Getting Started with SaberDesigner

Release 5.1

Avant! Corporation

GSS51-1

Registered in England No. 2372734
Registered Office: Aldwych House, Aldwych, London WC2B 4JP

Avant! Corp.
9205 S.W. Gemini Drive
Beaverton, OR 97008

Notice to Users: Read Before Using

Disclaimer

The Saber® simulator and all other software products offered to the customer (Customer) by Analogy, Inc. (Analogy Products) are software programs designed and engineered by Analogy® or its licensors. Analogy Products incorporate the latest ideas and theories in advanced technologies to assist an experienced, highly qualified engineer in the design of Customer's product. Analogy Products have been designed to provide limited analysis and are not intended to replace testing or other analysis of Customer's product. The advanced level engineer must use Analogy Products only as "tools" to help understand the potential performance of Customer's product while continuing to rely upon his or her own expertise.

Analogy Products are based on mathematical abstractions of real world environments or devices. Because Analogy Products work with abstractions, the use of Analogy Products can only provide an estimation of possible future performance of Customer's product. Customer acknowledges these limitations and agrees that it will not rely solely upon the results derived from any usage of Analogy Products in determining the final design, composition, structure, safety, reliability, or performance of any Customer product. Analogy will not be legally liable for any use of Analogy Products by Customer, including Customer misuse, errors in judgment or application of Analogy Products in the development or use of Customer's products.

Analogy does not represent that it has sufficient professional expertise in the application of Customer's product to determine the difference between the performance predicted by Analogy Products and the performance of any real product or device. Therefore, Analogy expects Customer will test Customer's product prior to any actual use. Where the failure of Customer's product may result in bodily harm, Customer must take additional measures, in accordance with the applicable standard of care, to ascertain that Customer's product conforms to Customer's specifications and standard engineering design practices. Regardless of end use application, any Customer product developed with the assistance of Analogy Products must be tested independently to confirm the results obtained from Analogy Products. Where a higher standard of care is required, Analogy recommends that the results of simulation obtained from Analogy Products be confirmed by outside independent professionals, including additional product testing and quality control.

Preface

SaberDesigner Tutorial

This tutorial provides an introduction to design creation and simulation with SaberDesigner, and covers only a few of its many features. To gain more familiarity with the application, you can repeat the tutorial and vary the simulation approach from that presented.

The SaberDesigner tutorial is divided into the following topics:

Viewing Manuals With SaberBook

SaberBook is Analogy's online documentation viewing tool based on the HyperHelp product from Bristol Technology Inc.

SaberBook can be started from either a UNIX shell command line (`saberbook`) or from an Analogy application such as SaberSketch using the **Help** menu item in the main session window.

On UNIX systems, before using SaberBook, you need to change the HyperHelp environment to the default Analogy settings by placing the following command in each user's start-up file, such as `.profile` or `.cshrc`:

`xrdb -merge` *saber_home*`/app-defaults/app-defaults`

If a user has modified HyperHelp background and foreground colors in their `~/.Xdefaults` file, the previous command will overwrite those definitions. To restore a user's custom changes, they need to add another line to their start-up file right after the previous `xrdb` command as follows:

`xrdb -merge ~/.Xdefaults`

Conventions

This manual uses the following conventions:

keywords	Keywords are reserved words such as the MAST language word **template** that are shown in the font style at left.
`~/tutorial/circuit`	File and directory paths appear in the `Courier` font.
placeholder	An item shown in *italic* in a command line indicates that you must supply the specific text to be used in its place.
bold	Menu items, Dialog box names, and button names are shown in **bold**.
emphasis	Emphasized text such as a manual titles and new terms appears in italic.
[ ]	Italicized square brackets enclosing text indicate optional entries.
Single Click	Press and release a mouse button once quickly.

Press and Hold	Press a mouse button and do not release it.
Double Click	Press and release a mouse button twice quickly.

The following convention is used in this manual to indicate a menu selection from a sub-menu as shown in the following example:

Analyses > Operating Point > DC Transfer

In this example, the **DC Transfer** menu item is selected by first selecting the item **Analyses** from a top-level menu, then selecting **Operating Point** from a sub-menu, and finally selecting **DC Transfer** from a second sub-menu.

Revision History

November 1999	Update text to reflect changes to the user interface.
March 1999	Update text to reflect changes to the user interface.
March 1998	Update text to reflect changes to the user interface.
October 1996	Total rewrite to include SaberSketch and move Frameway Integration tutorials to the *Getting Started Using Frameway Integrations* manual.
December 1995	First publication, accompanying Release 4.0 of SaberDesigner.

Table of Contents

chapter 1

Creating a Design Using SaberDesigner

The first part of this tutorial shows how to use SaberDesigner to create a single-stage transistor amplifier that includes load resistance and capacitance. You are shown the following tasks:

- How to use the Parts Gallery to find and place symbols
- How to use the Property Editor to modify property values
- How to wire the design

Before running this tutorial, make sure that SaberDesigner is properly installed and ready to run on your system (see your system administrator).

NOTE

For NT Mouse users: The left and right buttons on a two-button mouse should map to the left and right mouse functions respectively as described in this tutorial. If this tutorial specifies a middle mouse function, an alternate way is also described to complete the task.

Creating the Tutorial Directories

You need to create two directories to organize the data for the single-stage amplifier circuit you are creating.

1. Create (if necessary) a directory/folder called `avanti_tutorial`, where you will create the tutorial example.
2. Navigate to the `avanti_tutorial` directory.
3. Create a directory called `amp`.
4. Navigate to the `amp` directory.

Creating the Design Using SaberSketch

In this part you create the design for a single-stage transistor amplifier using SaberSketch.

1. Invoke SaberSketch using one of the methods as follows:

   (UNIX) On a command line, enter *install_home*`/bin/sketch`

   (Windows NT) **Start > Avant! >** *saberdesigner* **> SaberSketch**

   An empty schematic window appears.

2. Provide a name for the design as follows:

   a. Save the currently empty design by selecting the **File > Save As...** menu item. A Save Schematic As dialog box appears.

   b. In the File Name field, do one of the following:

   (UNIX) Enter the name `single_amp`.

   (Windows NT) Enter the complete path or browse to *your_path*`\avanti_tutorial\amp\single_amp`.

   c. Click **Save**.

3. Examine the SaberSketch work surface.

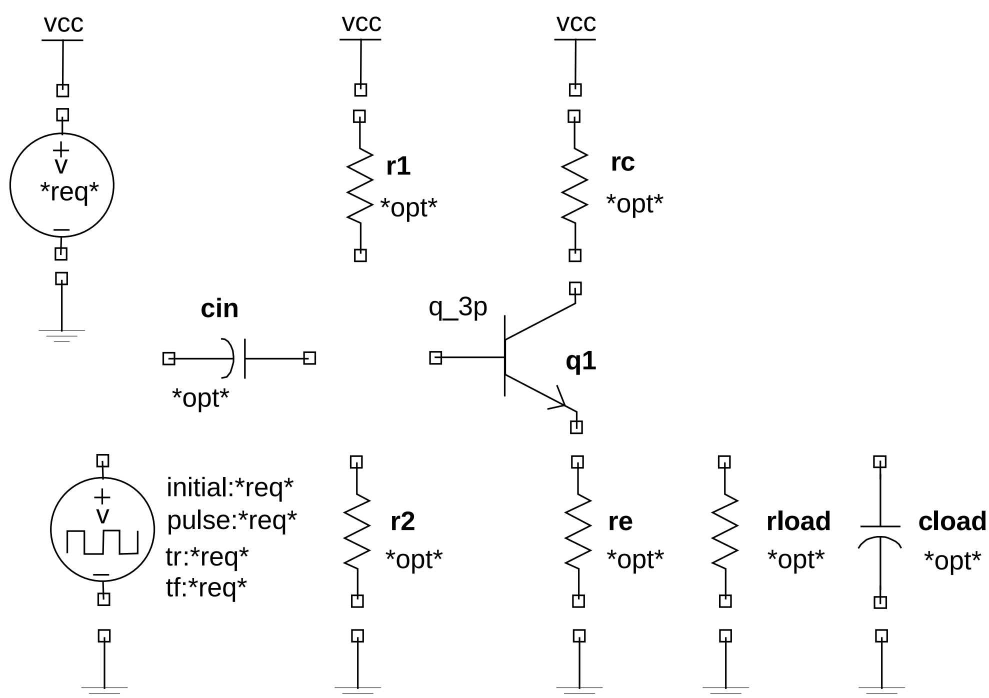
   a. Place the mouse cursor over any icon and hold it there. A text window is displayed to identify the icon. Also look in the Help field at the bottom of the work surface for information about the icon.

   b. Notice that a Schematic window named single_amp appears in the work surface.

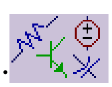
Placing the Parts

In this part of the tutorial you place symbols on the schematic block as shown in the following figure. Parts designators such as `r1` and `r2` have been added to the figure for reference.

For more information on the design parameters that were used with this amplifier, refer to Appendix A.



Single-Stage Transistor Amplifier Parts Placement

1. Find and place the **npn** transistor symbol as follows:
   a. Click the Parts Gallery icon. 
   The Parts Gallery dialog box appears.
   b. Set the following Parts Gallery fields:

Category Name	/
Search String	npn
Available Categories	Leave unselected for the remainder of this tutorial

c. Choose the **Options > Preferences** menu item. The Parts Gallery Preferences form appears.

d. Click the Search tab and select the settings as follows:

Search part by:	Part Name
Search match:	Containing
Ignore case when doing search	selected in the remainder of this tutorial

e. Click **OK**. If it disappears, double click the Parts Gallery icon to bring the Parts Gallery to the front.

f. In the Parts Gallery, click the **Search** button.

A list of all part descriptions that contain "npn" appear in the Available Parts list.

g. In the Available Parts list, select `BJT, 3 pin NPN`.

h. Place the **`npn`** symbol by clicking the **Place** button.

2. Find and place five resistor symbols as follows:

a. In the Parts Gallery, set the fields as follows:

Category Name	`/`
Search String	`res`

b. Choose the **Options > Preferences** menu item. The Parts Gallery Preferences form appears.

c. Click the Search tab and select the settings as follows:

Search part by:	Part Name
Search match:	Beginning with

d. Click **OK**.

e. In the Parts Gallery, click the **Search** button.

f. In the Available Parts list, select `Resistor (|)`.

g. Place five resistors by clicking the **Place** button five times.

h. Turn on the grid in the Schematic window by clicking , the Toggle Grid icon in the SaberSketch Icon Bar.

i. Position the five resistors in relation to the transistor as shown in the previous Parts Placement figure on page 1-3 as follows:

- One-by-one, position the mouse cursor on each resistor
- Press and hold the left mouse button
- Drag the part to the desired location and release the mouse button

j. In the SaberSketch Icon Bar, click the Zoom to Fit icon. Use the Zoom icons to help manage the size of displayed contents.

3. Find and place the **v_pulse** symbol using the following Parts Gallery settings and the previous Parts Placement figure on page 1-3:

Category Name	`/`
Search String	`v_pulse`

a. Choose the **Options > Preferences** menu item. The Parts Gallery Preferences form appears.

b. Click the Search tab and select the settings as follows:

Search part by:	Symbol Name
Search match:	Beginning with

c. Click **OK**.

d. In the Parts Gallery, click the **Search** button.

e. In the Available Parts list, select `Voltage Source, Pulse`.

f. Place the **v_pulse** symbol by clicking the **Place** button and then move it into position.

4. Find and place the **v_dc** symbol using the following Parts Gallery settings:

Category Name	`/`
Search String	v_dc

a. The Parts Gallery Preferences form settings you used in the previous search will work for this search. Click the **Search** button.

b. In the Available Parts list, select `Voltage Source, Constant Ideal DC Supply`.

c. Place the **v_dc** symbol by clicking the **Place** button and then move it into position.

5. Find and place the horizontal capacitor symbol as follows:

Category Name	`/`
Search String	`cap`

a. Choose the **Options > Preferences** menu item.

b. Click the Search tab and select the settings as follows:

Search part by:	Part Name
Search match:	Beginning with

c. Click **OK**.

d. Click the **Search** button.

e. In the Available Parts list, select `Capacitor (-)`.

f. Place the **ch** symbol by clicking the **Place** button.

g. Position the symbol as shown in the previous Parts Placement figure on page 1-3. Rotate the symbol 180 degrees as follows:

- Move the mouse cursor over the symbol to select it.
- Press-and-hold the right mouse button to bring up the popup Symbol Menu. Choose the **Rotate > 180** menu item and release the mouse button.
- Reposition the symbol as necessary.

6. Find and place the vertical capacitor symbol as follows:

a. In the Parts Gallery, use the same settings as you did for step 5, except in the Available Parts list, select `Capacitor (|)`.

b. Place and position the symbol as shown in the previous Parts Placement figure on page 1-3.

7. Find and place six ground symbols using the following Parts Gallery settings:

Category Name	`/`
Search String	`ground`

   a. The Parts Gallery Preferences form settings you used in the previous search will work for this search. Click the **Search** button.

   b. In the Available Parts list, select `Ground, (Saber Node 0)`.

   c. Place the **gnd** symbols by clicking the **Place** button six times.

   d. Position the symbols as shown in the Parts Placement figure on page 1-3.

8. Find and place three **vcc** symbols using the following Parts Gallery settings:

Category Name	`/`
Search String	`vcc`

   a. Using the same search settings that you used in the previous steps, click the **Search** button.

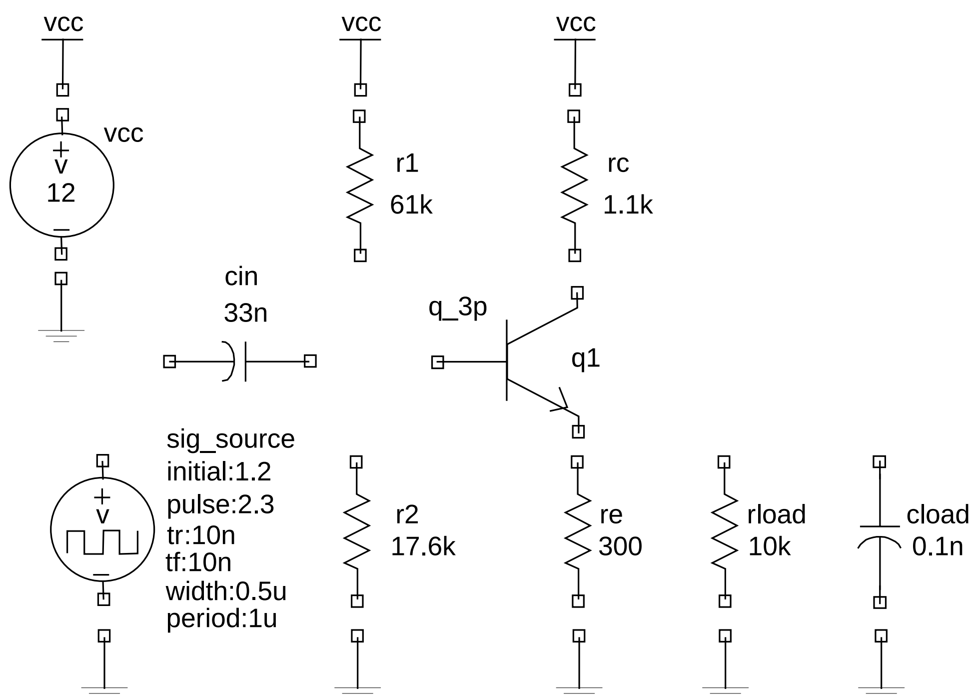
   b. In the Available Parts list, select `vcc`.

   c. Place the **vcc** symbols by clicking the **Place** button three times. Position the symbols as shown in the Parts Placement figure on page 1-3.

9. Close the Parts Gallery.

Editing the Symbol Properties

Once all the parts are placed on the schematic, you can put property values that are specific to this design on each symbol. The following figure shows many of the property values that you will add in this part of the tutorial.



Single-Stage Transistor Amplifier Symbol Property Values

1. For those symbols that have property value place-holders (`*opt*`) already displayed such as the resistor and capacitor symbols, you can change the value on each by doing the following:

   a. Position the mouse cursor at the end of the desired text field.

   b. Click the left mouse button.

   c. Backspace (use the backspace key) over the existing text to delete it.

   d. Type in the new text.

   e. For example, position the mouse cursor at the end of the text field for `rload` as shown in the previous Property Values figure on page 1-8 and replace the `*opt*` field with `10k`.

   f. Repeat steps a through d for each of the following components:

`r1`	`61k`
`r2`	`17.6k`

`re`	`300`
`rc`	`1.1k`
`cin`	`33n`
`cload`	`0.1n`

When you need to change properties values that are not visible, you must use the Property Editor as described in the next step.

2. To change properties on the **v_dc** symbol, do the following:

    a. Position the mouse cursor over the **v_dc** symbol to select it. It changes color to show it is selected. Also note in the Help field at the bottom that the symbol name (**v_dc**), along with an instance number (such as _7), is displayed.

    b. Press the right mouse button and hold it. The popup Symbol Menu appears.

    c. In the Symbol menu, move the mouse cursor over the **Properties...** item and release the mouse button. The Properties dialog box is displayed.

    d. In the Properties dialog box for the selected **v_dc** symbol, change the `ref` property to `vcc`.

    e. Change the `dc_value` property to `12`, which defaults to volts.

    f. In the Properties dialog box, click the **Apply** button.

    g. Notice that the Visibility Indicators (the circles to the right of the Value column) are set so that the Value is displayed for a number of properties. Experiment with these indicators by clicking on one of them and then clicking the **Apply** button to see what is displayed in the Schematic window. Repeat the process, clicking the indicator, followed by the **Apply** button to see what happens with each of the different settings.

    h. Close the Properties dialog box (click **OK**)

3. The procedure for using the Properties dialog box is slightly different for one of the properties on the `q_3p.q1` transistor as follows:

    a. Position the mouse cursor over the **q_3p** symbol to select it.

    b. Press the right mouse button and hold it. The popup Symbol Menu appears.

    c. In the Symbol menu, move the mouse cursor over the **Properties...** item and release the mouse button. The Properties dialog box is displayed.

d. In the Properties dialog box for the selected **q_3p** symbol, change the ref property to q1.

e. Move the mouse cursor to the (type=_n) value for the saber_model property. The cursor changes to a slanted arrow. This indicates that the property value is a either a structure or an array.

f. Click on the (type=_n) value. A STRUC model list box appears.

g. In the STRUC model list box, scroll down to the bf list item. Change the bf entry to 200.

h. In the STRUC model list box, click **OK**.

i. In the Properties dialog box, click the **OK** button to apply the change and close the Properties dialog box.

4. To change the properties on the rest of the symbols, repeat the following steps for each symbol listed in the corresponding table:

   a. Select one symbol at a time by moving the cursor over the symbol and then pressing the right mouse button and holding it. The popup Symbol Menu appears.

   b. In the Symbol menu, move the mouse cursor over the **Properties...** item and release the mouse button. The Properties dialog box is displayed.

   c. Fill out the fields in the Properties dialog box as shown in the following table. The Symbol references shown in the table corresponds to the references shown in the previous Property Values figure on page 1-8. Prior to changing the references, they have ref values such as v1, res1, c1 and so on.

      If a property appears in the Properties dialog box but is not listed in the table, do not change the default value.

   d. When you are finished entering the property values for each symbol, click the **Apply** button.

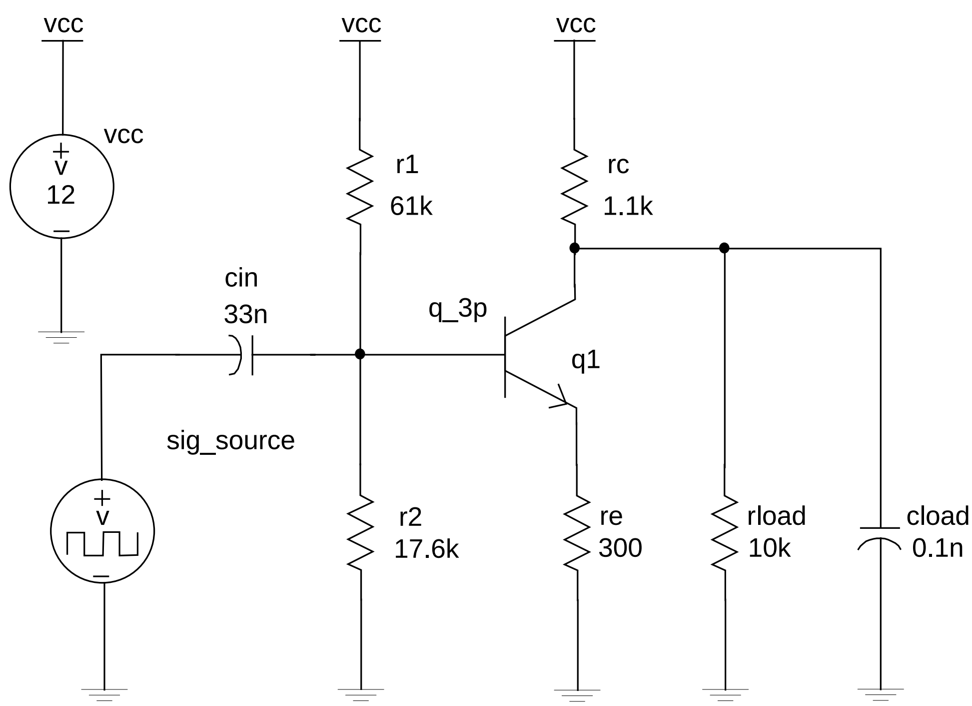
You can get more information about any of the properties on a selected symbol by choosing the **Help > Help on Part** menu item in the Properties dialog box. In the Properties dialog box, use the scroll bar if necessary to see additional properties that do not fit in the displayed list.

Symbol.Reference	Property Name	Value
v_pulse.sig_source	ref	sig_source
	initial	1.2
	pulse	2.3
	tr	10n
	tf	10n
	width	0.5u
	period	1u
	ac_mag	1
r.r1 (61k)	ref	r1
r.r2 (17.6k)	ref	r2
r.rc (1.1k)	ref	rc
r.re (300)	ref	re
r.rload (10k)	ref	rload
c.cin (33n)	ref	cin
c.cload (0.1n)	ref	cload

5. Close the Properties dialog box.

Wiring the Schematic

After the symbols are placed and the properties are set, you can wire the parts together as shown in the following figure.



Single-Stage Transistor Amplifier Wire Connections

In the following procedure, do not wire the intersection of `cin`, `q1`, `r1` and `r2` until specifically instructed to do so.

1. The simplest way to create a wire between two ports is as follows:
   a. Position the mouse cursor over the first port (start with the top of the **v_dc** symbol).
   b. Click the left mouse button.
   c. Position the mouse cursor over the second port (the **vcc** symbol above the **v_dc** symbol).
   d. Click the left mouse button again.
   e. Repeat steps a through d to connect each ground and vcc symbol to the associated part as shown in the previous figure.
2. Position the cursor on the bottom of `rc` and click the left mouse button. Move the cursor to the top of `cload` and click the mouse button again. Notice that a right-angle was formed above the `cload` symbol. By

repositioning the top segment of this wire, you can connect it to the collector of q1 as follows:

a. Move the mouse cursor to the wire segment that connects to rc.

b. Press-and-hold the left mouse button.

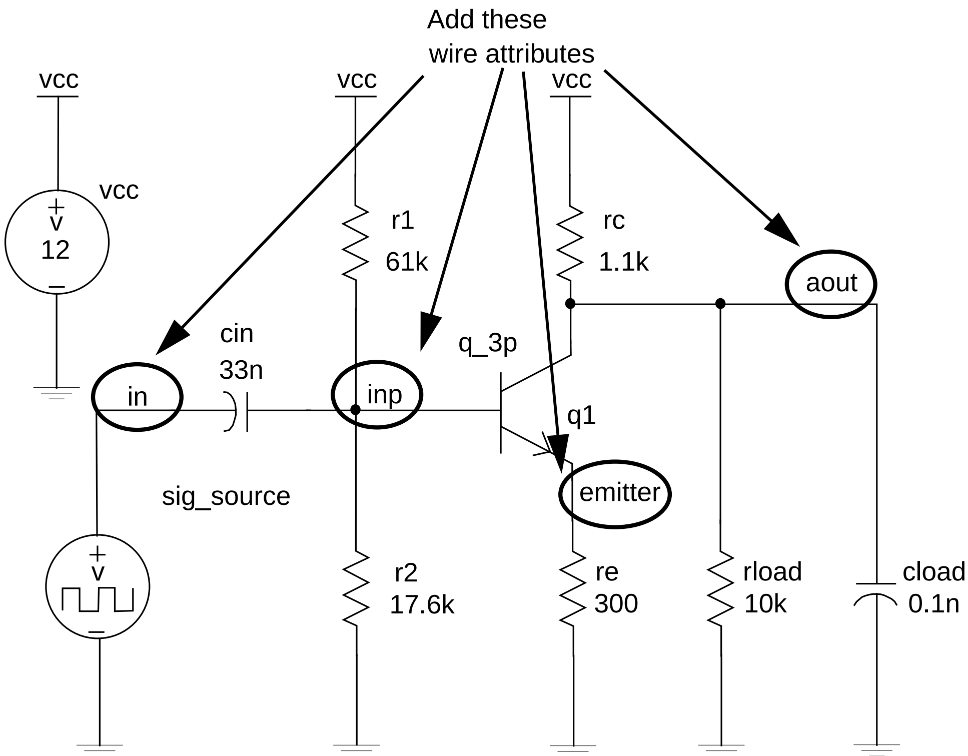
c. Drag the wire segment down until it contacts the collector of q1.

d. Release the mouse button.

3. Connect re to the q1 emitter.
4. Connect rload straight up to the wire formed from the cload-rc connection.
5. Connect sig_source to cin.
6. Connect the base of q1 to cin. Refer to the previous Wire Connections figure on page 1-12 as needed.
7. To connect r1 and r2 to the q1-cin intersection, you must draw two wires; one from r1 to the intersection, and one from r2 to the intersection. If you draw a wire from r2 to r1, there will be no connection point at the cin-q1 intersection.

Modifying Wire Attributes

If you do not label wires, SaberSketch automatically provides names for each wire, such as _n30. It can be helpful later on during design analysis if you label the wires with a meaningful designator that is easy to read and understand.



Single-Stage Transistor Amplifier Wire Attributes

Wires connected to vcc or ground are labeled by SaberSketch with vcc and gnd respectively. There is no need to change those designators.

To add the four wire attributes that are shown in the previous figure, do the following:

1. For the first wire attribute, move the mouse cursor to the desired wire to select it. It changes color.
2. Press and hold the right mouse button to display the popup Wire Menu.
3. In the popup Wire Menu, select the **Attributes...** item and release the mouse button. The Wire Attributes dialog box is displayed.
4. Change the value in the Name field to the desired text string. Refer to the previous Wire Attributes figure.
5. In the Wire Attributes dialog box, click **Yes** in the Display Name field.

6. Click the **Apply** button followed by the **Close** button.
7. Select the next wire by moving the mouse cursor to the desired wire and clicking the left mouse button. The existing wire name appears in the Wire Attributes dialog box Name field.
8. Repeat steps 4 through 7 until you have finished changing the wire attributes.
9. Close the Wire Attributes dialog box.
10. Save the design by clicking the Save icon. 

At this point you have a completed design that is ready to be analyzed. In the next part of the tutorial, you use SaberGuide to simulate the amplifier.

chapter 2

Analyzing the Design Using SaberDesigner

Once you have a design created with SaberSketch that is ready to simulate, you can access the SaberGuide Simulation Environment and analyze the design.

In this part of the tutorial you use SaberGuide to analyze the single-stage amplifier design that you created in the previous part with SaberSketch. These procedures show the following:

- How to perform a transient analysis that automatically performs the necessary DC analysis
- How to perform a small-signal frequency analysis
- How to vary the output load and analyze the results
- How to place probes and view simulation results on the schematic

Starting SaberGuide from SaberSketch

In the following procedure you start SaberGuide from SaberSketch, which already has the single-stage amplifier design open:

1. If the SaberGuide Icon bar is not visible, click the Show/Hide SaberGuide icon in SaberSketch. 
2. If the SaberGuide Transcript window is not open, click the Simulation Transcript icon. 

   The SaberGuide Transcript window contains the result of any command execution.

Performing a Transient and DC Analysis

In this part you perform a transient analysis. In the next part of this tutorial you display the results with SaberScope to check the output signal shape.

Before the transient analysis is executed, it is important to find the DC operating point of the circuit. The DC operating point is used as the first data point in the transient (time-domain) analysis. The DC analysis is initiated from the Time-Domain Transient (tr) Analysis dialog box.

Before any of the analysis can occur, a netlist must be created for the design. This is done automatically when you initiate the transient analysis.

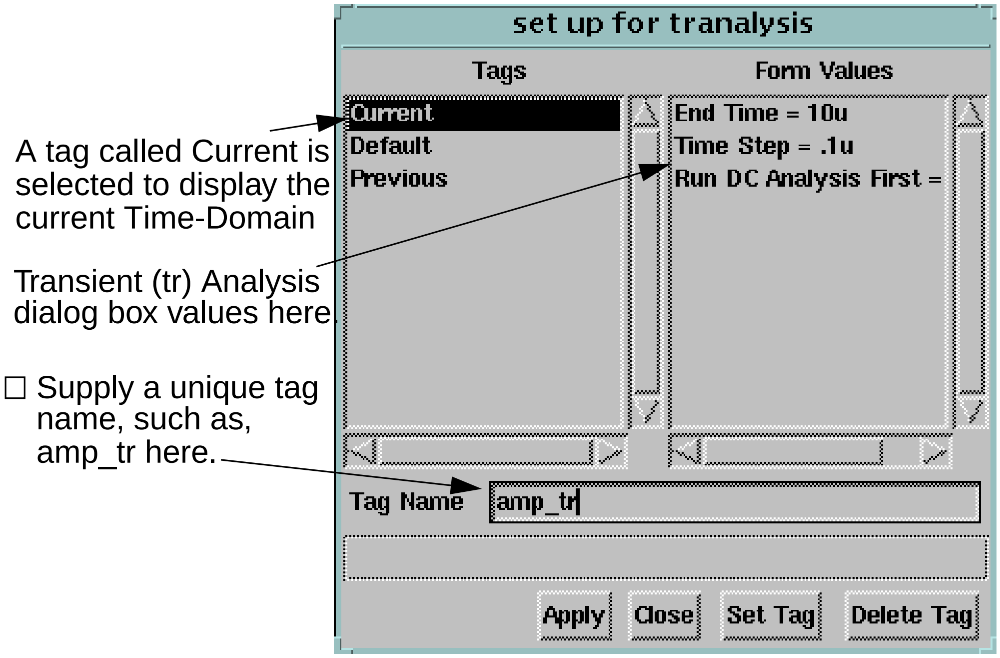
1. Click the Transient Analysis icon  to bring up the Time-Domain Transient Analysis dialog box. The Basic form is displayed.
2. Fill out the Basic form as follows:

End Time	`10u`	This value informs Saber to simulate the circuit for 10us.
Time Step	`.1u`	This value gives Saber a time increment to begin solving the circuit.
Run DC Analysis First	**Yes**	The DC Operating point must be determined before the transient analysis can be calculated.

   For general information on DC analysis, refer to Appendix B, DC Operating Point Analysis.

3. You use these transient analysis setup values later in the tutorial, so save them now as follows:

a. In the Time-Domain Transient Analysis dialog box, click **Defaults**. The set up for tranalysis dialog box appears as follows:



b. In the set up for tranalysis dialog box, enter a unique name `amp_tr` in the Tag Name field.

c. With the Current tag selected, click **Set Tag**.

The `amp_tr` tag name is added to the Tags list and the values you entered in the set up for tranalysis dialog box appear in the Form Values list.

d. Still in the set up for tranalysis dialog box, click **Apply**.

e. In the Time-Domain Transient Analysis dialog box, click the Input/Output tab.

f. In the Signal List field set to plot all by clicking **Select > All Signals** (`/...`).

g. Change the Include Signal Types field to the **All Variables** setting (to include both across and through variables). This will allow you to plot current (pin flow) and voltage data later on.

h. In the Time-Domain Transient (tr) Analysis dialog box, click **OK**.

i. A Yes or No dialog box may be displayed that informs you that the single_amp design will be saved before netlisting and asks if you want to continue. Click **Yes**. The netlister starts.

The values you entered in the Time-Domain Transient Analysis dialog box are not saved if you leave the application. To do a save-on-exit,

choose the following SaberSketch window menu item:
File > Configuration > Save on Exit.

4. Look in the SaberGuide Transcript window. (You should have opened this window from the topic "Starting SaberGuide from SaberSketch" on page 2-1, Step 2.)

   The transcript window displays the license information, followed by the specific commands that were executed after the netlister finished. You can scroll the window to see the text. This window first displays information such as the version of Saber being used, the specific licenses, and templates used by this design.

   The DC analysis (dc) execution time is displayed. In this step you did not fill out the Operating Point Analysis form, so default settings were used to determine the DC values for each node in the design. In most cases, the Saber simulator finds the operating point using the default settings in the Operating Point Analysis form. A file with a `.dc` extension is created to hold the results of the analysis. If you want to display the DC analysis results, do the following:

   a. In the SaberSketch window, choose the **Results > Operating Point Report...** menu item to display the Operating Point Report dialog box.

   b. In the Operating Point Report dialog box, click **OK**. This informs Saber to use the default settings to display the most recent DC operating point values in the Report Tool window.

   c. Refer to the DC results in the Report Tool window and the schematic to help analyze the results. If necessary, you can resize the window or use the scroll bar to view all of the results. The results are also displayed in the SaberGuide Transcript window.

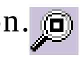
   d. Close the Report Tool window.

   Before you did steps a through c, the transient analysis (tr) execution time was displayed in the SaberGuide Transcript window; this indicates the completion of the transient simulation. A file with a “`.tr.ai_pl`” extension was created to hold the results of the transient analysis.

Placing Probes in the Design

One way to view simulation results is to place a probe on a wire or node in a design as described in the following steps:

1. In the Schematic window, move the mouse cursor to the `aout` wire to select it. The wire changes color.
2. Press and hold the right mouse button to display the popup Wire Menu.
3. In the popup Wire Menu, select the **Probe** item and release the mouse button. A Probe window is displayed and attached to the `aout` wire with a voltage waveform displayed.

   At this point it might be helpful to click the Zoom to Fit icon. 

   You can resize a Probe window by moving the mouse cursor to one of the corners and (grabbing) clicking-and-holding the left mouse button while repositioning the corner. You can also move the window around in the design similarly by grabbing any side of the window.
4. If the transient results are not already displayed in the Probe window, you can load a plot file as follows:
   a. Move the mouse cursor inside of the `aout` Probe window.
   b. Press and hold the right mouse button to display the popup Probe Menu.
   c. In the popup Probe Menu, select the **Open Plotfile...** item. An Open Plotfile dialog box is displayed.
   d. In the list box, select `single_amp.tr.ai_pl`.
   e. Click the **Open** button. Click **Yes** in the resulting dialog box. The output waveform is displayed in the Probe window.
5. Place a probe on the input (`in`) signal as you did in the previous steps.
6. From the results you can see that the shape of the output is being rounded by the circuit capacitance.
7. Delete the Probe windows on the input (`in`) signal as follows:
   a. Move the cursor inside the Probe window and click-and-hold the right mouse button to display the popup Probe Menu.
   b. Select the **Delete** menu item.
8. Plot the current flowing through the `cload` capacitor as follows:

a. Move the cursor to the Probe arrow currently on the `aout` signal (it turns color).

b. Click-and-hold the left mouse button and drag the arrow to the middle of the `cload` capacitor symbol. Release the mouse button. A Select Port & Signal window appears.

c. In the Select Port & Signal window, select the `p` pin.

d. Then change the Signal Type to `Through` and click **OK**. A current flow waveform is displayed in the Probe window.

e. Delete the remaining Probe window from the schematic. (See Step 7).

Performing AC Analysis and Invoking SaberScope

In this part of the tutorial you perform a small-signal frequency analysis to check the frequency response and gain of the amplifier. In addition, you invoke SaberScope.

1. Bring up the Small-Signal Frequency Analysis dialog box by clicking the Frequency Response icon. The Basic form is displayed. 

2. Fill out the form as follows:

Start Frequency	`100` (Hertz is the default unit.)
End Frequency	`10meg`
Plot After Analysis	`Yes - Open Only`

3. Without changing the other fields, click **OK**. The AC analysis execution time is recorded in the SaberGuide Transcript window. A file is created with a `.ac.ai_pl` extension to hold the results of the analysis.

   The last action recorded in the SaberGuide Transcript window was the invocation of SaberScope because you set the Plot After Analysis field to `Yes - Open Only`. SaberScope is displayed along with the Signal Manager window and a single_amp.ac Plot File window. From the Plot File window you can select and plot the signal of interest as described in the next topic.

Analyzing Waveforms with SaberScope

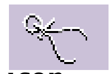
After the transient and AC analysis are complete, you can view the results with the SaberScope Waveform Analyzer.

1. To remove some screen clutter, iconify the SaberSketch window.
2. In the single_amp.ac Plot File window, select signal `aout` by left-clicking it. The signal is highlighted.
3. Plot the selected signal on the graph by either clicking the **Plot** button in the Plot File window, or by moving the mouse cursor to the Graph window and clicking the middle-mouse button. The `aout` waveforms are displayed in the Graph window.
4. In this tutorial you do not need the `Phase(deg):f(Hz)` waveform. To delete it from the Graph window, do the following:
   a. Move the mouse cursor to the `aout` signal name associated with the `Phase(deg):f(Hz)` plot. The `aout` signal name and the waveform changes color.
   b. Press and hold the right mouse button to bring up the Signal Menu.
   c. Select the **Delete Signal** item.
5. Even though you have already seen the results of the transient analysis, do the following to see how you can plot additional waveforms to the Graph window:
   a. In the Signal Manager dialog box, click the **Open Plotfiles...** button. The Open Plotfiles dialog box appears.
   b. In the File Name list, click on `single_amp.tr.ai_pl`.
   c. Click the **Open** button. A single_amp.tr Plot File window is displayed.
   d. From the single_amp.tr Plot File window, plot the `aout` and `in` signals as you did in steps 2 and 3. Two new waveforms are added to the graph window.
   e. Note that the `(V):t(s)` plots of the `in` and `aout` waveforms are identical to those displayed on the design in SaberSketch. Delete the `in` and `aout` waveforms when you have finished viewing them.
6. Look at the `aout dB(V):f(Hz)` (dB in volts versus frequency in Hertz) waveform in the Graph window.

   From the waveform you can see that the gain is about 10dB from about 2000 Hz to 75 kHz. The next part of this tutorial uses the Measurement Tool on this waveform to get some accurate readings on the gain and the frequency response.

Performing Measurements on a Waveform

The Measurement Tool within SaberScope provides a method of performing various measurements on a waveform. You check the bandwidth and gain of the single-stage amplifier output signal (aout) as follows:

1. Close the Plot File windows and the Signal Manager window.
2. In the Tool Bar located at the bottom of the SaberScope window, click the Measurement icon. 

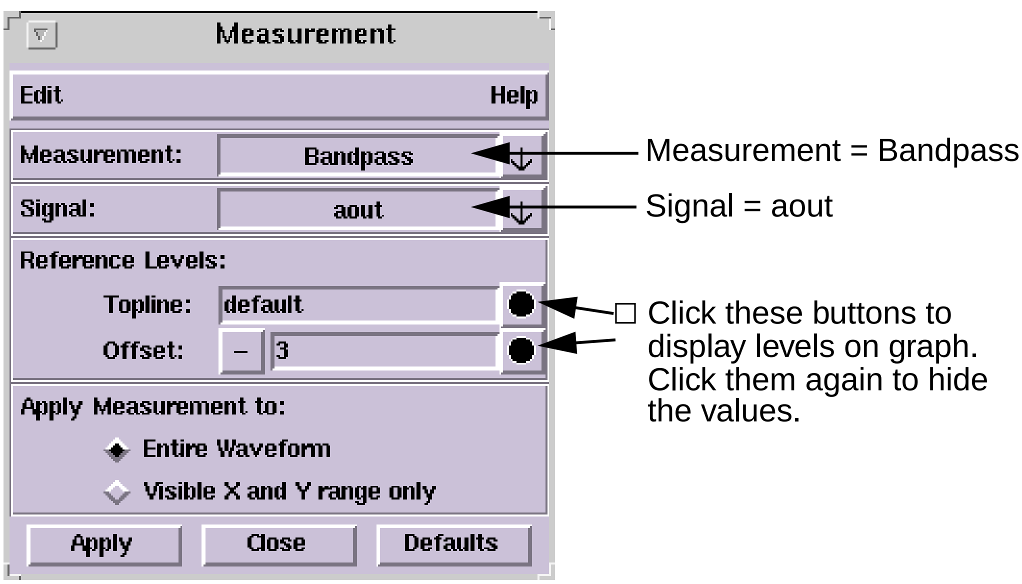
   The Measurement dialog box appears.
3. Select the Bandpass measurement as follows:
   a. Move the mouse cursor to the right of the Measurement field and press and hold the left mouse button on the down arrow  button.

   b. Move the mouse cursor down to the **Frequency Domain** menu.

   c. Select **Bandpass**.

   To summarize, choose the **Measurement > Frequency Domain > Bandpass** menu item.

   d. Because there is only one signal in the Graph window, aout should appear in the Signal field in the Measurement dialog box as shown in the following figure.



   e. If you want to see values displayed on the graph for Topline and Offset that are used in the bandpass calculation, click the visibility indicator buttons to the right of the perspective Reference Levels fields.

f. Click the **Apply** button. The bandwidth is displayed on the graph.

4. Select the Gain Margin measurement by doing the following:

   a. Choose the **Measurement > Frequency Domain > Gain Margin** menu item.

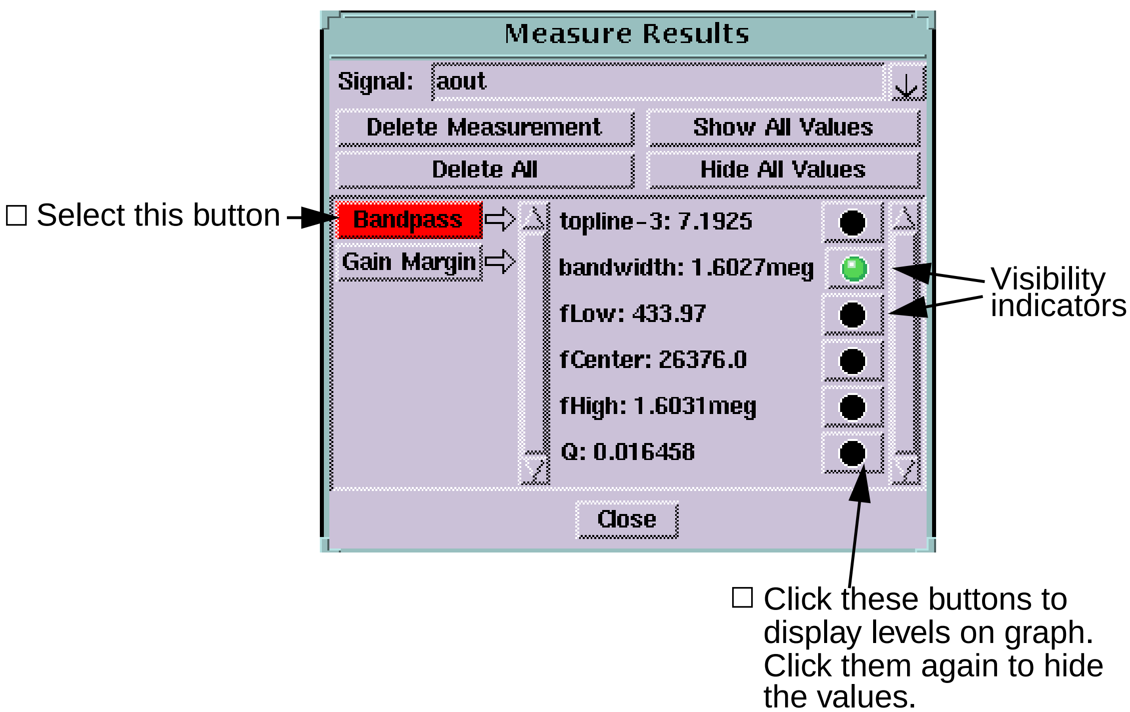
   b. Click the **Apply** button. The gain margin is displayed on the graph.

5. You can get more information about each of the measures you performed or control the amount of information displayed in the Graph window by using the Measure Results dialog box as follows:

   a. In the Graph window, move the mouse cursor to the aout signal name.

   b. Use the popup menu and choose the **Signal Menu > Measure Results...** item. A Measure Results dialog box appears.

   c. In the Measure Results dialog box, be sure the Bandpass item in the left column is selected as shown in the following figure:



   d. Notice in the Measure Results dialog box, in the right column, the different values that are available from executing the bandpass measurement.

   e. Click on the various visibility indicators to choose which values are displayed in the Graph window.

f. When you have finished exploring the Measure Results dialog box, close it.

Varying a Parameter

In this part of the tutorial you vary the value of the emitter resistor (re) from 200 to 400 ohms in 20 ohm steps while performing three analyses for each resistor value: DC operating point analysis, transient analysis, and small-signal analysis.

1. Iconify the SaberScope window and bring up the SaberSketch window.
2. Click on the Vary icon, . The Looping Commands dialog box appears.
3. In the Looping Commands dialog box, click the **vary** button. A Parameter Sweep dialog box appears.
4. Fill out the Parameter Sweep dialog box as follows:

Parameter Name	`r.re`	Specify the parameter using the primitive name (r) followed by the symbol reference (re)
Variation Type	**Step By**	

from
to
by
	`200`
	`400`
	`20`
		Causes Saber to vary the resistor from 200 ohms to 400 ohms in increments of 20 ohms.

Unit	_	You can leave this field as an underscore (_) or enter `ohms`.

   Click **Accept**.

5. Define which analysis to run on each swept parameter value as follows:

   a. In the Looping Commands dialog box, choose the **AddAnalysis > Within Loop(s) > Transient** menu item. A new button (**tranalysis**) appears in the Looping Commands dialog box.

   b. Click **tranalysis**. The Time-Domain Transient Analysis dialog box appears. If you have not closed SaberSketch since you began this tutorial, you see the values you entered earlier. Steps c through e show how to restore previously-saved settings.

   c. To recall the transient analysis settings you previously stored, click **Defaults...** in the Time-Domain Transient Analysis dialog box. The set up for tranalysis dialog box appears.

d. Select amp_tr in the Tags list. This is the name of the file you saved previously.

   Note that the saved transient analysis settings specified that a DC operating point would be executed prior to each transient analysis. A new operating point should be computed for each value of the re resistor.

e. In the set up for tranalysis dialog box, click **Apply**.

f. In the Time-Domain Transient Analysis dialog box, click the Input/Output tab.

g. In the Plot File and Data File fields, add the string “vary_” in front of the existing “tr” string so that it reads “vary_tr”. The results of this transient looping do not overwrite the previous plot files because you are specifying different file names.

h. If you do not need to view pin flow data for this vary operation, limit the data to across variables. This will produce a smaller plot file. Set the Include Signal Types field to **Across Variables Only**.

i. In the Time-Domain Transient Analysis dialog box, click **Accept**.

j. In the Looping Commands dialog box, choose the **AddAnalysis > Within Loop(s) > Small Signal AC** menu item. A new button (**acanalysis**) appears in the Looping Commands dialog box.

k. Click **acanalysis**. The Small-Signal Frequency Analysis dialog box appears. If you have not closed SaberSketch since you began this tutorial, you see the values you entered earlier.

l. In the Basic tab verify the following settings.

Start Frequency	100 (Hertz is the default unit.)
End Frequency	10meg
Plot After Analysis	No

m. Click the Input/Output tab.

n. In the Plot File and Data File fields, add the string “vary_” in front of the existing “ac” string so that it reads “vary_ac”. This creates a new name for the plot and data file and prevents the new results from overwriting the existing files.

o. In the Small-Signal Frequency Analysis dialog box, click **Accept**.

6. Execute the parameter sweep by clicking the **OK** button in the Looping Commands dialog box.

Observe the looping activity in the SaberGuide Transcript window. After the simulation has completed, you can display the results of the parameter sweep.

7. Exit the SaberSketch application (**File > Exit**). You can choose to save the design upon exiting.

Displaying the Parameter Sweep Results

In this part you load the plot files produced from the vary simulation and display them with SaberScope.

1. Bring up the SaberScope window.
2. From the SaberScope icon bar, click the Clear Graph icon  to remove previous waveforms.
3. If it is open, close the `single_amp.ac` Plot File window.
4. From the Signal Manager, click the **Open Plot Files...** button. The Open Plot Files dialog box is displayed.
5. Make sure that **Plot Files** is selected in the Files of Type field.
6. From the Open Plot Files dialog box, click on `single_amp.vary_ac.ai_pl`, then shift-click on `single_amp.vary_tr.ai_pl` file.
7. Click **Open**.
8. From the single_amp.vary_ac Plot File window, plot `aout`. The graph shows a multi-member waveform that shows the effect of varying the `r.re` resistor.
9. Delete the `Phase(deg):f(Hz)` waveform from the Graph.
10. From the `single_amp.vary_tr` Plot File window, plot the `aout` and `in` signals on the same X/Y axis as the previous `aout` signal by doing the following:
    a. In the `single_amp.vary_tr` Plot File window, select both the `aout` and `in` signals. (<Control>-click each item.)
    b. (3-button mouse system) Move the mouse cursor to the Graph window and click the middle-mouse button.

       (2-button mouse system) Do the following:

       a. Click the **Plot** button.
       b. In the Graph window, select both the new `aout` and `in` signals.

c. Move the cursor to the middle of the Graph, and click-and-hold the right mouse button to display the Graph menu.

d. In the Graph menu, select the following choice: **Selected Signals > Stack Region > Analog 0**.

11. To help manage the displayed data, do the following:
    a. Select the `dB(V):f(Hz) aout` signal.
    b. Choose the (popup) **Signal Menu > Members...** menu item. A Members Attributes dialog box is displayed.
    c. Click on the **Hide All** button. The results of the small-signal analysis are now hidden from the graph.
12. You can now observe the multi-member `(V):t(s) aout` signal as follows:
    a. Select the `(V):t(s) aout` signal.
    b. Choose the (popup) **Signal Menu > Members...** menu item. The Members Attributes dialog box displays the `aout` data.
    c. Click the **Hide All** button. The waveforms disappear.
    d. Click the **Loop Highlight** button. The graph loops through each waveform until you click the **Loop Highlight** button again to stop the looping.
    e. Experiment with the Member Attributes dialog box until you are comfortable with the various features.
13. Delete the `(V):t(s) aout` and `in` signals from the Graph window. The Member Attributes dialog box is now controlling the remaining `dB(V):f(Hz) aout` signal.
14. In the Member Attributes dialog box, click the **Show All Members** button.
15. Close the Member Attributes dialog box.

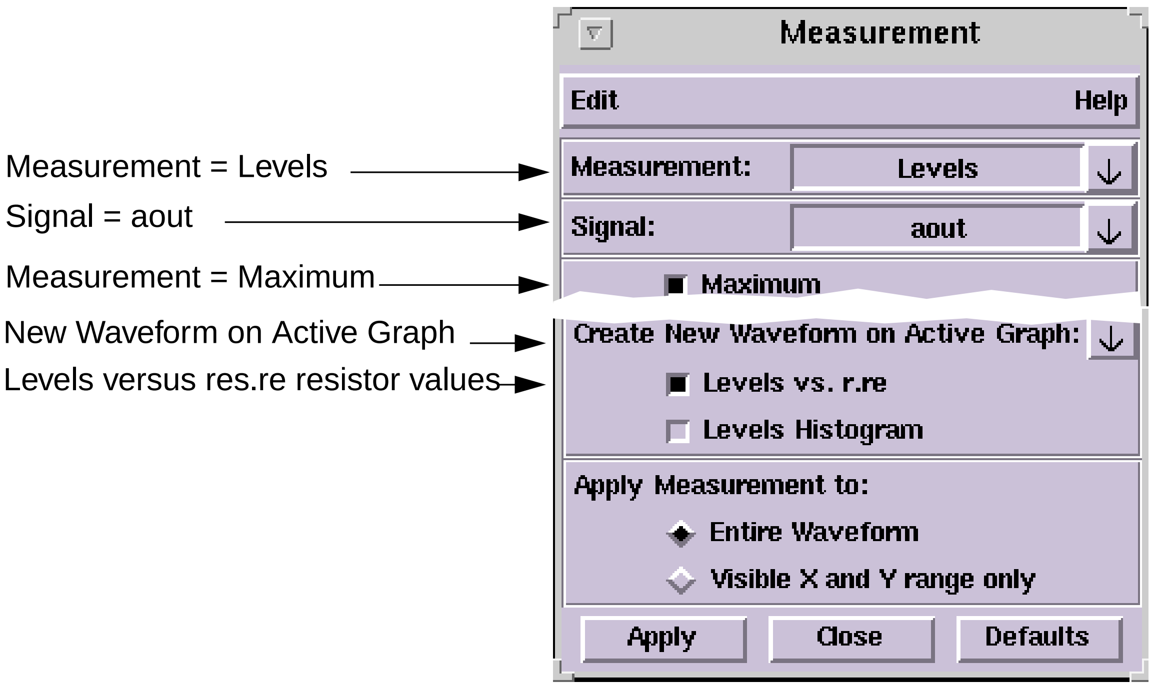
Measuring a Multi-Member Waveform

The Graph window shows how the output signal, aout fluctuates as you change the resistor value. What would be more useful is a graph that correlates the actual resistance value with the output voltage level.

NOTE

To perform a measurement on a multi-member waveform, you must have either the InSpecs Parametric Analysis Option or the InSpecs Statistical Analysis Option installed. Skip to the Conclusion *section if you do not have either of these options.*

1. Fill out the Measurement dialog box as shown in the following figure and described as follows:



   a. In the Measurement dialog box, the Measurement field, select **Levels > Maximum**.

   b. In the Signals field, verify that aout is selected (it is the only signal in the list.)

   c. Be sure that Create New Waveform on Active Graph is selected. (Using the down-arrow next to this field, you could also choose to create a new graph.)

   d. Be sure that Levels vs. r.re is selected.

   e. Apply the measurement to the entire waveform.

2. Click **Apply**.

   A new waveform for signal `Max(dB(aout))` has been created. The resistor value is plotted on the X-axis and the dB(V) level is plotted on the Y-axis.

   The graph appears as a continuous curve, when in actuality, you only simulated every 20 ohms, which you specified in the vary form. You have full control over how graphs are displayed, (shown in a later step).

3. Delete the un-needed waveform for `aout`.

4. To help analyze the new graph, display the Point Marker using the Measurement Tool as follows:

   a. In the Measurement dialog box, the Measurement field, choose the **General > Point Marker** menu item. (Check that the Signal field entry is `Max(dB(aout))`.

   b. Using the default values for the rest of the fields, click **Apply**. A marker appears on the `Max(dB(aout))` waveform.

5. Move the mouse cursor to the Point Marker. Left-click, hold, and pan the Point Marker until the Y-value equals a gain of 10. An emitter resistor value of around 307 (X-value) produces a gain of 10.

6. To get a feel for the graphing options available, do the following:

   a. Select the `Max(dB(aout))` signal name on the graph.

   b. With the mouse, right-click and hold in the graph area and choose the popup **Graph Menu > Selected Signals > Style > None** menu item.

   c. With `Max(dB(aout))` signal selected, again, right-click in the graph area and choose the **Selected Signals > Symbol > "square symbol."**

      In the resulting graph, a square represents each plotted data point from the analysis. You may want to try out some of the other options on your own.

7. Close SaberScope by choosing the **File > Exit** menu choice.

Conclusion

In the first part of this tutorial you were introduced to design creation using SaberSketch. You have seen how to choose and place parts and edit symbol and wire properties.

In the second part of the tutorial you have seen how to analyze a design. You have covered the concepts of establishing an operating point for a circuit. You performed a time-domain analysis and displayed the results on the design schematic much as you might probe an actual circuit with an oscilloscope. You evaluated the circuits frequency response in SaberScope and performed measurements on the waveforms, which greatly simplifies the design verification process.

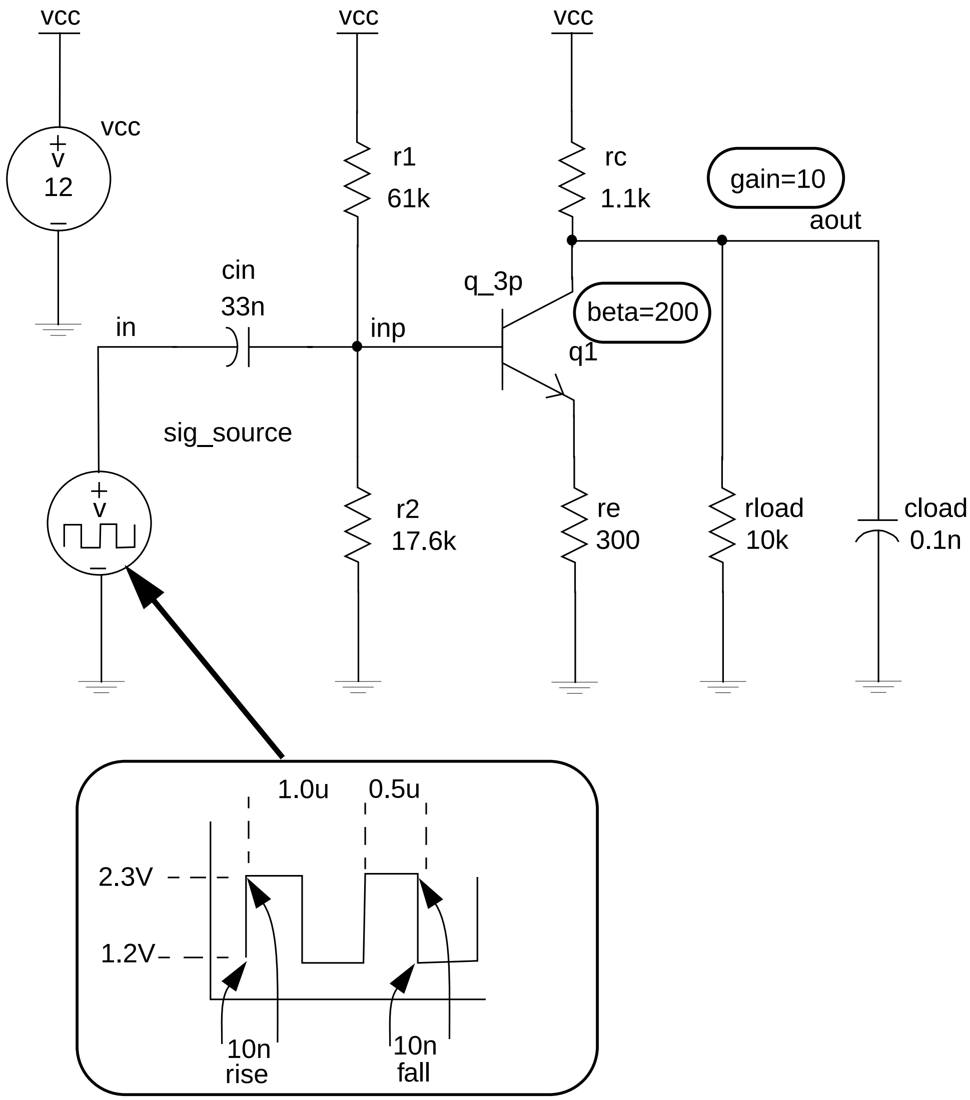
The procedures shown in this tutorial cover only a few of the many features in SaberDesigner. You may want to re-run through the steps, varying the simulation approach from that suggested in this tutorial.

appendix A

Amplifier Design Parameters

The transistor amplifier used in this tutorial is a text-book example. To create this example for SaberDesigner, some design parameters were set that determined which symbols were needed. For example, we wanted to set the beta of the transistor rather than use a characterized part. Therefore, we needed a symbol that would allow us to do so, such as the **q_3p** symbol allows.

The design needed an input signal source. The **v_pulse** symbol was chosen so that we could apply a square wave to the input. The vcc required a DC source, and the **v_dc** symbol was chosen.



Single-Stage Transistor Amplifier Design Parameters

appendix B

SaberDesigner Concepts

DC Operating Point Analysis

The DC operating point needs to be established prior to performing certain types of analyses (including time-domain and frequency-based simulations). The DC analysis determines a stable condition for the circuit when time = 0, effectively removing the dynamic elements in the design. For electrical circuits (as shown in this tutorial), the DC analysis solves the system with all capacitors opened and all inductors shorted.

The DC operating point analysis can be started prior to a small-signal or transient analysis from the SaberGuide Icon bar by clicking the Operating Point icon. It can also be specified to run when you fill out the form for the small-signal or transient analysis.

After you verify that the calculated DC operating point is correct, you are ready to proceed to checking the time-domain specification. The DC operating point is used as the first data point in the transient (time-domain) analysis run.