

# RFSim99 Help

## Table of contents

---

overview .....	4
RFSim99 Limitations .....	4
The Future .....	5
schematic editor .....	5
placing components .....	7
adding connecting wires .....	8
adding a ground connection .....	8
adding measurement ports .....	9
starting simulation .....	9
starting simulation .....	10
starting simulation .....	11
starting simulation .....	12
starting simulation .....	13
Match Network .....	14
Filter .....	16
tolerance analysis .....	16
S Parameter Transistor Design .....	17
An Auto Match Example .....	18
S Parameter Transistor Design .....	19
placing components .....	20
placing components .....	21
connecting wires .....	21
measurement ports .....	22
starting simulation .....	22
printing .....	23
setting schematic preferences .....	23
graph window .....	24
changing graph type .....	25
tune feature .....	26
tolerance analysis .....	26
stability circles .....	27
graph preferences .....	27
resistor model .....	28
capacitor model .....	29
inductor model .....	30
transmission line model .....	31
transformer model .....	32
S parameter files .....	33
op-amp model .....	34
match assistant .....	35
attenuator assistant .....	35
filter assistant .....	36
auto match .....	37
resonance reactance calculator .....	38
return loss VSWR calculator .....	39
signal level calculator .....	40
frequency wavelength calculator .....	40

thermal noise calculator .....	41
inductance component designer .....	41
capacitance component designer .....	42
transmission line component designer .....	43
coupler component designer .....	43
splitter component designer .....	44
matching complex impedances .....	45
analysis of balanced circuits .....	45
upper and lower tolerance limits .....	45
licence agreement .....	46
disclaimervent si> <sup>3</sup> / <sub>4</sub> H☐ .....	47

## overview

---

**RFSim99** is a linear S-parameter based circuit simulator offering the following features:-

- schematic capture front end
- one touch simulation....draw your circuit then just hit the simulate button
- analysis and display of full two-port S-parameters in Smith, polar, square grid or tabular formats
- 1 port and 2 port S-parameter file support
- tolerance analysis
- stability circles
- auto-match , for automatic complex match network generation
- match network, filter and attenuator design assistants
- circuit tune mode
- calculator for resonance/reactance, power, VSWR, wavelength etc.

[>> click here for Getting Started Tutorial<<](#)

[Software License Agreement](#)

[Disclaimer](#)

***Click on the 'Contents' or 'Index' buttons on the help window tool bar to find help on a specific topic.***

---

Created with the Personal Edition of HelpNDoc: [Full-featured EPub generator](#)

---

## RFSim99 Limitations

---

The following limits may not be exceeded in **RFSim99**:

- 100 nodes in circuit file
- 1000 wires
- 1000 components
- 50 S parameter files with up to 2000 lines in each file
- 500 graph points

**RFSim99** is a **linear frequency domain simulator** only.

Time domain performance of circuits may only be inferred by analyzing their frequency and phase responses with **RFSim99**.

For the purposes of linear frequency domain simulation, all voltage sources (such as power supplies) are regarded as an AC short to ground. For this reason no power supply or DC voltage source components are available in **RFSim99**.

For the purposes of simulation, real world power supplies may be realistically modeled as an impedance to

ground.

DC voltages in circuits are not analyzed ; if bias networks are to be analyzed then these may be included as impedances to ground.

**Non-Linear circuits may not be analyzed using *RFSim99***

[>> click here to continue <<](#)

---

Created with the Personal Edition of HelpNDoc: [News and information about help authoring tools and software](#)

---

## The Future

---

***RFSim2000 is coming***

**New features planned for RFSim2000:**

- MDI (Multiple Document Interface) schematic editor
- Schematic zoom and full complement of component orientations
- Simultaneous display of multiple graph outputs
- Optimisation
- Improved match assistant
- Improved filter assistant
- Improved tune facility
- Improved tolerance analysis
- Voltage and current source models
- Microstrip bends and end effects
- Undo

[>> click here to continue <<](#)

---

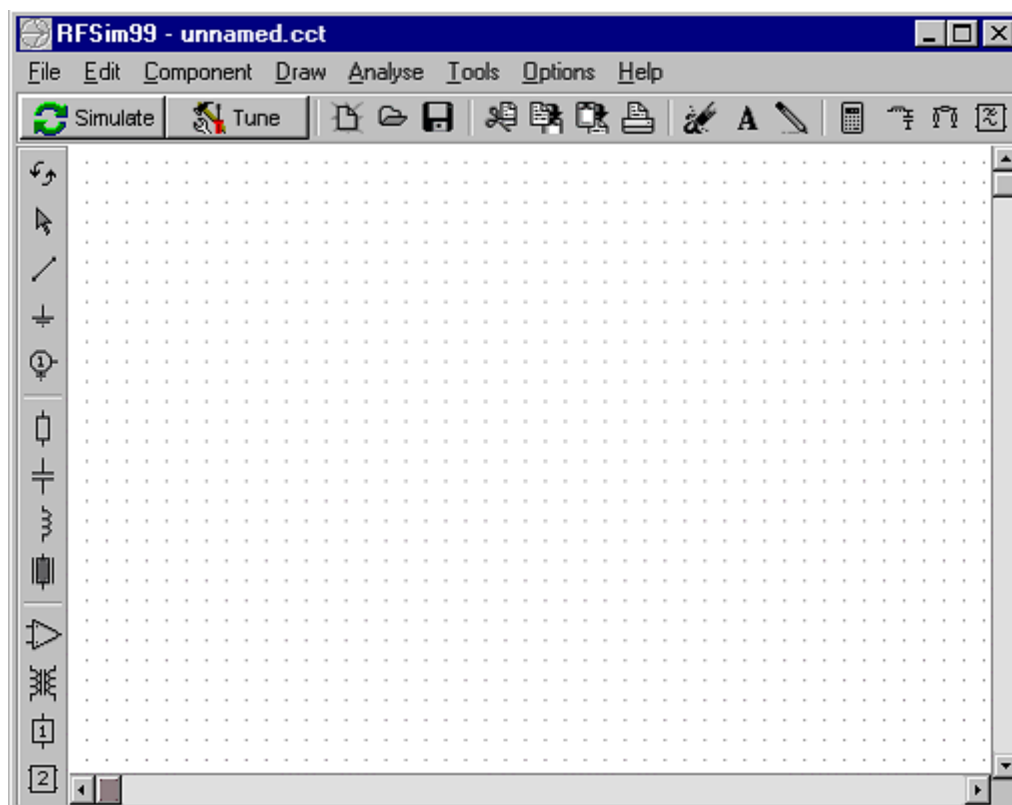
Created with the Personal Edition of HelpNDoc: [Easily create CHM Help documents](#)

---

## schematic editor






























---

After starting RFSim99 you will be presented with a blank schematic as shown below



All functions of the schematic editor may be found in the menu bar at the top of the window, however many of the more common menu items have associated tool bar buttons along the top, and down the left hand side of the window.

Button functions are outlined below

	simulate the schematic circuit		
	tune component values		
	create a new file		change component orientation
	open an existing file		select mode
	save the current file		connecting wire
	cuts to the clip board		ground connection
	copy to the clip board		measurement port
	paste from the clip board		resistor
	opens print/ print preview dialog		capacitor
	delete selection		inductor
	place graphical text		transmission line
	draw graphical line		op-amp
	show RF calculator		transformer
	launch match assistant		1 port S parameter block
	launch attenuator assistant		2 port S parameter block
	launch filter assistant		

In order to illustrate the operation of RFSim99 we will analyze a simple C/R filter circuit.

We will start by placing some components on the schematic.

[>> click here to continue <<](#)


---

Created with the Personal Edition of HelpNDoc: [Full-featured Help generator](#)

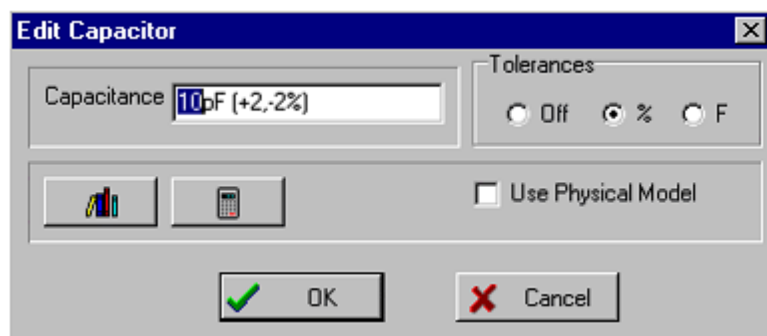
---


## placing components

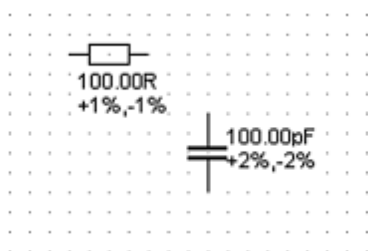
---

Pick a capacitor from the tool bar by clicking on  (or press 'c' on the keyboard), then place it in the middle of the schematic by pointing at the centre of the schematic page and clicking again.

When you are prompted to enter the value of the capacitor with the following dialog, enter 100pF and press OK.



Now pick a resistor from the toolbar  (or press 'r' on the keyboard). Before clicking on the schematic to place it, press the space bar to rotate the component, then place it to the left of the resistor. When prompted for the value, enter 100R. You should now have a schematic that looks something like that shown below.



Now we need to connect the components on the schematic.

[>> click here to continue <<](#)

### Related Topics:

[Placing Components](#)


Component Orientation


---

Created with the Personal Edition of HelpNDoc: [Easily create CHM Help documents](#)

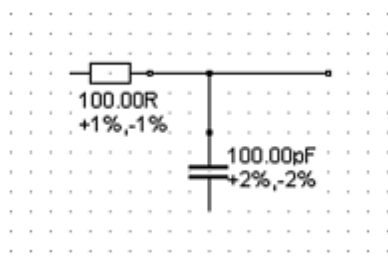
---

**adding connecting wires**

Click on the wire icon  (or press 'w' on the keyboard), the cursor will now have changed to a cross to indicate wire drawing mode.

Place the cursor on the right hand end of the resistor wire and click, then drag to the top of the capacitor and release.

Repeat the process until the schematic looks like that shown below:



Now we need to add a ground connection for the capacitor.

[>> click here to continue <<](#)

**Related Topics:**

[Connecting Wires](#)

---

Created with the Personal Edition of HelpNDoc: [Free EBook and documentation generator](#)

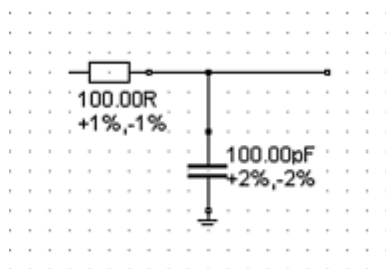
---

**adding a ground connection**

Click on the ground icon  (or press 'g' on the keyboard) to pick up a ground connection for placement.

Move the cursor so that the ground symbol connects to the bottom of the capacitor as shown below:





All that we have to do now before we can simulate the circuit, is to connect some measurement ports.

[>> click here to continue <<](#)


---

Created with the Personal Edition of HelpNDoc: [Free iPhone documentation generator](#)

---

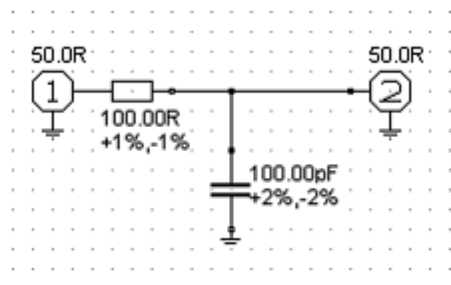
## adding measurement ports

---

Click on the measurement port icon  (or press '1' on the keyboard) to pick up measurement port 1 for placement.

Move the cursor so that port connection makes contact to the resistor wire as shown below.

Click on the measurement port icon {bmc bm40.WMF} again to pick up port 2 (or press '2' on the keyboard) and place this port so that it connects to the capacitor wire as shown below:



If your circuit now look like that shown above then you are ready to start the simulation.

[>> click here to continue <<](#)

### Related Topics:

[Measurement Ports](#)


---

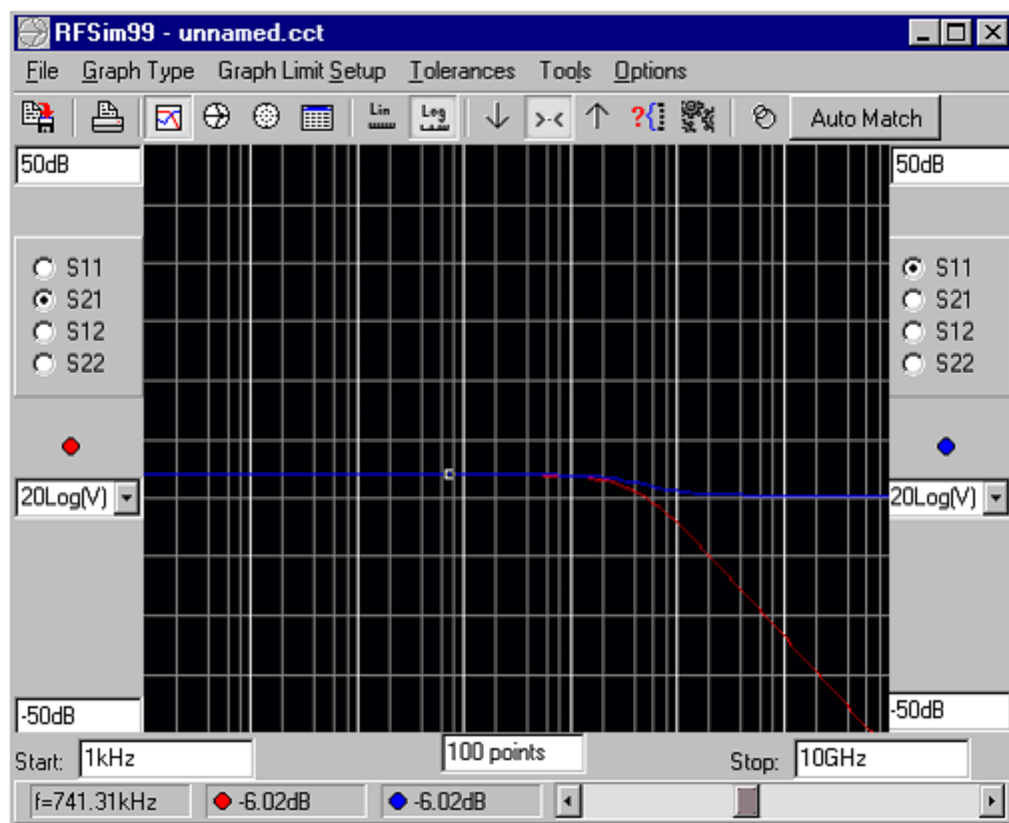
Created with the Personal Edition of HelpNDoc: [Generate Kindle eBooks with ease](#)

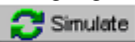
---

## starting simulation

---

Click on the  button. As long as there are no connection errors on the schematic circuit then the simulator will run and a dB-logF plot of the circuit response will be displayed in a separate window.



If there are connection errors associated with the schematic then these will be highlighted with circles to indicate any un-connected component pins. Fix these errors and press the  button to proceed.

[>> click here to continue <<](#)

## Related Topics:

### Simulation

Created with the Personal Edition of HelpNDoc: [Write eBooks for the Kindle](#)

## starting simulation

Now that the graph output window is visible, it is possible to adjust the frequency range and vertical scale limits by changing the values directly in the edit boxes.

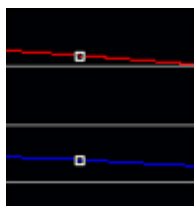
For each of the vertical axes of the square grid, it is also possible to select the data to plot (S11,S21,S12,S22) and the manner of display (dB voltage, dB power, linear magnitude, phase).



To change between log frequency and linear frequency on the x axis, just click on the buttons on the toolbar



The small white square on each of the two traces is a measurement cursor controlled by the scroll bar in the bottom right corner of the graph window.



As the scroll bar position is adjusted, the markers move over the frequency sweep range and parameter values for each of the traces are shown in the bottom left of the window.

To change the display format to polar click on the polar {bmc bm50.WMF}button on the toolbar.

>> click here to continue <<

### Related Topics:

[The Graph Window](#)

[Getting Started – The Smith Display](#)

[Getting Started – The Polar Display](#)

[Getting Started – The Numeric Display](#)

---

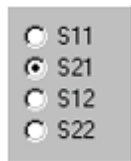
Created with the Personal Edition of HelpNDoc: [Easy EPub and documentation editor](#)

---

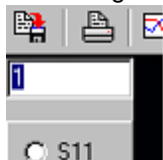
## starting simulation

---

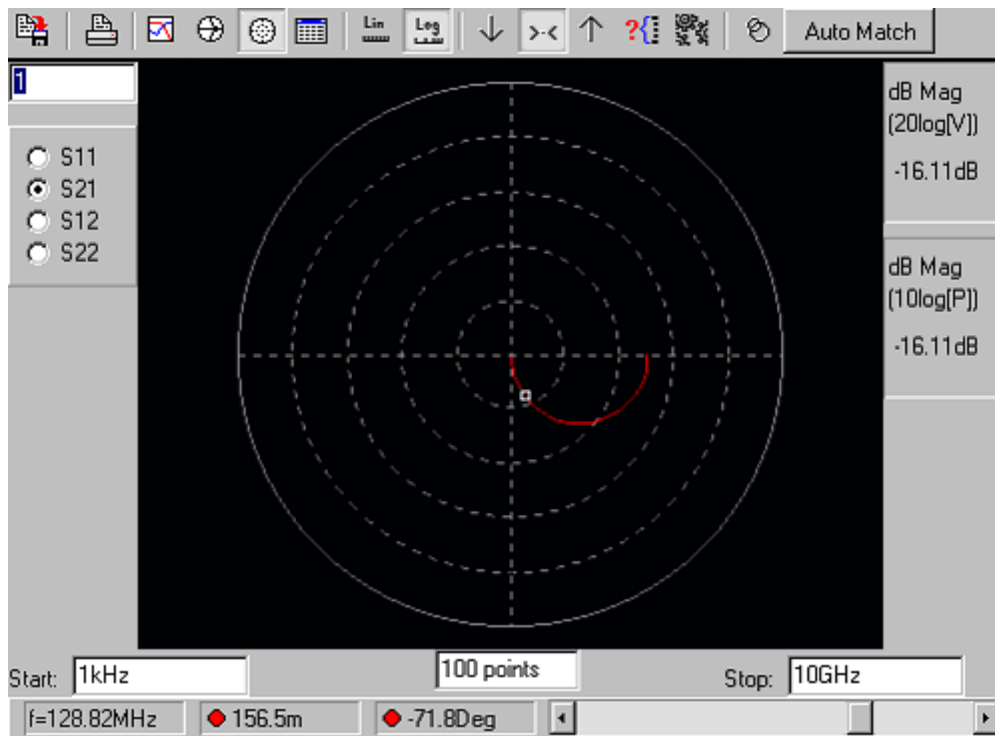
Select S21 as the displayed parameter,



and change the polar scale to 1



your plot should now resemble that shown below.



The marker and marker read out operate in the same way as the square grid, with magnitude and angle displayed in the bottom bar of the window.

In addition to the magnitude/phase display, there are also two other helpful marker readings in the right hand border of the window. These are the dB magnitude values at the cursor frequency in the form of  $20\text{Log}(v_o/v_i)$  and  $10\text{Log}(p_o/p_i)$ .

Let's now have a look at Smith chart display of impedance values

[>> click here to continue <<](#)

### Related Topics:


[The Graph Window](#)

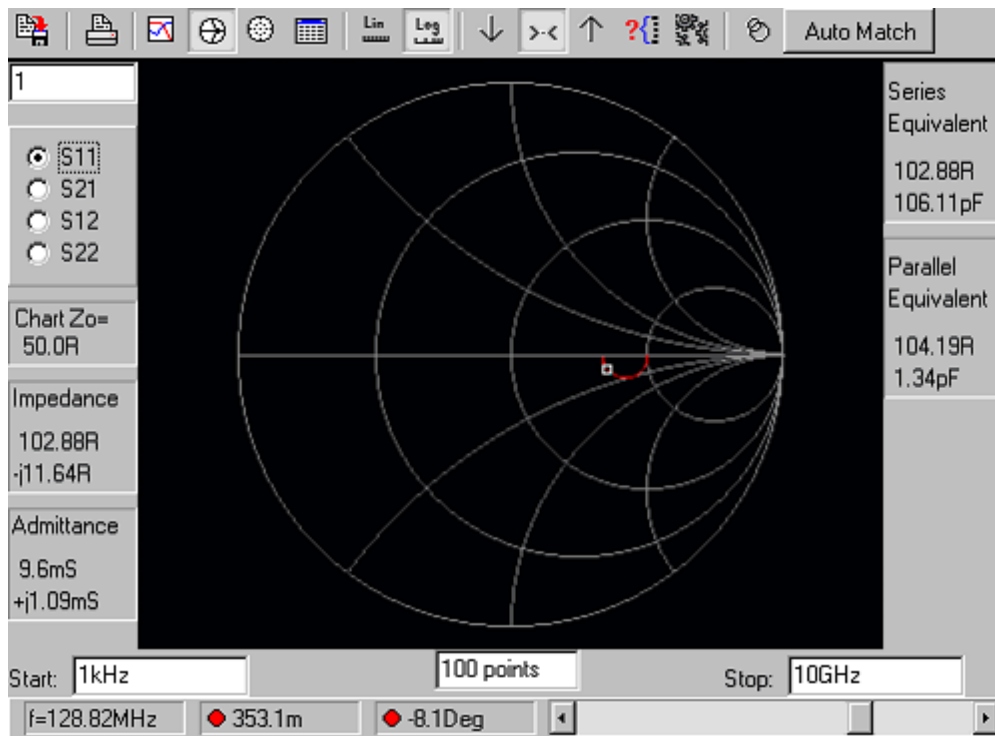
[Getting Started – The Square Grid Display](#)

[Getting Started – The Smith Display](#)

[Getting Started – The Numeric Display](#)

## starting simulation

Press the smith chart button  and select  as the displayed parameter and your plot should resemble that shown below.



As you can see, there are many similarities between the polar and smith chart windows, but there are also some important differences.

On the marker control bar, the magnitude and angle of the reflection coefficient can be read directly.

On the left hand margin are shown the complex impedance and admittance values at the current marker frequency.

On the right hand margin can be found series and parallel equivalent circuits for the impedance seen looking into the circuit at the marker frequency.

We shall now have a look at the numerical output window.

[>> click here to continue <<](#)

### Related Topics:

[The Graph Window](#)

[Getting Started – The Square Grid Display](#)

[Getting Started – The Polar Display](#)

[Getting Started – The Numeric Display](#)

Press the numeric output window  button on the tool bar.

Your display should now resemble that shown below.

Freq	S11 Mag	S11 Ang	S21 Mag	S21 Ang	S12 Mag	S12 Ang	S22 Mag	S22 Ang
1kHz	0.500	-450u°	0.500	-1.35m°	0.500	-1.35m°	0.500	-4.05m°
1.175kHz	0.500	-528.704u°	0.500	-1.586m°	0.500	-1.586m°	0.500	-4.758m°
1.38kHz	0.500	-621.173u°	0.500	-1.864m°	0.500	-1.864m°	0.500	-5.591m°
1.622kHz	0.500	-729.815u°	0.500	-2.189m°	0.500	-2.189m°	0.500	-6.568m°
1.905kHz	0.500	-857.457u°	0.500	-2.572m°	0.500	-2.572m°	0.500	-7.717m°
2.239kHz	0.500	-1.007m°	0.500	-3.022m°	0.500	-3.022m°	0.500	-9.067m°
2.63kHz	0.500	-1.184m°	0.500	-3.551m°	0.500	-3.551m°	0.500	-0.011°
3.09kHz	0.500	-1.391m°	0.500	-4.172m°	0.500	-4.172m°	0.500	-0.013°
3.631kHz	0.500	-1.634m°	0.500	-4.902m°	0.500	-4.902m°	0.500	-0.015°
4.266kHz	0.500	-1.92m°	0.500	-5.759m°	0.500	-5.759m°	0.500	-0.017°
5.012kHz	0.500	-2.255m°	0.500	-6.766m°	0.500	-6.766m°	0.500	-0.02°
5.888kHz	0.500	-2.65m°	0.500	-7.949m°	0.500	-7.949m°	0.500	-0.024°
6.918kHz	0.500	-3.113m°	0.500	-9.34m°	0.500	-9.34m°	0.500	-0.028°

As you can see, S parameter values can be read directly from the display. Use the vertical scroll bar to see results across the whole simulation frequency range.

If you wish to save the results in standard S parameter file format, then this can be accomplished by using the 'save results to S parameter file' option in the file menu.

[>> click here to continue <<](#)

### Related Topics:

[The Graph Window](#)

[Getting Started – The Square Grid Display](#)

[Getting Started – The Smith Display](#)

[Getting Started – The Polar Display](#)

Created with the Personal Edition of HelpNDoc: [Produce Kindle eBooks easily](#)

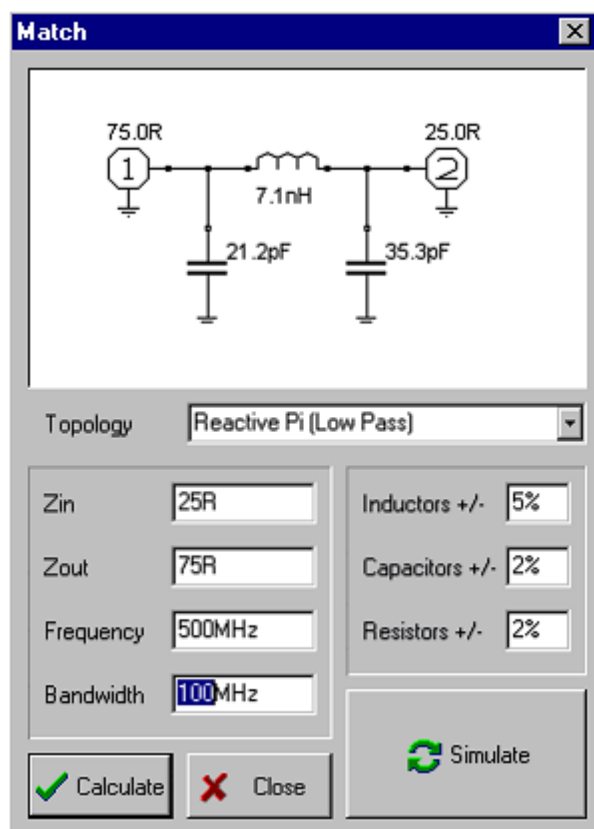
## Match Network

Design a low pass PI match network to match 25R to 75R at a center frequency of 500MHz and with a bandwidth of 100MHz.

Start the match assistant by clicking the match button on the tool bar {bmc bm59.WMF} or selecting 'match' from the 'tools', 'design' menu.

- Select 'Reactive Pi (Low Pass)' in the 'topology' section.
- Edit the values for input and output impedances (25R for  $Z_{in}$  and 75R for  $Z_{out}$ )
- Type in the center frequency of 500MHz
- Type in the bandwidth required (100MHz)
- Press the 'calculate' button.

The match network is shown in the preview window with associated component values.



To simulate the match network, press the 'simulate' button.

*Note: Two element complex matches can be easily handled by using the Auto Match feature. Simultaneous complex matching can also be accomplished with Auto Match.*

[>> click here to continue <<](#)


### Related Topics:

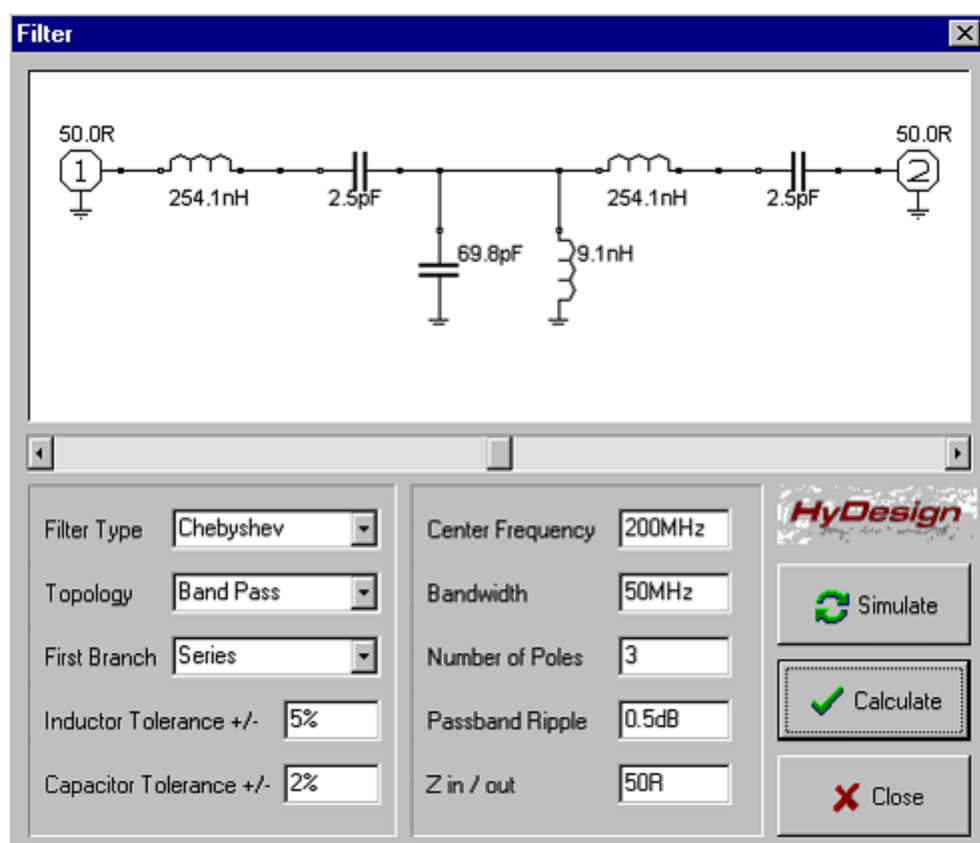
[An Auto Match Example](#)

[Hints & Tips. Using the Match Assistant to match complex impedances](#)

## Filter

Design a fourth order Chebyshev bandpass filter with 0.5dB ripple, centered on 200MHz and with a bandwidth of 50MHz.

- Launch the filter assistant by clicking the filter button  on the tool bar or by selecting 'filter' from the 'tools','design' menu.
- Fill in the required design parameters outlined above.
- Press the calculate button: The filter topology and component values are now displayed in the circuit preview window.



- Press the 'simulate' button to analyze the filter performance in more detail.

[>> click here to continue <<](#)

### Related Topics:






[The Filter Assistant](#)

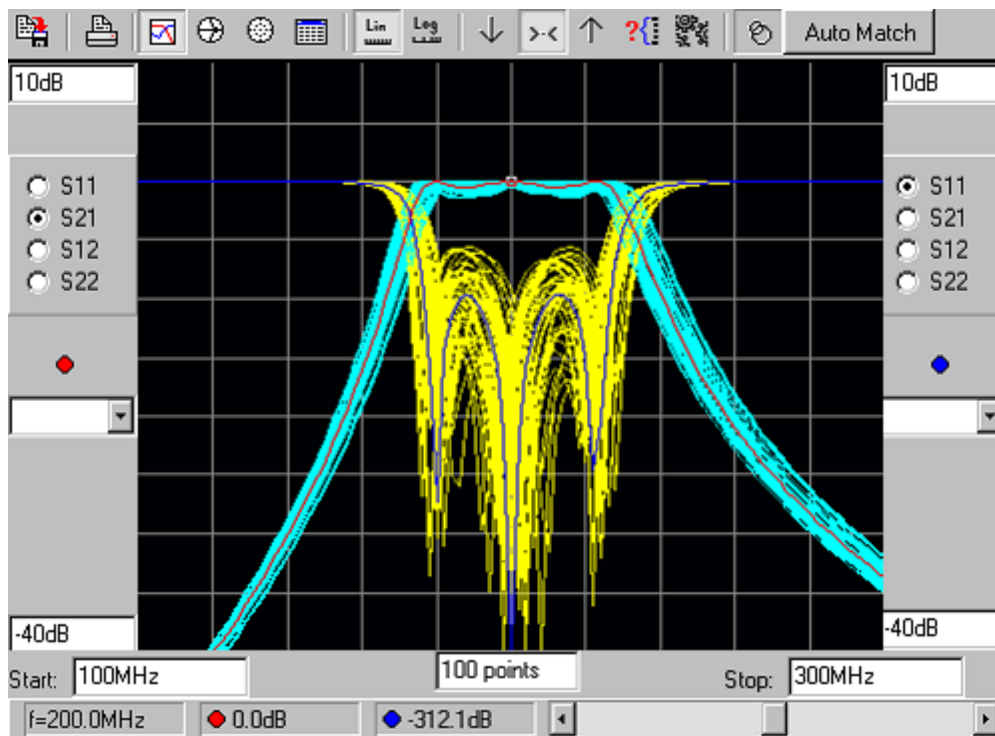
Created with the Personal Edition of HelpNDoc: [Free Qt Help documentation generator](#)

## tolerance analysis

- Load the 'tolerance.cct' file from the examples directory.



- Following simulation, press the low limit button . The plot now shows the response of the circuit if all the tolerated components are set at their low tolerance limit.
- Press the nominal button . The plot now shows the response of the circuit if all the tolerated components are set at their nominal value.
- Press the high limit button . The plot now shows the response of the circuit if all the tolerated components are set at their high tolerance limit.
- Now repeatedly press the random button . The plot now shows the response of the circuit if all the tolerated components have their values set randomly between the specified tolerance limits.
- Now press the montecarlo button . A dialog opens allowing specification of the number of sweeps to carry out. On pressing 'OK' the analysis starts and the traces for the specified number of random variations start to accumulate on the graph (press 'Esc' to abort the run). On completion the plot shows the spread of response of the circuit if all the tolerated components have their values set randomly between the specified tolerance limits. The nominal traces are plotted on top of the monte carlo traces in a different colour for reference.



[>> click here to continue <<](#)

## Related Topics:

### Tolerance Analysis

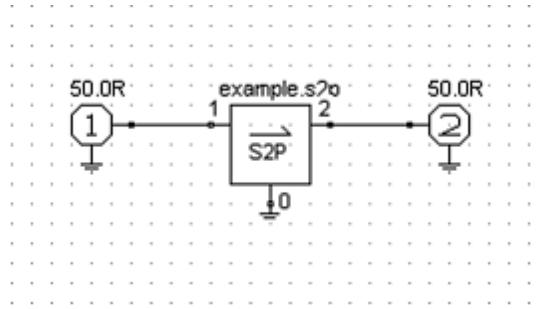
Created with the Personal Edition of HelpNDoc: [Free EBook and documentation generator](#)


## S Parameter Transistor Design

- Pick the 2 port S parameter component from the component palette and place it on a blank schematic.
- Press the 'load file' button and select the 'example.S2P' file in the examples directory – this will load in

an example two port S parameter file.

- Place measurement ports on input(1) and output(2), and a ground connection(0).  
The schematic should now look like that below:



- Press the  button.
- A 'Simulation frequency range outside S parameter file range, Autoscale?' message will appear. This is to warn that the frequency range covered in the S parameter file is less than that currently set up for simulation. Answering yes automatically scales the frequency range of the simulation to the frequency limits of the S parameter file. Answering 'No' allows simulation outside the frequency range covered by the S parameter data using linear interpolation extended from the last two points in the data file. Interpret results from simulations outside the range of the s data files with caution!
- Answer 'Yes' to the autoscale message and the simulation results will be shown in the normal way.

Note: S parameter files may be toleranced by specifying a percentage variation on magnitude and angle.

[>> click here to continue <<](#)

#### Related Topics:

[S parameter Files](#)

---

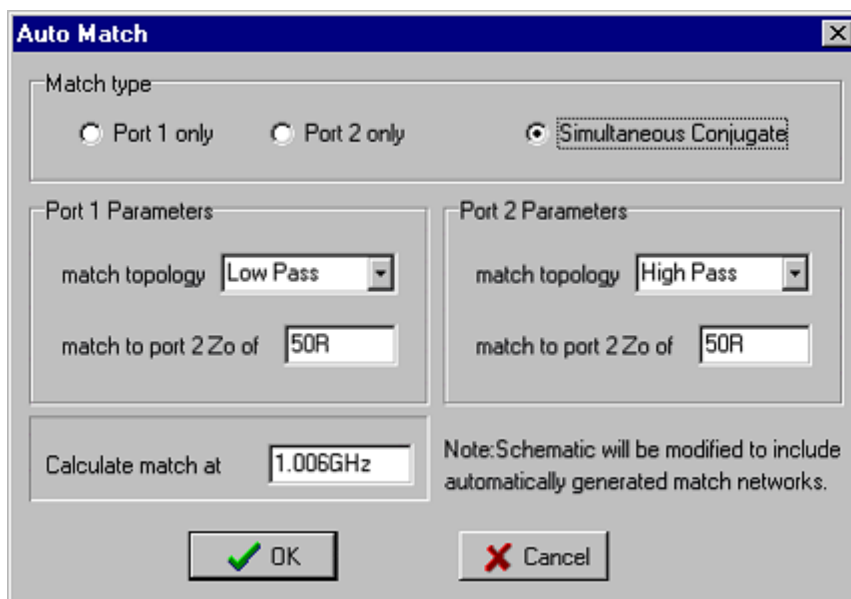
Created with the Personal Edition of HelpNDoc: [Easily create Help documents](#)

---

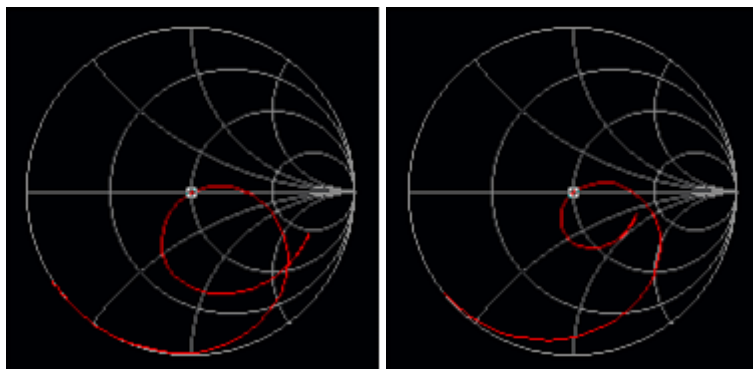
## An Auto Match Example

---

- Load the 'automatch.cct' file from the examples directory.
- Following simulation, move the graph cursor to 1GHz and then press the 'Auto Match' button.  
The dialog shown below will appear:



- 'Simultaneous two port match' will automatically be selected and the match frequency will be set to match that of the graph cursor (1GHz).
- Select 'Low Pass' as the port 2 topology and then press the 'OK' button.
- The match networks required are added to the schematic and the circuit is re-simulated.
- You can see that both S11 and S22 have been simultaneously matched with a high degree of accuracy.



0 [>> click here to continue <<](#)

# 1 Related Topics:


2 [The Auto Match Feature](#)

3 [A Match Network Example](#)

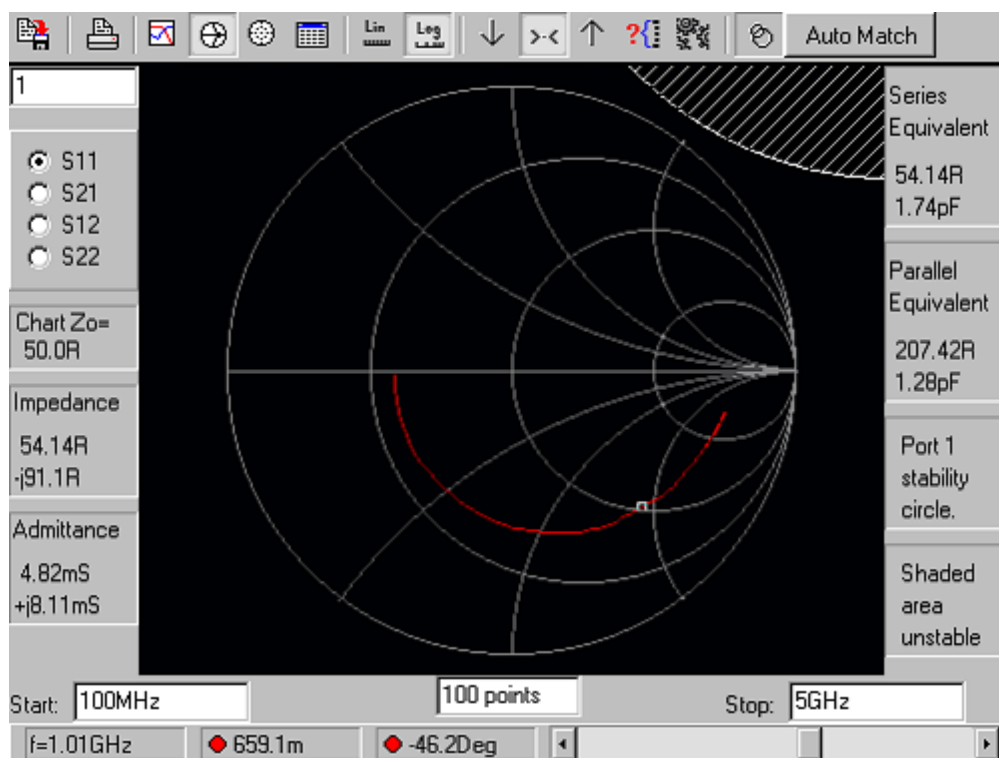
4 [The Match Assistant](#)

Created with the Personal Edition of HelpNDoc: [Generate Kindle eBooks with ease](#)

## S Parameter Transistor Design

- Load the 'stability.cct' file from the 'examples' directory.
- Following simulation, press the stability circle button  to turn on stability circles.

- When S11 or S22 are displayed in smith/polar format, the relevant stability circle is displayed for the current marker frequency, with the unstable region shown shaded.



- Move the graph cursor to examine input or output stability at different frequencies.
- When S11 is selected then the port 1 (input) stability circle is shown. When S22 is selected then the port 2 (output) stability circle is shown.

### Related Topics:

#### Stability Circles

Created with the Personal Edition of HelpNDoc: [Easily create HTML Help documents](#)

## placing components

To place a component on the schematic, pick the desired component from the tool bar by left clicking on it with the mouse, the component will then attach it's self to the mouse pointer. Move the mouse to the required position for the component and press the left mouse button again to release it.

Keyboard short cuts are also available for certain components. These are listed below:

- 'r' resistor
- 'l' inductor
- 'c' capacitor
- 'x' transmission line
- 't' transformer
- 'o' op-amp

's' S parameter two port  
'g' ground connection

Pressing one of the keys relating to a keyboard short cut will cause the desired component to attach it's self to the mouse cursor ready for placement.

A component may also be selected for placement from the 'component' menu in the schematic editor window.

#### Related Topics:

[Getting Started – Placing Components](#)

[Connecting Wires](#)

---

Created with the Personal Edition of HelpNDoc: [Easy to use tool to create HTML Help files and Help web sites](#)

---

## placing components

---

- **During Placement**

Following selection of a component for placement from the tool bar, and before the component is actually placed, the orientation may be changed by pressing the space bar on the keyboard.

- **After placement**

After placement, orientation may be changed by first selecting the desired component (by single clicking with the left mouse button while pointing at it), and then pressing the space bar on the keyboard.

*Note: Orientation can also be changed by clicking the right mouse button on a selected component and selecting 'orientation' from the pop-up menu.*

#### Related Topics:

[Placing Components](#)

[Getting Started – Placing Components](#)

---

Created with the Personal Edition of HelpNDoc: [Produce online help for Qt applications](#)

---

## connecting wires

---

To place connecting wires on the schematic, first select the wire icon {bmc bm76.WMF} from the tool bar. This causes the editor to enter wiring mode and the cursor then changes shape to a cross. Wires can now be placed by clicking at the desired start and end points on the schematic. Wires are not chained (unless the 'chain connecting wires' option in the preferences window is checked) and the editor stays in wiring mode until the right mouse button is pressed, the 'Esc' key is pressed or another tool bar button is selected.

An alternative method of entering wiring mode is to press 'w' on the keyboard.

#### Related Topics:

[Getting Started – Adding Connecting Wires](#)

---

Created with the Personal Edition of HelpNDoc: [Generate Kindle eBooks with ease](#)

---

## measurement ports

---

At least one measurement port must be placed in-order to be able to simulate a circuit.

The measurement ports provided are set by default to 50R characteristic impedance (although this is user controlled).

The effect of placing a 50R measurement port is much akin to connecting a piece of 50R test equipment to a circuit on the bench. The test circuit will be 'loaded' by the measurement port, but this is a necessary situation in order to carry out normal S parameter analysis.

If a port impedance is set very low (eg. 0.1R) then it may appear as a voltage source to the test circuit where as if it is set very high(eg. 1MR) then the port may appear as a voltage probe.

In either of these situations it will most probably be appropriate to use 20log(V) plotting rather than 10Log(P) as voltage gains, not power gains are likely to be of interest.

To place a port on the schematic, click on the measurement port icon on the toolbar \$measure port\$ and a port will attach it's self to the mouse pointer.

Place the port on the schematic and connect to it in exactly the same way as any other component.

#### Related Topics:

[Getting Started – Adding Measurement Ports](#)

---

Created with the Personal Edition of HelpNDoc: [Easily create eBooks](#)

---

## starting simulation

---

Once the test circuit has been completely drawn, and measurement ports have been placed and connected, then simulation may be carried out by pressing the {bmc bm77.WMF}button.

If there are errors in the schematic circuit that prevent the simulator from running then an error message will be displayed indicating the nature of the problem.


Usually errors will be due to unconnected component ends, and these will be highlighted in the schematic by red circles.



#### Related Topics:


[Getting Started – Starting Simulation](#)


## printing

Printing of the schematic or simulation result windows is accomplished in the same way.

In either window there is a print button  on the tool bar (and also a print command on the menu). Pressing the print button will cause a print preview window to open up showing the scale and orientation of the print on the current paper selected in the default printer.

To change the scale of the print relative to the paper use the  and  buttons.

To change the printer settings then press the  button.

To proceed with the print press the print  button.

Print colours are defined in the associated preferences for Schematic or Graph output. If it is desired to print in grey scale or black & white, then this must be set up in the colour preferences dialog for the window in question.

### Related Topics:

[Setting Schematic Preferences](#)

[Setting Graph Preferences](#)

## setting schematic preferences

Selecting 'Schematic Editor Preferences' from the 'Options' menu displays a dialog allowing configuration of the schematic editor.

When the 'chain connecting wires' option is checked , connecting wires are drawn chained together until either the 'Esc' key is pressed , or the right mouse button is clicked. If this box is left un-checked then each wire must be drawn separately with a beginning and end.

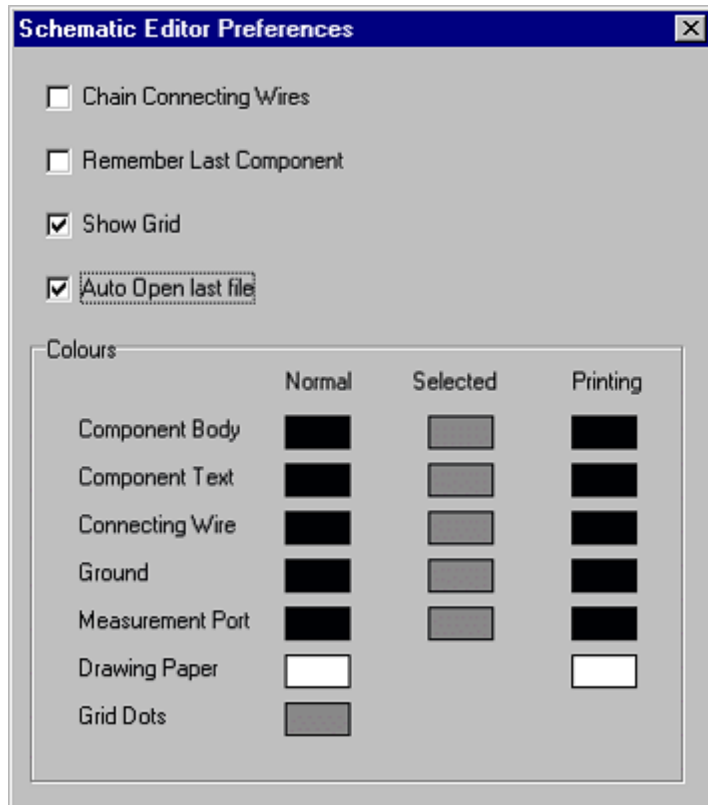
When the 'remember last component' option is checked , the cursor will stay in component placement mode after a component has been placed. If this box is left un-checked then the cursor reverts to selection mode after component placement.

When the 'show grid' option is checked , a component placement grid is displayed in the schematic window in order to aid schematic drawing.

When the 'auto open last file' option is checked , RFSim99 re-loads the last edited file on startup.

The 'colours' section allows user defined colours for the various elements of the schematic (both drawing and printing).

To change a colour, click on the colour box and select the new desired colour from the colour pick dialog.



#### Related Topics:

[Setting Graph Preferences](#)

---

Created with the Personal Edition of HelpNDoc: [Easily create Help documents](#)

---

## graph window

---

The graph output window is launched when the 'simulate' button is pressed in the schematic editor.

All simulation results including numerical table output are shown in the graph window and selection of the various output formats and setting of plot limits is accomplished by either pressing the tool bar buttons, or using the pull down menus.

Further features available in the graph output window are:

- **Auto Match**

The 'Auto Match' feature allows matching circuits to be automatically generated to match the complex impedance at the current marker frequency to the measurement port impedance. Auto Match can also carry out a full simultaneous conjugate match.

- **Stability Circles**



Input and output stability circles can be plotted in smith and polar formats when measuring S11 or S22



of a two port network. The circles plotted for the current marker frequency and are shaded to indicate the unstable region.

- **Tolerance Analysis**



Circuit production spreads can be analyzed for components in the schematic that have tolerance information. The tolerance analysis functions allow the effect on the circuit of upper, lower, random and nominal tolerances to be seen. A multiple sweep function is also provided to allow examination of likely production spreads over a number of units.

#### Related Topics:

[The Auto Match Feature](#)

[Stability Circles](#)

[Tolerance Analysis](#)

---

Created with the Personal Edition of HelpNDoc: [Free Qt Help documentation generator](#)

---

## changing graph type

---

Changing graph limits such as start and stop frequencies and upper and lower Y axis limits is easily accomplished by just typing the new values into the appropriate edit box in the graph window.

Use the top tool bar buttons to change graph type:



Square Grid



Smith Chart



Polar



Tabular Numeric

and Sweep Type:



Linear Frequency

{bmc bm92.WMF}`      Log Frequency

#### Related Topics:

[Getting Started – Starting Simulation](#)


[Getting Started – The Square Grid Display](#)

[Getting Started – The Polar Display](#)

[Getting Started – The Smith Display](#)

## tune feature

The tune feature may be used once a circuit has been successfully simulated.

To select tune mode press the  button in the schematic editor window; the button stays down and the cursor now shows 'Tune' .

When in tune mode a single click with the cursor on a component opens up the appropriate component edit window. On changing a parameter and pressing 'enter' or 'OK', the circuit is re-simulated and the component edit box remains open to allow further tuning.

When tuning is complete for the current component then a click on the 'cancel' button closes the edit box and allows selection of another component for tuning.

To leave tune mode press 'Esc' , click on the location with no component or press the 'tune' button again.

## tolerance analysis

When the schematic circuit under simulation contains toleranced components then the tolerance analysis features of the simulator may be used. The functions of the various buttons are listed below:



set all toleranced components to their low tolerance limit



set all toleranced components to their nominal value



set all toleranced components to their high tolerance limit



set all toleranced components to a random value between the specified tolerance limits



run a user specified number of random sweeps and accumulate the results to show circuit response spread

### Related Topics:

[Getting Started – A Tolerance Analysis Example](#)

## stability circles

---

RFSim99 can display input and output stability circles for a two port network. In order to show stability circle relating to port 1 (normally the input stability circle) the graphical result display must be set to measure S11 in smith chart or polar format. Similarly to display the port 2 circle (normally the output) the display must be showing S22 in smith or polar mode.

The stability circles are shown for the current marker frequency and the unstable region is shown shaded.

To set RFSim99 to stability circles press the  on the top toolbar of the graphical output window.

### Related Topics:

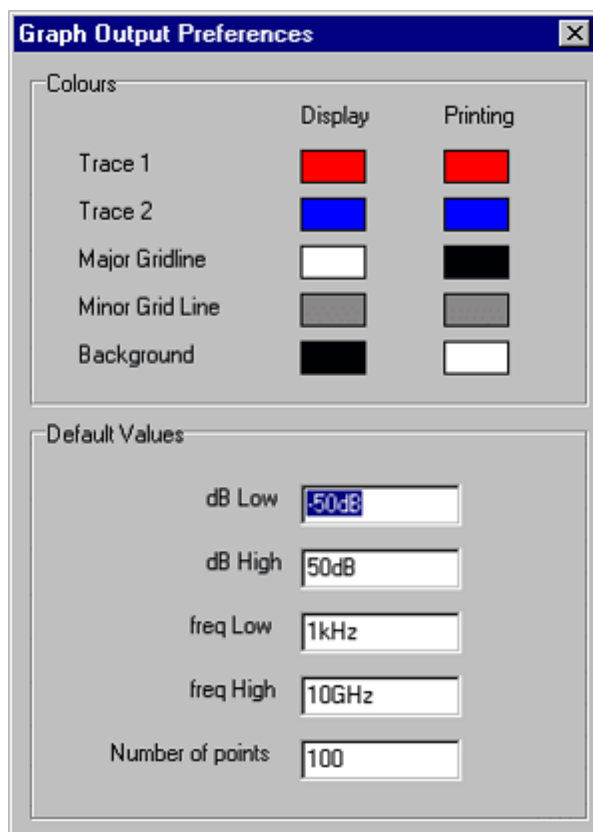
[Getting Started – A Stability Circle Example](#)

## graph preferences

---

The graph preferences window allows setting of the display and printing colours, and also the default limits for simulation frequencies and Y axis limits.

The graph preferences dialog can be accessed from the options menus of both the schematic editor and the graph output window.



To change a colour, click on the colour box and select the new desired colour from the colour pick dialog.

*Note: If black and white , or grey scale printing is desired , then this can be accomplished by setting the appropriate printing colours in this dialog.*

#### Related Topics:

[Schematic Preferences](#)

---

Created with the Personal Edition of HelpNDoc: [What is a Help Authoring tool?](#)

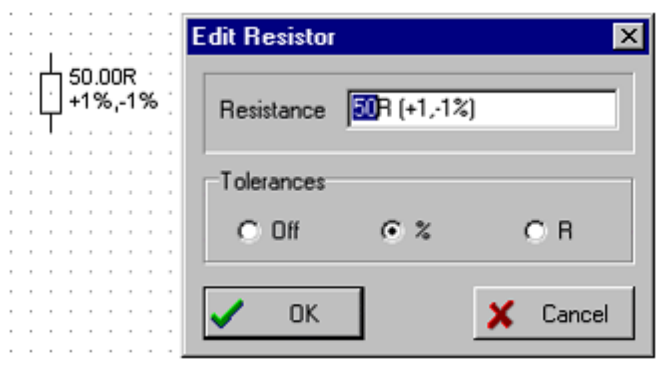
---

## resistor model

---

The resistor component model is very simple. Apart from the resistance, it is possible to define absolute and relative tolerances for a resistor.

The resistor symbol and edit box are shown below




---

Created with the Personal Edition of HelpNDoc: [What is a Help Authoring tool?](#)

---

## capacitor model

---

The capacitor model supports two levels of complexity:

### 1/ **Simple**

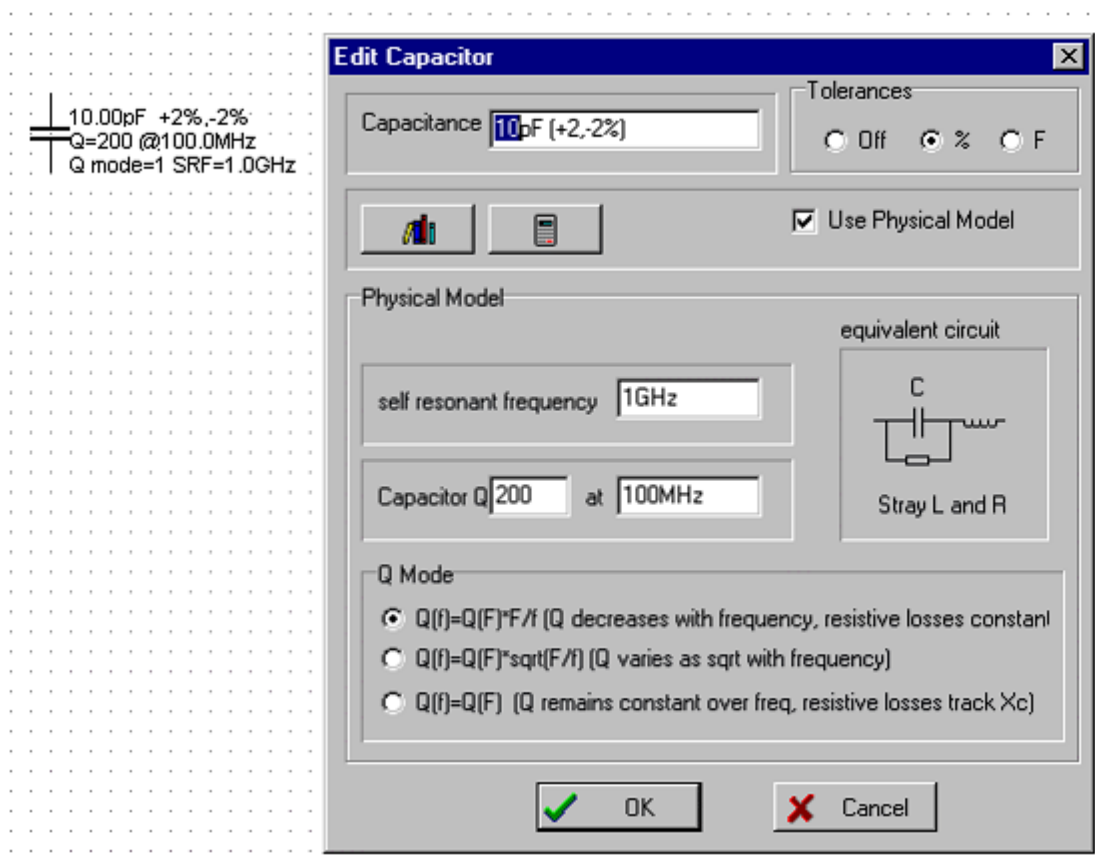
In the simple model (use physical model un-checked), the component is a simple perfect capacitor with optional relative/absolute associated tolerance.

### 2/ **Physical**

In the complex model (use physical model checked) , the capacitor is modelled as having a self resonant frequency and resistive losses.

The resistive losses are defined in terms of Q (ie.  $XC/R$ ) at a given frequency (F), and the Q variation with frequency may be selected from one of three modes.

The Physical capacitor symbol and edit box are shown below:



### Related Topics:

[The Calculator – Resonance / Reactance](#)

Created with the Personal Edition of HelpNDoc: [What is a Help Authoring tool?](#)

## inductor model

The inductor model supports two levels of complexity:

### 1/ Simple

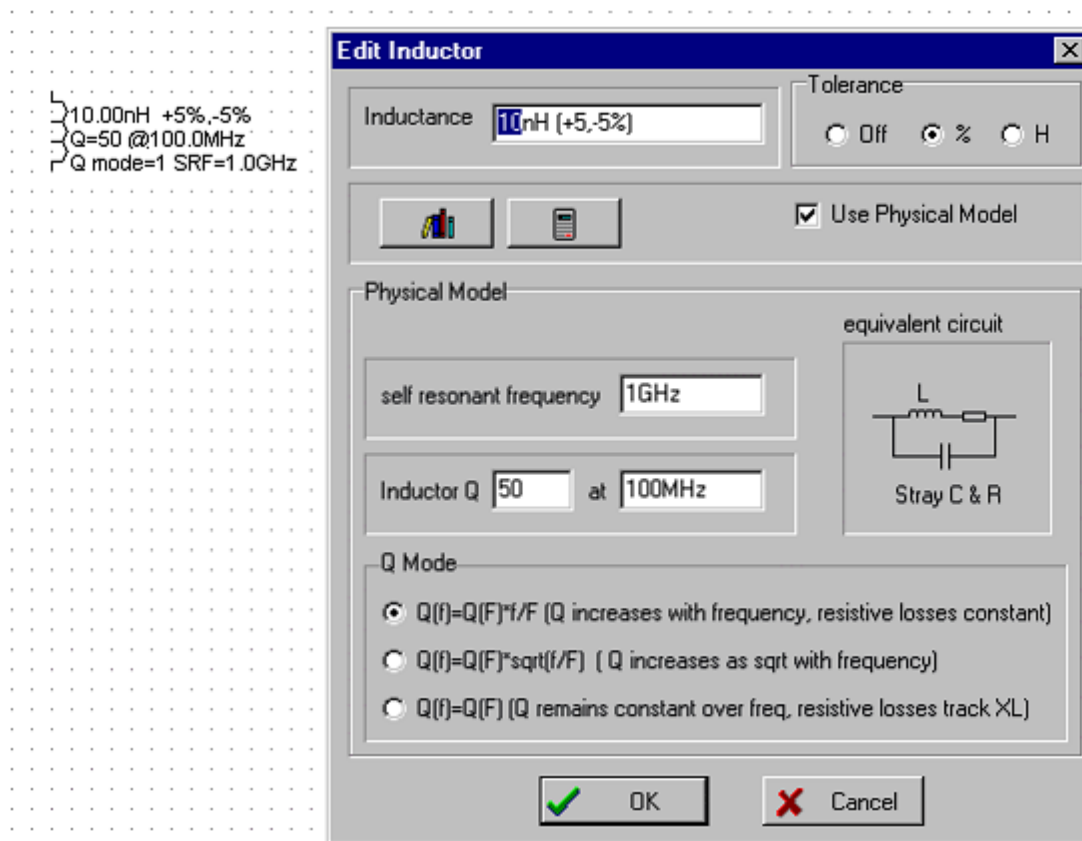
In the simple model (use physical model un-checked), the component is a simple perfect inductor with optional relative/absolute associated tolerance.

### 2/ Physical

In the complex model (use physical model checked), the capacitor is modelled as having a self resonant frequency and resistive losses.

The resistive losses are defined in terms of Q (ie.  $XL/R$ ) at a given frequency (F), and the Q variation with frequency may be selected from one of three modes.

The inductor symbol and edit box are shown below:



#### Related Topics:

[The Calculator – Resonance / Reactance](#)

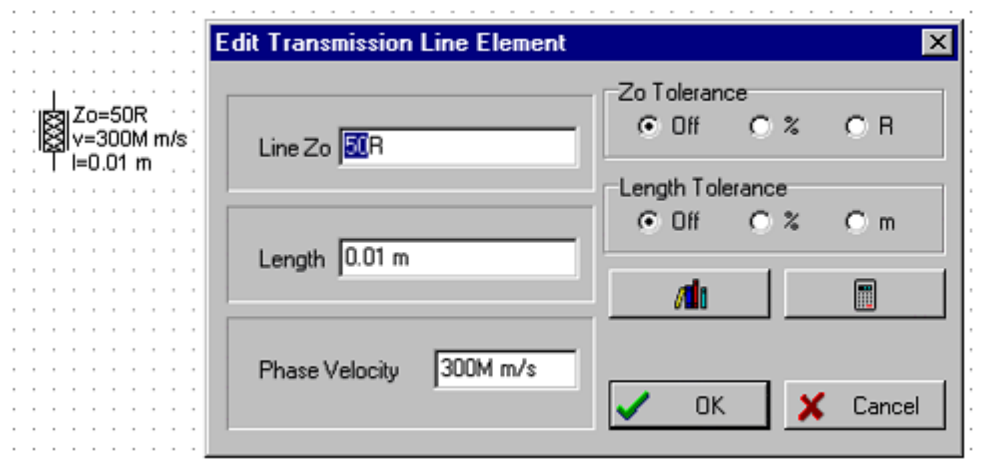
Created with the Personal Edition of HelpNDoc: [Generate EPub eBooks with ease](#)

## transmission line model

The transmission line component is modelled as a simple lossless transmission line with a given length, phase velocity and characteristic impedance.

Both the length and the impedance may be tolerated parameters, and the tolerance limits may be specified in absolute or relative terms.

The transmission line symbol and edit box are shown below:



### Related Topics:

[The Calculator – Frequency / Wavelength](#)

Created with the Personal Edition of HelpNDoc: [Easy EPub and documentation editor](#)

## transformer model

The transformer model supports two levels of complexity:

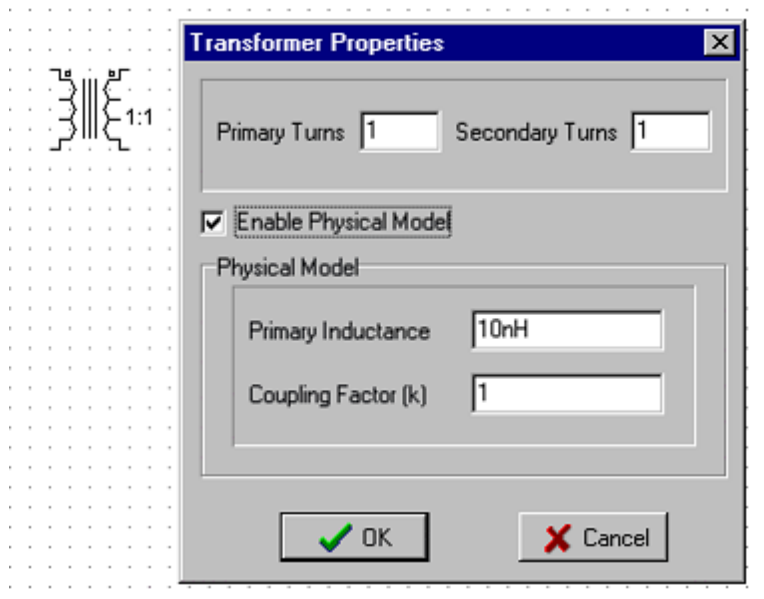
### 1/ Simple

In the simple model (use physical model un-checked), the component is a simple transformer with a specified number of primary and secondary turns.

### 2/ Physical

In the complex model (use physical model checked) , the primary inductance and the coupling factor may be specified

The transformer symbol and edit box are shown below:





**Related Topics:**

[Hints & Tips – Analysis of Balanced Circuits](#)

---

Created with the Personal Edition of HelpNDoc: [Free help authoring environment](#)

---

## S parameter files

---

One and two port S parameter blocks are defined as S parameter files in industry standard text file format.

S parameter files are loaded into the component by using the 'load' button in the component set-up dialog and selecting the desired file using a standard windows file selector box.

S parameter files may be toleranced by specifying magnitude and angle variations in percent.

S parameter files in CITIFILE format (as generated by some network analyzers) are also supported, and these are loaded in the same way as normal S parameter files. Change the 'files of type' setting in the file selector dialog to show CITIFILE \*.d1 & \*.d2 files, then select them in the normal way. CITIFILES are converted to normal mag/angle files internally.

**S parameter File Format:**

Example file:

```
! comment
! comment
# MHz MA
100.000000 0.761 -11.117 6.353 172.781 0.042 17.781 0.399 -10.694
147.875764 0.755 -13.089 6.313 170.999 0.042 16.957 0.396 -11.262
218.672415 0.746 -16.013 6.253 168.361 0.043 15.735 0.391 -12.095
323.363503 0.734 -20.342 6.167 164.455 0.044 13.923 0.385 -13.319
478.176250 0.715 -26.752 6.039 158.667 0.045 11.230 0.374 -15.104
707.106781 0.700 -35.543 5.854 149.727 0.047 12.392 0.359 -20.270
1045.639553 0.662 -49.290 5.592 137.581 0.051 14.863 0.342 -27.471
1546.247474 0.628 -67.779 5.214 120.473 0.058 14.860 0.305 -37.676
2286.525260 0.571 -93.770 4.700 96.553 0.070 12.149 0.225 -50.712
3381.216689 0.498 -131.763 4.060 64.171 0.085 2.033 0.077 -81.637
5000.000000 0.452 168.726 3.187 17.202 0.101 -17.798 0.198 114.195
```

The comment lines (with the !) can contain any text  
 The format specifier line (with the # ) defines the data format

Valid frequency key words are:

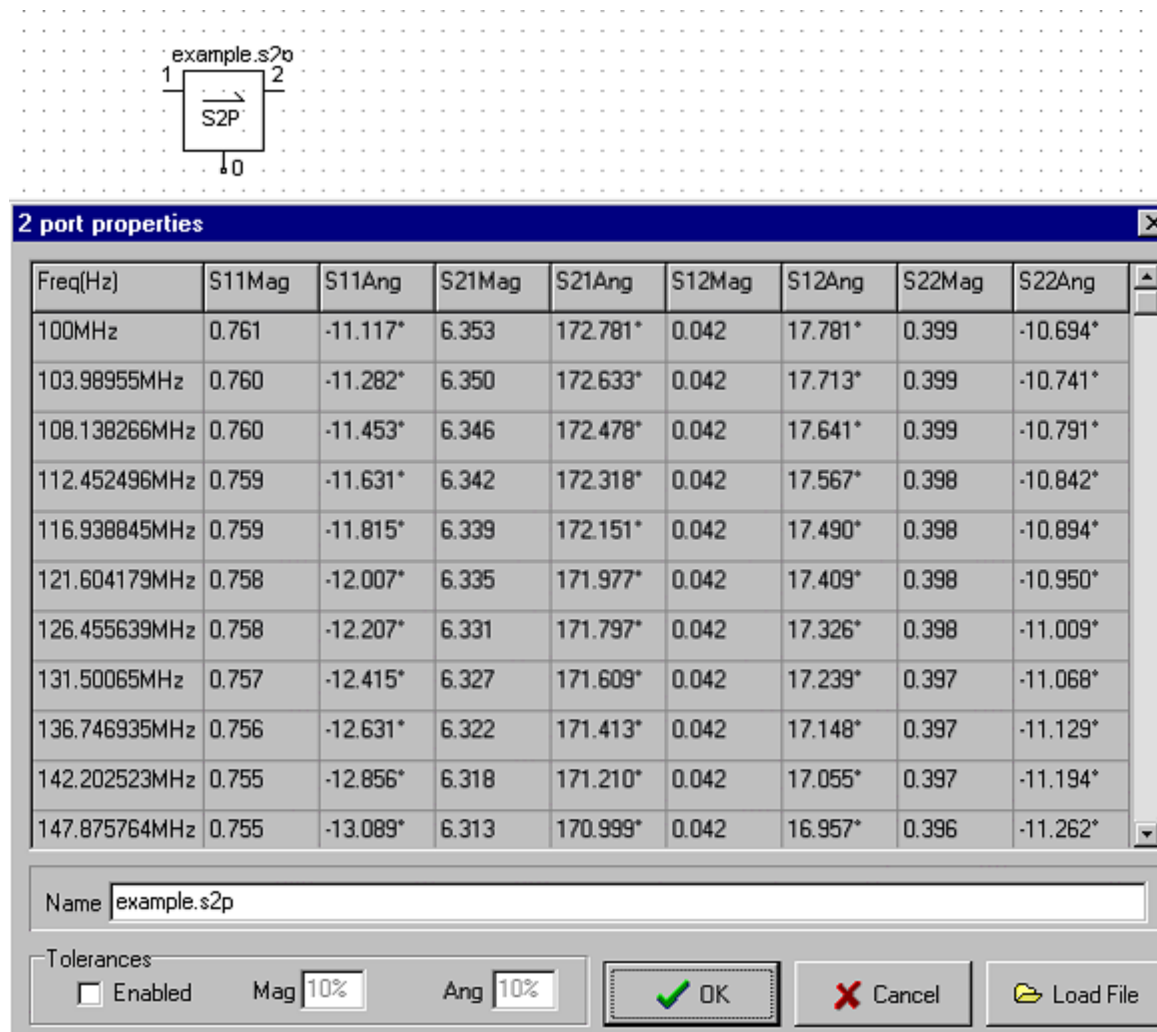
Hz.....Hertz  
 MHz....Mega Hertz  
 GHz....Giga Hertz

Valid format key words are:

MA.....Magnitude/Angle  
 RI.....Real/Imaginary

All S parameters are assumed to be measured in a 50R system.

The 2 Port S parameter symbol and edit box are shown below:



The diagram shows a 2-port S-parameter symbol on a grid. The symbol is a square with two input ports labeled '1' and '2' on the left, and one output port labeled '0' on the right. The symbol is labeled 'example.s2p' and 'S2P'.

The '2 port properties' dialog box is shown below the symbol. It contains a table of S-parameter values for various frequencies. The table has columns for Freq(Hz), S11Mag, S11Ang, S21Mag, S21Ang, S12Mag, S12Ang, S22Mag, and S22Ang. The values are as follows:

Freq(Hz)	S11Mag	S11Ang	S21Mag	S21Ang	S12Mag	S12Ang	S22Mag	S22Ang
100MHz	0.761	-11.117°	6.353	172.781°	0.042	17.781°	0.399	-10.694°
103.98955MHz	0.760	-11.282°	6.350	172.633°	0.042	17.713°	0.399	-10.741°
108.138266MHz	0.760	-11.453°	6.346	172.478°	0.042	17.641°	0.399	-10.791°
112.452496MHz	0.759	-11.631°	6.342	172.318°	0.042	17.567°	0.398	-10.842°
116.938845MHz	0.759	-11.815°	6.339	172.151°	0.042	17.490°	0.398	-10.894°
121.604179MHz	0.758	-12.007°	6.335	171.977°	0.042	17.409°	0.398	-10.950°
126.455639MHz	0.758	-12.207°	6.331	171.797°	0.042	17.326°	0.398	-11.009°
131.50065MHz	0.757	-12.415°	6.327	171.609°	0.042	17.239°	0.397	-11.068°
136.746935MHz	0.756	-12.631°	6.322	171.413°	0.042	17.148°	0.397	-11.129°
142.202523MHz	0.755	-12.856°	6.318	171.210°	0.042	17.055°	0.397	-11.194°
147.875764MHz	0.755	-13.089°	6.313	170.999°	0.042	16.957°	0.396	-11.262°

The dialog box also includes a 'Name' field with the value 'example.s2p', a 'Tolerances' section with 'Enabled' checked, 'Mag' set to 10%, and 'Ang' set to 10%. There are 'OK', 'Cancel', and 'Load File' buttons at the bottom.

### Related Topics:

#### Getting Started – An S Parameter Example

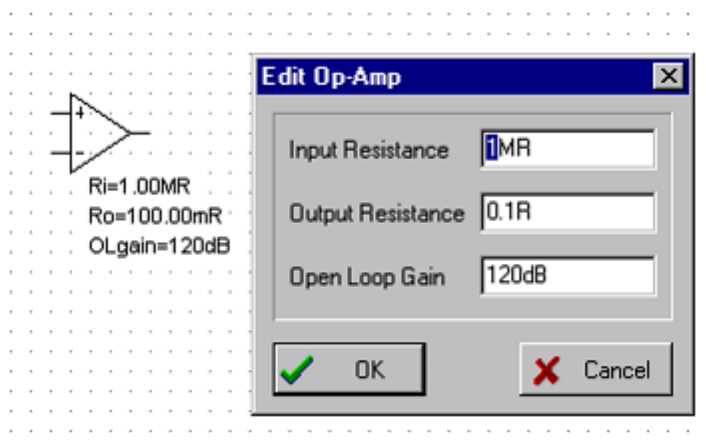
Created with the Personal Edition of HelpNDoc: [Free PDF documentation generator](#)

## op-amp model

The op-amp model is very simple and consists only of open loop gain and input/output impedances. None of the parameters are tolerated.

The op-amp model is purely theoretical and therefore has no upper frequency limit.

The op-amp symbol and edit box are shown below:




---

Created with the Personal Edition of HelpNDoc: [Create cross-platform Qt Help files](#)

---


## match assistant

---

The match assistant allows automatic generation of match networks for matching real impedances using reactive L, reactive Pi, reactive Tee and minimum loss R topologies.

The assistant provides options for High/Low pass response selection and input and output impedances. Toleranced components values are supported and it is possible to specify the approximate bandwidth desired when designing Pi and Tee circuits.

When designing L topology circuits, the bandwidth box indicates the approximate 3dB bandwidth of the match with the current impedance transformation ratio.

The schematic for the circuit is shown in the preview window and this can immediately be simulated by pressing the  button.

### Related Topics:

[Getting Started, A Match network example](#)

[Hints & Tips, Matching complex impedances using the Match Assistant](#)

---


Created with the Personal Edition of HelpNDoc: [Full-featured Documentation generator](#)

---

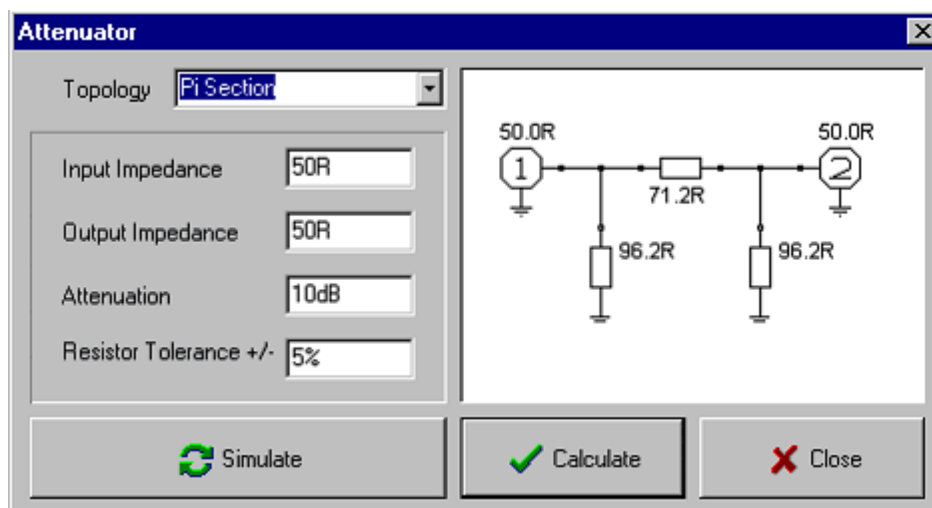
## attenuator assistant

---

The attenuator assistant is a convenient way of designing resistive attenuator pads in Pi, Tee and bridged-Tee topologies.

To design an attenuator simply select the attenuator type from the Topology list, then fill in the desired input/output impedances and attenuation and press the  button.

The schematic for the circuit is shown in the preview window and this can immediately be simulated by pressing the {bmc bm110.WMF}button.



Created with the Personal Edition of HelpNDoc: [Full-featured multi-format Help generator](#)

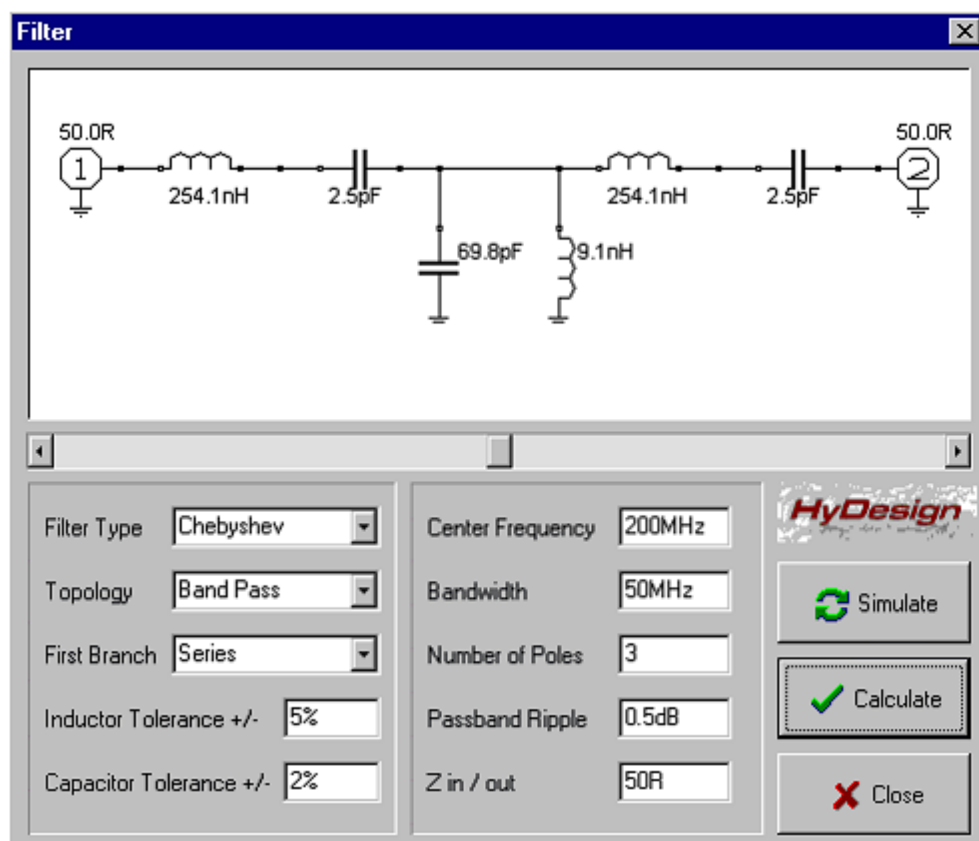
## filter assistant

The filter assistant allows automatic generation of high-pass, low-pass and band-pass filter networks with Chebychev and Butterworth characteristics.

The user can specify:

- Filter type (Butterworth or Chebychev)
- Topology (high/low/band pass)
- First branch (series or parallel)
- Center Frequency (for band-pass only)
- Bandwidth
- Number of poles (order of filter)
- Ripple (Chebychev only)
- Input/Output impedances
- Inductor and Capacitor tolerances

Once the desired performance characteristics have been entered, a press on the {bmc bm112.WMF} button causes the prototype circuit to be displayed in the preview window. Pressing the {bmc bm113.WMF} button passes the circuit to the schematic editor and triggers simulation. Tuning of values and tolerance analysis is then a simple matter.




### Related Topics:

[Getting Started, A Filter example](#)

Created with the Personal Edition of HelpNDoc: [Free help authoring environment](#)

## auto match

RF Sim99 can automatically calculate simple matching networks required to match a complex impedance from a simulation result. Simultaneous conjugate match networks can also be automatically generated for 2 port networks.

Pressing the  button in the graph output window causes the Auto Match dialog to open as shown below:

By default the match frequency will be set to the current marker frequency, but this can be overridden.

Following selection of the required match type and topology just press the 'OK' button and the match network(s) will be automatically added to the schematic and the circuit will be re-simulated.

**Notes:**

1/ There are certain occasions when a theoretical simultaneous conjugate match is not possible due to the particular input and output impedances of the network. In this case RFSim99 displays a warning message allowing a 'best attempt' match or 'cancel'.

2/ When the automatically generated match network produces components with negative values this is usually because the wrong topology was picked for that particular port. Re-running Auto Match with the alternative topology will usually produce a non-negative result.

**Related Topics:**

[Getting Started – Aauto Match Example](#)

---

Created with the Personal Edition of HelpNDoc: [Generate EPub eBooks with ease](#)

---

## resonance reactance calculator

---

The resonance/reactance section of the calculator allows rapid calculation of reactance, inductance, capacitance or resonant frequency.

Entering a value into an edit box causes it to 'lock'.

When the 'enter' key is pressed the un-locked parameters are calculated from the locked ones.

Parameters may be locked or un-locked by pressing the lock buttons {bmc bm117.WMF} next to the edit boxes.

When too many values are locked, the oldest locked value is automatically un-locked in order for calculation to proceed.

The screenshot shows the 'RF Calculator' window with the 'Resonance/Reactance' section selected. The fields are as follows:

Parameter	Value	Lock Icon
Resonant Frequency	100MHz	Locked
Reactance (XLXC)	50R	Locked
Inductance	79.877nH	Locked
Capacitance	31.831pF	Locked

Created with the Personal Edition of HelpNDoc: [Easily create HTML Help documents](#)

## return loss VSWR calculator

The return loss <> VSWR section of the calculator allows conversion between the following parameters:

- Return Loss
- VSWR
- Transmission Loss
- Reflection Coefficient

Just type the value for conversion into the appropriate box and press return, all other parameters are automatically calculated.

The screenshot shows the 'RF Calculator' window with the 'Return Loss/VSWR' section selected. The fields are as follows:

Parameter	Value
Return Loss	10dB
VSWR	1.925 : 1
Transmission Loss	0.458dB
Reflection Coeff.	0.316

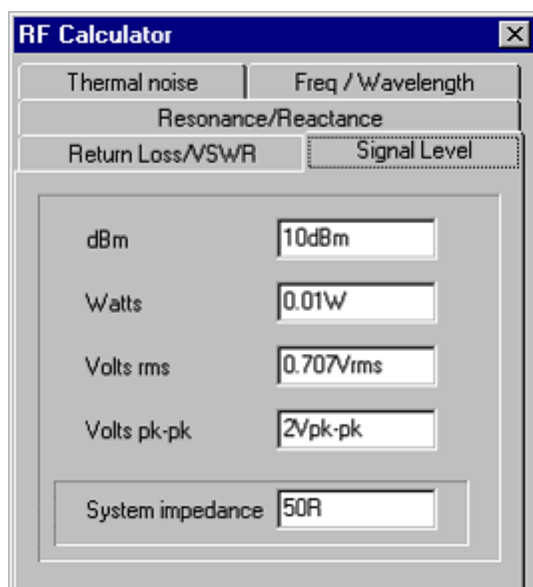
## signal level calculator

The signal level section of the calculator allows conversion between the following parameters:

- dBm
- Watts
- Volts RMS
- Volts Peak to Peak

Just type the value for conversion into the appropriate box and press return, all other parameters are automatically calculated.

In order to carry out conversions between power and voltage a system impedance must be specified; this is by default set to 50R but may be user specified.



The screenshot shows a window titled "RF Calculator" with a close button (X). It has four tabs: "Thermal noise", "Freq / Wavelength", "Resonance/Reactance", and "Signal Level". The "Signal Level" tab is selected. Inside this tab, there are five input fields arranged vertically. The first four fields are labeled "dBm", "Watts", "Volts rms", and "Volts pk-pk" respectively, with values "10dBm", "0.01W", "0.707Vrms", and "2Vpk-pk". The fifth field is labeled "System impedance" with a value of "50R".

## frequency wavelength calculator

The Freq <> Wavelength section of the calculator allows conversion between the following parameters:

- Phase Velocity
- Physical Length
- Delay
- Phase Shift

Just type the value for conversion into the appropriate box and press return, other parameters are automatically calculated.



The screenshot shows the 'RF Calculator' window with the 'Resonance/Reactance' section selected. The 'Thermal noise' tab is also visible. The input fields are as follows:

Parameter	Value
Phase Velocity	300M m/s
Physical Length	0.01 m
Delay	0s
Phase Shift	0 Deg at 100MHz

Created with the Personal Edition of HelpNDoc: [Full-featured Documentation generator](#)

## thermal noise calculator

The thermal noise section of the calculator allows calculation of thermal noise in dBm and Watts from the noise bandwidth and temperature.

Just type the value for conversion into the appropriate box and press return, other parameters are automatically calculated.

The screenshot shows the 'RF Calculator' window with the 'Thermal noise' tab selected. The input fields are as follows:

Parameter	Value
Temperature	27degC
Bandwidth	1Hz
Noise Power (W)	4.14E-21W
Noise Power (dBm)	-173.83dBm

Created with the Personal Edition of HelpNDoc: [Free Qt Help documentation generator](#)

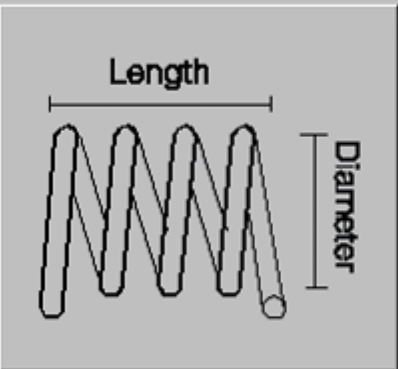
## inductance component designer

The inductor designer allows analysis/design of the following types of inductor:

- Air cored wire wound coil
- Printed square spiral
- Short circuit stub

**Inductor**

Air Cored Inductor | Printed Spiral Inductor | Printed Short cct stub



Length: 10 mm

Diameter: 3 mm

Number of turns: 5

L = 19.8nH

#### Related Topics:

[The Calculator – Resonance / Reactance](#)

Created with the Personal Edition of HelpNDoc: [Easily create Qt Help files](#)

## capacitance component designer

The capacitor designer allows analysis/design of a simple parallel plate capacitor following specification of plate area, separation and dielectric constant.

**Capacitor**

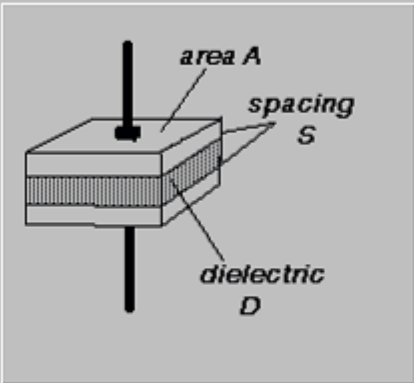


Plate Area (A): 10 sq mm

Plate Spacing (S): 0.085 mm

Dielectric: Air Er=1

Dielectric Er (D): 1

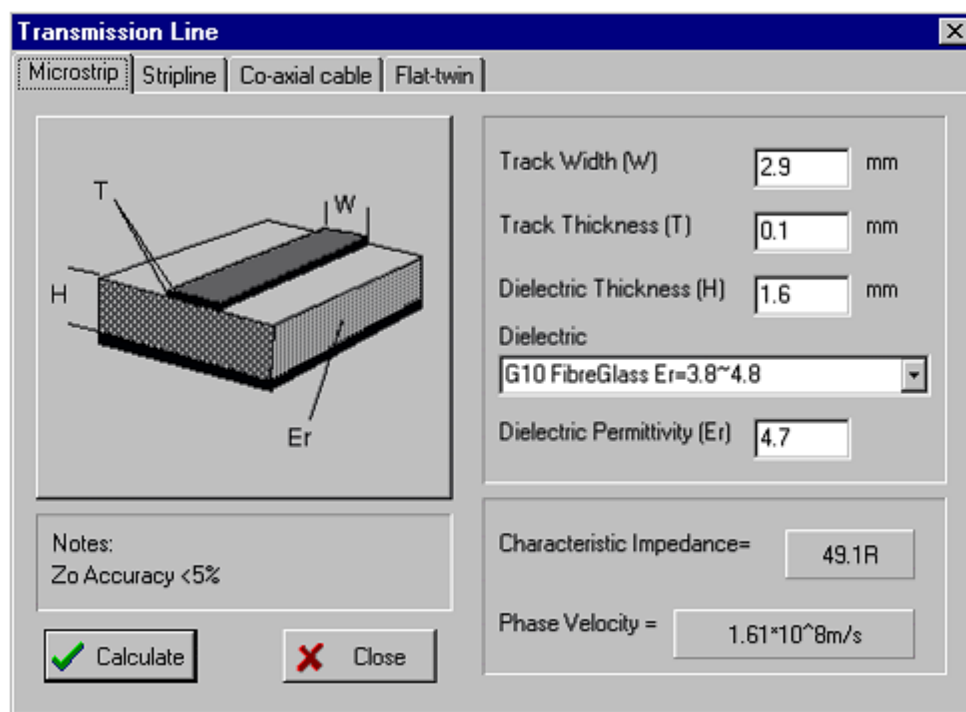
Capacitance = 1.0pF

**Related Topics:**[The Calculator – Resonance / Reactance](#)Created with the Personal Edition of HelpNDoc: [What is a Help Authoring tool?](#)

## transmission line component designer

The transmission line designer allows analysis/design of the following types of transmission line:

- Microstrip
- Stripline
- Co-axial
- Flat twin

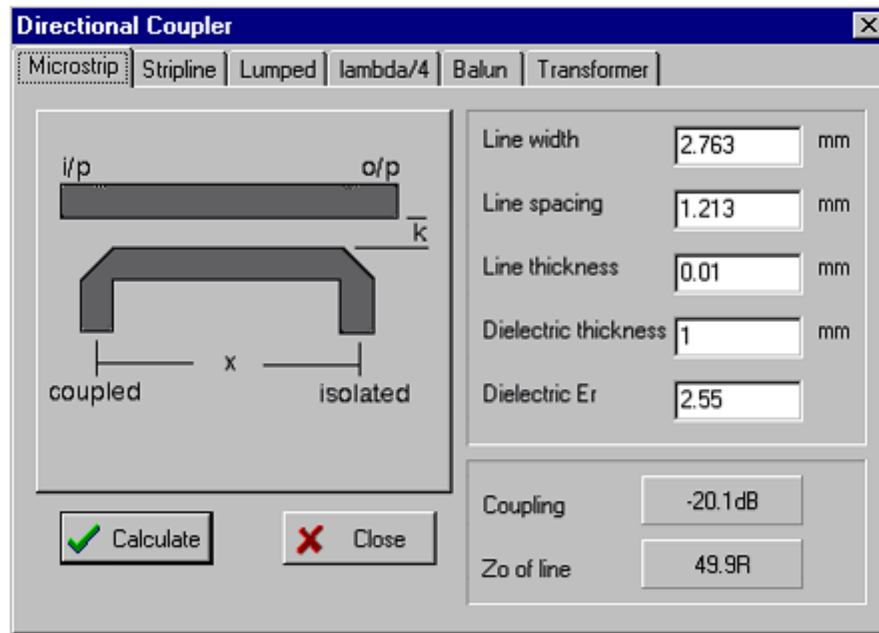
**Related Topics:**[The Calculator – Frequency / Wavelength](#)Created with the Personal Edition of HelpNDoc: [Produce online help for Qt applications](#)

## coupler component designer

The coupler designer allows design/analysis of the following types of coupler:

- Microstrip
- Stripline
- Lumped
- Lambda/4

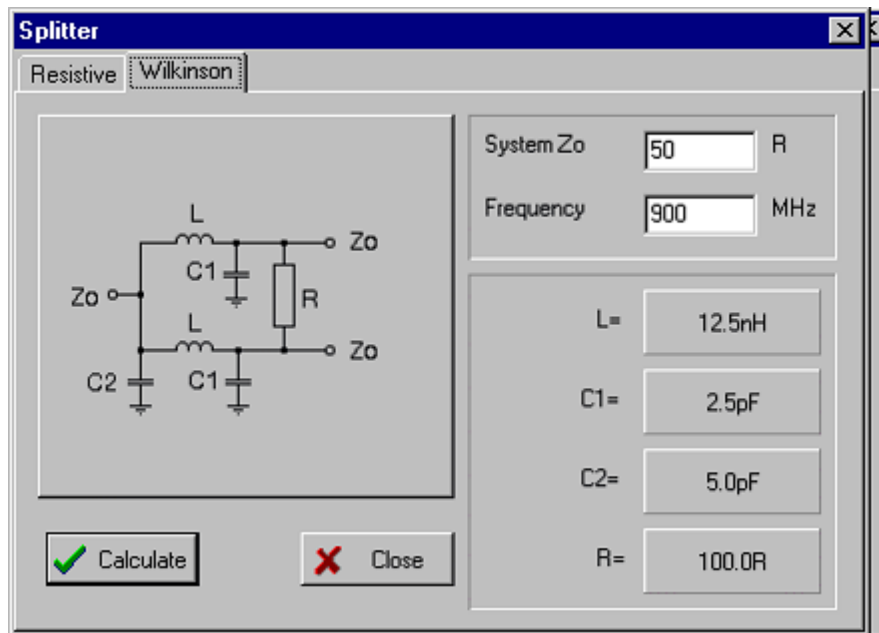
- Balun
- Transformer



Created with the Personal Edition of HelpNDoc: [Free help authoring tool](#)

## splitter component designer

The splitter designer allows quick design of either resistive or wilkinson splitter/power dividers.



Created with the Personal Edition of HelpNDoc: [Produce Kindle eBooks easily](#)

## matching complex impedances

If a match from a real to a complex impedance is required then proceed as follows:

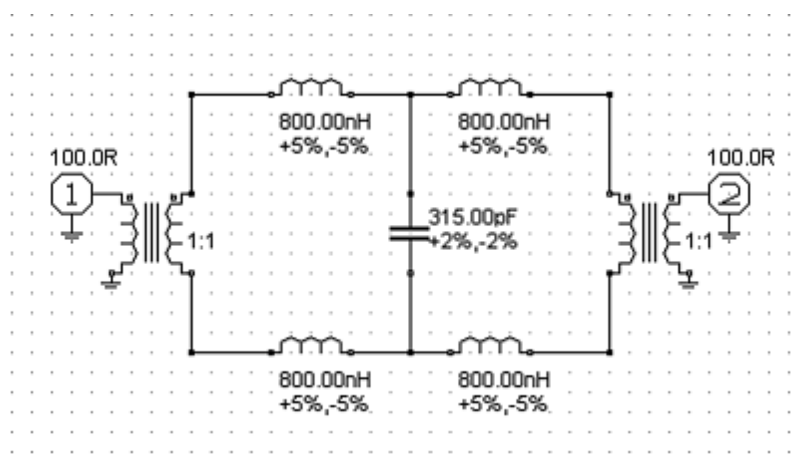
- Choose a topology that will allow absorption of the imaginary part of the complex impedance into the match network (eg. if the impedance to be matched is capacitive then choose a match network topology with a parallel inductor across the capacitive load).
- Use the match assistant to calculate the match network required to match between the real impedance to the real part of the complex impedance.
- Absorb the imaginary part of the complex impedance into the match network

Created with the Personal Edition of HelpNDoc: [Generate Kindle eBooks with ease](#)

## analysis of balanced circuits

It is possible to analyze balanced circuits quite simply in RFSim99 through use of the transformer component.

Set the ports to the required balanced  $Z_0$  and then use 1:1 transformers at each end of the balanced circuit as in the example below



This shows a balanced 10MHz butterworth filter with 100R balanced impedances.

### Related Topics:

[The Transformer](#)

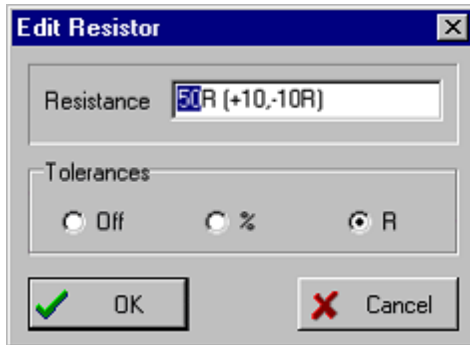
Created with the Personal Edition of HelpNDoc: [Produce online help for Qt applications](#)

## upper and lower tolerance limits

If it is required to define a component with a positive minimum tolerance and a negative maximum tolerance than this can be achieved by simply editing the tolerance parameters to reflect this.

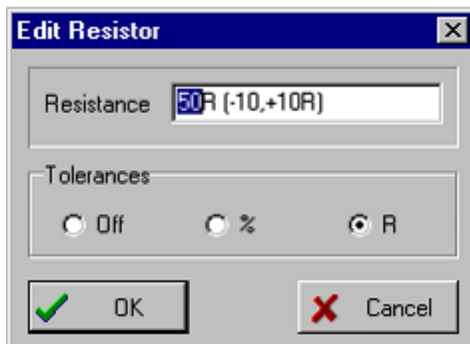
Normal Tolerance : negative min, positive max

This resistor would be 40R at min tolerance and 60R at max tolerance.



Reversed tolerance: positive min, negative max

This resistor would be 60R at min tolerance and 40R at max tolerance.




---

Created with the Personal Edition of HelpNDoc: [Free EPub producer](#)

---

## licence agreement

---

### HYDESIGN SOFTWARE LICENSE AGREEMENT for RFSim99

This license statement constitutes a legal agreement ("License Agreement") between you (either as an individual or a single entity) and HyDesign Ltd for the software product ("Software") identified above, including any software, media, and accompanying on-line or printed documentation.

By installing, copying or otherwise using the software you agree to be bound by all of the terms and conditions of the license agreement.

Upon your acceptance of the terms and conditions of the License Agreement, HyDesign Ltd grants you the right to use the Software in the manner provided below.

This Software is owned by HyDesign Ltd and is protected by copyright law and international copyright treaty and you must treat this Software like any other copyrighted material.

Except as provided in the License Agreement, you may not sell, rent, lease, modify, translate or sublicense the Software, media or documentation.

You acknowledge that no title to the intellectual property in the Software is transferred to you. You further

acknowledge that title and full ownership rights to the Software will remain the exclusive property of HyDesign Ltd, and you will not acquire any rights to the Software except as expressly set forth above. You agree that any copies of the Software will contain the same proprietary notices which appear on and in the Software.

You agree that you will not attempt, and if you are a corporate entity you will use your best effort to prevent your employees and contractors from attempting, to reverse compile, modify, translate, or disassemble the Software in whole or in part.

THIS SOFTWARE AND THE ACCOMPANYING FILES ARE PROVIDED "AS IS" AND WITHOUT WARRANTIES AS TO PERFORMANCE OF MERCHANTABILITY OR ANY OTHER WARRANTIES WHETHER EXPRESSED OR IMPLIED NO WARRANTY OF FITNESS FOR A PARTICULAR PURPOSE IS OFFERED

The entire risk as to the quality and performance of the Software is with you. HyDesign Ltd does not warrant, guarantee, or make any representations regarding the use of or the results of the use of the Software in terms of corrections, accuracy, reliability, or otherwise, and you rely upon the Software and results solely at your own risk. HyDesign Ltd does not warrant that the function contained in the Software will meet your requirements or that the operation of the Software will be uninterrupted or error free.

In no event shall HyDesign Ltd be liable for any damages whatsoever (including, without limitation, damages for loss of business profits, business interruption, loss of business information or other pecuniary loss) arising out of the use of or inability to use this product.

---

Created with the Personal Edition of HelpNDoc: [Generate EPub eBooks with ease](#)

---

## disclaimervent si>¾H

---

THIS SOFTWARE AND THE ACCOMPANYING FILES ARE PROVIDED "AS IS" AND WITHOUT WARRANTIES AS TO PERFORMANCE OF MERCHANTABILITY OR ANY OTHER WARRANTIES WHETHER EXPRESSED OR IMPLIED NO WARRANTY OF FITNESS FOR A PARTICULAR PURPOSE IS OFFERED.

The entire risk as to the quality and performance of the Software is with you. HyDesign Ltd does not warrant, guarantee, or make any representations regarding the use of or the results of the use of the Software in terms of corrections, accuracy, reliability, or otherwise, and you rely upon the Software and results solely at your own risk. HyDesign Ltd does not warrant that the function contained in the Software will meet your requirements or that the operation of the Software will be uninterrupted or error free.

In no event shall HyDesign Ltd be liable for any damages whatsoever (including, without limitation, damages for loss of business profits, business interruption, loss of business information or other pecuniary loss) arising out of the use of or inability to use this product.

---

Created with the Personal Edition of HelpNDoc: [Benefits of a Help Authoring Tool](#)

---