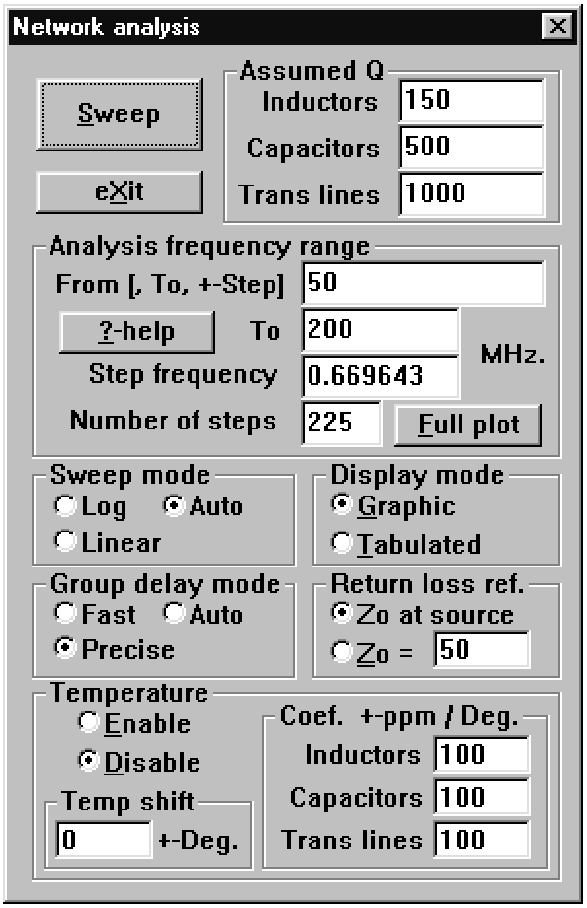
Network analysis

The main analysis dialog box (Not power analysis) is used to control the frequency range and resolution of the internal network analysis module.



Assumed Q

Sets the assumed quality factors for the three types of components. All of the same type of components in the network being analyzed are assumed to have the same Q.

Analysis frequency range

Four edit boxes are provided. The fist must have at least the starting frequency of the sweep. You can also key the entire analysis range and step into this box in this format:

`Start, Stop, +-Step`

Note that each item is separated by a comma. The last item can either be the step frequency or the number of frequency points as a negative number.

The bottom to boxes in the frequency range area shows the step frequency and the number of points. You only need to set one and the other is calculated for you. The maximum number of points is 1000.

Sweep mode

(*) Auto mode will sweep log if the stop frequency is 20 times the start frequency or more. **(*) Log** or **(*) Linear** only can be selected. These settings are recorded with each design ".spk" file.

Display mode

(*) Graphic mode:

When analysis is operating in the graphic mode, a graph of the analysis data is displayed immediately. The [**Full plot**] button will set the correct number of points to fill the plot.

(*) Tabulated mode:

In the "Tabulated" display mode, the analysis data is displayed in numerical form in a window with vertical scroll capability. The same scroll window is used from the Utilities, Hard copy review to screen feature.

Group delay mode:

The calculation of group delay requires the phase change between two frequency points. The delay calculated becomes the average delay between these two points. The closer these two frequency points are, the more precise the average will be to the delay at both single points. To precisely calculate the delay at a number of single points in frequency requires that two frequency points very close together be calculated for each point. If enough points are calculated however, the points can be considered close enough to yield a good approximation of the delay. PCFILT provides both a "**Fast**" delay mode where delay is calculated from phase change between adjacent frequency points, and a "**Precise**" mode where an extra unseen point is calculated just below each main point. In this mode, the unseen frequency point is F - F1 / 1 x 10^7 below the main point. F1 is the start frequency of the analysis run. This method is a factor of 2 slower than the "Fast" mode but yields a very close approximation of the actual delay at the main point if needed.

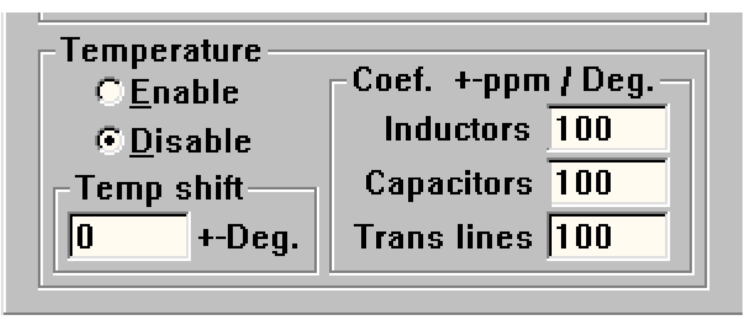
A third delay mode, "**Automatic**", will select the "Fast" mode if 50 or more frequency points are analyzed. If 49 or less points are analyzed, the "Precise" mode is selected.

The fast and automatic modes were developed when the program was being used on a computer having an 8086 CPU running at a blazing clock speed of 5 MHz! With the speed of modern computers there is not much need for the extra speed. It is recommended that the delay mode be left in the (*) Precise mode.

Impedance reference

Under some conditions it is desirable to display the return loss and Smith chart reference impedance in terms of a different impedance than the impedance of the source end of the filter. An example of this would be when evaluation a multiplexer to be used as a loudspeaker crossover network. An audio amplifier can be considered a voltage source having a very low source impedance but you would like to display the impedance seen by the amplifier in terms of the impedance the amplifier is rated to operate into, such as 4 or 8 Ohms. The normal (default) setting is **(*) Zo at the source**. If a different impedance reference is desired it may be set into the edit box associated with the **(*) Zo = [__]** option. The impedance is continuously being preset to that of the termination impedance of the network in memory until you select it. It will then remain at whatever impedance you set it to be.

Temperature shift analysis



When enabled, each part is shifted in value by the amount dictated by the temperature coefficient and temperature shift set. All of the capacitors, inductors and transmission lines in the network will have the coefficients specified for each.

(*) Enable:

This turns the temperature analysis function on. Once you turn it on, it will stay on.

(*) Disable:

This turns temperature analysis off. It is the default setting when the program is first started.

Temp shift

Set the temperature change from ambient temperature in the edit box provided. This NOT the actual assumed temperature, it is the temperature difference from ambient temperature. The values of each part in the network as displayed by the circuit editor is assumed to the value at a temperature shift of zero Degrees. If this is assumed to be 20 Deg C, then to analyze the network at 0 degrees C would require a setting of -20 Degrees. Similarly, to see the results of a temperature of 100 Deg C would require a temperature shift setting of +80 Deg.

Coef. +-ppm / Deg.

Set the temperature coefficient of each of the three part types in units of parts per million per degree. The actual temperature scale can be centigrade or Fahrenheit so long as the temperature shift is set assuming the same temperature scale.

The coefficient of transmission lines is that of the line electrical length. The line impedance is not changed with temperature.

The three coefficients you have set are recorded in the design file when the design is saved. The temperature shift setting however is not recorded.

Other good stuff to be aware of:

The Automatic delay mode status (on/off) is recorded in the ".spk" specifications file. If the automatic mode is recorded as OFF in the ".spk" file, the "Precise" mode will be the default. The mode recorded will be that set after the last analysis run.

When "Tabulated" display is selected, the Precise delay mode is forced. If the "Graphic" mode is returned to after a tabulation, the Delay mode will be set to either Automatic or Precise mode as was last set in the "spk" file, the Fast mode will not return automatically.

When in the "Automatic" delay mode, if the actual mode selected is the "Fast" mode (>49 points), The delay readout on the plot will show (Fast) just after the delay reference level, otherwise the display is "Precise". The first analysis point is ALWAYS done using the "Precise" method no matter what delay mode is selected.

Transmission line Q

Because the analysis module is normally concerned with the analysis of filters, forward transmission lines as well as stubs have been related to losses with respect to resonator Q. This leads to considerable confusion since the loss of a forward line in the "ladder" analysis scheme used must be related to dB / length, such as dB per foot. This also leads to varying insertion loss results when analyzing transmission line filters with different resonator lengths through the passband using the same Q factor. The question of whether this is, or is not a problem has NOT been resolved. The following correlation is used and seems to work well with filters having 90 Degree resonators:

$$Qt = \frac{L \; Pi \; Log10(e)}{18 \; dB} = .0758 \frac{L}{dB}$$

```
Where:
 L = Length in Degrees.
Pi = 3.141.....
 e = 2.7183....
```

Finding the Zo AT a point in a network

Analyzing the Zo AT a point in a network rather than looking INTO it, as is normally the case when using the "`Report —> ─┘`" branch, can be done by using some fancy branch shuffling to "fold" the filter back on itself bringing the point we want to analyze to a "common". As an example, here is a standard L-C nodal type bandpass filter in which it is desired to know the impedance at the second section.

```
 0 ┌─ Termination ─┐           50 Ohms
   │               │
 1 │               C       9.2896 pFd.
   │               │
 2 ├────── C ──────┤       18.435 pFd.
 3 ├────── L ──────┤       87.455 nHy.
   │               │
 4 │               C       2.1814 pFd.
   │               │
 5 ├────── C ──────┤           27 pFd.
 6 ├────── L ──────┤       82.796 nHy.
   │               │
 7 │               C       1.6422 pFd.
   │               │
 8 ├────── C ──────┤           27 pFd.
 9 ├────── L ──────┤        84.27 nHy.
   │               │
10 │               C       1.6422 pFd.
   │               │
11 ├────── L ──────┤       82.796 nHy.  \ Analyze to find the impedance
12 ├────── C ──────┤           27 pFd.  / at this point in the filter.
   │               │
13 │               C       2.1814 pFd.
   │               │
14 ├────── L ──────┤       87.455 nHy.
15 ├────── C ──────┤       18.435 pFd.
   │               │
16 │               C       9.2896 pFd.
   │               │
17 └──── Source ───┘           50 Ohms
18  Fc = 99.626 MHz.
```

After "folding" using the circuit editor >Insert and >Move commands:

```
 0 ┌─ Termination ─┐           50 Ohms
   │               │
 1 │               C       9.2896 pFd.
   │               │
 2 ├────── C ──────┤       18.435 pFd.
 3 ├────── L ──────┤       87.455 nHy.
   │               │
 4 │               C       2.1814 pFd.
   │               │
 5 ├────── C ──────┤           27 pFd.
 6 ├────── L ──────┤       82.796 nHy.
   │               │
 7 │               C       1.6422 pFd.
   │               │
 8 ├────── C ──────┤           27 pFd.
 9 ├────── L ──────┤        84.27 nHy.
   │               │
10 │               C       1.6422 pFd.
   │               │
11 ├────── L ──────┤       82.796 nHy.
12 ├────── C ──────┤           27 pFd.
   └─────>         └─── to Common A ───>    Insert port termination here.
13 ┌── port term ──┐           50 Ohms  <-- (Zo equals old source Zo.)
   │               │
14 │               C       9.2896 pFd.  <-,   This is the source end of the
   │               │                      | filter with the branches in
15 ├────── L ──────┤       87.455 nHy.    | REVERSE order. This can be done
16 ├────── C ──────┤       18.435 pFd.    | using the Insert or Move circuit editor
   │               │                      | commands.
17 │               C       2.1814 pFd.  <-'
   │               │
18 │               │<─── Common A       <-, Add these two new branches.
19 │   Report ──> ─┘                    <-'
20 └──── Source ───┘        517.1 Ohms  <-- Value determined by iteration.
21  Fc = 99.626 MHz.
```

The source Zo set (at branch 20) is not important to the analysis of the Zo looking into the "Common" at branch 18. Its value can be changed to roughly the mean value determined by the analysis if desired. Only if the Source Zo is changed, as shown here, will the forward attenuation and return loss roughly resemble a power splitter as might be expected. Forward loss will be around 3 dB with no component losses. For this purpose however, only the value is of interest.

The `Report ——> ┘` branch (19) is included here for clarity but is not necessary because the analysis module normally calculates and records complex impedance looking into the source of every network for use with the Smith chart feature of the analysis plot module.

Q Exception branches

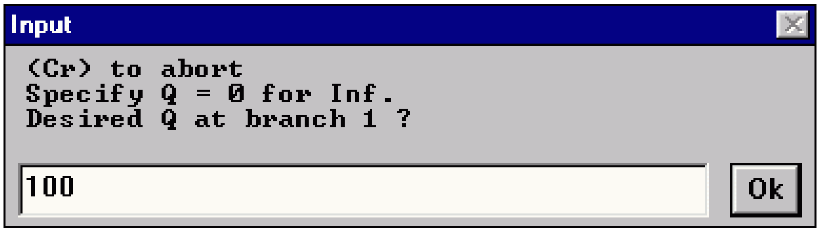
It often happens that specific elements in a network will have Q values different from that assumed by the analysis for all of the inductors and capacitors. For example, with quartz crystals, whose equivalent inductance Q might be several orders of magnitude higher than wound inductors. PCFILT provides a method to specify the Q factor of any specific branch by adding a **Q exception** branch ahead of that branch. Another example of this would be the use of an all-pass group delay equalizer in the center of a bandpass filter to act as an amplitude equalizer. The finite Q of the inductors will generate a rise of attenuation at the point of maximum group delay. With the correct inductor Q, the passband of the bandpass can be flattened. Below, a group delay equalizer is shown in a bandpass. The circuit editor “Q Exception” feature will be used to insert two Q exception branches to allow the Q of both inductors to be specified or iterated on by the optimization module to flatten the passband.

Any number of Q exception branches can be added to the network. Each will reference a single branch somewhere after it (higher branch number). Multiple branch networks like the all-pass equalizer cannot be broken up, so all the Q exception branches will be inserted ahead of the equalizer. In this case, the two Q exception branches will be inserted between branches 0 and 1 to reference the inductors at branches 1 and 4. A Q exception branch can reference branches as many branches ahead as necessary. All the inserting and referencing will be done automatically.

```
 0  ┌- Termination ─┐          50 Ohms
    │            ┌──┴──┐
 1. │            L     C     6.7295 nHy.       745.41 pFd.
 4  ├─── L ── C ─)─────┤     931.76 nHy.       2.6967 pFd.
 3  │            │     C     745.41 pFd.
    │            └──┬──┘
    │               │
 6  │               C        18.447 pFd.
    │               │
 7  ├──────── C ────┤        70.629 pFd.
 8  ├──────── L ────┤        27.968 nHy.
    │               │
 9  │               C        6.7865 pFd.
    │               │
10  ├──────── C ────┤        79.513 pFd.
11  ├──────── L ────┤        27.968 nHy.
```

Begin by highlighting the value of the inductor at branch 1 using the mouse left button. Select the “**Q exception**” item on the [**MIS1:**] edit menu.

The standard “Input” dialog box will ask you for the value of Q you would like to assign to branch 1.

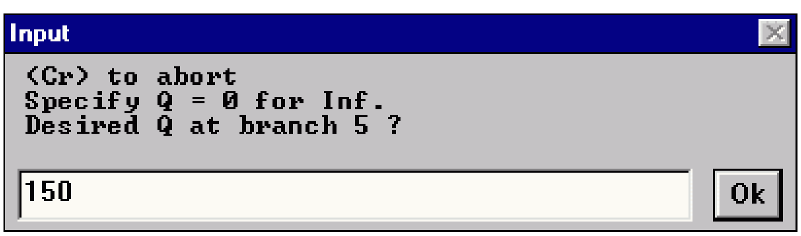


The results will look like this:

```
 0  ┌─ Termination ─┐           50 Ohms
 1  │  Q branch 2   │         Q = 100.0
    │            ┌──┘
 2. │            L    C      6.7295 nHy.      745.41 pFd.
 5  ├── L ─ C ─)─────┤       931.76 nHy.      2.6967 pFd.
 4  │            │    C      745.41 pFd.
    │            └──┐
 7  │               C        18.447 pFd.
    │               │
 8  ├──────── C ────┤        70.629 pFd.
 9  ├──────── L ────┤        27.968 nHy.
    │               │
10  │               C        6.7865 pFd.
    │               │
11  ├──────── C ────┤        79.513 pFd.
12  ├──────── L ────┤        27.968 nHy.
```

The newly inserted Q exception branch becomes branch 1 and indicates that it references branch 2.The assumed Q of the inductor at branch 2 is 100. This value can be changed with the “Change” menu option of the circuit just like any other branch value.

In order to reference the Q of the second inductor at what has become branch 5, another Q exception branch is required. Highlight the value of branch 5 and select “**Q exception**” again.



Take carful note of the branch numbering of the all-pass network. The branch numbers are NOT displayed consecutively.

```
 0 ┌─ Termination ─┐           50 Ohms
 1 │  Q branch 6   │       Q = 150.0
 2 │  Q branch 3   │       Q = 100.0
   │             ┌─┴──┐
 3.│             L    C    6.7295 nHy.       745.41 pFd.
 6 ├─── L ── C ─)────│    931.76 nHy.       2.6967 pFd.
 5 │             │    C    745.41 pFd.
   │             └─┬──┘
   │               │
 8 │               C       18.447 pFd.
   │               │
 9 ├─────── C ─────┤       70.629 pFd.
10 ├─────── L ─────┤       27.968 nHy.
   │               │
11 │               C       6.7865 pFd.
   │               │
12 ├─────── C ─────┤       79.513 pFd.
13 ├─────── L ─────┤       27.968 nHy.
```

As with the other assumed Q inputs to the analysis module, a Q of 0 (zero) means infinite Q.

Q exceptions function for inductors and capacitors only, no transmission lines.

Including equalizer data in analysis

A Method to include the file data used to design amplitude or group delay equalizers into the analysis has been provided. This will add the data in a disk file to the results of analysis across the frequency range of the data in the file. Linear interpolation is used to determine data between frequency points provided in the file. To do this a special branch must be added to the network using the circuit editor "Insert" feature. The branch is identified as "**|- EQ Data -|**" on the insert menu. Insert it anywhere near the network which will be responsible for doing the correcting for the data in the file. When the EQ Data branch is inserted you will be asked to define if the data in the file is group delay or amplitude ([A] or [D]). When the first analysis sweep is done a prompt will request the file name containing the data unless an equalizer was just designed using a data file. In this case that data file will be used. The default file name is **kbd_data.eqd** which is the same one used by the equalizer module. In the event you wish to specify a different file at any time simply press the **[eXit]** button on the analysis dialog box. The next time an analysis sweep is done a prompt will appear asking for the name of the file to include. When doing optimization, the same procedure will allow the file to be changed. The analysis dialog box can be called from the main menu when the optimization window is open. Press [Exit]. The file name prompt will appear when the optimization next calls the analysis module. This may seem cryptic but it eliminates the need for a separate command to specify the file name. It is actually quite convenient.