

# Handbook for AdLab/AdLabPlus 2.5x

## Contents

### 1 Overview

#### 1.1 New in this Version

#### 2.1 Usage of Smith Chart Tool

#### 2.2 Background Information to the Smith Diagram

#### 2.3 Matching Networks Examples

#### 2.4 Reflection Factor and SWR Table

#### 3.1 Case Study Broadband Amplifier Design

#### 3.2 Case Universal Matching Network

#### 3.3 Case Study Match for RF Power Amplifiers4.1 Hints on the AdLab Tools

#### 5.1 How to Get the Full Version and Support

#### 5.2 Information on the Restrictions of the Shareware Version

## 1 Overview

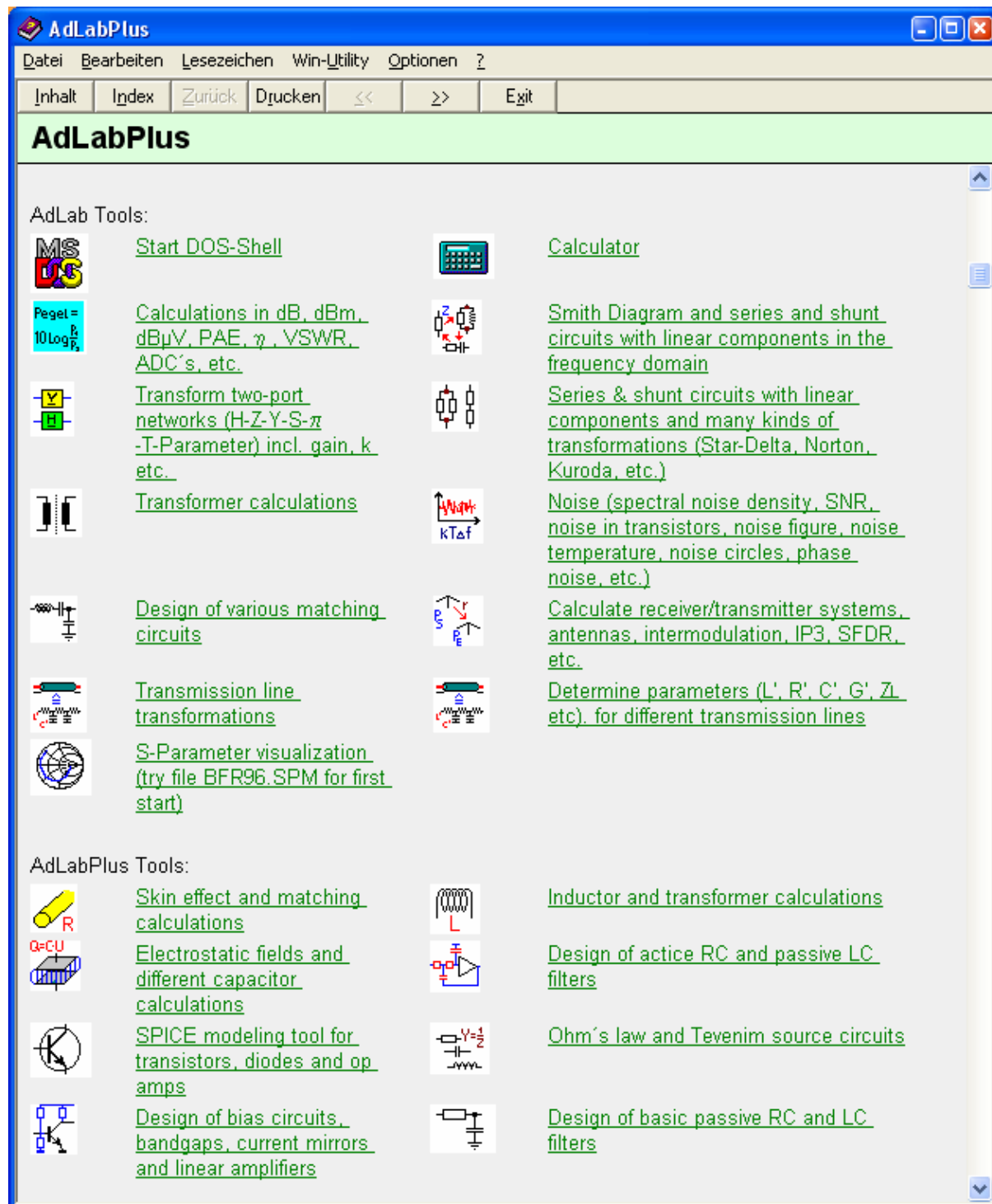
*AdLab* is a collection of electronic related programs with many high-level features. For the installation of AdLab, simply extract all the files (preferable in a new directory) and double click ADLAB.HLP (or ADLABP.HLP for AdLabPlus, extract AdLabPlus.ZIP after AdLab.ZIP). To put it on the desktop, drag the *AdLab*-icon from the explorer to the desktop.

With the special *AdLab* tools you can perform a number of operations. Most of them are time consuming and error prone if done manually. By letting the tools do "all" the work for you, you the designer can spend more time designing rather than calculating.

The electronic Smith chart program (also available separately) is a good example for this. In the help file you will find many information which is also valid for the other *AdLab* tools. With the Smith chart tool you can design matching networks in a similar way to the conventional Smith diagram, but there is a number of new features included. For instance you can:

- use elements including its parasitics, like series resistors or stray capacitances
- also include transistors and S-parameter blocks
- easily modify, rearrange, load and save your circuits
- export S-parameter of the total circuit
- export the total circuit to other simulators for optimization or extension
- overlay circles for constant Q, constant VSWR, etc.
- perform frequency sweeps and a Monte-Carlo analysis
- display power gain or return loss vs frequency
- get more information like stability factor, MAG, available gain, VSWR, etc.
- transform mixed parallel and series RLC combinations
- copy to clipboard or print all relevant data

That's the first page of *AdLabPlus* show after start-up:



## 1.1 New in This Version

### Improvements in Csmith:

Even though CSmith V. 1.0 was a full-featured Smith chart program. There was some work done to make it even better. In spite of this the current version of CSmith is still compatible with earlier versions.

#### Changes from version 1.0 to 1.1 :

- Some people asked if it is possible to include the circles which connect the different impedances at each element. This was not so easy because CSmith includes also the parasitic elements, i.e. the circles don't lie on the admittance or impedances Smith chart circles. CSmith also has elements like transistors or S-parameter blocks in which it makes no sense to connect the element impedances by a circle. In spite of these difficulties this feature is now implemented!
- Because CSmith has elements like transistors or S-parameter blocks in which it makes no sense to connect the element impedances by a circle you can suppress the connecting lines for these elements..
- For loading circuit files not only a file browser is implemented but also a recursive file searcher.
- The number of turn ratio was named  $\ddot{u}$  in the 1<sup>st</sup> version which is common in Germany. Now it has the letter  $v=n1/n2$ .
- Very few hints, items, etc. were still in German for Version 1.0. Now I think everything is in English. The only exception - due to compatibility - is the INI file.
- It works also for version 1.04 but it was not documented: You can specify a circuit file on the command line which will be loaded after startup. Also in the Explorer you can drag a circuit file by mouse over the CSmith program.

#### Changes from version 1.1 to 1.15 :

- Saving circuits works good, but in version 1.1 there was an error in saving circuits with transformers. This bug is fixed.
- For matching networks with quarter wave length line stub lines there is no change in the impedance. This creates an error message in version 1.10. Error checking is now improved.
- CSMITH.EXE is renamed to CSERPAR.EXE to give a better integration to *ELEKTA* and *AdLab*.
- In the 1<sup>st</sup> tab page you can load the intermediate impedances from the Smith chart. Now this is possible also for the generator. Astonishingly this makes the working much easier if you use a complex impedance as the generator impedance.

#### Changes from version 1.15 to 1.21 :

- The Smith chart can be copied to clipboard or can be saved as BMP file. These functions are available via standard menu and context pop-up menu.
- Now you can place a text below the Smith chart for instance for documentation.
- In the previous versions the labels at the Smith chart have a wrong sign.
- If no printer is installed the former version crashes. This bug is removed.
- The bubble help for the element input field was not always up-to-date. Now it is.
- Now you can mark a point in the Smith chart via mouse click and context pop-up menu.

#### Changes from version 1.21 to 2.0 :

- Last time I had to design a large matching network. So I increased the number of possible elements to 12. This also results in a larger program window.
- The sweep steps for the generator reflection coefficient are optimized to speed up operation.
- Now there is also a graphical output for the MAG vs frequency. This is helpful for amplifiers, but also for matching networks, to see what is really achievable.
- On the 1<sup>st</sup> page I added a graphical output for the voltage across a transmission line which shows the reflexion.
- Correction of some minor errors. For instance the length tolerance in the Options dialog was not updated correctly in earlier versions.

#### Changes from version 2.0 to 2.1 :

- Drag & Drop together with the MS Explorer is now implemented. Simply open the Explorer (or Filemanager) and drag a CSP file to the CSmith window.
- You can set the graphic output window to „always stay in front“. This is very useful if you want to trim the circuit elements.
- Now you can export the automatically created matching circuit from the 1<sup>st</sup> page to a circuit file.
- A minor bug is removed: The spin buttons to tune the series resistance of a transformer are now enabled.
- The tabular out is now more flexible, e.g. you may display either  $|r|$ , VSWR or return loss.
- The autoscale function in the graphical output is now enhanced. Formerly it works only for the transducer gain and rounding was not always correct.
- Now CSERPAR will automatically a link to all CSP files. So if you double click a CSERPAR circuit file the program will be started immediately.
- If you work with a circuit file (e.g. you load or save such a file) the filename will be put to the Windows Start button. If you click to Start - Documents you will see the recent files and you can start CSERPAR very comfortable.
- In version 2.0 an error occurs if you insert a transistor although appending works correctly. This error was due to initialization of some transistor parameters and is now fixed.
- Calculation of the maximum theoretical bandwidth due to Fano on the 1<sup>st</sup> page.
- Now you can jump to the AdLab homepage or write an email to the author within the program (look at the Help menu).

#### Changes from version 2.1 to 2.25 :

- Much more information is included in the help file on matching and Smith diagram usage.
- In the older versions by default no elements will be optimized (in Aplac). Now the default is set to Optimization=True for capacitors and inductors. This is more realistic and helps to reduce user entries.
- Now there are three kinds of optimizations available : The usual pass-band optimization and an optimization based on a tolerance scheme for for pass and stop band. This is very useful for filters, e.g. you can design the pass-band in CSERPAR and then optimize also the stop-band in Aplac. The 3<sup>rd</sup> optimization type is transistor model optimization on S-parameters loaded from a file. This is for instance useful if you have S-parameters in a small frequency range. Then the optimized model will give a good extrapolation.
- Saving of S-parameter data is improved, now you can specify the frequency range and the stepsize.
- In 2.1 the Fano calculation leads to an error if the phase is zero. This bug has been removed.\*
- When you delete the last element (load) in version 2.1 this leads to an error. This bug is removed.\*
- Now it's possible to abort the frequency sweep in the Smith chart.\*
- The export of automatic L-match wasn't correct for  $R_s > Z_0$  (elements interchanged). This has been corrected.\*

\* by suggestion of Mr. Eric van Slee

#### Changes from version 2.25 to 2.29 (Juli 2001) :

- Sometimes the connection of the different impedances work not 100% correct. So I add a checkbox in the Options window to enable straight line interconnection as in version 1.0.
- When writing S-parameter, Aplac or circuit files backup files will be created if needed.
- The table now also shows the phase of the reflexion coefficient.  
The column which corresponds to the selected element is now marked and the impedance in the Smith chart also.
- The graphical VSWR output on the 1<sup>st</sup> page is now correct. In the previous version it was a sine wave shape, but this is only an approximation.
- Switching between the tabs and to „Z(Generator) from 1<sup>st</sup> page“ was enhanced.\*
- The output for optimum load and generator impedance in the More window was wrong. It was correct in the Two-Port program VIERPOL.\*
- To have access to ALL AdLab tools I added ANPASS and ANTENNE to the Tool item of the CSmith menu.

\* by suggestion of Mr. Eric van Slee

#### Changes from version 2.29 to 2.3 (October 2001) :

- I increase the window size to increase the maximum number of elements to 14. That's good for multi-stage transistor amplifiers or large filters.
- Often an element such as a non-ideal bias-T is needed. Such element is now available in the Smith chart (with Series L and R and blocking capacitance to ground with parallel R).
- Menu item in the File menu to create an icon on the desktop for CSmith/SERPAR.
- The frequency sweep in the Smith chart is much improved. Now it has more points, you can select a linear or logarithmic sweep and the point density and labels for the frequency can be placed automatically.
- In the graphical output now the output impedance is available. This was previously possible only when using CSERPAR and SPARAM together.
- In the previous versions you can copy, paste & insert the circuit elements. Now a move function is available. That's nice for potentially unstable amplifiers to insert damping resistors.
- The output of MAG is enhanced, if  $k < 1$  MSG is shown.
- Now you can suppress the output for the -1dB and -3dB level line in the graphic menu.

#### Changes from version 2.3 to 2.31 (January 2002) :

- The Smith chart size is increased by more than 20% to get a better overview for matching network design. I also added some labels to the chart.
- On the 2<sup>nd</sup> page left to the table I added a button to switch between polar and Cartesian format.
- To find the current element faster in the Smith chart, the arc of the current element is now made thicker.
- Now you can also insert gain blocks defined by input and output resistance and power gain. This is useful for amplifier design if no transistor data or S-parameters are available.
- The window for the table on VWSR, return loss, etc. was still in German. I translated it now. I also add two buttons, one for email and one for webpage access.

#### Changes from version 2.31 to 2.4 (April 2002) :

- Integration to AdLabPlus.
- E-Mail connection in registration dialog.
- Output for damping of a transmission line corrected and added output for mechanical length in status bar.
- I changed the Smith chart background color from silver to white. This more useful if you want to copy the chart to your own documents (Word files etc.) or if you want to make photo copies from a printout.
- For the graphical Monte-Carlo output I marked the normal curve (nominal design) with a wider pen and a darker color. I also increased the number of runs to 10 because today's pcs are much more powerful.
- More information, like VSWR and return loss at the input port, will be shown on the 'More'-window for the Smith chart. I also rearranged the output to give the user a better overview.
- The function „Always stay in front“ is very useful for the graphic window, e.g. if you trim matching elements. In the past this function was coupled with the autoscale function. Now the coupling is removed, which is an advantage if you have modified the axis scales, etc.
- Export of the Smith chart to Jpeg file. It's typically much smaller than a BMP file with slightly lower quality.

#### Changes from version 2.4 to 2.41 (July 2002) :

- The table output für VSWR was not always updated correctly after activating the graphic window. This small bug has been fixed.

To Version 2.42 (October 2002) :

- I put a nice Smith chart with some explanations to the help window.
- I changed some colors in the Smith chart to improve contrasts.

To Version 2.5 (January 2003) :

- Many new menu items have been added to improve Smith chart usage.
- In the Smith chart page now you can display nearly all the circuit element values in the schematic view. There is a button to toggle between the output table view and the circuit elements. I also changed the background color to white get make it nicer for printing and exporting.
- The bubble helps might bother the experienced user, so you can disable them now in the Options menu.
- For the Smith chart I added a button to append an element from buffer.
- Many improvements have been made within the Monte-Carlo window (e.g. printing is now available, axis scaling is enhanced, the output has now the more common line diagram and the number of runs has been increased to 1000).  
Now you can also enter limits for transducer gain to obtain yield information and Cpk values.
- Flickering of the cursor cross in removed in all charts.
- The Search Circuit dialog features now a larger output window to display also very long pathes.

To Version 2.51 (April 2003) :

- Help for Smith chart has been improved.
- In capacitor or inductor elements you can now also define loss by parallel resistance (instead of series resistance only. This allows e.g. better modeling of magnetic loss in coils.
- An error when appending circuits has been corrected.

### Improvements in the other tools:

To Version 1.15 : The previous version based on 16-bit Windows, now all programs are 32 bit. There are also many other improvements, for instance :

- |            |  |
|------------|--|
| • ANPASS   | More topologies, calculation of bandpass filters and class E amplifiers                            |
| • ANTENNE  | Calculation of mixer spurious, intermodulation products and phase noise requirements in TX systems |
| • DBRECHNE | VSRR calculations  |
| • LBELAG   | Export function and definition of lines by L', C, etc.   |
| • LTRANSF  | Output for equivalent circuits   |
| • NOISE    | Noise circles and calculation of noise parameters  |
| • SERPAR   | Filter denormalization   |
| • TRAFO    | Symmetrical transformers   |
| • VIERPOL  | Table output, area factor, feedback, Smith chart, transmission and ABCD parameters and much more!  |

To Version 1.21 :

- |           |   |
|-----------|---|
| • General | Better translation from German to English                             |
| • LBELAG  | Export function for SPICE and APLAC, Model includes series resistance |

To Version 2.0 :

- SPARAM A complete new tool to visualize S-parameter data (Smith chart, Bode plot, k-factor and much more). After starting the program load a file with Touchstone or Supercompact format like BFR96.SPM from the file menu.
- Help Case study on a universal RF matching network
- General Minor corrections

To Version 2.1 :

- DBRECHNE Damping and voltage drop for PAE calculations now adjustable.
- NOISE Output for base current in transistor noise calculations.
- VIERPOL If you load a s-parameter file it will be also displayed in the Windows Start menu under Recent Documents. In the tabular output you now also get the optimum generator and load impedances.

To Version 2.25 :

- LBELAG & LTRANSF Direct coupling of the programs: If you design a TRL in LBELAG you can start LTRANSF and you automatically get the correct values for  $Z_0$ ,  $f$ , length, etc.
- LTRANSF Now you can also export a lumped model to CSERPAR. This has advantages when you want to insert a stub e.g. with an end capacitance or inductance.
- ANTENNE The old table entry for the "Intermodulation Calculation" has an error, it wasn't possible to enter new frequencies. This is corrected and also the autoscaling has been much improved.\*
- SPARAM Some bugs has been removed and now the program can handle larger SPM files.\*
- DBRECHNE New page for dB calculation on digital-to-analog and analog-to-digital converters including SINAD, THD, etc.
- Help Case study on match for RF power amplifiers

\* special thanks to Mr. Eric van Slee for his hints

- ANPASS Export to CSERPAR is now with optionally with E-series values for all circuit elements.
- NOISE Calculation of oscillator jitter (and its SNR influence) and output für the power of the minimum detectable signal.
- DBRECHNE Calculation of VSWR uncertainties for power measurements

To Version 2.30 :

- CSERPAR Again, many things! See CSMITH.HLP
- ANTENNE Now you can enter the antenna temperature\*. This is useful for non-terrestrial links.
- DBRECHNE Output field for peak-to-peak voltage at the 1<sup>st</sup> page and for power gain at the 2<sup>nd</sup> one.
- SERPAR Now correct US-style resistor symbols in all pages. Window size has been increased.
- Help Presentation of the key factors in RF power amplifier design, showing also the usage of some AdLab tools..

\* special thanks to Mr. Luitjens Popken for his hints

To Version 2.31 :

- CSERPAR Again, many things! See CSMITH.HLP
- LBELAG Some value are not stored correctly in the INI file. I corrected this and rearranged the output fields to get a better overview.
- ANPASS Now designing a double-L type section is implemented. This type of matching network is quite common for RF power amplifiers.  
A bug is removed in the series LC tank matching configuration.  
For the circuit export to CSERPAR a comment is included.
- ANTENNE Extra output field for the output intercept point OIP3.
- DBRECHNE More output fields for the VSWR calculations (e.g. short-circuit VSWR and ratio of forward and reserve traveling power)

To Version 2.40 :

The previous version based on some older ELEKTA tools. For AdLabPlus many extensions have been made, for instance :

- AFILTER Some small bugs have been removed and the design of an LC-bandpass diplexer is included. Also now it is possible to design 2<sup>nd</sup> order highpass / lowpass diplexer.
- COILS Includes all transformer calculations of TRAF0 from AdLab, but also all coil calculations of ELEKTA. I also added the inductance calculation of an wire over a ground plane.
- LEITER Includes now also the calculation of matching behavior of resistors and transistors.
- ESTATIK Features now the calculation on triplate and microstrip transmission lines and also the design of a sample & hold circuit (Drop-rate, hold-step, etc.).
- MODEL This is totally new and combines modeling on diodes, different transistor types and op amps. Very powerful are the tuning modes for many parameters and the graphical outputs.
- OHM Combines two smaller programs for Ohm's Law and source equivalent circuits.
- RLCFILT Combines the two programs for RC and LC filters and has also some extensions.
- TRANSISTOR This is a very powerful program for the design of transistor bias circuits, current mirrors, bandgap circuits and linear amplifiers including a graphical output for MAG and stability factor.

To Version 2.41 :

- SPARAM Graphical output also characteristic impedance vs frequency.
- LEITER Output also for conductivity  $\sigma=1/\rho$ .
- TRANSISTOR For current mirrors the area ratio should be much smaller than the DC beta of the transistors, so some checks are needed. These are now improved to get less confusing warnings.
- AFILTER The output for bandstop gain was wrong (especially above the resonance frequency). This has been corrected.
- OHM The help was extended and for the time domain calculations I added a graph to give a better understanding.
- COILS Now you can calculate the mutual inductance between straight wires and a rectangular loop.
- ESTATIK Output for area of an integrated capacitor also in mm<sup>2</sup>.

To Version 2.42 :

- DBRECHNE Completely new tabpage for calculation of oversampling ADC/DAC's. Additional output for collector/drain efficiency.
- LBELAG Bond wire calculation included.
- ANTENNE Due to a typing error the receive field strength was calculated to small\*. This has been corrected.
- Some When the calculated output is not a valid number a hint was shown, like 'not needed' or 'infinite', sometimes this hint was cutted in the past due to too small output fields. Now the programs switch internally to a smaller font to prevent this.
- TRANSISTOR More design hints on each circuit.
- COILS Calculation of square-shaped spiral inductors using Wheeler's formula.
- ESTATIC The supply current of switching logic gates will be estimated on a new tabpage.
- AFILTER Startup speed and autoscaling in graphical output is improved. Also now you also have a complete output for group delay in LC ladder filters.
- Some In some tools the color sceme was not consistent in all Windows environments.

\* special thanks to Mr. Jouko Tenhunen

Version 2.42 was created using Delphi 7 instead of Delphi 3. The executable files become significantly larger, so I create the AdLab files by using packages, i.e. common parts are now included in shared

files. In version 2.5 I also find some inconsistencies in the background colors of some windows, which become aware when I changed the default button color. To improve user interface I also put some nice icons to important menu items and made a lot of minor improvements. To decrease memory occupation I used more dynamic child windows instead of static ones. In 2.5 you can also export graphics in JPEG format in all the tools. Note that the quality is a bit worse compared to BMP. However, the file size is often more than 10 times lower

#### To Version 2.5 :

- Some I added some menu items for features already available from buttons to improve user interface.
- DBRECHNE Some German notations have been corrected and I added some more hints.
- ANPASS A proposal for best matching networks have been implemented. Also unneeded input field have been removed depending on selected tap page. I also improved the optimization to get only positive component values.
- NOISE I made the window a little bit larger and I also corrected the output on the 1st page for the corresponding RC filter bandwidth (it's  $2/\pi$  times the noise bandwidth).
- AFILTER Improved menu for calculation of filter order, it's now both for high- and low-pass filters and also for cascaded passive PC filters.  
Also a small bug in LC filter calculation has been removed (calculation was wrong for  $N=2$  if you previously design a higher order filter) and the user interface is improved (better recalculation).  
From the LC-Filter page you can now also create 2<sup>nd</sup> order subfilters. So the LC-Filter page will also work as a starting point for an active filter design.
- LEITER The graphical output for cylindrical conductors was not always correct. This has been correct and all current densities are now normalized to the maximum.
- OHM By accident a lot of the spin buttons don't work, I correct this.
- RLCFILTER By accident no startup link was implemented for RLCFILT in the old help help. Now there is! I also added a completely new page for Twin-T, Wien bridge and RC bandpass filters.
- ESTATIC The S&H tool will now also calculate the frequency drift which is related to drop rate when a VCO or FM demodulator is involved. For the calculation of microstrip or strip lines there is now a button for more detailed calculations with transmission line tool.
- SYS In 2.42 this program was included accidentally. 3 of the tab pages work already fine, but the multistage PA designer was not fully implemented.

#### To Version 2.51 :

- VIERPOL In 2.5 an error message occurred when the program was started from ADLAB.HLP.
- ESTATIC I correct the formulas in the CMOS page and add an output field for the effective resistance and extend the help text.
- RLCFILTER Additional Q calculation for LC tank circuits.
- Help Some extensions in help file, update of many pictures.

## 2.1 Usage of Smith Chart Tool

The program is divided in two tab pages.

1<sup>st</sup> page :

It is related to the transformation of mixed parallel and series connections of RLC elements. It is very helpful if you get an impedance (e.g.  $10\Omega + j5\Omega$  from the Smith chart section or by hand calculations) and you now want to get an equivalent circuit for this impedance at a given frequency. Also you can get a simple L-section matching network from any impedance to the characteristic impedance  $Z_0$  of your system.

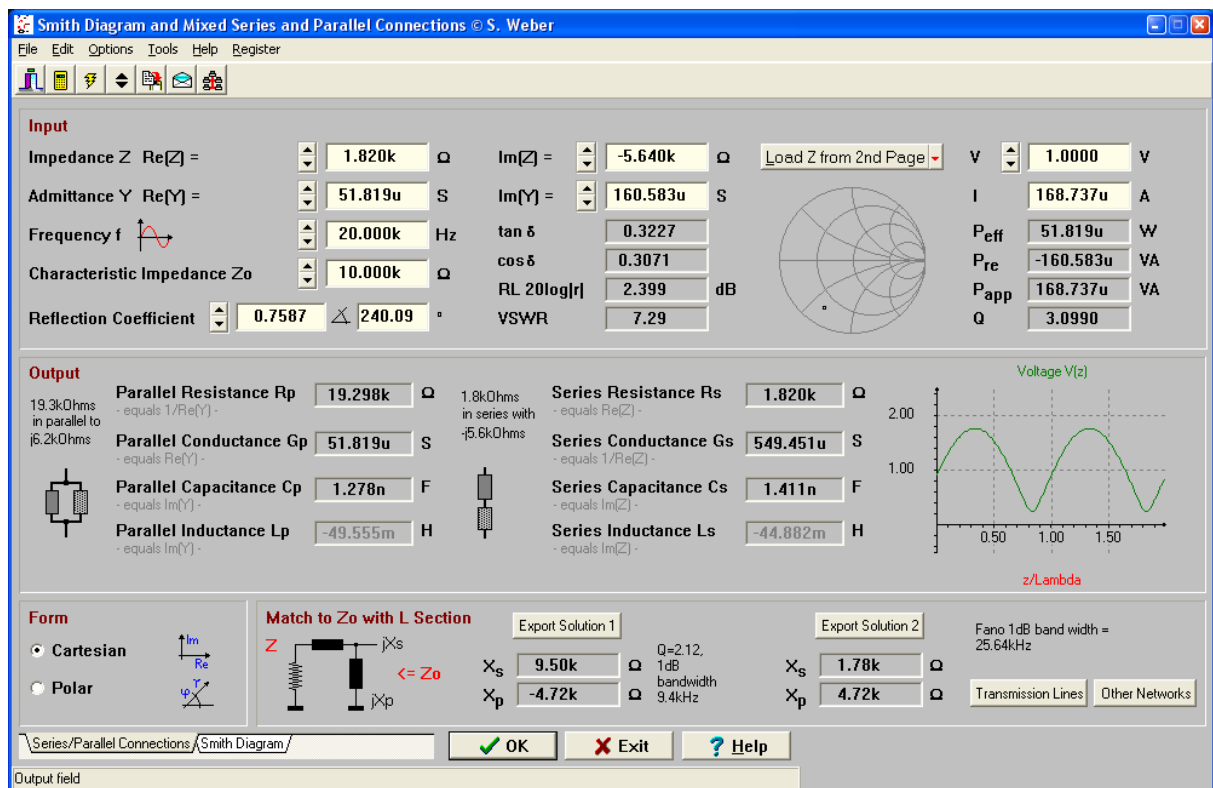


Figure : 1<sup>st</sup> page

2<sup>nd</sup> page :

Here you can design any kind of matching network (or amplifier, filter, etc.) using an electronic Smith diagram. All blocks can be specified including their parasitic elements, which is very important for high frequencies.

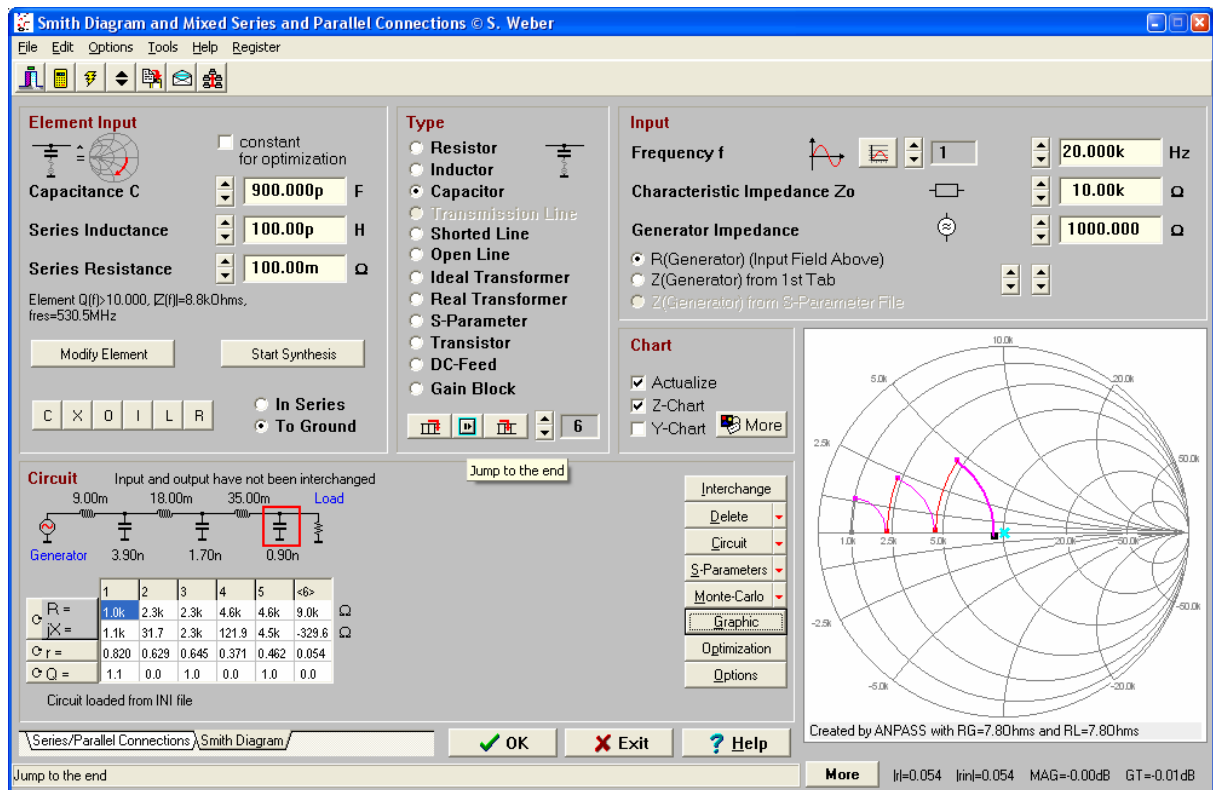


Figure : 2<sup>nd</sup> page (with 6<sup>th</sup> order lowpass as an example circuit)

### Circuit Entry :

You can load a already defined circuit into the program by specifying it on the command line, e.g. `cserpar wbmach.csp` (look at the example files). Another possibility doing this is to press the item load on the Circuit button. To define a new circuit or to add a new element you have to go to the load (at the right end of the circuit) by pressing the End button (the button with the right arrow symbol). Then you have to decide on the connection (in series or shunted to ground) and the type of the new element. After this you can modify the element values and insert it by pressing either the Insert (new element inserted on an intermediate position) or Append (insert the new element at the end) button. CSERPAR can work with up to 12 blocks, but if you export the S-parameters of the circuit and import them as a block you can calculate much larger circuits.

### Editing of the Circuit :

Often you need to modify your circuit to get the performance needed. The element values can be adjusted very easily at the input fields or using the spin buttons. To remove a block you can choose between the actual, the last or all the blocks. The element type cannot be changed directly, for doing this please remove the element and insert a new one. Above the circuit there are also some small buttons to copy (C), cut (X), overwrite (O) or insert (I) blocks. This is very similar to the standard windows clipboard function. Of course it's also possible to save or load circuits. There is also an append circuit function available which is useful for two-stage amplifiers or filters.

### **Hints on Output :**

The output of the circuit impedance (at the load) is directly on the Smith chart. If you want to take a look at the input side, then simply interchange the input and output by pressing the Interchange button. There is also a frequency sweep available by pressing the small button near the frequency input field. CSERP can also display the power gain and the return loss vs frequency, which is very important for broadband matching.

### **Hints on Optimization :**

Sometimes it's not easy to achieve optimum performance by hand. This is very often the case for active or broadband circuits. Here optimization is the best tool. CSERP creates an optimization netlist for the well-known APlac simulator. Of course it's possible to extend or to modify the circuit in APlac, e.g. to include the bias networks. The APlac syntax is very similar to the syntax of other RF simulators like Touchstone or SuperCompact.

### **Hints on Monte-Carlo Analysis :**

This feature is available in the Smith chart and in the graphical output for power gain and return loss. In the chart you can either perform each run separately or a number of runs which will result in a cloud of impedances. There is also a summary information on mean value, sigma, minimum and maximum available.

### **Options :**

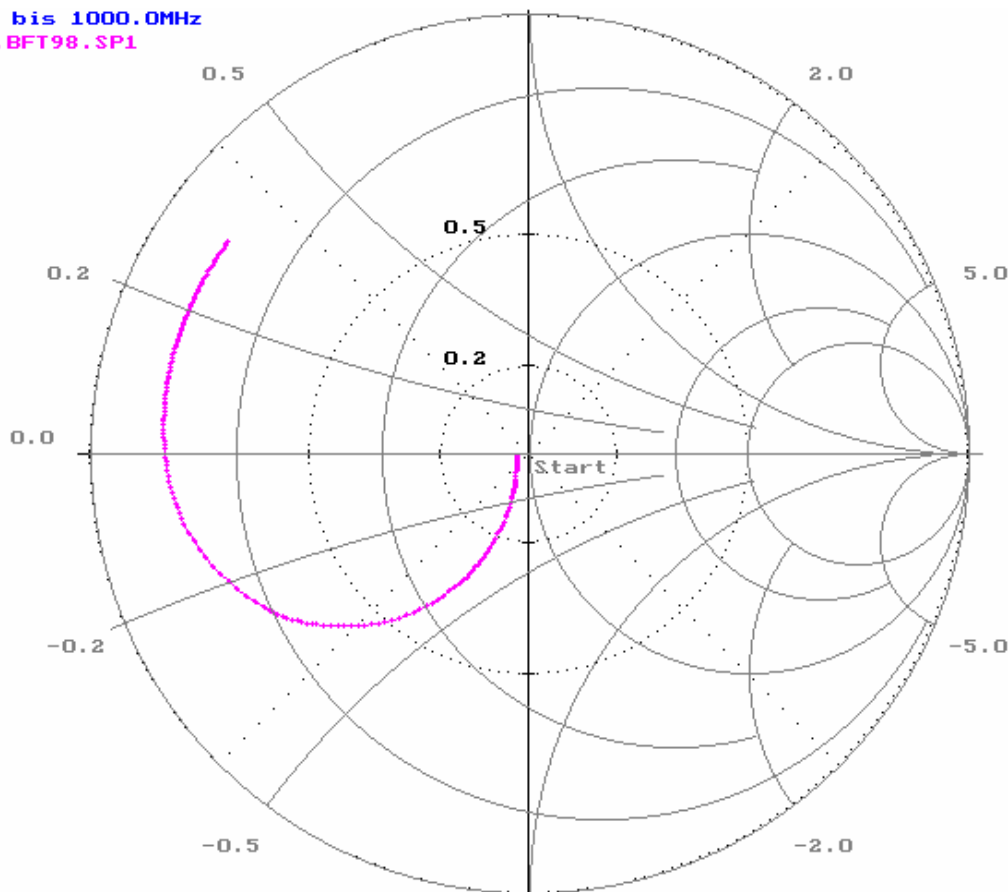
In the options menu you can define the tolerances for Monte-Carlo analysis and some DC parameters for transistors (if they are present in the circuit). Also the graphical output on the Smith diagram can be influenced, e.g. the connecting lines for the intermediate impedances.

## 2.2 Background Information to the Smith Diagram

In general, the reflection factor  $r$  is a complex number and for impedances with a positive resistance it usually has a magnitude between zero and one. If all passive impedances were to be displayed graphically, an infinite plane would be required in the direct case. For the reflection factor, though, a unit circle would suffice. This particularly elegant graphical representation was developed in 1939 by Phillip H. Smith and bears his name, the Smith diagram. The lines of constant value for the real and imaginary components of  $Z$  are displayed in the diagram so that the impedance is easily determined and no further calculations are necessary. With a bit of practice the Smith diagram can easily be used to design transformation networks with inductors, capacitors, transmission lines, etc.

### Example :

$S_{11}$  bis 1000.0MHz  
+ ..BFT98.SP1



The figure shows the the input reflection factor  $r=S_{11}$  of an RF transistor (BFT98) at two different currents. The starting point is at a low frequency (300kHz) and the end point at a high frequency (1 GHz). Such charts are often found in the spec sheets of semiconductor manufacturers. Here are a few guidelines for interpreting these diagrams:

### Orientation in a Smith diagram:

1. The matched point  $Z=Z_L$  (generally  $50\Omega$ ) lies exactly at the origin  $r=0$ . It is often the goal to reach this impedance by using matching networks.
2. The short circuit case  $Z=0$  lies at  $r=-1$  and the open circuit case  $Z=\infty$  at  $r=1$ .
3. All real impedances lie on the x-axis (real axis) including the case with real  $r$ .
4. Capacitive impedances  $Z=1/j\omega C$  lie under the real axis and all inductive impedances  $Z=j\omega L$  lie over the real axis.
5. All RC networks result in semicircles, which can overlap. All LC networks result in full circles since the impedance of a resonator (or a transmission line) can be inductive as well as capacitive.

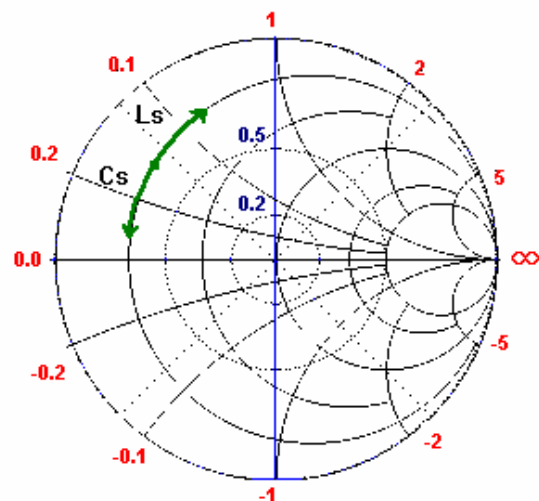
This information makes it simple to interpret transistor characteristics. The input impedance is real at low frequencies and around middle values ( $50\Omega$  in this case). The transistor capacitances  $C_{BE}$  and  $C_{BC}$  then have their impact as the curve moves down and toward a short circuit  $r=1$ . The inductances of the package come to bear at high frequencies and cause the curve to move up. It can also be seen that the base series resistance prevents reaching the short circuit point. Finally, it can be seen that the bias point of the lighter curve corresponds to the lower collector current, since this is the case with the higher impedance.

## 2.3 Matching Networks Examples

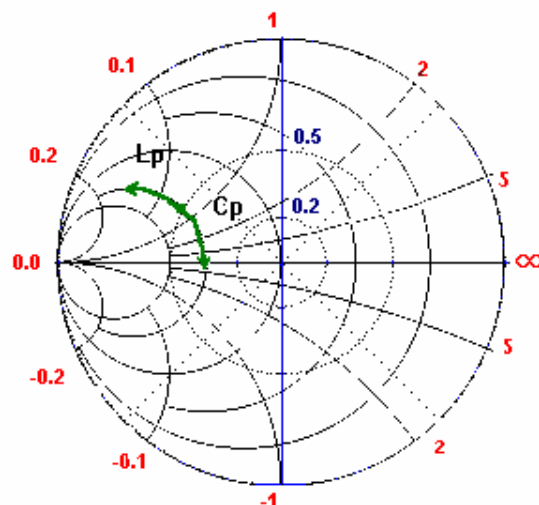
Matching networks can be dimensioned fairly straightforward, using a Smith Diagram. Impedances  $Z$  are entered in the diagram based on their reflection factor  $\Gamma$ , and the position makes it easy to determine whether the impedance is capacitive or inductive, and whether it is high or low relative to the characteristic impedance.

If an additional impedance is placed in parallel or in series with another, the total impedance is changed. This fact can be implemented for an impedance transformation. If only reactive elements are used, then the transformation is loss free, just as in the case of an ideal transformer. A transformation with capacitors, inductors and transmission line segments usually results in a narrowed band, however. The effects (and the calculation) can be seen quite nicely in a Smith diagram since all passive components (all those with  $\text{Re}(Z) > 0$ ) lie inside the unit circle.

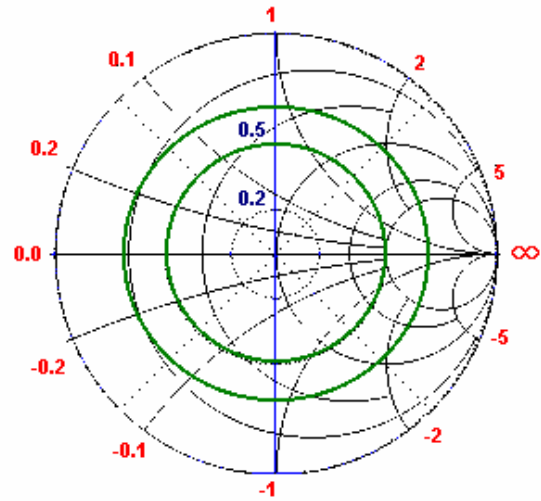
Transformation with series-L or C :



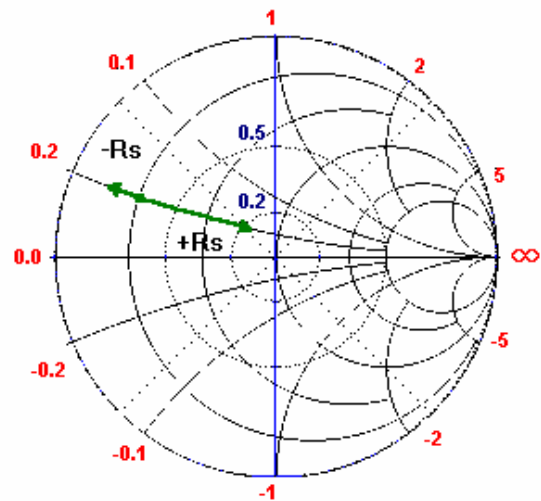
Transformation with parallel L or C :



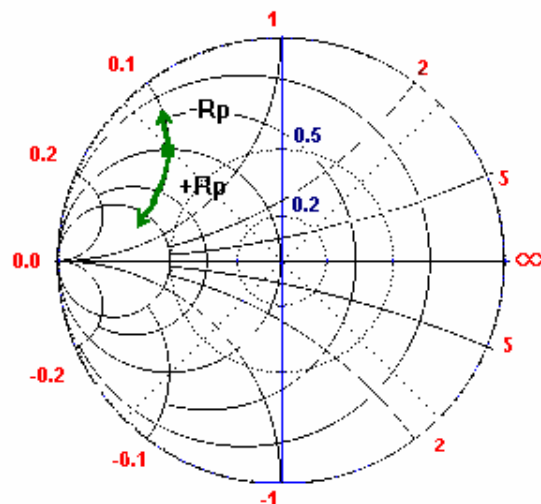
## 2 Transformations with transmission line ( $Z=Z_L$ ) :



## Transformation with series R :



## Transformation with parallel R :

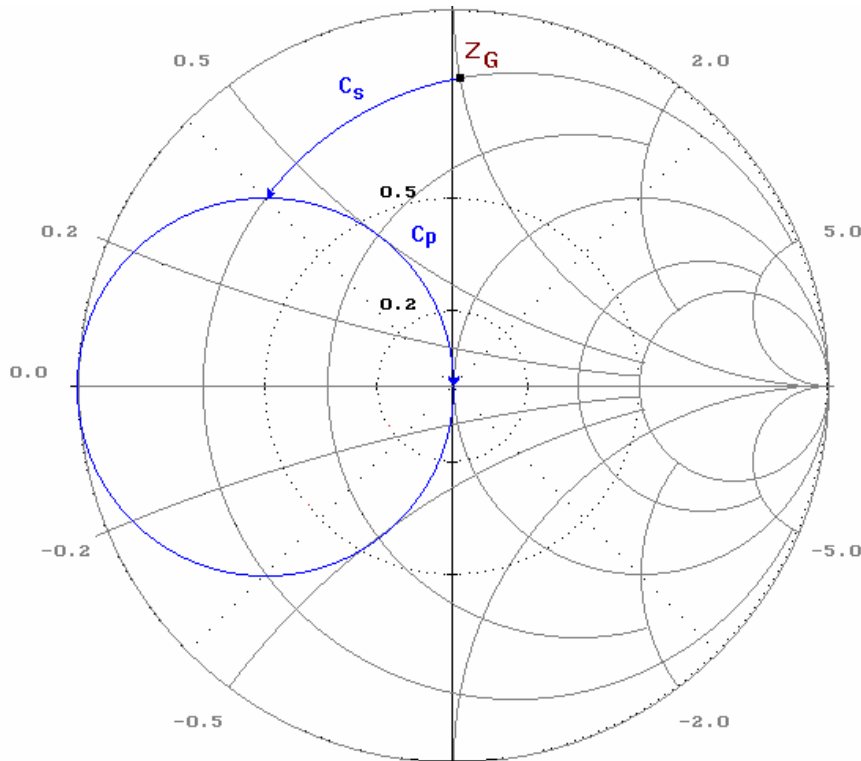


In order to transform onto the  $50\Omega$  point, it is necessary practically in every instance to choose a combination of a series and a parallel network. This is because it is not possible to change the resistive component of  $Z$  with a reactive series network and thus a parallel network is necessary.

### Example :

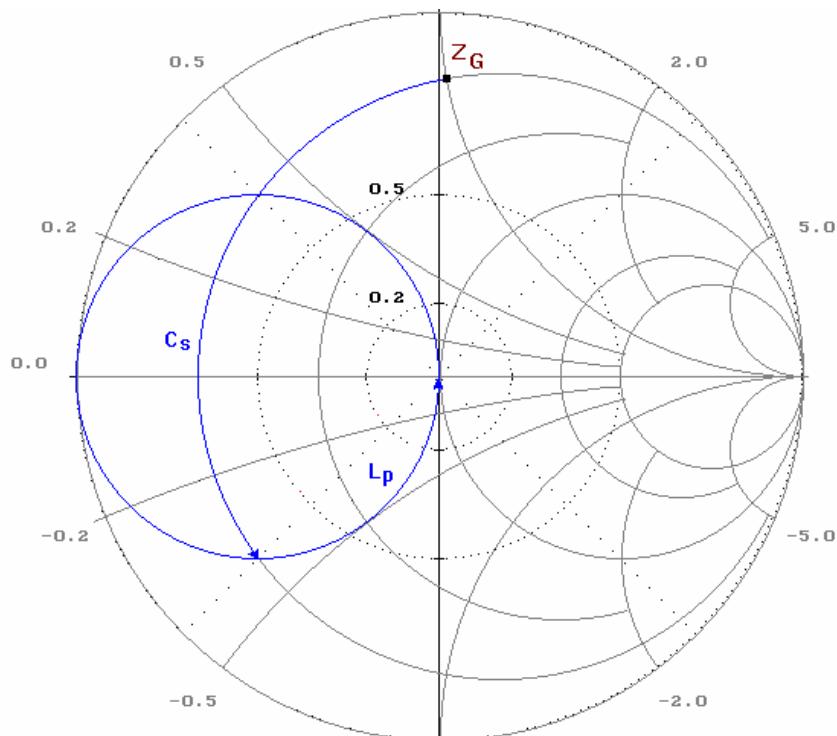
An inductive generator with  $Z_G=10\Omega+j50\Omega$  is to be matched to line impedance  $Z_L=50\Omega$  at  $f=1\text{GHz}$  in order to avoid reflections.

a) One possibility is to use a series capacitance  $C_s$  in conjunction with a parallel capacitance  $C_p$  as the matching network.



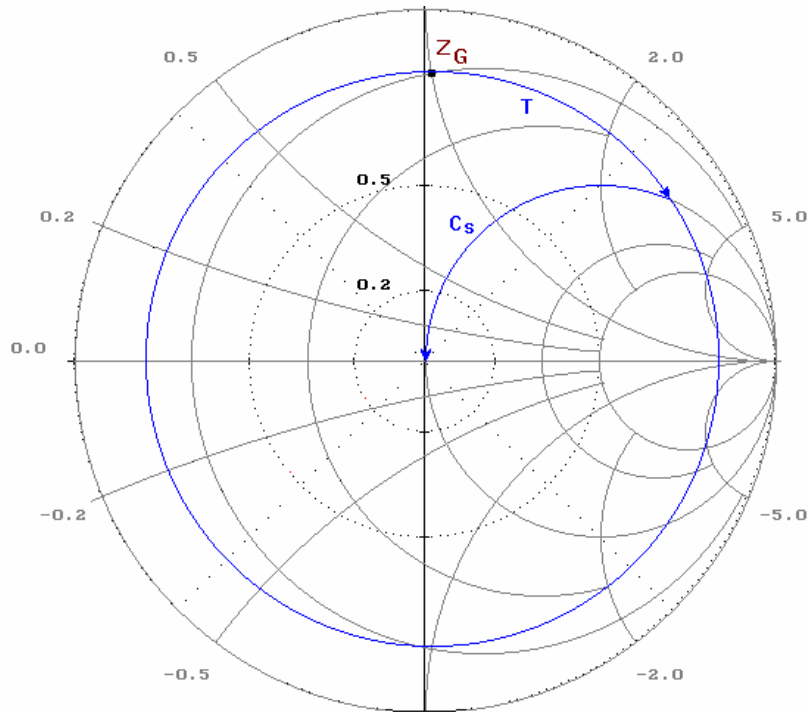
The  $X_C$  values normalized to the line impedance can be determined from the Smith diagram or calculated using CSmith.

b) A second possibility is to increase the impedance of the series capacitor until the total impedance is capacitive and then to include a parallel inductance  $L_p$ .



c) It is also possible to use transmission line segments, i.e. a series line T with  $Z_L=50\Omega$  together with a series capacitance. The reflection factor remains constant for a transmission line with the line

impedance. Thus one moves in a circle about the origin of the Smith diagram until the real value  $Y=1/Z=20\text{mS}=1/50\Omega$  is reached. The remaining imaginary component can that be compensated using a simple parallel capacitance.



There are many other possibilities, including using a  $\lambda/4$ -line or transformers.

If no power matching ( $r=0$ ) is possible when coupling two stages, the gain is reduced by reflections of the transmitted power. This decrease is greater the more the load impedance varies from the complex conjugate of the generator impedance.

**Loss due to mismatch :**  $L = -10 \cdot \log(1 - |r|^2)$   
 (Feldtkeller relation)  $= -10 \cdot \log(4 \cdot \text{VSWR} / (1 + \text{VSWR}^2)^2)$

where  $r = (Z_L - Z_G^*) / (Z_L + Z_G)$

r	Loss	t <sub>transmitted power</sub>	VSWR	Return Loss
0	0	100%	1	$-\infty$
0.05	0.01dB	99.75%	1.1	26dB
0.1	0.044dB	99%	1.22	20dB
0.2	0.177dB	96%	1.5	14dB
0.3333	0.5dB	88.89%	2	9.5dB
0.5	1.25dB	75%	3	6dB
0.9	7.2dB	19%	19	0.9dB
0.99	17	1.99%	200	0.09dB
1	$\infty$	0%	$\infty$	0dB

The reflection factor  $r$  therefore does not describe a real line impedance but rather the complex conjugate of the generator impedance  $Z_G^*$ . This relation is valid for when the stages are connected and does not consider other losses such as transmission resistance, etc.

## 2.4 Reflection Factor and SWR Table

If transmission lines are not terminated with impedance  $Z_L$ , then a portion of the signal is reflected at the end of the line.

$$r = (Z - Z_L) / (Z + Z_L) \quad (\text{Reflection factor})$$

and

$$\text{SWR} = V_{\max}/V_{\min}$$

$$= (|V_h|+|V_r|)/(|V_h|-|V_r|)$$

$$= (1+|r|)/(1-|r|) \quad (1 \leq \text{SWR} < \infty)$$

Examples :	Terminal impedance Z	Reflection factor r	SWR	Return loss
Short Circuit	0	-1	$\infty$	$\infty$
Open Circuit	$\infty$	+1	$\infty$	$\infty$
Matching	$Z_L$	0	1	0dB
-	$1.1 \cdot Z_L$	0.047	1.0986	26.55dB
-	$2 \cdot Z_L$	$1/3=0.333$	2	9.54dB
-	$3 \cdot Z_L$	$2/5=0.4$	2.333	7.96dB
-	$10 \cdot Z_L$	0.8182	10	1.74dB
-	$100 \cdot Z_L$	0.9801	99	0.17dB
-	$0.9 \cdot Z_L$	-0.052	1.11	25.68dB
-	$Z_L/2$	$-1/3=-0.333$	2	9.54dB
-	$Z_L/3$	$-2/5=-0.4$	2.333	7.96dB
-	$Z_L/10$	-0.8182	10	1.74dB
-	$Z_L/100$	-0.9801	99	0.17dB
-	$jZ_L$	j	$\infty$	$\infty$
-	$-jZ_L$	-j	$\infty$	$\infty$
Resistive Load	real	-1...1	0... $\infty$	0.. $\infty$
Reactive Load	imaginary	$ r =1$		0dB
Capacitive Load	$1/j\omega C$	$ r =1, \text{Im}(r)>0$	$\infty$	$\infty$
Inductive Load	$j\omega L$	$ r =1, \text{Im}(r)<0$	$\infty$	$\infty$
Passive Load	$\text{Re}(Z)>0$	$ r <1$	-	-
Active Load	$\text{Re}(Z)<0$	$ r >1$	-	-

### 3.1 Case Study Broadband Amplifier

To present the usage of the Smith chart tool CSERPAP let us design a Si bipolar single stage 5GHz IC broadband amplifier. Normally a Smith chart is used for the design of narrow band matching networks, but CSERPAP offers you much more functionality. For a narrow band LNA it's often possible to get a gain of about 15dB at 2GHz. But for a broadband amplifier (especially at 5GHz) this is not realistic because you want to suppress resonances. For narrow band amplifiers the maximum stable gain MSG of the transistor gives a good orientation (e.g. 18dB for the BFP520 at  $I_C=2mA$  and  $f=2GHz$ ), but for broadband amplifiers the magnitude of  $|S_{21}|$  (about 15dB for the BFP520) is a better performance indicator. So we can define the goal parameters for our study:

- Gain : 12dB
- Frequency Range : 1MHz to 5GHz (-3dB)
- Return Loss : >10dB

As a transistor we may choose a well-known RF transistor like the BFP520, but as an IC designer it is more interesting to evaluate which transistor parameters (transition frequency  $f_T$ , feedback capacitance  $C_{BC}$ , transconductance  $G_m$ , etc.) are really needed. As a starting point we can take a look at a transistor with  $f_T=25GHz$ ,  $G_m=160mS$  (this equals  $I_C=4.5mA$  with no series feedback),  $f_{max}=30GHz$  and  $C_{BC}=0.2pF$ . If you add a transistor in CSERPAP with these parameters, you can see that the transistor is not stable at 5GHz ( $k=0.44$ ) and that its MSG is about 14.1dB.

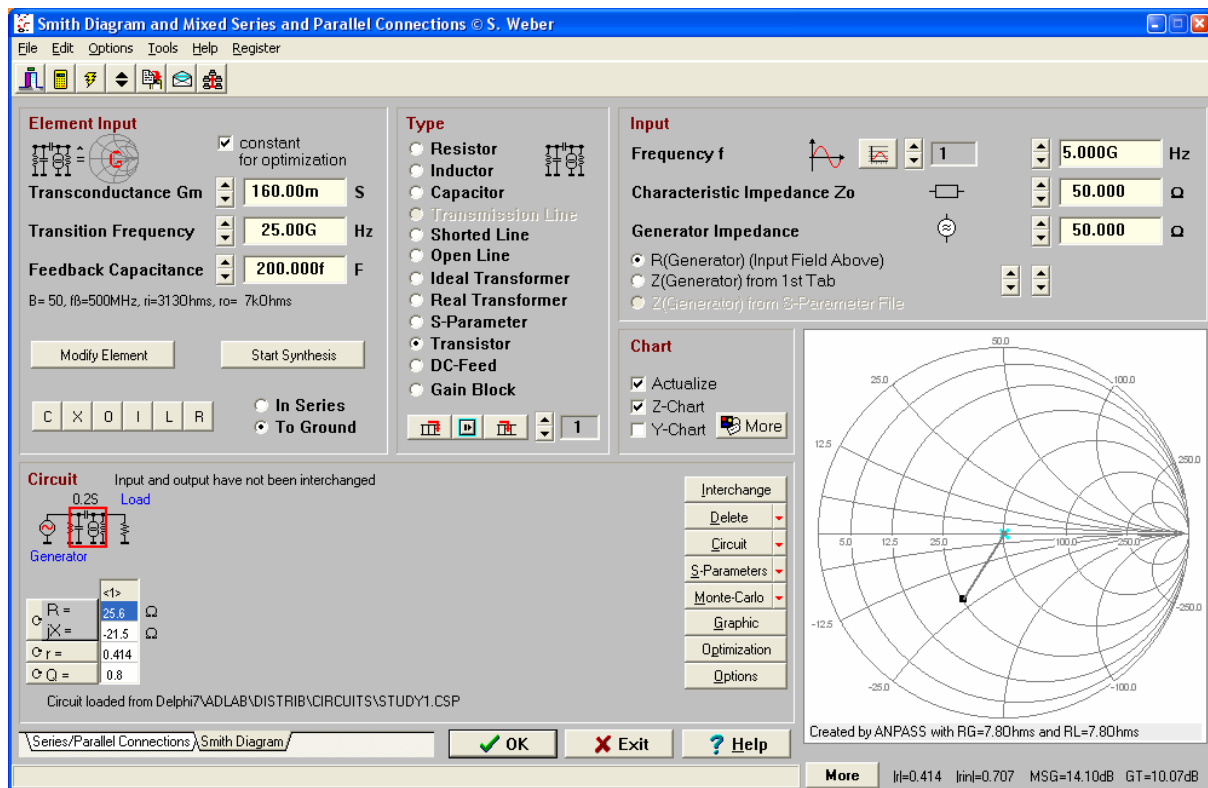


Figure 1 : CSERPAP with the new defined transistor alone

The impedances are pretty high at both the input and the output (at DC), so it's a good idea to use a series LR element as a shunt or to optimize the frequency response (and tolerance behavior) by using shunt feedback. A series feedback (with parallel capacitor) is in this respect not well suited (at least in this operating point). Because the MSG is large enough the transistor is (at least in principle) suited for this circuit and you can go on with the circuit design. Concerning the elements available in Si IC technology we have to mention that only low-Q inductors ( $Q_{max}$  about 3..10, due to high substrate parasitics) are available. Therefore it's better to use only a resistive shunt feedback (and not a floating inductor, due to high impedance level). At low impedance nodes or at the input or output series inductances (e.g. bond wires) can be used without drawbacks.

Do to feedback inside and outside the transistor there is a interaction between output and input. Another reason why the circuit design is not easy, is that the MSG is large enough, but the gain without matching is too low  $G_T=S_{21}=10.07\text{dB}$ , which is 2dB less than our goal. With a shunt feedback using  $R_{fb}=1\text{k}\Omega$  (press Modify Element button) the DC gain will decrease from 22.6dB to 19.6dB and there is also an improvement in return loss and k-factor. For luck, the gain decrease at 5GHz is much smaller (about -0.4dB, with an ideal floating inductor even better values are achievable). If you take a look at the frequency response, you will see that the -3dB corner frequency is at 1.75GHz.

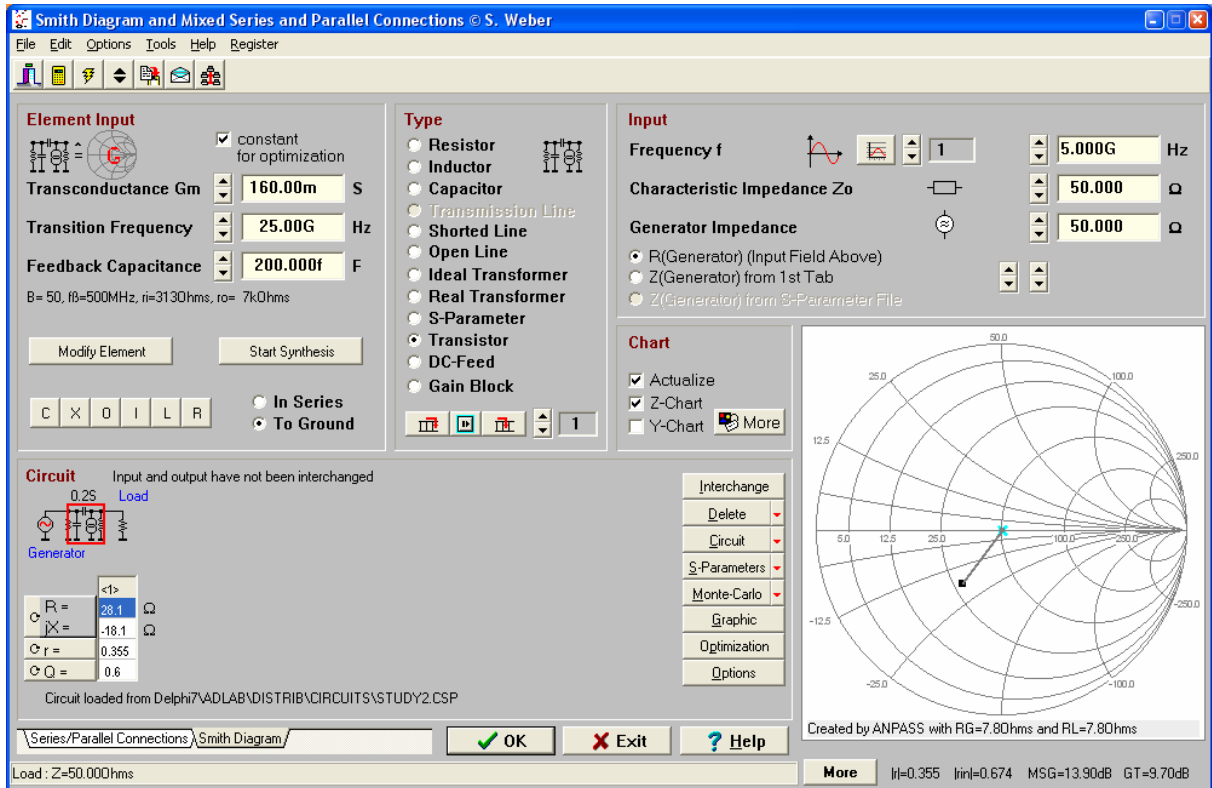


Figure 2 : CSERPAR with the transistor including feedback

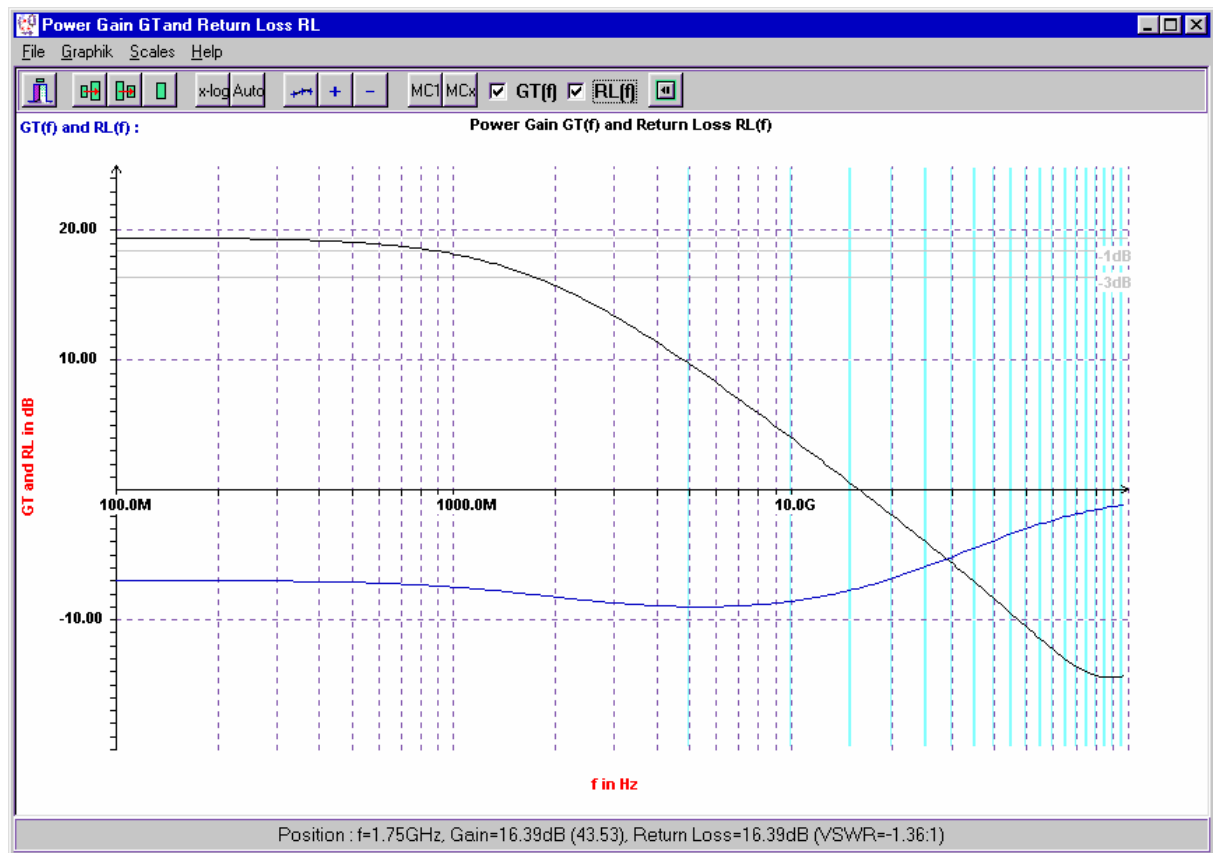


Figure 3 : Graphical output of the frequency response

By shunting the input and output with series RL elements you can now try to improve stability, matching and frequency response of the power gain. If you take a look at the return loss you will see that there is some margin for doing this. By iterative element values optimization by hand (use the cursor keys or the spin buttons) at both low frequencies and at about 3 to 5GHz you can get a better performance. Unlike narrow band amplifiers its not possible to state that the design process will result in a full-spec broadband amplifier. For instance in this case we have to accept some minor deviations at the input return loss and for the upper cutoff frequency. But the frequency response is quite smooth which will result in good time domain characteristics (like small overshoot).

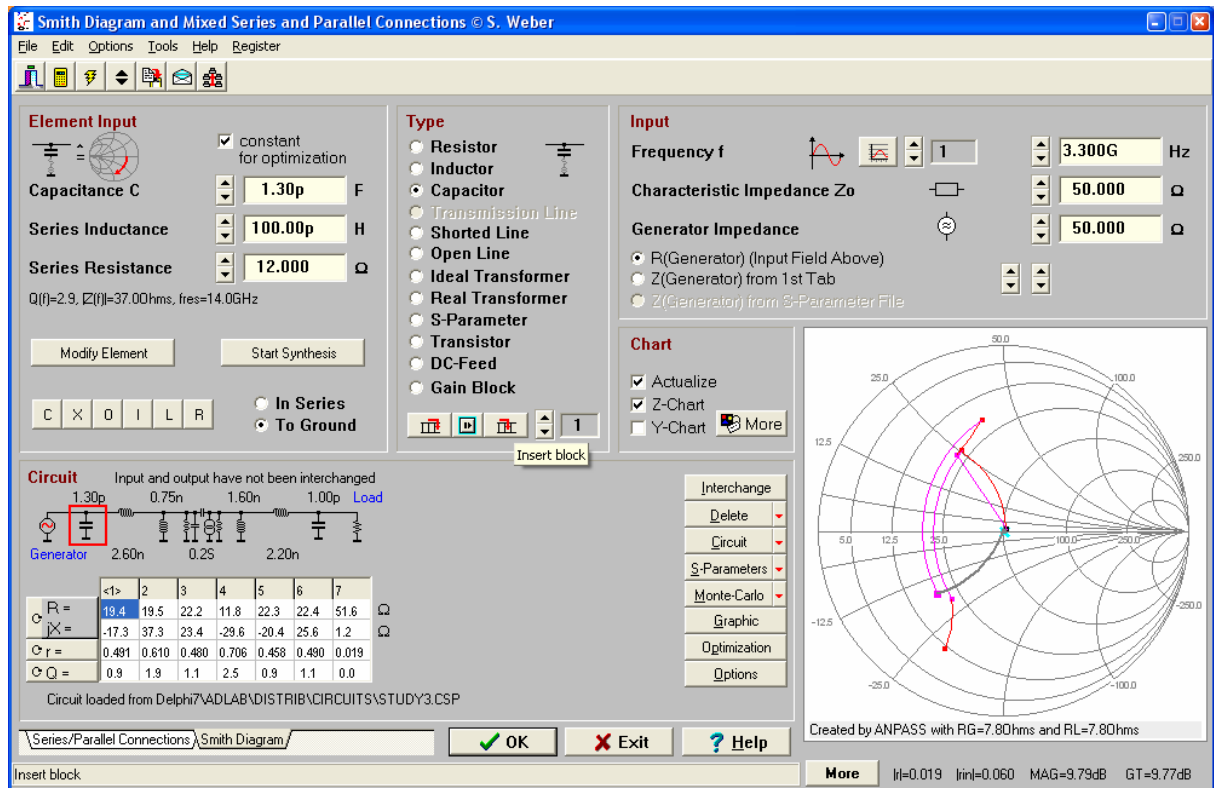


Figure 4 : The total circuit

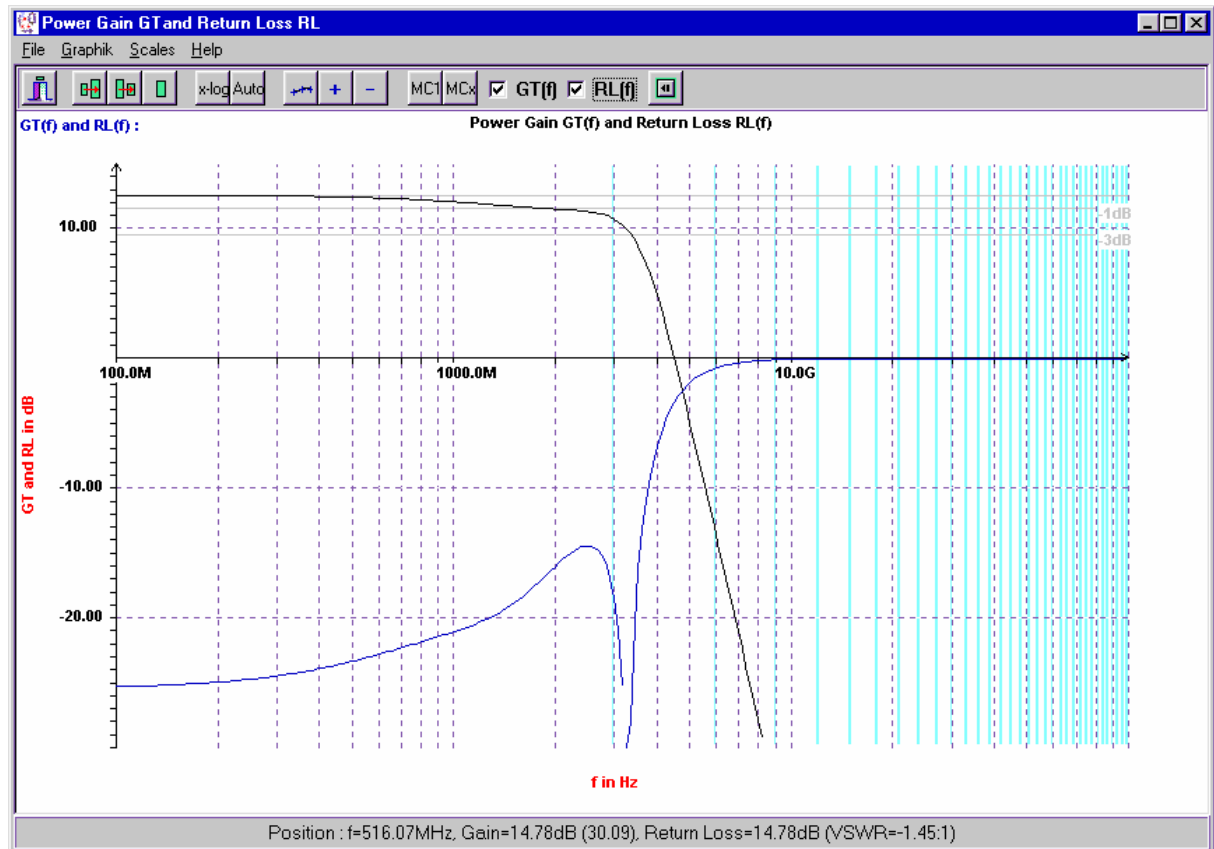


Figure 5 : Graphical output of the frequency response

Some ways to improve the circuit are to modify the operating point or the feedback, to use a better transistor or to apply numerical optimization. Also the influence of the emitter inductance is interesting.

For differential amplifiers (often used in measurement equipment) the emitter is on virtual ground, i.e. the effective ground inductance is zero. But for single ended amplifier this is completely different, even 100pH will have an impact on frequency response.

The optimization is perhaps the most flexible way to improve the circuit. This can be done very easily in Smith chart program : At first you have to decide (look at the checkbox in the input field) which elements should be optimized and which should be fixed. Then press the Optimization button. A new window appears and you have to decide on the frequency range and optimization goals (like minimum return loss or k-factor). Then press the Optimization button and you will get an Aplac file which specifies the whole circuit and the complete simulation and optimization. Using Aplac we can achieve nearly all our design goals without using a faster transistor. But now around 3.5GHz the k-factor slightly smaller than one.

```
AplacVar C1=0.69 OPT Min=0.1 Max=2
AplacVar C7=0.76 OPT Min=0.1 Max=2
AplacVar L2=1.6 OPT Min=0.1 Max=4
AplacVar L3=5.6 OPT Min=1 Max=10
AplacVar L5=5.7 OPT Min=1 Max=15
AplacVar L6=1.2 OPT Min=0.1 Max=4
```

```
AplacVar Cbe=0.82 $ OPT Min=0.2 Max=2
AplacVar Le=0.01 $ OPT Min=0.001 Max=1
AplacVar Rpgk=410 OPT Min=300 Max=3000
AplacVar Gm=0.16 $ OPT Min=0.1 Max=0.2
```

```
$ Charecteristic impedance Zo=50.00Ohm
```

```
Cap C1 1 0 1p C=C1 L=0.10n R=12.00
```

```
Ind L2 1 2 1n L=L2 R=0.10 C=10.00f
```

```
Ind L3 2 0 1n L=L3 R=95.00 C=10.00f
```

```
DefModel Transistor4
```

```
+2 1 2
```

```
+PARAM 4 Cbe Rpgk Gm Le
```

```
Default Cbe=0.82
```

```
Default Rpgk=1k
```

```
Default Gm=160m
```

```
Default Le=0.001
```

```
Cap Zpgk 1 2 10U R=Rpgk L=1.000P
```

```
Ind Zsgk 0 E 1.000N G=1.000K C=1.000F
```

```
Res Rx 1 B 5.500 L=10.000P
```

```
Res Rc Ci 2 1.000M L=10.000P
```

```
Res Re E Ei 1.000M L=Le*1N
```

```
Cap Cr B Ci 0.200P
```

```
Cap Cin B Ei 1p C=Cbe G=3m
```

```
Res Rout Ci Ei 7.2K
```

```
Gm Vccs Ci Ei B Ei Gm
```

```
DefEnd
```

```
Transistor4 Q4 2 3 Cbe=Cbe Rpgk=Rpgk Gm=Gm Le=Le
```

```
Ind L5 3 0 1n L=L5 R=95.00 C=10.00f
```

```
Ind L6 3 4 1n L=L6 R=0.10 C=10.00f
```

```
Cap C7 4 0 1p C=C7 L=0.10n R=0.10
```

```
DefNPort Nport 2
```

```
+ 1 0 (50.00,0) 4 0 (50.00,0)
```

```
Sweep "S-Parameters"
```

```
+ LOOP 201 FREQ LIN 100.00M 4G
```

```
+ Window 0 smith
```

```
+ Window 1 polar autoscale
```

```
+ Window 2 autoscale grid
```

```
Show Window 0 RI S(1,1)
Show Window 0 RI S(2,2)
Show Window 1 RI S(2,1)
Show Window 1 RI S(1,2)
Show Window 2 Y MagdB(S(2,1))
Show Window 2 Y MagdB(S(1,2))
EndSweep
```

```
OptimMethod=Gradient OPT_FTOL=0.0001u OPT_XTOL=0.0001m
Sweep "Gradient Optimization"
+ LOOP 51 FREQ LIN 0.1g 5g
  Goal MagdB(S(2,1)) EQ 12 Weight=2
  Goal MagdB(S(2,2)) LT -10

  Goal MagdB(S(1,1)) LT -10
EndSweep
```

```
$ Print optimization variables
Print OPTIMIZE
```

```
Sweep "S-Parameters optimized"
+ LOOP 201 FREQ LIN 100.00M 5G
+ Window 0 smith
+ Window 1 polar autoscale
+ Window 2 autoscale grid
+ Window 3 autoscale grid
+ Window 4 autoscale grid
+ Window 5 autoscale grid
Show Window 0 RI S(1,1)
Show Window 0 RI S(2,2)
Show Window 1 RI S(2,1)
Show Window 1 RI S(1,2)
Show Window 2 Y MagdB(S(2,1))
Show Window 2 Y MagdB(S(1,2))
Show Window 3 Y MagdB(S(1,1))
Show Window 3 Y MagdB(S(2,2))
Show Window 4 Y MagdB(S(2,1))
Show Window 5 Y S_K
EndSweep
```

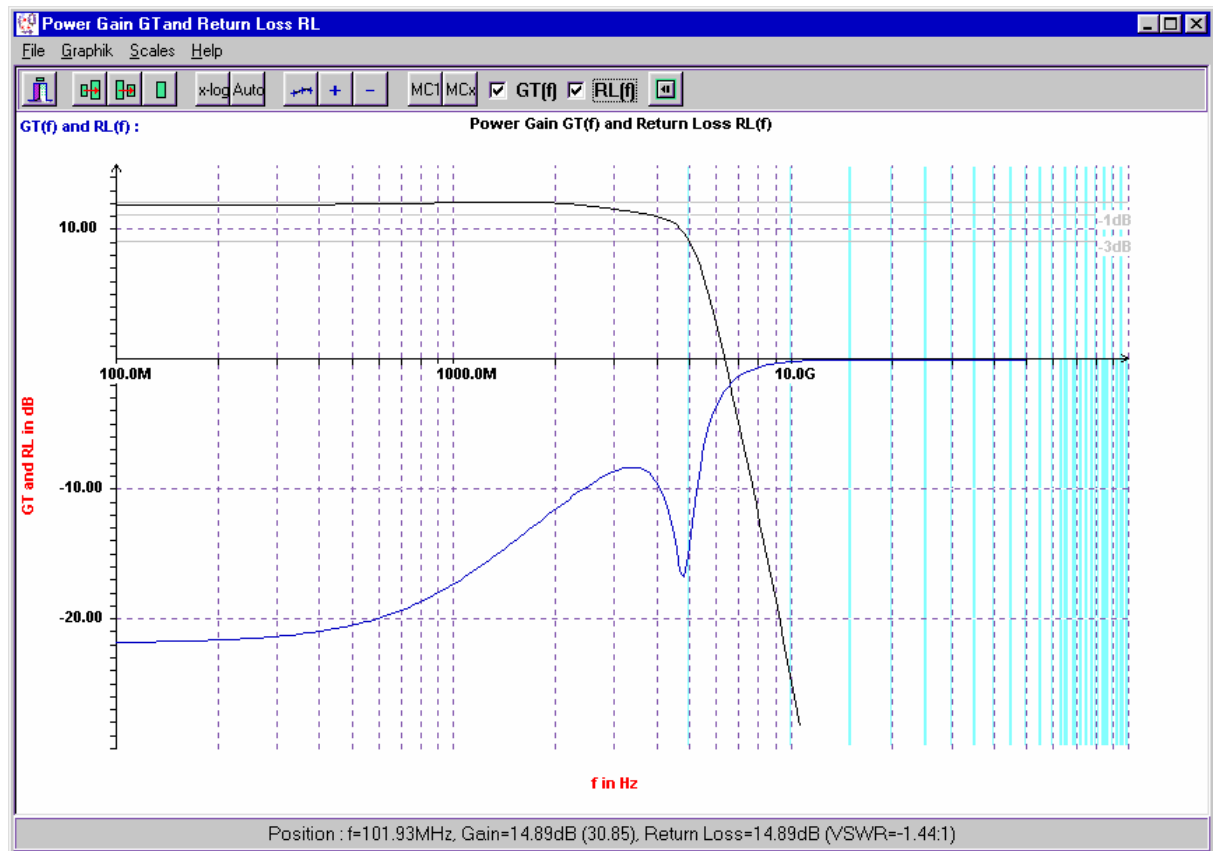


Figure 6 : Frequency response after Aplac optimization

With nonlinear simulators like Aplac or PSpice you can also design the bias circuitry. Even simpler is the use of another ELEKTA tool called TRBIAS, because it presents and calculates different circuit topologies. You can also use the more powerful TRANSISTOR from AdLabPlus.

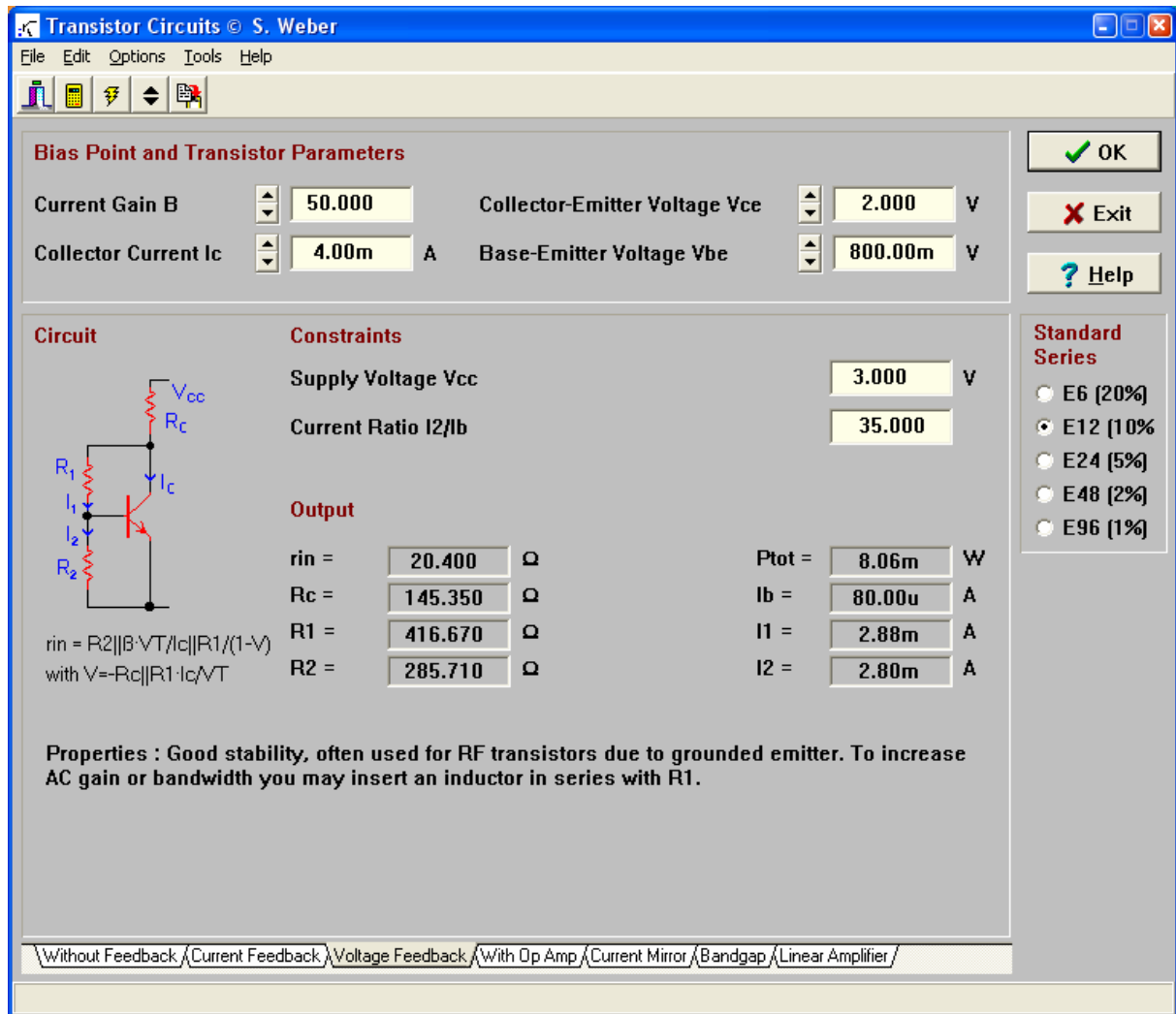


Figure 7 : Simple bias circuit for a broadband amplifier

If you want to start a design based on S-parameter (not on a transistor equivalent circuit model), then you can evaluate the S-parameters using VIERPOL. This program will show you, e.g. important parameters like stability factor, maximum available and stable gain, the gain circles and much more. For LNA designs you should take a look at NOISE.

## 3.2 Case Study Universal RF Matching Network

An universal RF matching network is described and as an example it is designed for the 70cm amateur band (around 440MHz). It is easy to use and can be realized using standard elements. A lowpass structure is chosen which results in good rejection of harmonics. A useful effect in practice. DC signals will be transferred 1:1, if this is not needed a series blocking capacitor can be added. To achieve really every impedance three trimming elements are used. Due to better availability trimming capacitors are chosen. Figure 1 shows the total circuit. It is an elliptic 3rd order lowpass filter.

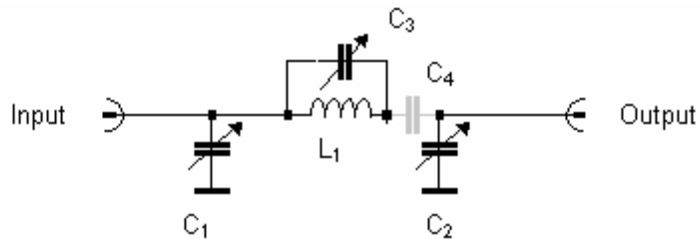


Figure 1 : The circuit (C<sub>4</sub> is optional)

### Design

The circuit is very similar to the equivalent circuit of a transmission line. And  $\lambda/4$  lines are well known matching networks. If you calculate the element values of a  $\lambda/4$  line in the used frequency band you already get good starting values for the described circuit. For the 70cm band you get the following values (see figure 2) :

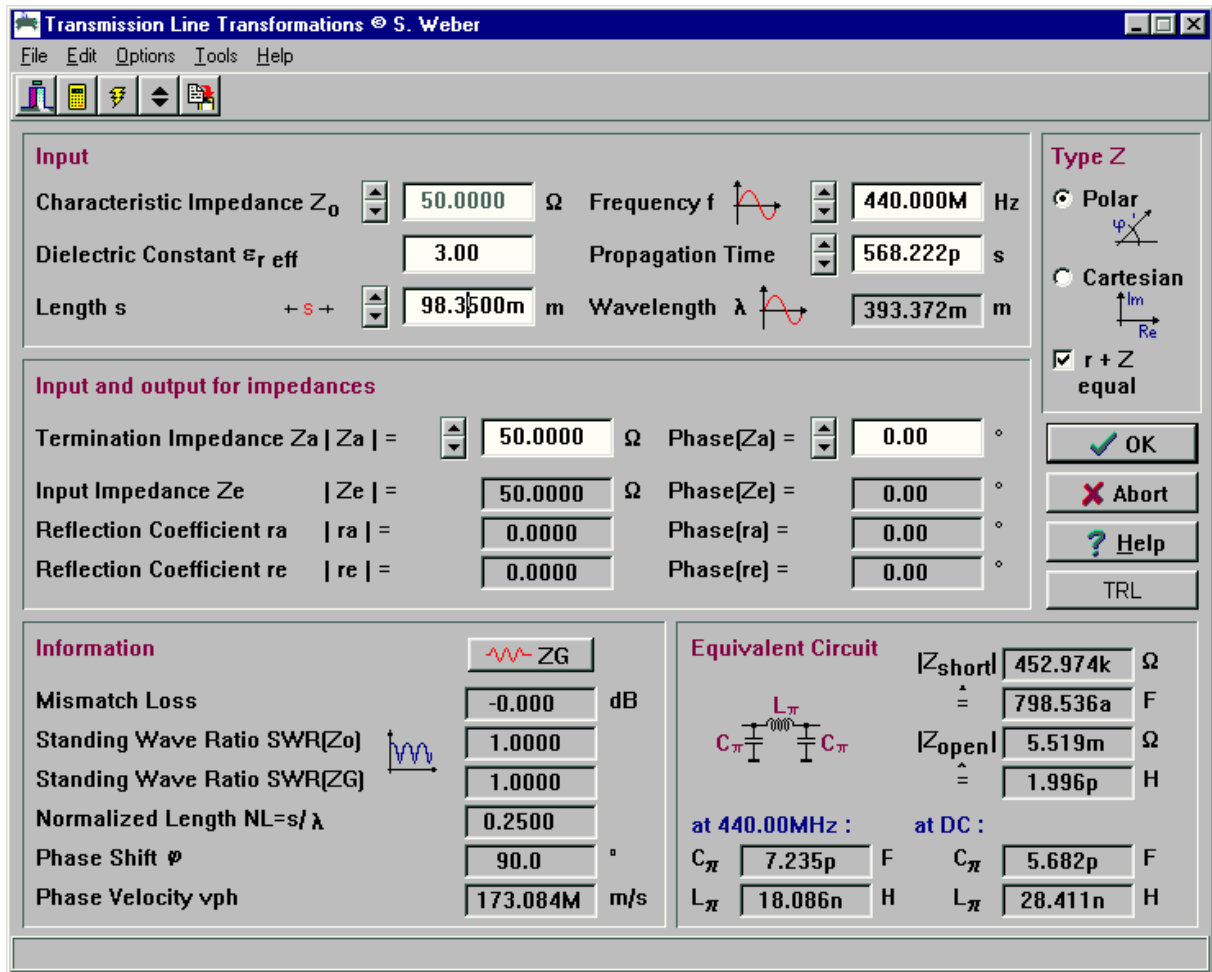


Figure 2 : Equivalent circuit of a 50 $\Omega$   $\lambda/4$  line at 440MHz (calculation with LTRANS :  $L_1=18.08nH$ ,  $C_1=C_2=7.235pF$ ) Transmissionline transformations

The trimmer  $C_1$  and  $C_2$  should cover the calculated values (7.2pF) (maximum value about 3 times larger).  $L_1$  should be selected two times smaller (e.g. 8.2nH instead of 18nH), because the parallel circuit with  $C_3$  gives a larger effective inductance (figure 3).  $C_3$  may be as large as  $C_1$  or  $C_2$ .

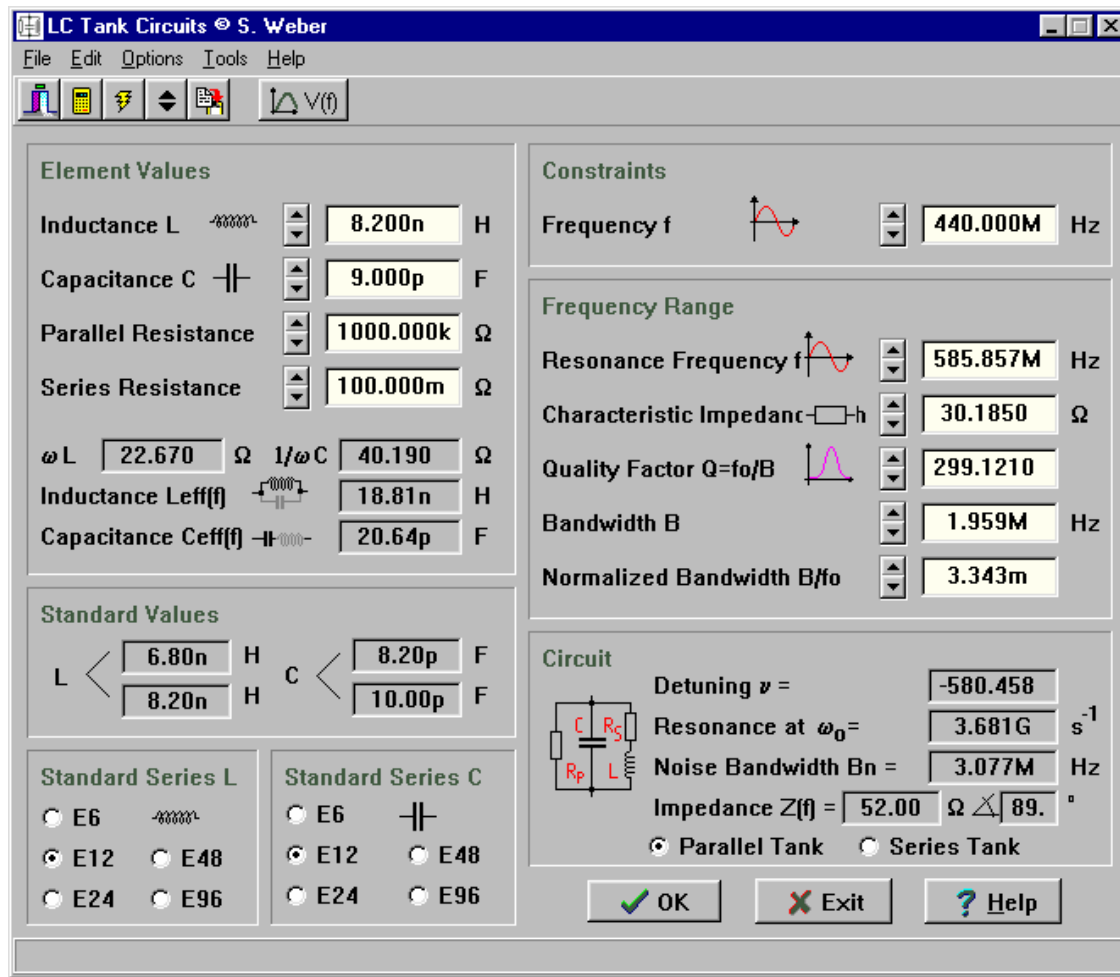


Figure 3 : The effective inductivance of the parallel LC circuit (calculation with LCFILT from ELEKTA Professional : 8.2nH || 9pF results in 18.8nH at 440MHz). You can also use the more powerful RLCFILT from AdLapPlus)

$C_4$  should be at least  $5 \cdot C_1$ . To move to another band, e.g. the 2m band or FM radio you may follow the same steps or you may simple scale all elements. For 100MHz FM radio multiply all capacitors and inductors by 4.4 (440MHz:100MHz).

### Trimming

You may think the circuit is not easy to adjust with its 3 trimming elements. But you can get a good feeling using the Smith chart. If all trimmers are near the center position you will see that the circuit behaves like a  $\lambda/4$  transmission line. A 50Ω source will result in an output impedance of 50Ω. This position is a good starting point. (figure 4).

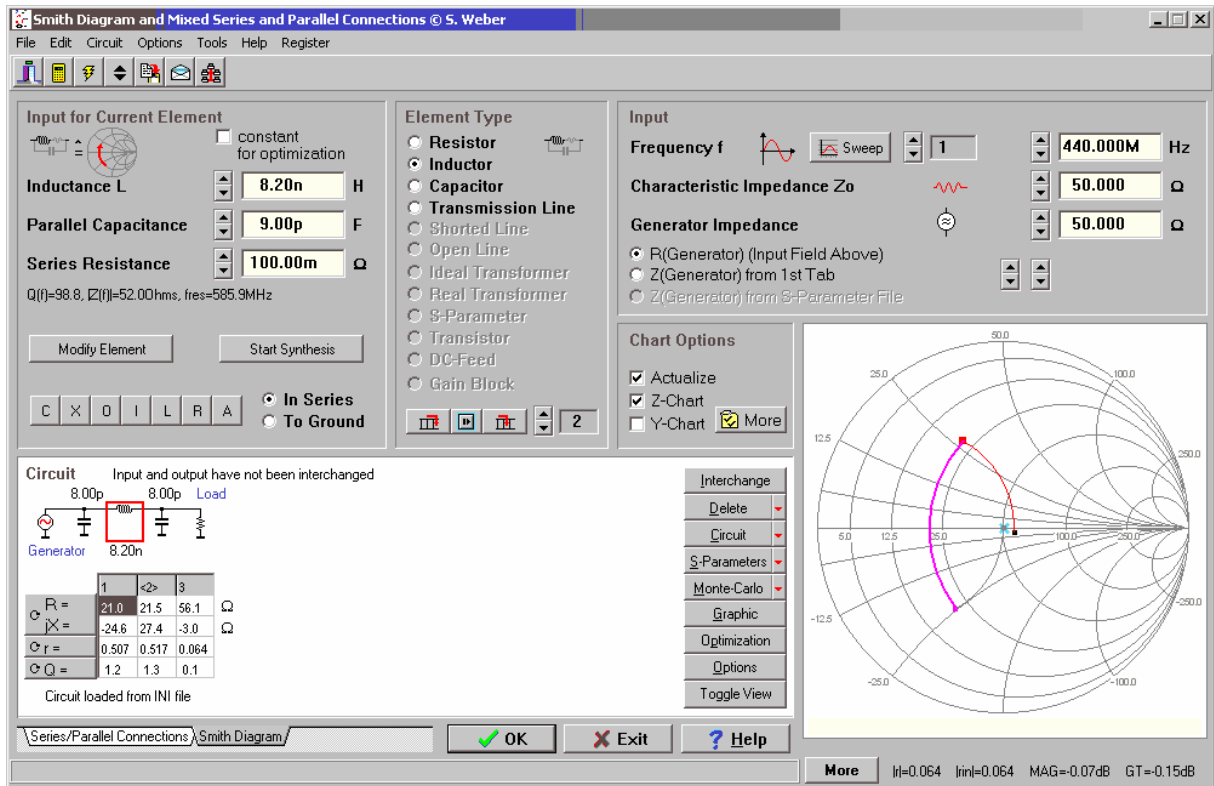


Figure 4 : Neutral position of the circuit (calculated by CSmith)

### Matching network design

But the frequency response is not periodic as for the transmission line. It shows lowpass characteristic (figure 5). The 2nd harmonic will be attenuated by 14dB.

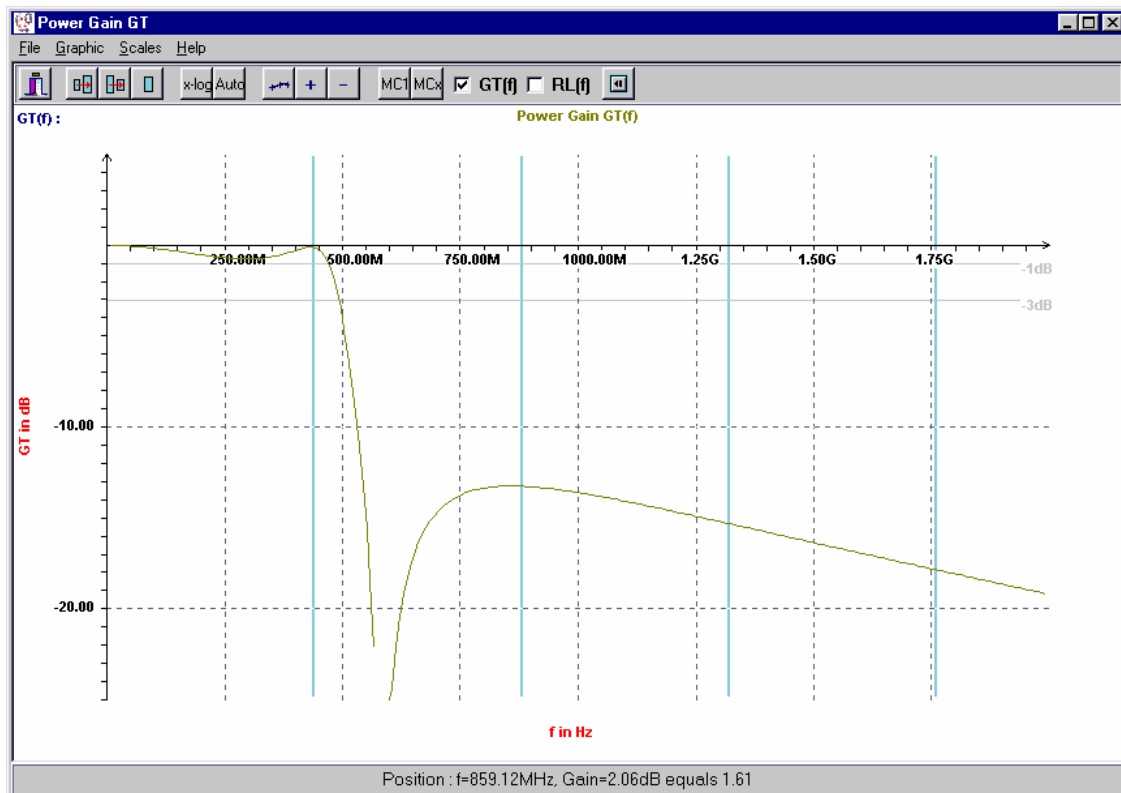


Figure 5 : Frequency response in the neutral position (CSmith)

In the Smith chart program you may easily change the circuit values. You will see that  $C_3$  has a strong influence on the real part of the impedance and  $C_1$  and  $C_2$  have a strong impact on the imaginary part. You should start trimming with the trimmer having the strongest impact. If you already know your circuit impedances (e.g. because you already measured them with a network analyser) you may first adjust the trimmers in the Smith chart and make only fine tuning in the lab.

The circuit bandwidth in figure 5 is pretty large, but if you use more extreme trimmer positions or circuit impedances it will be lowered. Figure 6 shows the matching of a  $8\Omega$  generator to a  $50\Omega$  load (VSWR 6.25:1). A small remaining mismatch of 1.45:1 must be tolerated because the trimmers don't achieve  $0pF$ . But the damping is only  $0.5dB$  (instead of  $3.3dB$  without matching network). The  $-1dB$  bandwidth is now about  $200MHz$  and the harmonic rejection is very good especially above  $1.5GHz$ .

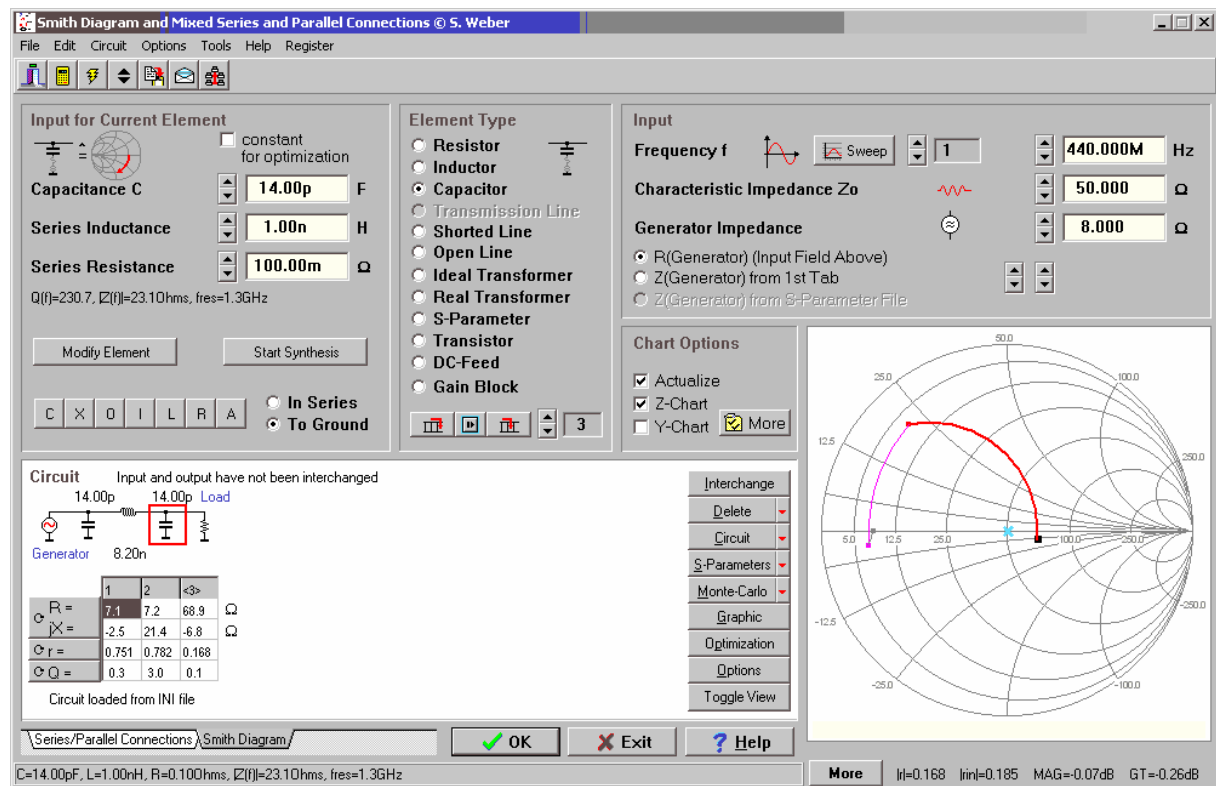
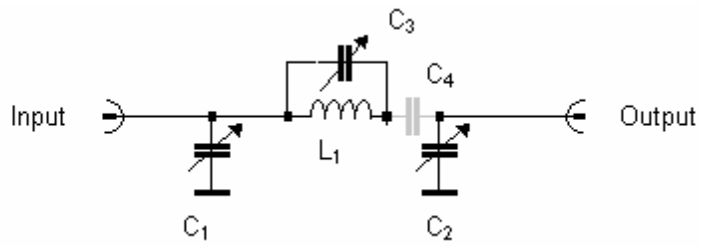


Figure 6 : Circuit trimmed for matching from  $8\Omega$  to  $50\Omega$  (CSmith)

For many impedances many trimmer combinations are valid to achieve the matching condition (at the fundamental frequency). But these combinations might differ in bandwidth, harmonic rejection or amplifier efficiency. If  $C_3$  is large a steeper cutoff will result due to the stop band zero of the parallel LC tank. Small capacitor values will result in a more flat frequency response and also the circuit will be less sensitive to tolerances.

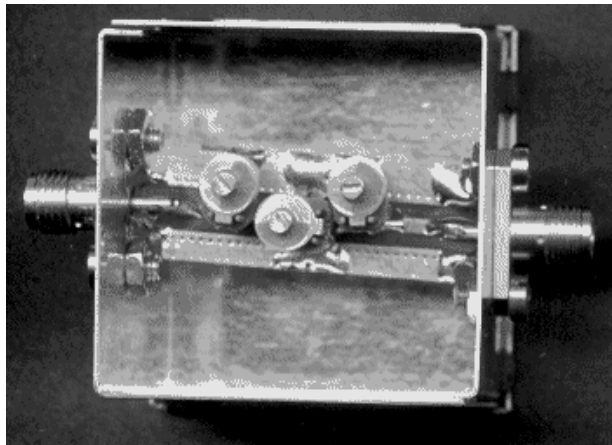
## Application

Very often matching networks are used for RF amplifiers or antennas. The first application was to achieve the matching between a small 70cm mobile and a RF booster. Without matching the mobile cannot drive the amplifier strong enough, so the output power was low and sometimes even the automatic RX - TX switch (RF VOX) doesn't work (to low drive power). The matching box was simply connected between the mobile and the booster and the trimmers are adjusted for maximum antenna power. At extreme trimmer positions the matching might be very bad and you have to be careful because many RF amplifiers might be destroyed in the presence of a large VSWR.



$L_1 = 8.2\text{nH}$  (air coil : 3 turns, 0.5mm cooper wire, inner diameter 2.5mm)  
 $C_1=C_2=C_3=1..22\text{pF}$   
 $C_4=470\text{pF}$

Figure 7 : Matching network with the values for the 70cm amateur band



### 3.3 Case Study Match for RF Power Amplifier

The S-parameters are only a good characterization for small signal circuits. Power amplifiers are very nonlinear and the S-parameters will depend on power level. Despite this S-parameters measured at the input port at the power level also used in the application are a very good starting point for the design of the input matching network. Even more critical is the output of an RF power amplifier. Power match based on small-signal S-parameters will result in highest small-signal power gain but for RF power amplifiers the output power itself and the power efficiency are much more important. So the question is: What impedance should be applied at the amplifier output to get a given output power with best efficiency? Many people are using an impedance tuner to search for the best match by hand. This will lead to a complete different design procedure then typical used in small-signal amplifiers! Also here with some theory a faster way is possible.

Let us consider a concrete design problem:

Design a matching network for a DECT power amplifier. The 1.9GHz digital European cordless phone standard allows 24dBm (250mW) at the antenna. In reality some loss occurs in the TX filter and antenna switch, so the PA should deliver app. 26dBm. When we want to use a balanced PA (with push-pull output stage) like the Infineon PMB6819 (low cost Silicon device with integrated system functions), we need a good match and a BALUN.

The output power depends also on supply voltage (due to  $P=V_{rms}^2/R_L$ ) and best efficiency (app. 45%) can be expected when the PA is deep in the compression. This is no problem because DECT uses GFSK modulation with constant envelope. The datasheet recommends a supply voltage of 2.4V (2 NiCd cells), so we can calculate the load impedance  $R_L$  using the AdLab tool ANPASS (assume that the transistor saturation voltage is app. 0.2V).

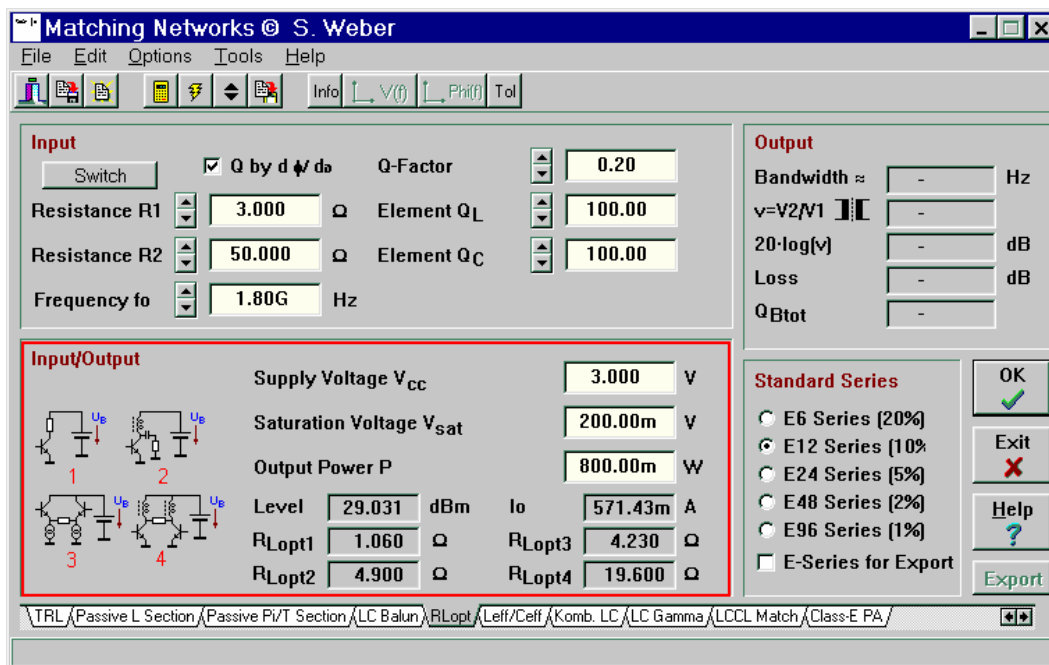


Figure 1 : Calculating R<sub>Lopt</sub> via ANPASS

The result is a reel value for the impedance (20 $\Omega$ , so 10 $\Omega$  for each side) which is not truly realistic with real world transistors and finite package inductances. So ANPASS delivers the correct value for an idealized PA in class-A operation (hints available on bubble help). For class-B operation used in the PMB6819 a higher value of R<sub>Lopt</sub> is a bit better for higher efficiency (say 11 $\Omega$ ). Using CSERPAR we can start with the corrected value as the generator impedance and we can add the transistor output capacitance (app. 3pF) and the pin inductance (app. 0.8nH and a small package capacitance) by hand.

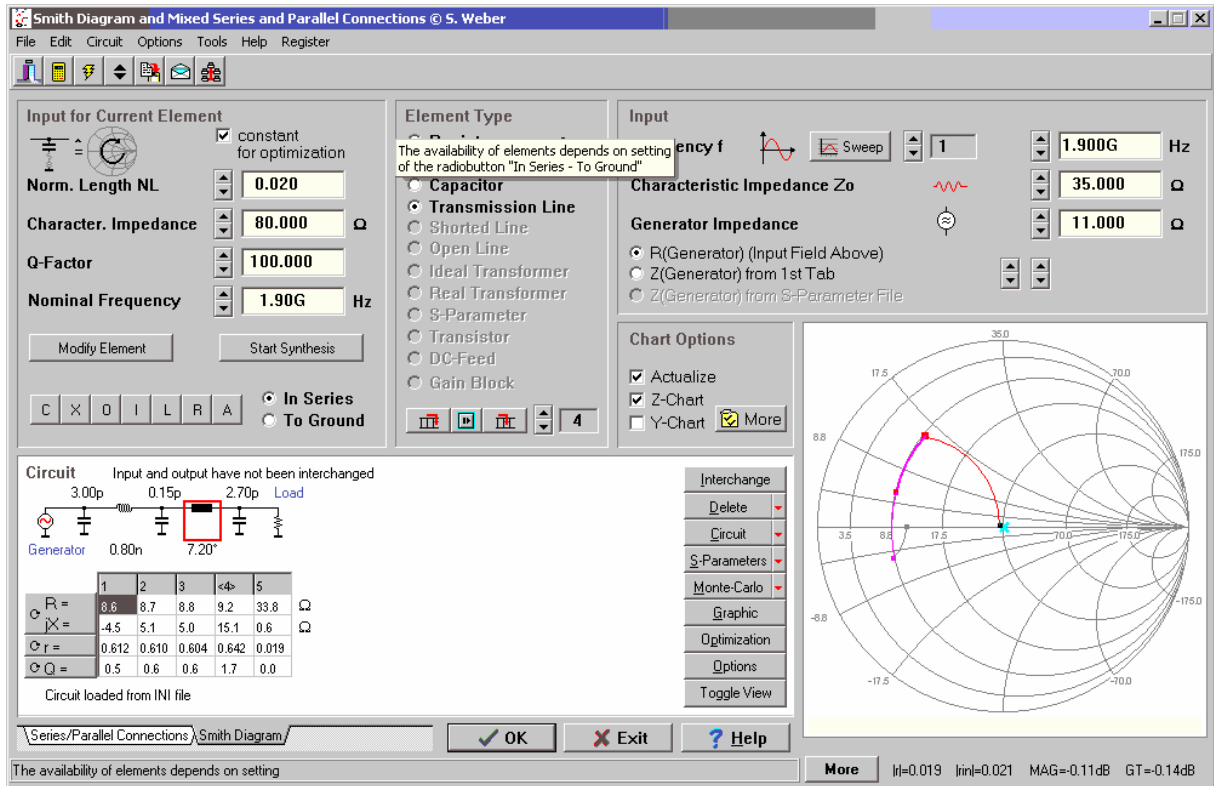


Figure 2 : PA modeling in CSERP and the matching network to 35Ω. Note: The end capacitor has a series inductances of 0.7nH as a typical SMD component.

What we need now is a match from the transistor output to the BALUN. A BALUN transforms the differential PMB6819 output to a single-ended output (which is normally 50Ω). A BALUN can be designed using ANPASS. One open question is the intermediate BALUN input impedance. It's a good idea to take an intermediate impedance value (say 35Ω) so that the match is distributed over the first matching network and the BALUN. This often gives the largest bandwidth and low tolerances.

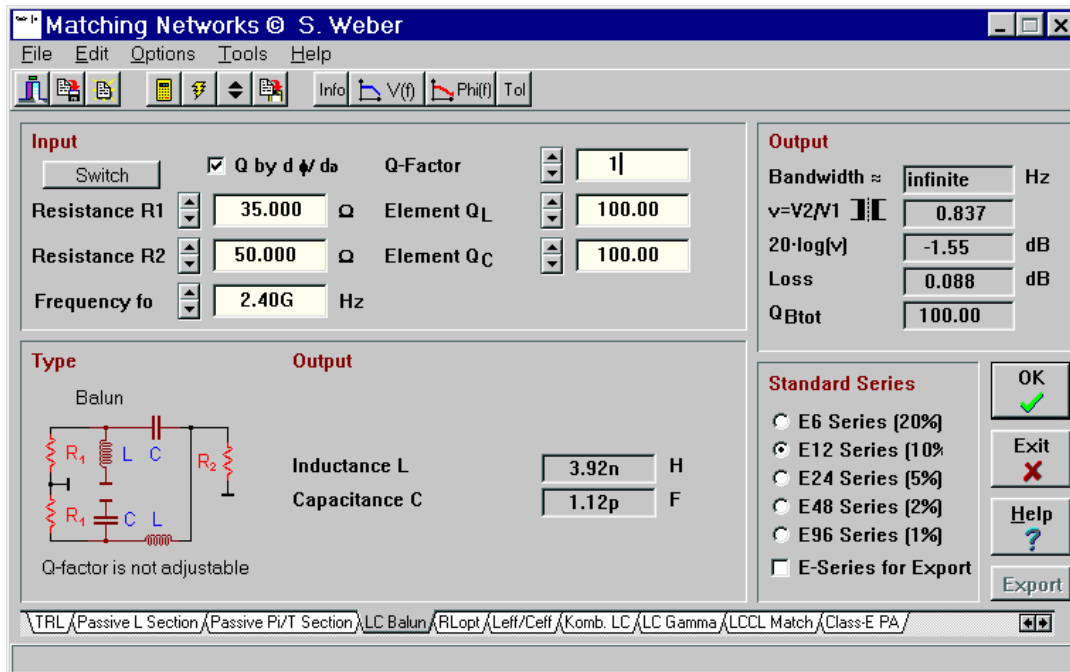


Figure 3 : BALUN design using ANPASS

The resulting circuit is very close to what is shown in Infineon’s PMB6819 data sheet. Of course in reality some tweaking is always needed in 1.9GHz circuits due to component parasitics and modeling inaccuracies. Also the impedance at the harmonic frequency is not unimportant due to large signal operation. This behavior is known as harmonic matching, but it is not easy to get advantage from this behavior at a GHz power amplifier.

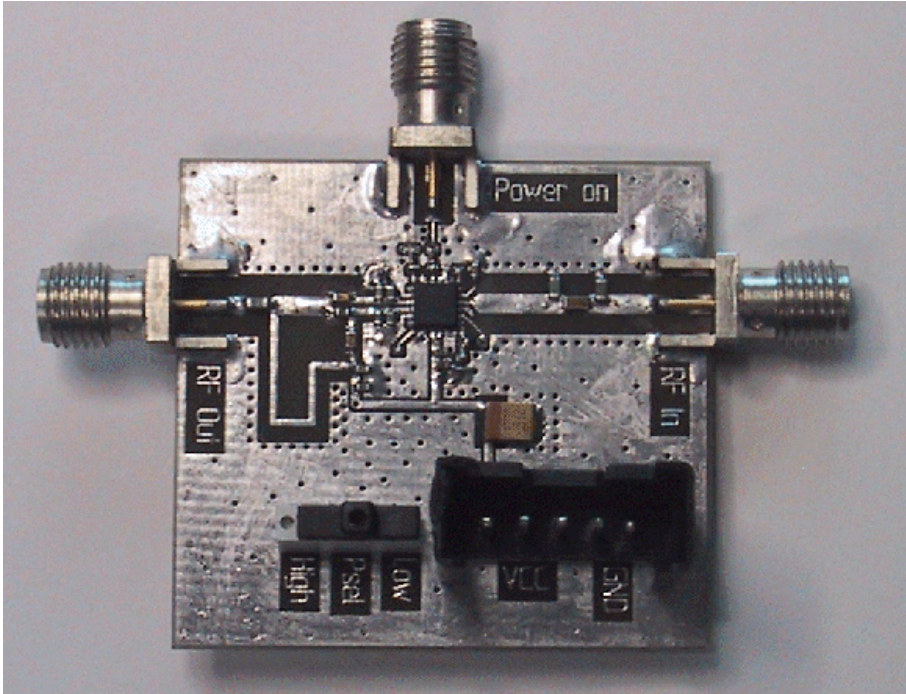


Figure 4 : PCB of the 2.45GHz PA. The L-shaped line is a  $\lambda/4$  DC feed.

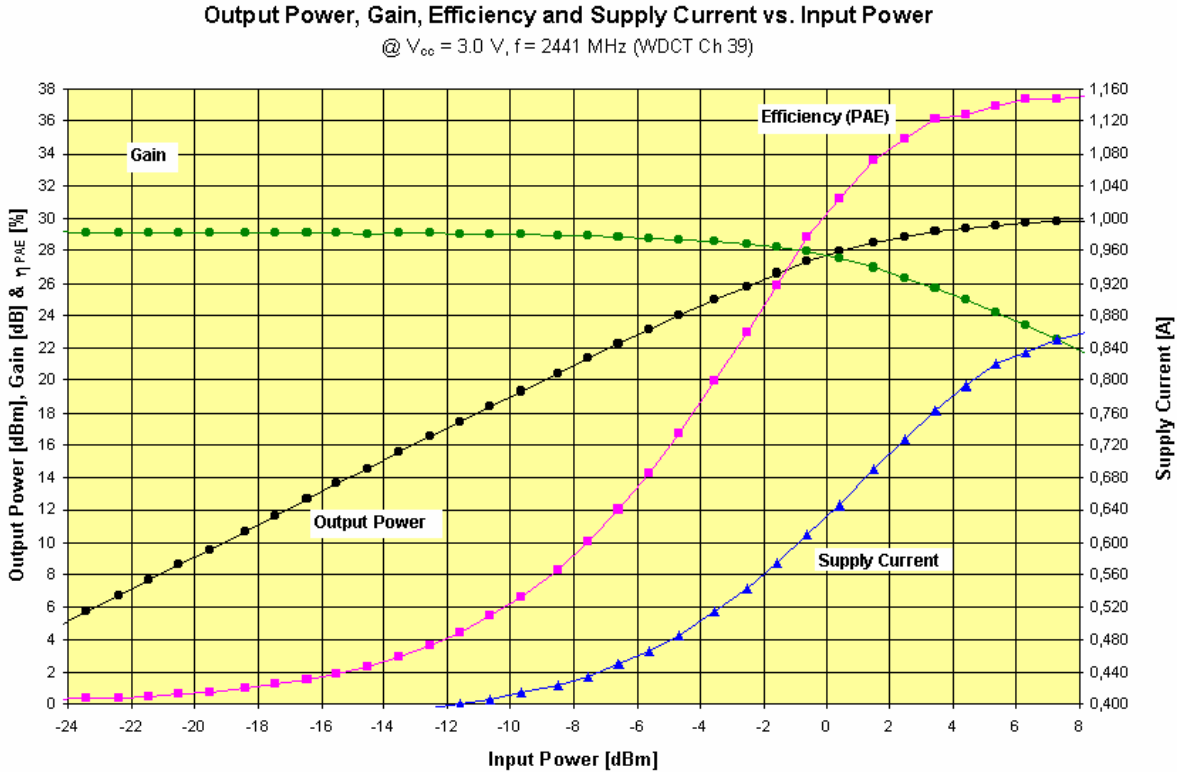


Figure 5 : Measurement results (some improvements are still possible)

Careful biasing and supply bypassing is needed because any RF PA will not create trouble only at the operation frequency. Especially at lower frequencies they often become unstable. In practice the transistor should "see" no too extreme impedances at all his 3 terminals all over its active frequency range. Often damping resistances are necessary. It is very interesting to see that bypassing with high-Q capacitors is in many frequency regions much worse compared to caps with lower Q, hence larger series resistors. Very important is the minimization of any series inductance, sometimes you need 3 or 4 capacitors with well-chosen values.

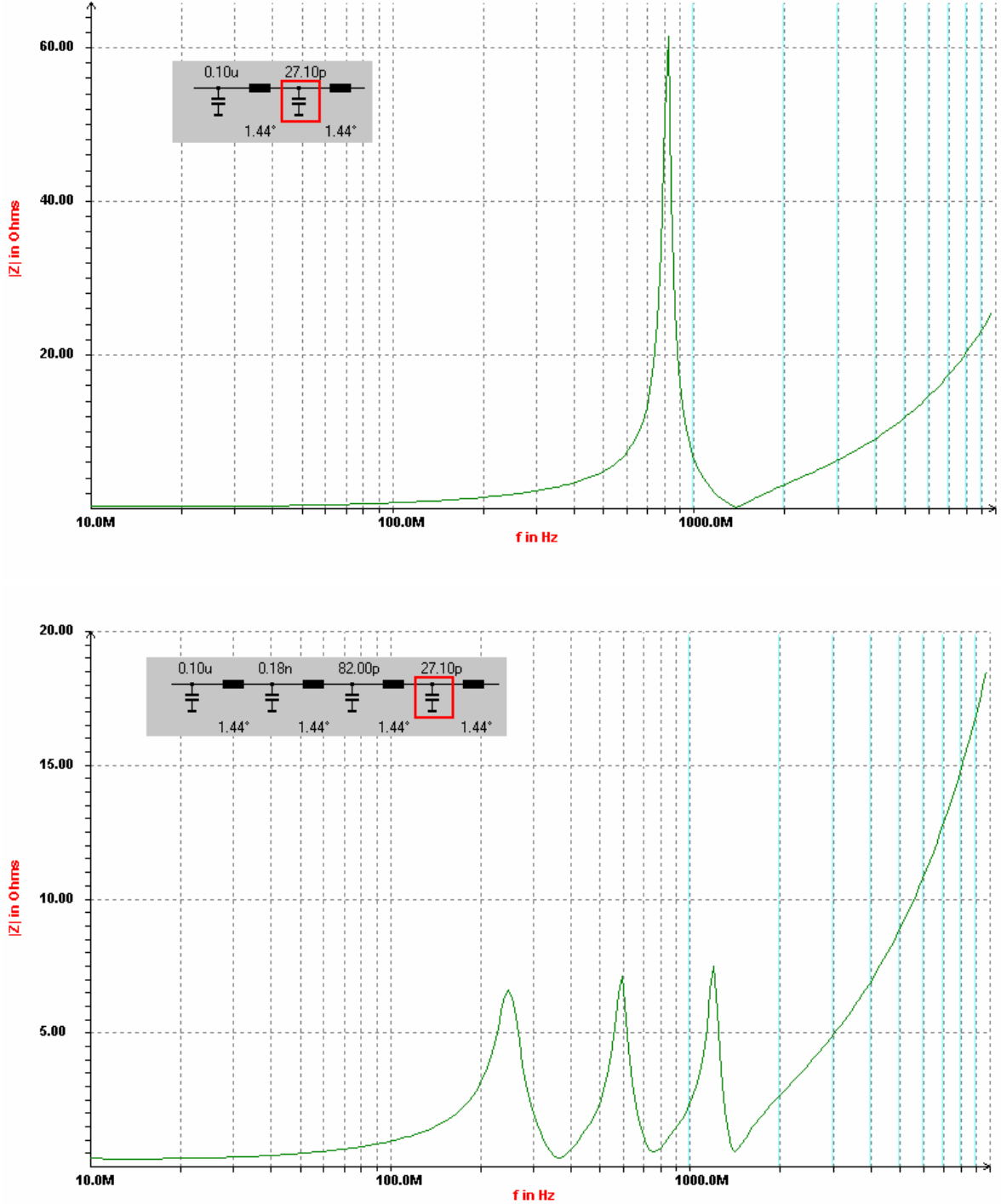


Figure 6 : CSMITH impedance graph of a supply bypass network with 2 high-Q capacitors and 4 standard capacitors (lower graph).

Some people say simulating RF power amps is nearly impossible, but this is not true. With careful modeling you can increase accuracy step-by-step. The remaining errors should be finally smaller than 1dB in output power and gain. To not overlook any aspect you should always ask yourself what you calculate really close enough to reality.

Topics	Influence	Comments
Transistor models	May have a large influence, especially on interstage matching!	Gummel-Poon may be sufficient for Si, but not in all cases. High current/low voltage region is critical, also quasi-saturation and breakdown!
Capacitances to substrate	Often a low influence (not for transistor or MOS-C capacitances)	This is different to low power/high impedance designs.
Series resistors	Medium influence. Look also at the on-chip MOS capacitances	Reduces gain
Series inductances	Large influence! Not only as feedback in BJT emitters stages	Changes frequency response
On-chip coils	Medium influence. A peak Q of 5..10 is realistic for Si technologies. Include the lines to the coil.	Modeling is not too difficult, but Q is limited for typical Si technologies
Package model	Strong influence due to series inductances	Not easy to model
Substrate model	Medium influence on bias and RF performance	Difficult to model, important for mixed mode designs
PCB and external components	Large influence	Grounding and crosstalk are difficult to model
Bypassing and biasing	Large influence on stability and linearity	Don't optimize only at the operation frequency

Table 1 : Summary of key factors in modeling for RF power amplifiers.

- [1] <http://www.weberconnect.com/adlab2.htm>
- [2] Class E - a New Class of High-Efficiency Tuned Single-Ended Switching Power Amplifier, N.O. Sokal, A.D. Sokal, IEEE JSSC, vol. SC-10, no.3, pp. 168-176, June 1975
- [3] Modeling for Si-Bipolar Power Amplifiers, Dr. S. Weber, AACD Workshop Copenhagen 1998

## 4 Hints on the AdLab Tools

The basic usage of all programs is very similar to CSERPAR. Some of the tools are quite complex, so you will need some time to evaluate all features.

Sometimes it may happen that one of the tools crashes on start up. In this case delete the corresponding INI file e.g. CSERPAR.INI or VIERPOL.INI in the Windows directory.

### **NOISE :**

#### **Input of two-port parameters :**

You may specify them by hand or you can load them from a file.

#### **Apply series feedback for LNA design (page Noise Circles) :**

By using inductive feedback for many transistors it is possible to get the impedances for noise and power matching close together. In detail you can see the response to inductive series feedback by pressing the More button.

#### **Hints on optimization :**

The optimum for best system noise performance also depends on the noise figure of the following stage. By pressing the Optimization button you can start the search for the optimum generator impedance. With series feedback may may get even better results. CSERPAR also features an export function to Aplac.

### **ANPASS :**

This program is basically for the synthetization of matching networks. But there are also other calculations included, e.g. resistive damping elements or optimum load impedance for power amplifiers. Note: ANPASS includes the loss of the elements used, but e.g. not its parasitic inductance or capacitance. For luck this is possible with CSERPAR and there is an export function from ANPASS to CSERPAR.

### **VIERPOL :**

With VIERPOL you can transform two-port parameters, e.g. from S-parameters to Y-parameters. But you can also analyze the two-port by calculating its maximum stable gain or its stability factor. So if you want to design an RF amplifier you should first analyze the active device using VIERPOL. After this you can move to CSERPAR or NOISE.

### **ANTENNE :**

It's not only for basic antenna calculations but also for wireless links or intermodulation calculations.

## 5.1 How to Get the Full Version and Support

If you like the program please register your shareware. Then you get the newest full featured version and e-mail support. If you received the key code, please start the Smith chart program CSERPAR.EXE and go to the upper right menu item Registration. A window will appear and you have to enter your name, adress and the key.

For details look at the files README.TXT or README.HTM and ORDER.TXT.

Note: CSmith is part of AdLab and of course AdLabPlus features all AdLab tools including CSmith.

### Author :

Stephan Weber, Therese-Giehse-Allee 23, 81739 Munich, Germany

E-mail: [support@weberconnect.com](mailto:support@weberconnect.com)

Homepage : <http://www.weberconnect.com>

If you have further questions or suggestions on the program functionality, please write an e-mail.

## 5.2 Restrictions of the Shareware Version

AdLab is a collection of programs. In the shareware version there are some minor restrictions, i.e. some buttons or checkboxes are disabled.

- All programs            The total count of program calls is limited 100.
- CSERPAR                S-Parameter import functions are removed  
                              The circuit can contain only up to 8 elements (instead of 12)
- ANPASS                 No export of the circuit to CSERPAR, less circuit topologies
- ANTENNE                No restrictions
- DBRECHNE              No restrictions
- LBELAG                 No export
- LTRANSF                Characteristic impedance is fixed to 50Ω
- NOISE                  No graphic for VCO noise, no load and save of noise files
- SERPAR                 No restrictions
- TRAFO                  No restrictions
- VIERPOL                No feedback circuit can be applied, area scaling factor is fixed to one
- SPARAM                No output for impedances and gain (MAG, MSG, etc.)
- AFILTER                No graphical output for phase shift, group delay and step response, only for gain vs frequency.
- COILS                  Parallel capacitance for resonance frequency calculations is fixed to 1pF.
- LEITER                 Within calculation of current distribution only the infinite plate is available.
- ESTATIK                In the field calculation save/load of charge configuration to file is disabled.
- MODEL                 Trimming the parameters IKF, ISE and ITF is disabled.
- OHM                    No restrictions!
- RLCFILT                No series tank circuit and no graphical output for LC tanks.

- TRANSISTOR No export and no graphical output within amplifier page.