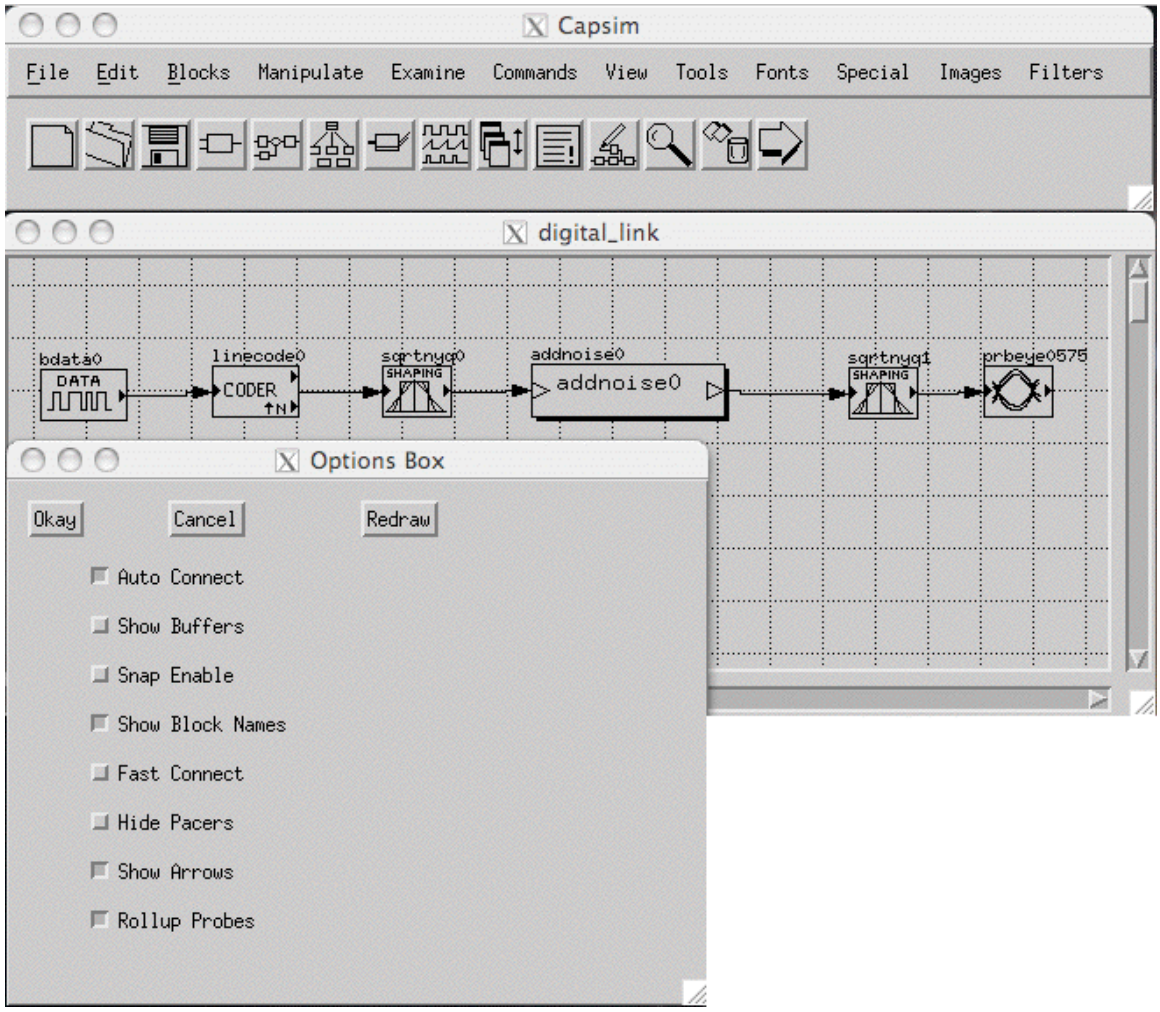


Figure 65 Showing Block Names

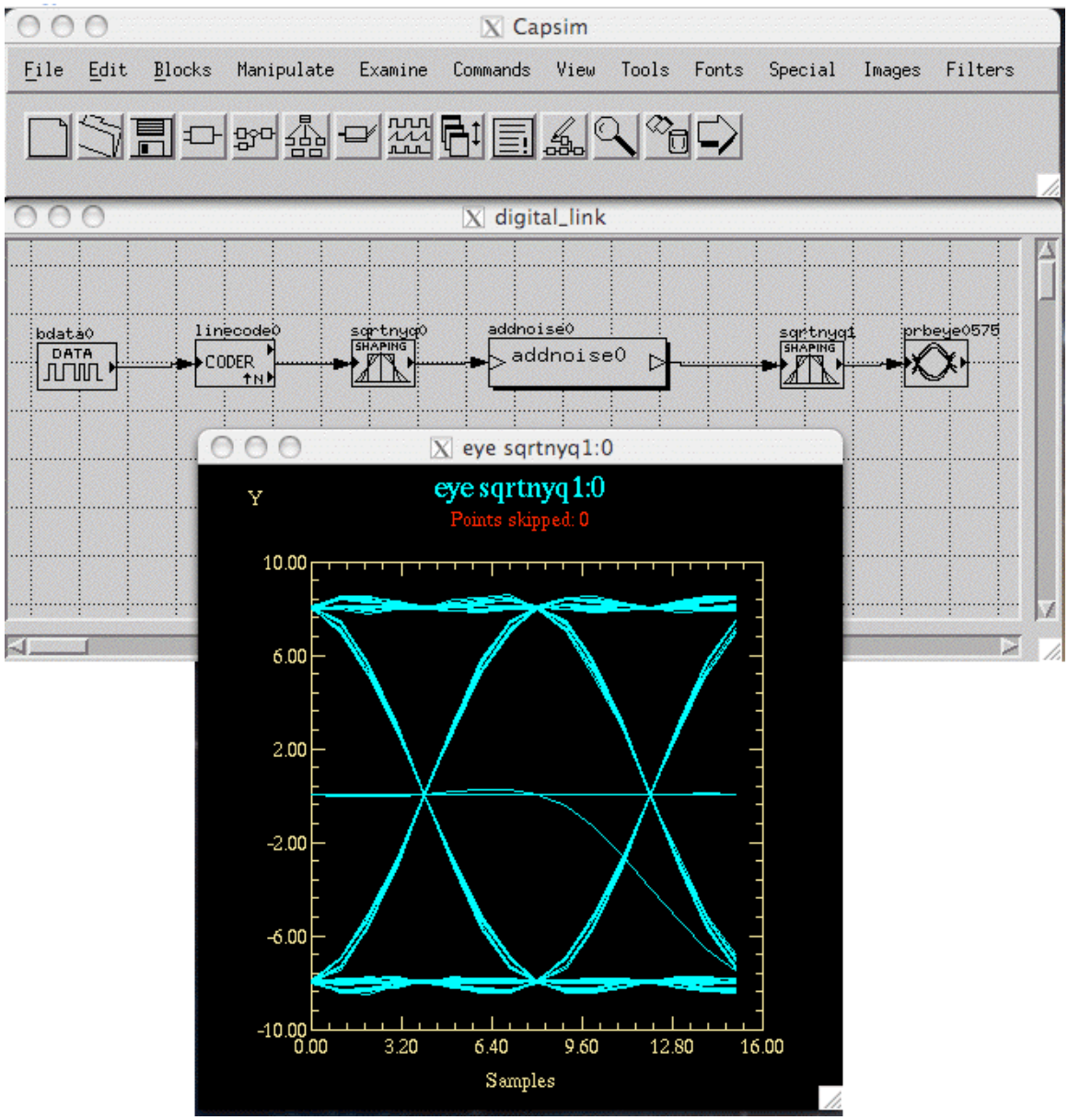


Figure 66 Eye Diagram

In order to see what is going on, let us plot the signals in the system. Rather than creating individual plots with the plot probe, we will use the *logic analyzer* probe. This probe is not limited to digital signals as the name implies but can plot any signal.

Add a *logic analyzer* probe to the output of the *bdata* block, the *linecoder* block, and the second *sqrtnyq* block as shown in Figure 68. When you run the simulation you will not get the stacked plots. To display the logic analyzer probes, select "Stack Selected Plots"

from the View pulldown menu. The stacked plots should popup as shown. You can also use the logic analyzer icon from the toolbar .

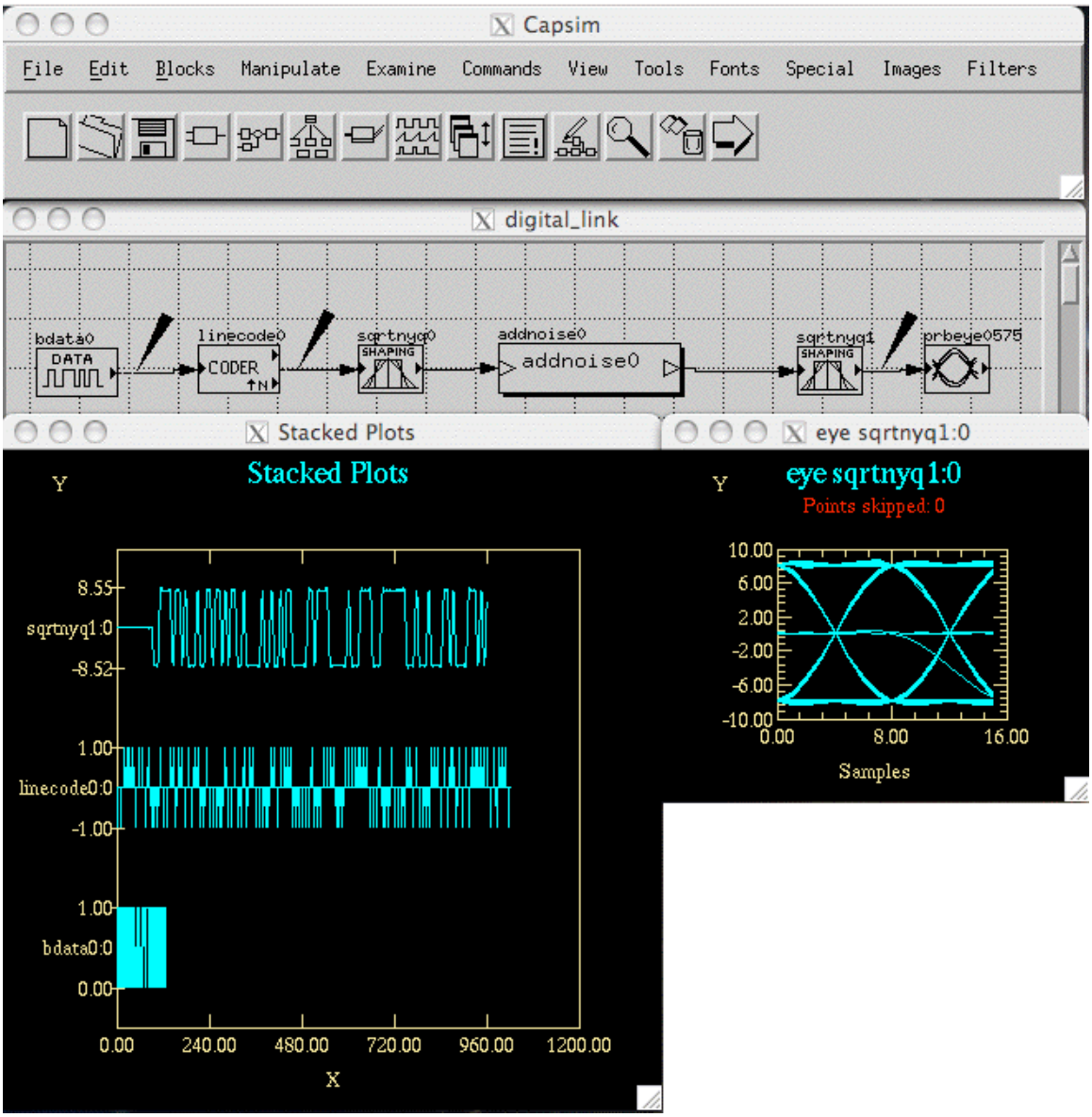


Figure 67 Stacked Plots Showing Multirate Sampling

Using the logic analyzer probe and stacked plots you can easily see what is going on. In particular the multi-rate sampling process and pulse shaping is clearly demonstrated.

Try changing the roll off factor and adding noise to see the effects on the eye diagram. Note that the rolloff factor for both Nyquist filters should be the same. Use a global argument to set them.

Multiplot Block and the Logic Analyzer Probe

Capsim provides many plotting options for the visualization of simulation results. The *multiplot* block and the logic analyzer probe are useful when a number of signal have to be compared. Build the topology shown in Figure 69. Multiple signal generators are connected to the *multiplot* block (*wave, pulse, sine* and *gauss*). The *multiplot* block supports auto fan-in and will create a plot with overlaid curves where each curve represents samples gathered from the input buffers. Set the parameters of the *pulse* block as shown in Figure 70. Set the "Period in samples" and the "Pulse width in samples" as shown. Also set the parameters in the *wave* blocks as shown in Figure 71.

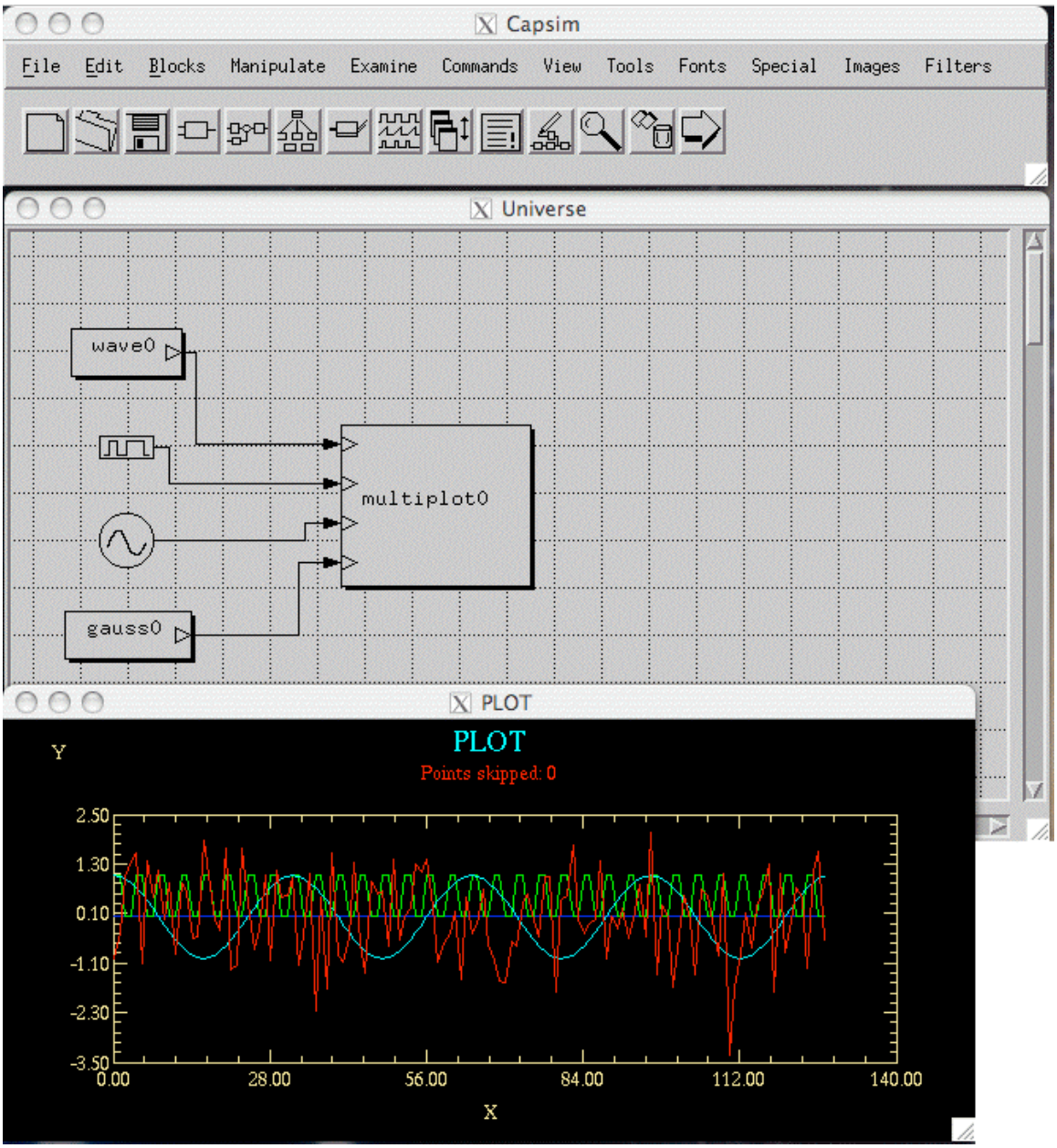


Figure 68 Multiplot

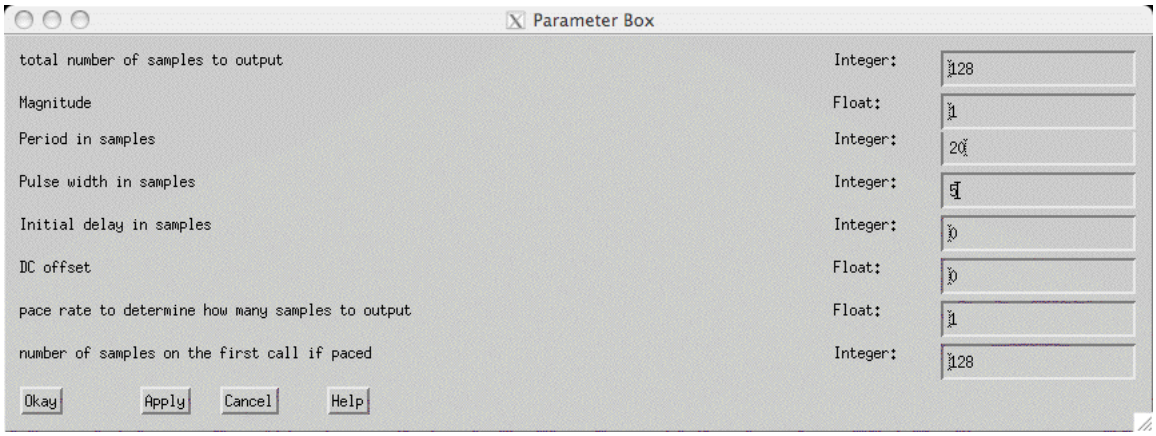


Figure 69 *pulse* Block Parameters

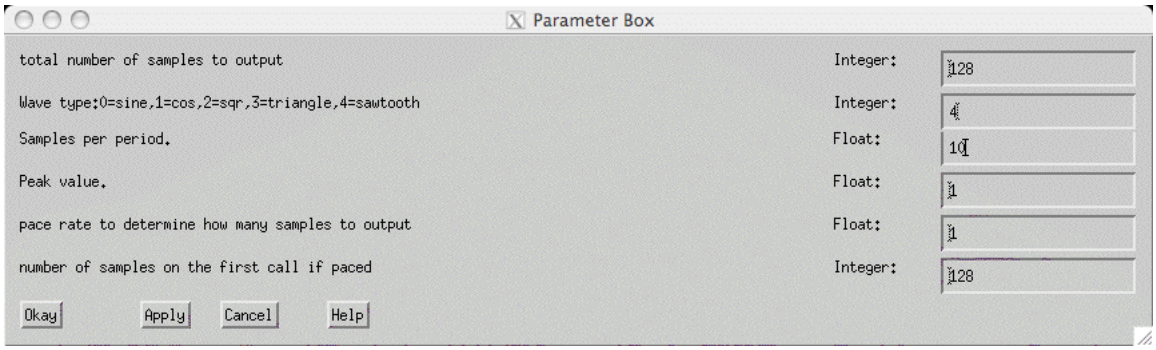


Figure 70 *wave* Block Parameters

Overlaid plots are one way to look at multiple data sets. Another is to use the logic analyzer probe. Add a logic analyzer probe to each connection as shown in Figure 72.

When you run the simulation, click on the logic analyzer icon or select "Stack Selected Plots" from the View pulldown menu to see the stacked plots. Note that all curves share the same x axis but have independent y axis.

In the simulation in Figure 72, you may have to set the parameters to get exactly what appears in the plots, particularly the wave blocks. Try different settings.

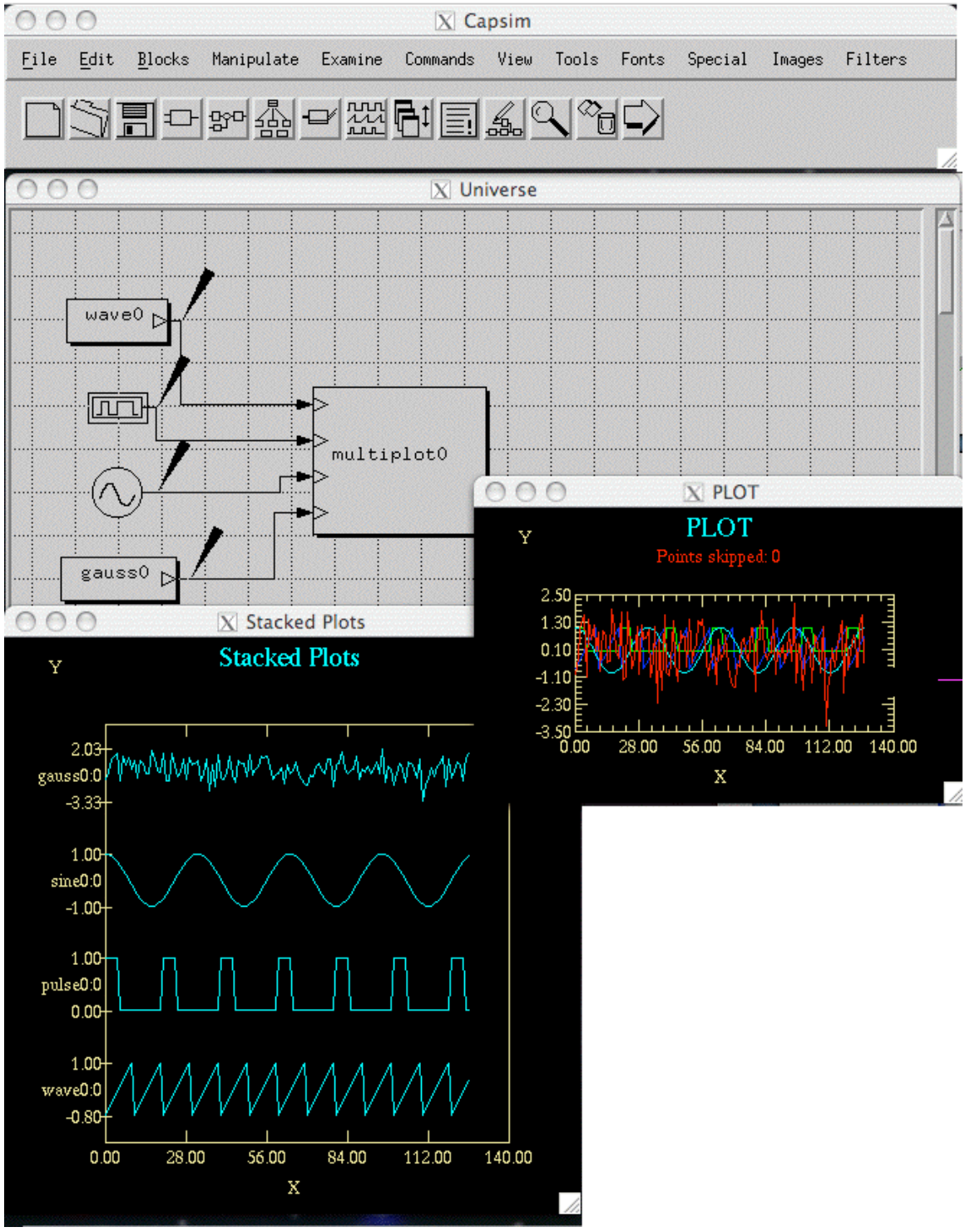


Figure 71 Results of Simulation with Logic Analyzers

Complex Buffers and the Complex Plot Block

This block diagram will illustrate complex buffers and the *cxplot* block which is used as a probe block to plot complex samples. Build the topology shown in Figure 73.

Complex buffers in Capsim transfer complex floating point samples. Each sample is one unit consisting of a real part and an imaginary part. The block *cxmakecx* will convert two floating point buffers connected to its input into a single complex buffer. Buffer 0 is the real part and buffer 1 is the imaginary part. The block *cxmakereal* converts a complex buffer into two real buffers. The real part is output on buffer 0 and the imaginary part on buffer 1.

The probe block *cxplot* is connected to complex buffers and provides a mechanism to view both the real and imaginary parts.

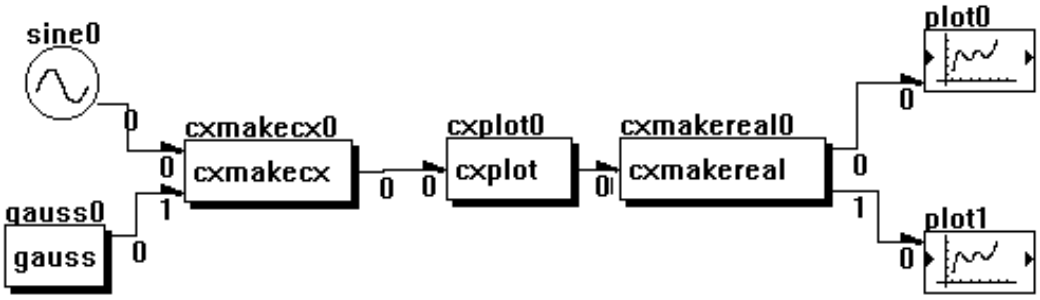


Figure 72 Topology to Illustrate Complex Buffers

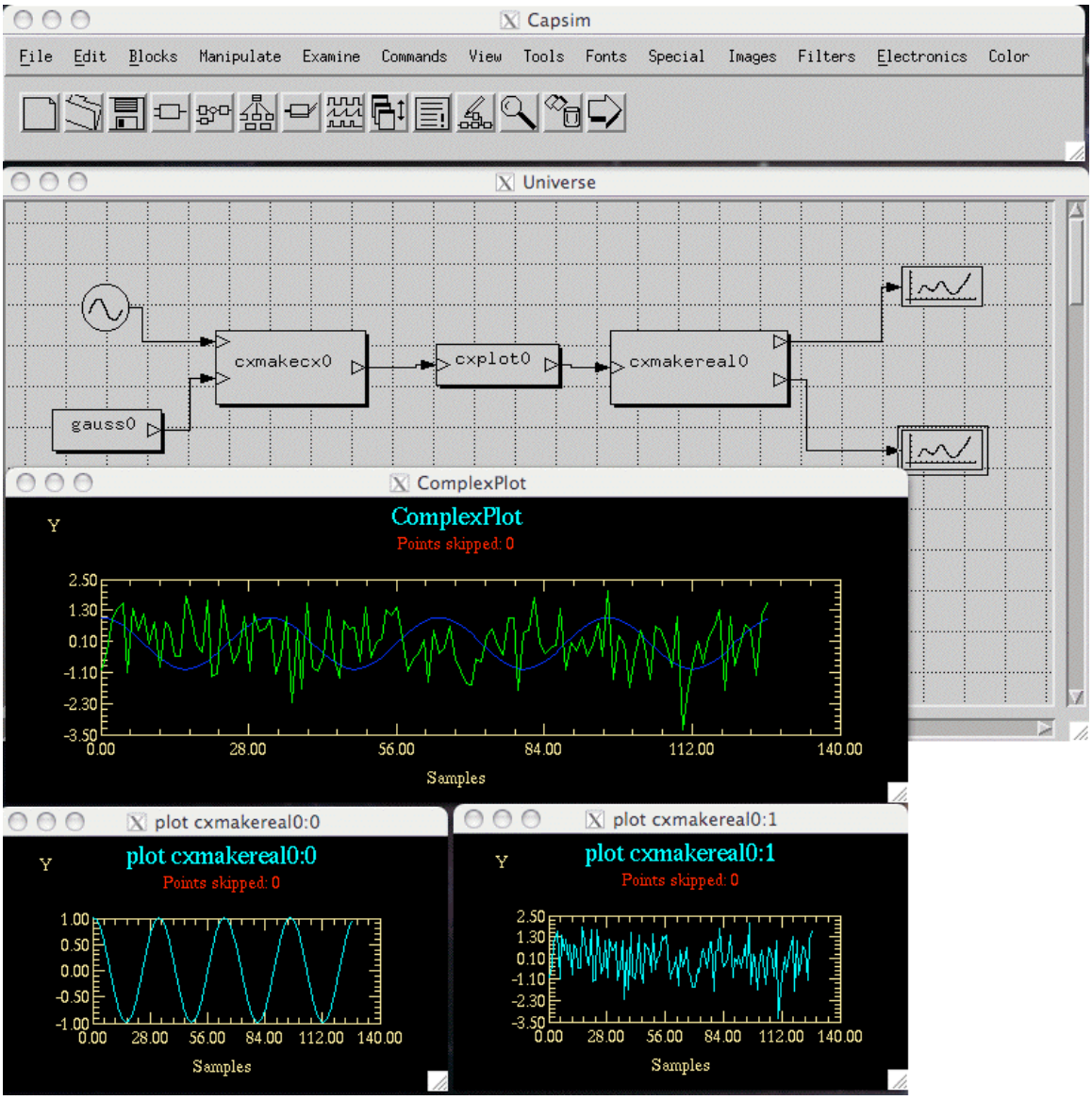


Figure 73 cxplot Plots both the Real and Imaginary Parts of Complex Buffers

After building the block diagram in Figure 73, run the simulation. There is no need to change any parameters. The plots shown in Figure 72 will appear. Note that the *sine* wave is the real part and the gaussian samples are the imaginary part of the complex buffer at the output of *cxmakecx*. Here they are one unit. *cxplot* produces an overlaid two curve plot of the real and imaginary parts. *cxmakereal* then separates the real and imaginary parts into two buffers which are connected to plot blocks. The original sine wave and gaussian process are recovered.

Pole Zero Analysis

Included with Capsim is an integrated pole-zero analysis package. This package allows you to analyze pole-zero diagrams from data stored in files, or to create pole-zero diagrams from scratch. The impulse response and frequency response can be computed. Poles (zeroes) can be added or deleted from the pole-zero diagram or they can be moved.

Open the tutorial for filter design, shown in Figure 75, and click on the *iirfil* block to popup the block menu. Select parameters. The parameters are shown in Figure 76. Note that by default the *iirfil* block designs a lowpass filter. Look at parameter 11. This parameter allows you to specify a name for the designed filter. The default is *tmp*. When the filter is designed, the filter poles and zeroes will be stored in *tmp.pz*, the cascade sections in *tmp.cas* and the filter data in *tmp.dat*. Since we have already run a simulation, the file *tmp.pz* must exist. If you are in doubt run the simulation again.

Figure 74 IIR Filter Impulse Response

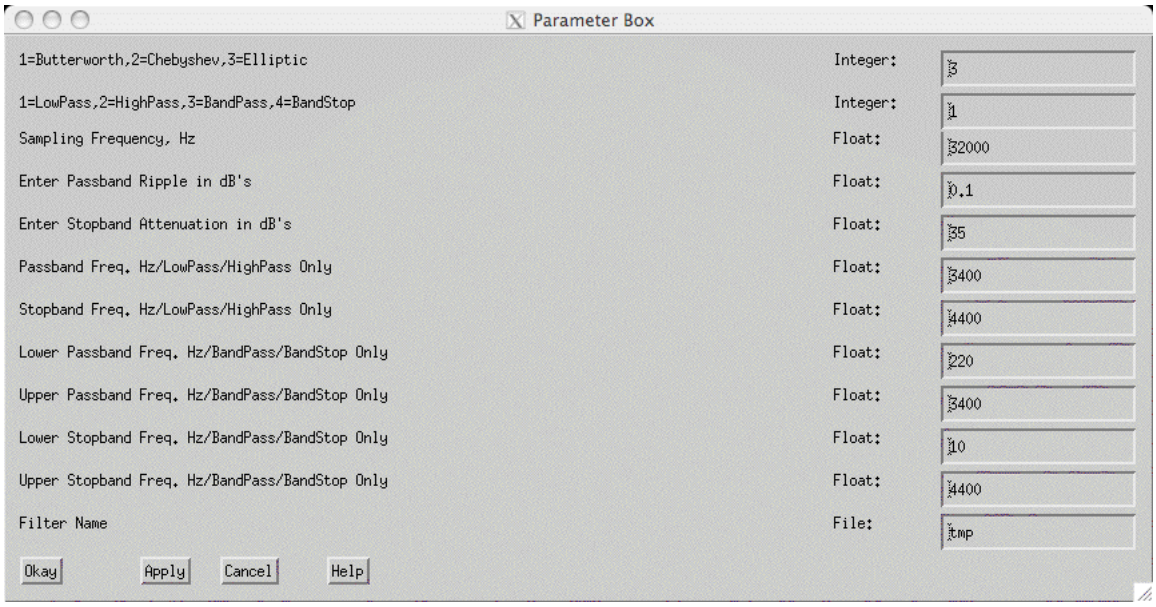


Figure 75 *iirfil* Default Parameters

When you run the simulation, the poles and zeros are stored in a compact form in the file *tmp.pz*. The contents of this file for the default low pass filter is shown in Figure 76. Note the sampling rate of 32000 Hz and the normalization factor of 37.285336. To obtain a pole-zero diagram of the IIR filter, click on "Open Pole Zero File" in the Filter Design Pulldown menu in the Capsim menu bar, see Figure 77. The file *tmp.pz* is probably the only file on the list with *.pz. Select it and click on OK. You should get the pole-zero diagram shown in Figure 78.

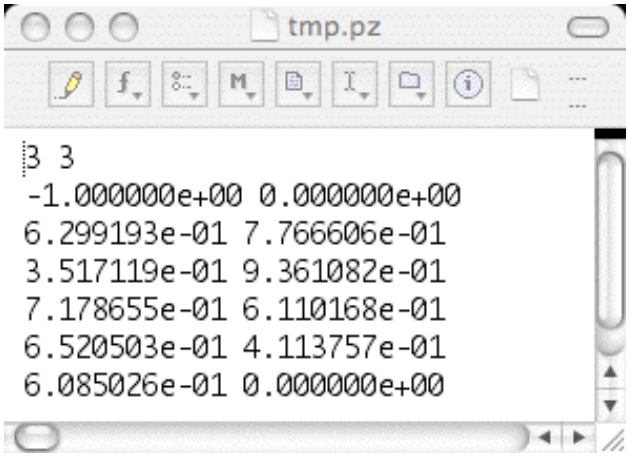


Figure 76 tmp.pz File Contents

Figure 77 Filter Design Pulldown Menu

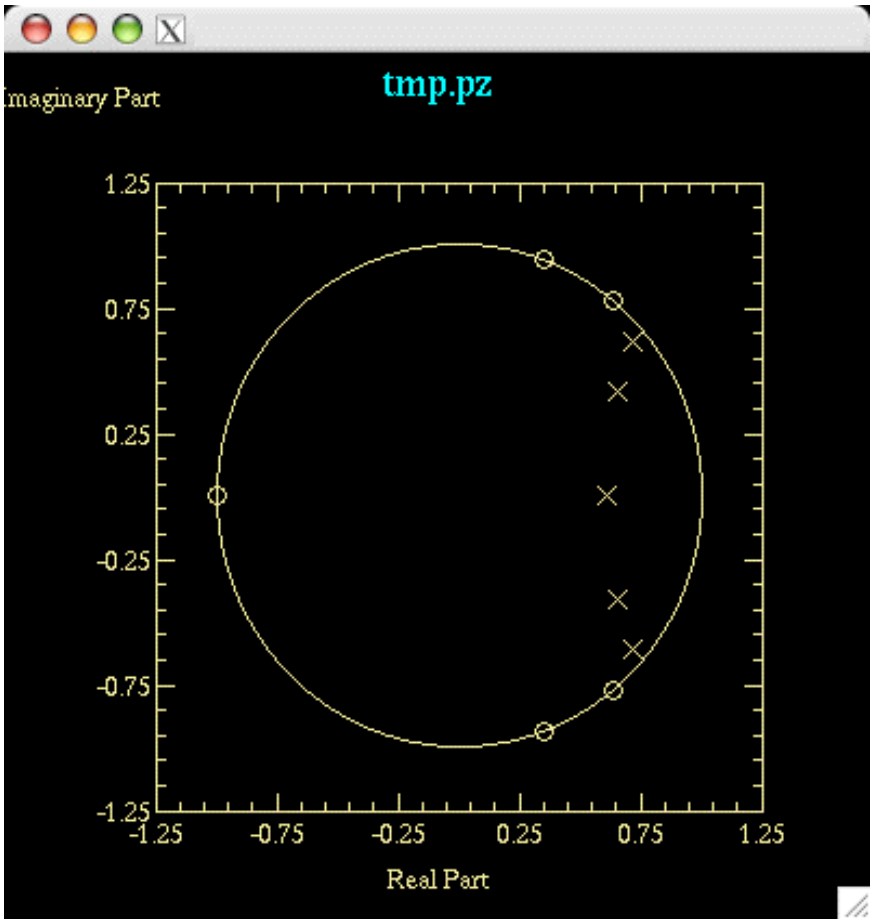


Figure 78 Pole-zero Layout for tmp.pz File. Also Shown is the Pole-zero Toolbar.

To calculate the impulse response and frequency response select the Tools ... menu item from the Filter Design pulldown menu. The tool box shown in Figure 79 will appear.

Simply click on the calculator icon (next to last by the trash can) and the plots will appear. See Figure 79.

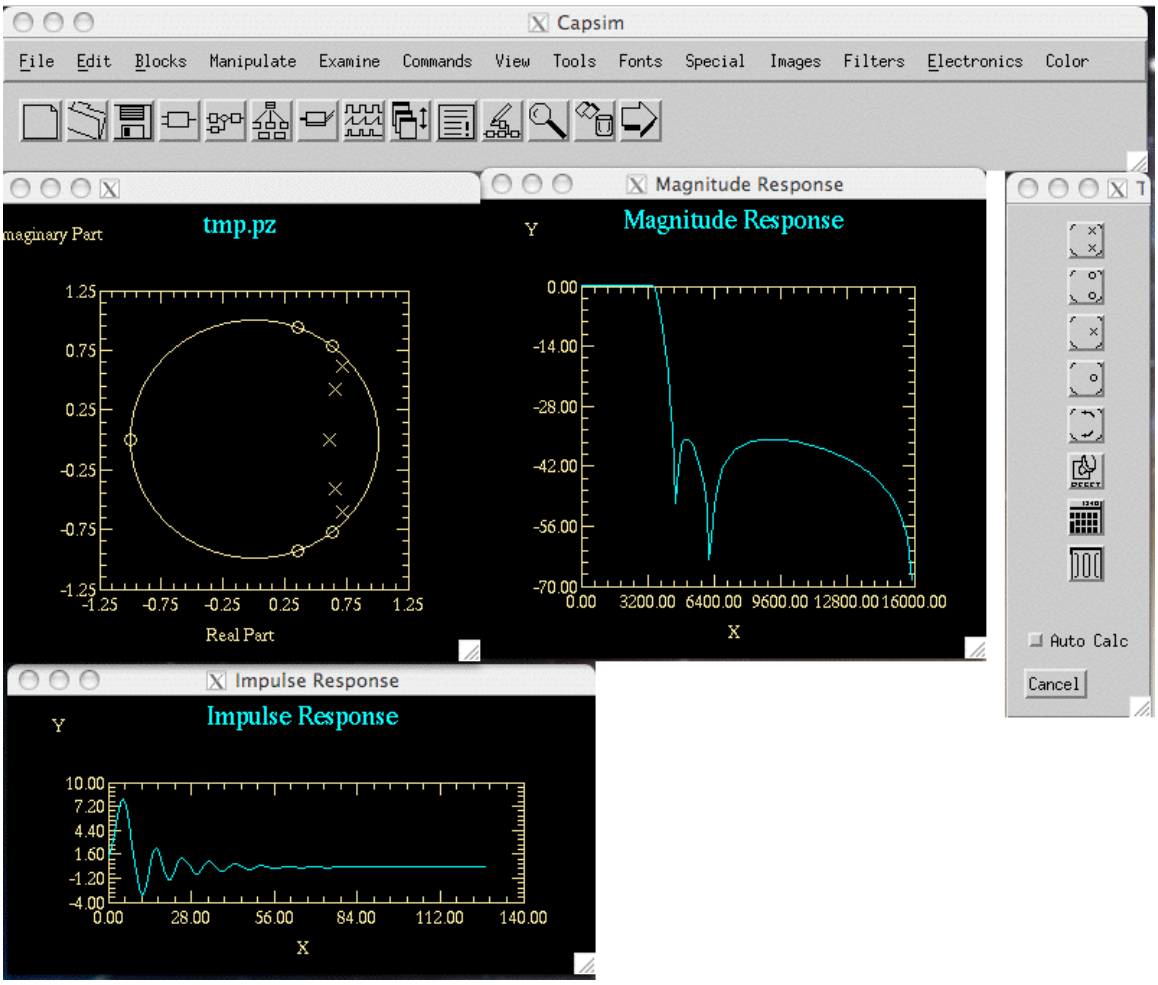


Figure 79 Pole-Zero Analysis After Calculation

Place the plots in a corner. You may also resize them. You can now edit the pole-zero diagram using the tool box. For example you can move pole/zeros and see their effect on the impulse and frequency response. To get the group delay, select Options in the Filter pulldown menu. You can also change the number of points used to compute the responses. You can also save the pole-zeroes in a file and generate various IIR filter structures (normalized lattice, direct form, and cascade form). Capsim has blocks that read these structures and simulate the filters. These are *nl*,*casfil*,*dffl* for normalized lattice, cascade and direct form filters.

Poles and zeroes are stored in a compressed format. That is complex poles/zeroes are specified only once omitting the conjugate. Real poles/zeroes are distinguished by having zero imaginary parts. The number of poles/zeroes specified in the file is equal to the

number of real poles/zeroes plus the number of complex pairs. Figure 76 shows the contents of the *tmp.pz* file.

To reset the pole-zero analysis mode (which you enter when you select any of the tools in the tool box), click on the RESET icon . This will allow you to modify plots. Do this if you get an error message, since you have to leave pole-zero editing mode once entered from the tool box.

Note: To move a pole-zero first select the move icon . Next click the mouse with the cursor as close as possible to the pole-zero you intend to move. This will select it. Next move the cursor to where you want to move the pole/zero and click the mouse. Click and the pole or zero will move to that location. To trash pole-zeroes select as described, but then click the trash icon.

Appendix A: Block Popup Menu Description

The following is a description of the options in the block popup menu:

(1) Connect an Input:

To connect an input to the block. The cursor will turn into a question mark. Select the block which will be connected to THIS block.

If you click on an empty area of the workspace, an INPUT TERMINAL will be created.

(2) Connect an Output:

Use this option to connect the block to another block. Again the cursor will change to a question mark so that you can click on the desired block.

If you click on an empty area in the workspace, an OUTPUT TERMINAL will be created.

(3) Change Parameters:

This menu option will bring up a parameter dialog box for the block. You can change the parameters by typing into the text fields of each parameter. You can also use the TAB key to move between parameter text fields (if modal is on, see graphics help).

(4) Move Block:

An outline of the block will appear so you can move the block to a new location. Connections will also be moved.

(5) Move Blocks:

Same as using select and move from the edit pulldown menu. See edit help.

(6) Replace:

This option allows you to replace the block with either a block or a HBlock. A block or HBlock list will appear. Once a block has been selected, click on Okay and the block will be replaced in place. Connections are preserved.

(7) Delete(Repair Connections):

This option will remove the block but will repair connections. Thus if blocks A, B and C are connected, using this option on block B will remove it and connect A and C together.

(8) Examine Source Code:

This option will create a window and display the block source code or HBlock topology. A path must exist to the BLOCKS directory or the directory in which the block source code resides. Use the command:

path s ../BLOCKS

for example. This command can be placed in the .capsimrc file so that it executed on blocktup, or it can be entered via the Capsim Line Command Option from the Special Pulldown menu.

(9) Info:

This option will display information about the block. For example, its parent HBlock, input/output connected blocks, etc.

(10) Rename:

This option will rename the block. Normally when blocks are created and placed on the workspace, their names are automatically generated. This is the block model name followed by a number indicating the instance of the model. For example if three sine blocks have been placed on the workspace, their names will be

sine0, sine1, sine2

Using this option you can name the block Oscillator for example.

(11) Duplicate:

This option will create another block of the same type. An outline of the block will appear so you can place it at a new location in the workspace. Click on the left mouse button once its location has been determined.

(12) Make Contiguous:

In many cases, after connecting and disconnecting connections from a block, the buffer numbers become non-contiguous.
For blocks such as adders and nodes and many others, the buffer numbers are not significant. So by using this option all buffer numbers are made contiguous.

This option should not be used for blocks in which buffer numbers are significant. For example, in an adaptive filter, buffer 0 may the the error, buffer 1 the desired response, etc.

(13) Down One Level:

This option is valid for HBlock blocks. It allows you to create

a new workspace window displaying the HBlock topology.
You can view and edit the HBlock. To make a workspace the current workspace, click the left mouse button in an empty are of the workspace.

(14) Exit:

This option will popdown the menu without doing anything in the HBlock workspace

Appendix B: Connection Popup Menu Description

Remove Connection:

This option will remove the connection between the blocks. There is no undo command.

Redraw (Same Numbers):

This handy option allows you to redraw the connection. A pencil cursor will appear. Start at the beginning block. Click the Mouse. Move to the location you want a line, click the mouse and so forth. At the ending block double click the mouse to terminate.

Renumber (Same Line):

This option will allow you to change the buffer numbers of a connection. This may be necessary to force the connection to correspond to its functionality per the block documentation. You may also want to explore using the Make Contiguous option in the block popup menu.

Name Signal:

Names the signal. Displays the signal name in the workspace. Uses the name in probes and logic analyzers to identify the signal.

Insert a Block:

Breaks the connection and inserts either a block or HBlock between two blocks.

Probes:

Adds a probe to the connection. A probe dialog box will appear. Multiple probes can be added. Use this option to change a probe's parameters also.

Delete Probes:

Deletes all probes in a connection.

Pace Option:

Brings up the pace option dialog box. This allows you to designate the connection as a pacer. This is only a graphical effect. That is no actual pacing is done. The connection will appear as a dashed line. By using the Options Dialog Box from the special pulldown menu, pacers can also be hidden.

Make Orthogonal:

Makes connections orthogonal. Only works with connections with less than four lines.

Exit Menu:

Exit the popup menu without doing anything.

Appendix C: Command Line Graphic Settings

Various graphic options can be set using the “graphic” command.

For example to display block names you use

graphic name on

The graphic command can be invoked in .capsimrc or through the command line option in the Special pulldown menu.

Here are the options:

Turn auto connect on or off:

graphic auto on(off)

Turn fast connect on or off:

graphic fast on(off)

Display block name, connector buffer or arrows on connections:

graphic name on(off)
graphic buffer on(off)
graphic arrow on(off)

Hide or show pacers:

graphic pacer on(off)

The following changes how dialogs appear and also is used in DecWindows and Open look where we need to set graphic modal on.

With Motif window managers you want graphic modal off.

graphic modal on(off)

This will allow you to use the apply button without popping down the dialog.

However, to use the TAB key to move between text fields you will need to set modal on.

Most of the above options can be set using the Options dialog box. However, the commands allow you to setup defaults using the *.capsimrc* file.