Gunthard Kraus, DG8GB

Possibilities and limits of circuit simulation for radio amateurs

Revised version of the lecture to the UKW conference 2009 in Bensheim

1.0

Sense and purpose of circuit simulation

Everyone who has already developed circuits knows the questions that torment those who do this work:

- How do I find out how the circuit functions?
- How do I create the data required?
- Will I find the optimum?
- What things that I have not considered could affect the results?
- Will the circuit work "cleanly" without oscillating?
- Do component and manufacturing tolerances affect the results?
- Do I have alternatives if the solution chosen leads to a dead end?
- etc.

Circuit simulation can answer most of these questions before a prototype is manufactured or a soldering iron is touched. However exact knowledge of the desired circuit behaviour is absolutely necessary. Working with the simulation as well as any changes are ALWAYS the responsibility of the developer

- no program can replace your specialised knowledge (even if the advertisement suggests otherwise).

The use of circuit simulation always means precious work time is saved. Also alternative ideas can be examined quickly or a prototype can be optimised.

2.0

The different types of simulator program

The differences are:

- Simulators that work in the time Domain. These are in principle all SPICE and PSPICE programs. "SPICE" stands for Simulation Program with Integrated Circuit Emphasis and "PSPICE" is simply SPICE for the PC.

  They provide all the information about stability and starting behaviour. The response to signals with arbitrary waveforms is determined by splitting the signal behaviour into very short, but practically linear sections. Additionally all nonlinearities in the circuit are identified.

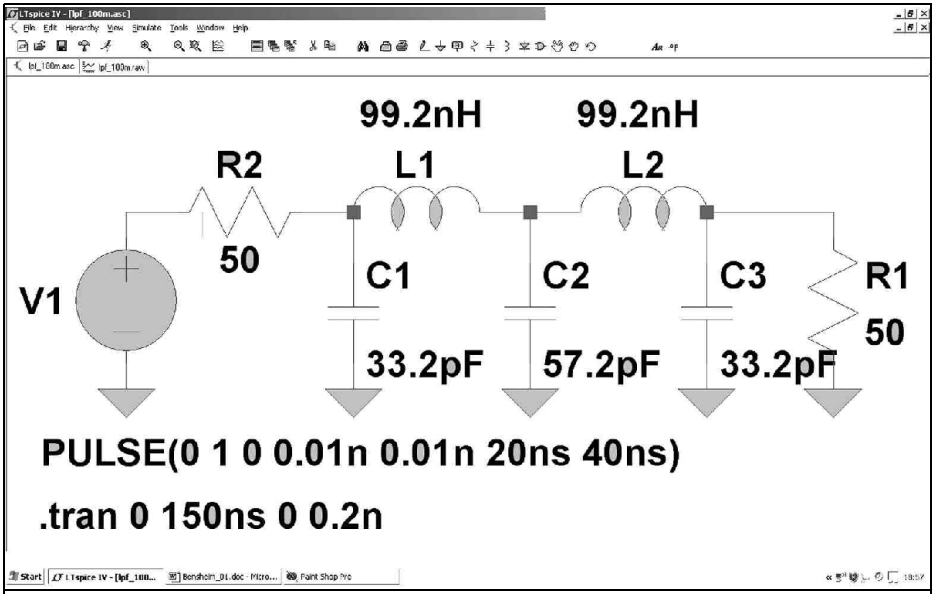

Fig 1: This is how it looks: a screen without complications, only with the circuit and the necessary text inputs (Properties, Controls and Commands).

Other options are the AC-Sweep (wobulator to determine the frequency response), noise simulation and the FFT (Fast Fourier transformation) for the analysis of a signal in the frequency domain (spectral representation). Thus they offer the most comprehensive possibilities, but the simulation accuracy always depends on the quality of the SPICE models. These are supplied with the software or by the component manufacturers. In reality they specify exactly how the simulation and reality will agree.

- Simulators that work in the frequency domain, these often use S-Parameters (some also convert Y-Parameters to S-Parameters). So vector network analysers can be used to check the simulation or to gain the necessary component properties for the simulation itself. This achieves a good agreement between simulation and prototype. A disadvantage with this in the RF and microwave area (to over 100GHz) is that it is principally a linear simulation (a small signal system). Information about distortion with large signals is usually only available with cheats that the program may offer (e.g. using harmonic balance) and these are normally secrets of the software vendors.
- EM simulators use the distribution and the relationships of the electrical and magnetic fields around "structures" for analysis. They can be divided in two groups:

  For things with surfaces (Patch antennas, couplers, stripline filters etc.) The complete structures are divided into very small rectangular or triangular "cells". The individual fields for each cell are computed and finally all added together.

  For wire antennas (or similar wire objects) these are known as "NEC". The structure is also divided into small elements, in this case short pieces of wire with linear current and power distribution. These elements

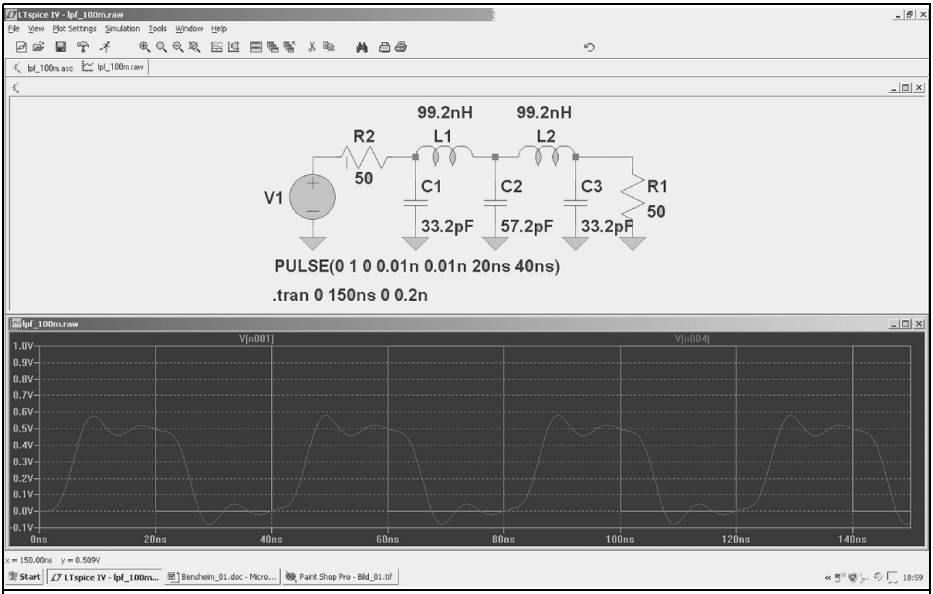

Fig 2: The result of the simulation. Additionally to the circuit the desired signal is shown when the measuring point is clicked.

can be examined more easily to calculate the total behaviour of the arrangement by addition of all individual calculations.

3.0

SPICE simulation for the radio amateur

The situation is very much relaxed because the restriction "for the amateur" usually means reduced price. It is not a problem to spend €50,000 on simulation software, these are enormous machines that can be used to design the associated printed circuit board and examine the EMV behaviour etc. For the private person money is replaced by time. In practice that means a reduction of comfort as well as a separating the simulation from the PCB CAD. There are already affordable solutions for PCB design that include EMV simulation and also a simple SPICE simulation (e.g. Target).

The large SPICE software manufacturers always supply free demonstration versions of their products, but these make you grind your teeth in the middle of a simulation with a message like: "all components used" or "MEMORY restricted in this version". Above all because, unfortunately, the design started can never be transferred to a more complex (and more expensive) program variant.

A program for amateur purposes (or training purposes!) should have the following features:

- Little or no cost
- No restrictions or limitations, thus fully functional
- An easy import of new component models from the Internet
- A good user group removing the

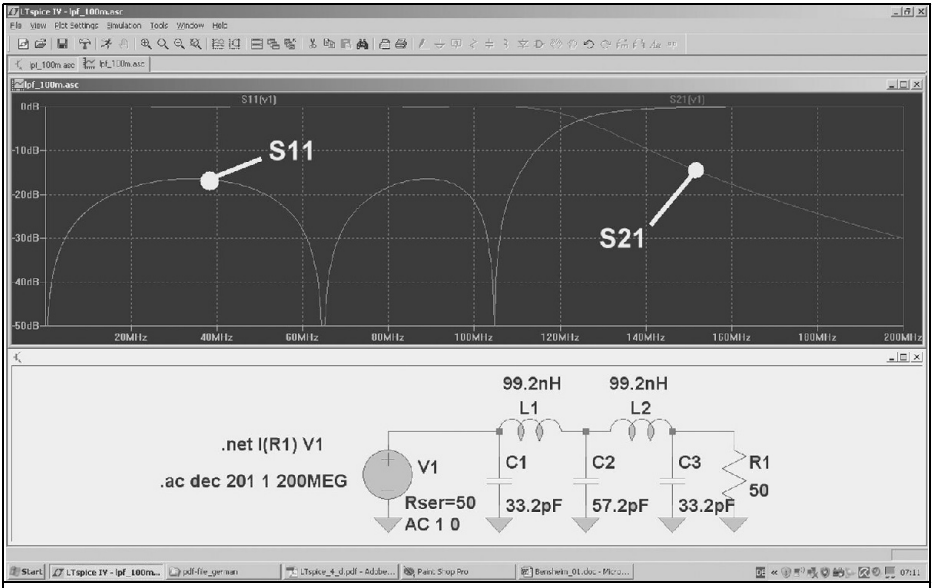

Fig 3: The simulation of S-parameters requires a different spice directive and a new simulation command.

dependency on expensive support for questions or problems solving

Fortunately the company Linear Technologies has made a free version of their PSPICE software LTSpice IV available to engineers on their homepage [1]. There is no problem to download the software, even the registration before download can be refused. Importing new models from the Internet is problem free and there is an easy to use symbol editor.

The only drawback in using the software is that it has a very simple user interface and it takes some time to become familiar with the mixture of keyboard and mouse commands required.

Note: there is a Tutorial (in German or English) for this program on my homepage [2] that can be downloaded free of charge. It contains an extensive collection at practical projects for training.

The use of LTSpice will be demonstrated using the example of an 110MHz Tschebyschef low pass filter for a characteristic impedance of 50Ω driven with a 25MHz square wave signal. The screen shown in Fig 1 is produced.

Beside the circuit there is the facility to include additional text lines this requires some explanation:

The line “Pulse…” below the circuit defines a square wave signal with Vmin = 0V and Vmax = 1V. The rise and fall times are 0.01ns with a pulse length of 20ns and a period of 40ns. The signal begins immediately at the time t = 0s.

The line “.tran… “ defines the time period, in the time domain, that will be simulated. The time period is 0 to 150ns that will be divided into maximum 0.2ns time steps.

After the simulation is started the screen is divided and an empty result diagram

appears for the simulation (Fig 2). Clicking the mouse on the point of interest in the circuit immediately displays a diagram of the signal at that point (from the initially starting point of the circuit). Using different mouse clicks and submenus the result can be adjusted until the desired results are achieved (e.g. several diagrams, different values or different colours for the curves…).

For an S-Parameter simulation the internal resistance of the source (50Ω) must be set in the "Property" menu and an AC Sweep programmed with a SPICE simulation COMMAND (Wobulator from 1Hz to 200MHz with 201 points per decade, decade sweep). The Spice directive ".net I (R1) V1" specifies an S-Parameters display. The final result is shown in Fig 3.

4.0

S-Parameter simulation for the radio amateur

4.1. Some basic information about S-Parameters

System descriptions and computation start from approximately 1MHz upwards with the highest frequencies (today more than 100GHz) easily measured. These require a completely different model:

Everywhere in the system the same system resistance is used (usually 75Ω for entertainment electronics or video technology and 50Ω for most other areas). This defines the internal resistance of the signal sources, the output resistance of the modules used, the characteristic impedance of all cable connections and all terminating resistors.

The principle of this technique is perfect power matching (Ri = Ra) everywhere in the system.

Using suitable measuring instruments (directional couplers) the deviations of the ideal resistance are measured at the input and output (technical term: "Ports") to calculate the "reflection factors" (S-Parameters S11 and S22).

The power output can be calculated from the input power multiplied by the "forward transmission factor" (S-Parameter S21).

The effect of a component from the output back to the input is the "reverse transmission factor" (S-Parameter S12).

A complete system has many components connected in line. Using suitable software the total behaviour can be easily determined.

4.2. Using PUFF

This well-known DOS program available from [9] is probably the standard program used by radio amateurs for nearly 20 years and has not retired yet. It is low priced, has minimum memory requirements, high stability and simulation accuracy. It is simple to operate using key commands with the most important basic models available. Clear screen layout makes it the RF pocket calculator for the developer making it easy to quickly test new ideas. In addition it has been upgraded again and again and even made compatible with Windows XP.

Note: my homepage [2] contains all documents required and help (including a detailed list of the control commands with explanations and a guidance plus a "PUFF_XP-Servicepack" for trouble free operation under Windows-XP - sorry but only in German.

The complete PUFF simulation of the

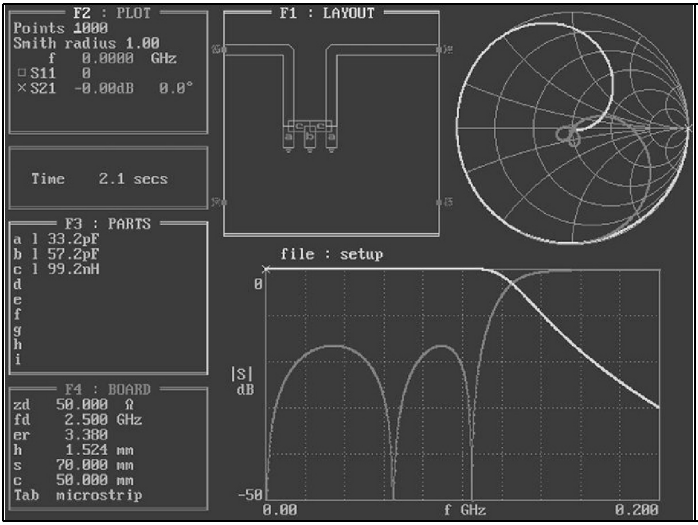

Fig 4: The Puff screen showing the material data entered and the simulation results entered on the same page.

110MHz low pass filter is shown in Fig 4:

The screen shows component input, results by Smith Chart or graph, PCB layout and most importantly the material data.

The disadvantages are not too bad: both input and output of diagrams is dated (particularly expressions). Using PCX output for diagrams and screen shots helps. Important models for failure areas are missing e.g. for micro strip lines etc. It has not been changed for Windows Vista. Nevertheless for fast testing of an idea, the speed and operation are unbeatable with the same accuracy as large expensive simulators. The restrictions must simply be accepted.

4.3. ANSOFT Designer SV

This uses the most modern simulation techniques. It is one of the best Windows programs with all conceivable facilities. Hundreds of components and failure area models cover all conceivable applications up to waveguide. An integrated filter calculator saves the search for a suitable external filter calculation program. The complete linear simulation of the expensive ANSOFT product is included without restrictions for study purposes in the Student Version (SV), it is free of charge and can be downloaded from [4].

This is a well thought out marketing strategy; if students learn to know and love such a program they will want to use the same with greater security in a later job. Thus the limitations are set accurately where it would cause problems to a company that tries to use it secretly. The direct creation of a draft PCB is missing, the very good, and patented, nonlinear simulator is missing and the automatic optimiser is restricted.

However this is ideal for amateur radio use but I have some nostalgia for PUFF. A complex program such as ANSOFT Designer SV with so many fascinating possibilities clearly requires more training and operator effort. The input and simulation are separate and some work is needed to produce the output required. There is nothing that cannot be shown; there are stability values, S-Parameter, noise figures etc.…

Note: an extensive Tutorial (in German or English) can be downloaded free of

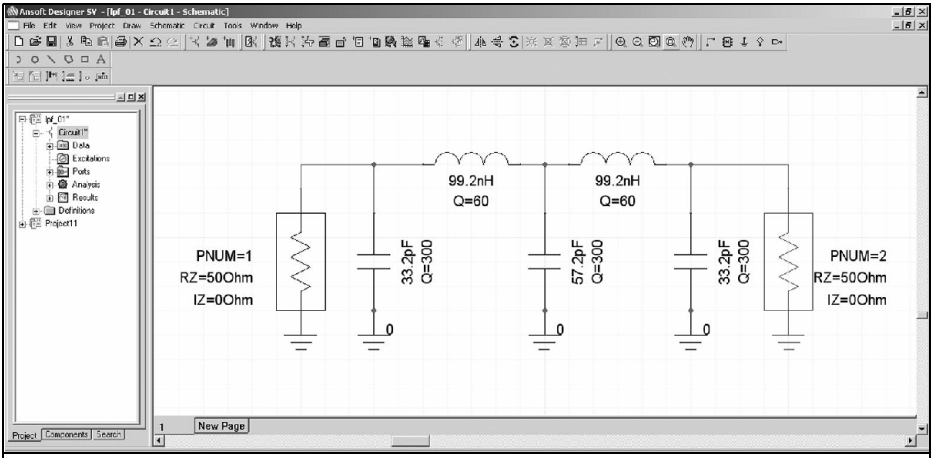

Fig 5: The input screen for an ANSOFT project, the designer must spend time producing this data.

charge from my homepage [2]. It contains many a projects with active or passive components. The first part of a project is shown in Fig 5 i.e. the input of the circuit for the simulation. A screen display of S11 and S22 in graph form as shown in Fig 6 can be displayed if required after the simulation under "Results"

5.0

EM simulation for the radio amateur

5.1. Simulation of flat structures with SONNET Lite

The comments about the good marketing

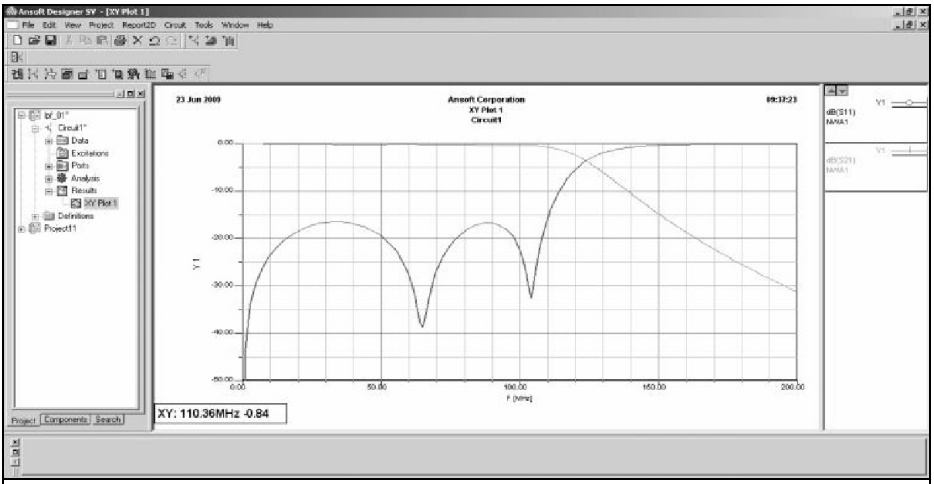

Fig 6: The results after simulation shown after choosing the required display.

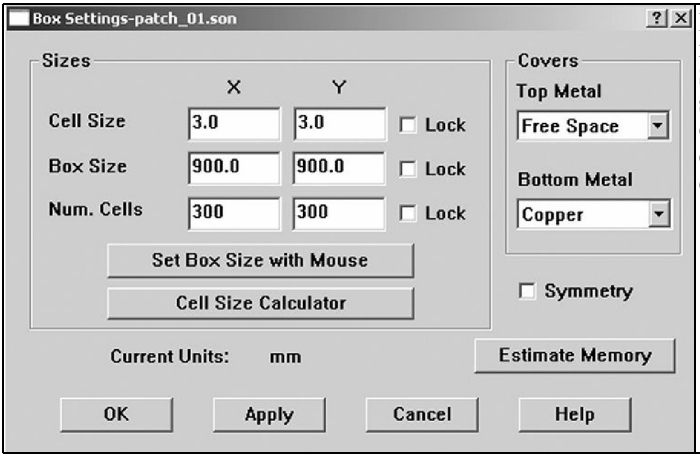

Fig 7: The most important part of a SONNET simulation is correct definition of the box characteristics.

strategy by ANSOFT also apply to the Sonnet Company. With this program a definite limitation is made for the maximum useable storage, that gives somewhat greater inaccuracies in the final result of the simulation but these are still quite acceptable.

The following applies:

- If you just download the free program and install it you get 1Mb of storage.
- If you register on line after installation and request a free licence you can suddenly use 16Mb.

Since improvements in the simulator are incorporated in the Lite version it becomes more efficient saving on the use of storage.

Training for this subject is clearly more difficult than for PSPICE or PUFF. It can be done by specifying the surfaces, the individual cells sizes and the different materials used. Then the signal injection must be made correctly. Again and again the storage used must be controlled.

Note: There is a list of my publications in UKW Berichte (The German version of VHF Communications Magazine – generally all articles published in UKW Berichte appear 2 issues later in VHF Communications Magazine, if you cannot find an article there is a full index for VHF Communications Magazine on the web site [9]) on my homepage [2].

An interesting program, SONNET Lite 9.51 was published in VHF Communications Magazine issue 3/2004. It contains exact instructions for using the program and two different patch antenna simulation sample applications. This guidance can still be used for the most modern program version.

For training and assistance with problems there are a great many examples, application notes and tips on the homepage [5], the FAQ section is really first class, or on the Internet.

Generally everything is great, but one thing does cause pain; the simulation of the radiation patterns as well as the antenna gain computation are intentionally missing. Unfortunate….

As an example the simulation of a square patch antenna made from FR4 material for the frequency f = 435MHz will be used. The input impedance of the antenna is determined relative to the patch edges. The following must be considered:

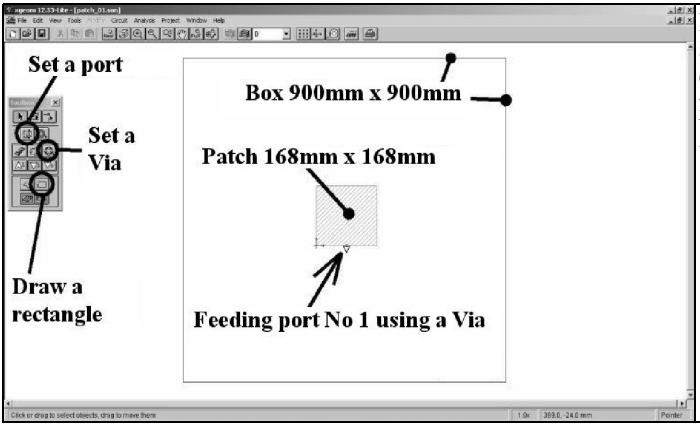

Fig 8: The antennas structure designed with a feed port Using a Via all inside the box.

SONNET is only an analyser and NOT a design program. It only tests the finished ideas of the user.

Thus the basic design of the antenna must be produced e.g. with the free DOS program "patch16" that can be found easy on the Internet (enter "patch16.zip" into a search engine). It is excellent because it is simple and easy to use. An A4 (210mm x 297mm) double-sided copper clad FR4 laminate board 1.52mm (0.06 inch) thick is used. The dielectric constant is 4.35 and the dissipation factor tand is 0.01. The Patch program gives 6.61inch x 6.61inch = 168mm x 168mm as antenna dimensions to be taken forward to SONNET for simulation.

The PCB material is entered with the metals involved and "the box" data as shown in Fig 7. SONNET uses a rectangular box, with the PCB to be analysed inside, as basis of the simulation. The field distribution in such a box is well known and calculable. The user must program the details.

The first step is to enter the box dimensions (with half a wavelength distance between the structure to be examined and the walls in each direction). The length and width as well as the number of cells that the structure is divide into must be specified. The cell width or length may not be more than 5% of the wavelength otherwise the accuracy suffers. It should not be much less as 1% for the same reason. The optimum is between 1 and 2%.

The screen shown in Fig 8 is the SONNET screen after completion of the input work; it should then be stored under a suitable name.

Next the sweep frequency range is programmed - here a reasonable setting is 400 to 500MHz. Now the simulation can start but as usual with large programs there is NOTHING to see. Programming the presentation of the results has its own procedure (View Response/Add to graph) and produces the screen shown in Fig 9.

The resonant frequency of the antenna can be seen easily and is still approximately 7MHz too low. But it cannot be seen from this presentation if the input impedance (made up from the parallel connection of the resonant element, the radiation resistance and the losses of the PCB material) is larger or smaller than 50Ω. A Smith Chart is required and there is a button "graph" in the menu border for this purpose. When this is selected there is a question; "Cartesian" or "Smith", the remainder is automatic.

The actual input impedance is not shown

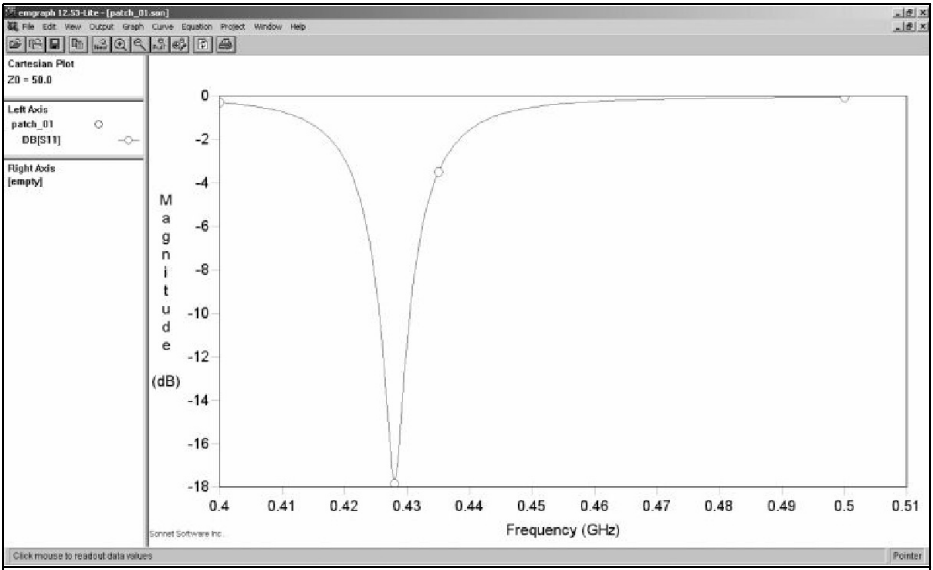

Fig 9: The first result to show is the reflection factor S11 that shows the resonant frequency of the antenna.

in Fig 10; it takes some of work to determine from the input impedance at the feed point:

We have to use the reflection factor r ("Mag") of approximately +0.128. The following operation is performed using pocket calculator:

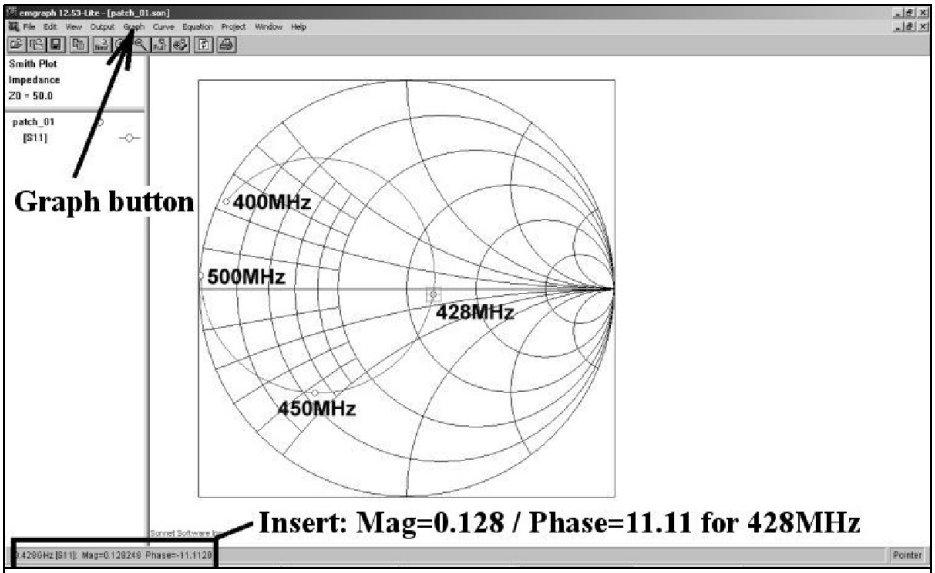

Fig 10: The Smith Chart is indispensable to calculate the exact input impedance - see text.

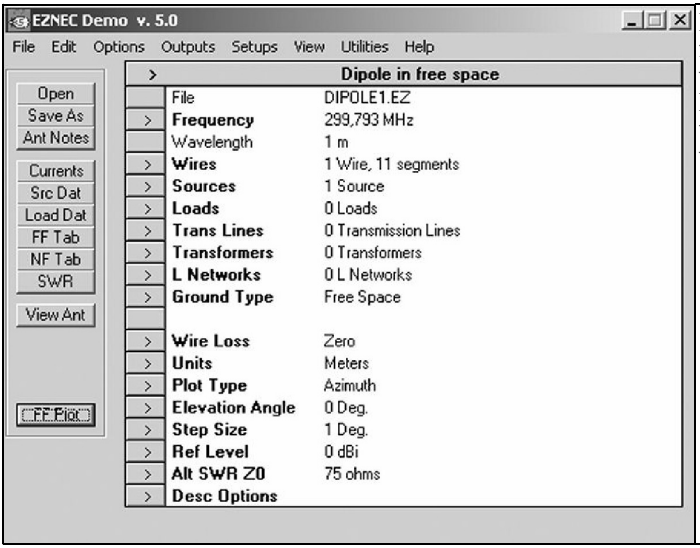

Fig 11: The input form of EZNEC will automatically load the last design that can just be reworked.

$$R_{IN} = 50\Omega \cdot \frac{1+r}{1-r} = 50\Omega \cdot \frac{1+0.128}{1-0.128} = 64.67\Omega$$

To see the pure radiation resistance at the patch edges the simulation must be repeated with the PCB losses set to zero and the metal losses set to “lossless”. Only then is the radiation lossless.

The great thing is that experience shows that the simulation agrees quite exactly with the results of measurement made on a prototype. The resonant frequency is always somewhat high (usually between 1 and 3%), that is unfortunately typical for all EM simulators.

But this isn't a serious problem: manufacturing a sample PCB determines the deviation from the desired value and dimensions are corrected using a pocket calculator. If the dimensions are larger this results in a larger wavelength and therefore a lower frequency.

We can only wait and hope for the calculation of the radiation patterns for the antenna…

5.2. Simulation of wire antennas with NEC

The Lawrence Livermore laboratory developed a simulation method for wire antennas called NEC (Numerical Electric Code) in 1981 for the Navy. The antenna is divided into very short pieces = “segments”, where the current and the voltage change is (nearly) linear (see SPICE). Thus amazingly exact simulation can be accomplished.

The standard used in a great many free application software packages is NEC2.

The development continued and the weaknesses of NEC2 e.g. wrong computation of structures close to or crossing wires buried in the soil were corrected by NEC4, but:

NEC4 was, until recently, not available and was considered in former times as secret. Today it is available outside of the USA but costs a lot of money (normally between $2,000 and $4,000).

The private individual (small purse) must be content with NEC2 and the only problem is the choice from the selection

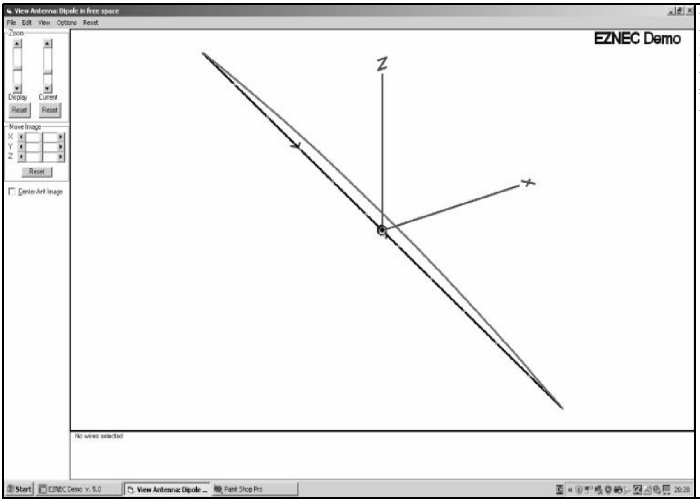

Fig 12: The antenna view presented by EZNEC.

available on the Internet. It is reasonable to select something that:

- Is already in common use
- Has good input and output facilities
- Has an easy to understand output
- Is regularly and well maintained
- Gives Near and Far Field simulations
- Has many examples, help and support on a forum.
- Has a good manual and help
- Does not cost too much

For these reasons many radio amateurs and antenna developers have used EZNEC by Roy Lewallen, [8] for years. It has a fair current price of $89 with direct Download and payment by credit card.

Fig 11 shows a view of the operator interface of the newest version 5.0. The data is for a simple 299.793MHz half-wave dipole that is one of the examples supplied. This only shows a row of buttons and screen with input lines. The reason for this is:

EZNEC does not use the philosophy that shows an empty sheet when starting, but always shows the last loaded example. Simply retrieve a suitable antenna from the archives and change the appropriate lines of the menu. For investigation your own antennas you should collect a large supply of different simulated antenna types. This proceeding is somewhat differently, i.e. re-engineering, but it is not bad.

Fig 12 shows the screen after clicking on “View Antenna “. The antenna wires are shown in a coordinate system arranged in moveable segments using the controls located on left of the screen.

Clicking on “FF Plot” (FF stands for Far Field) produces the radiation pattern (Fig 13), the cursor can be used to read off the curve. A different elevation view can be selected under “Setups” in the main menu, there is even the even the option “Near Field”.

Tip: For those who would like to see the radiation pattern in 3D and in colour, get yourself the program “4nec2” from the Internet.

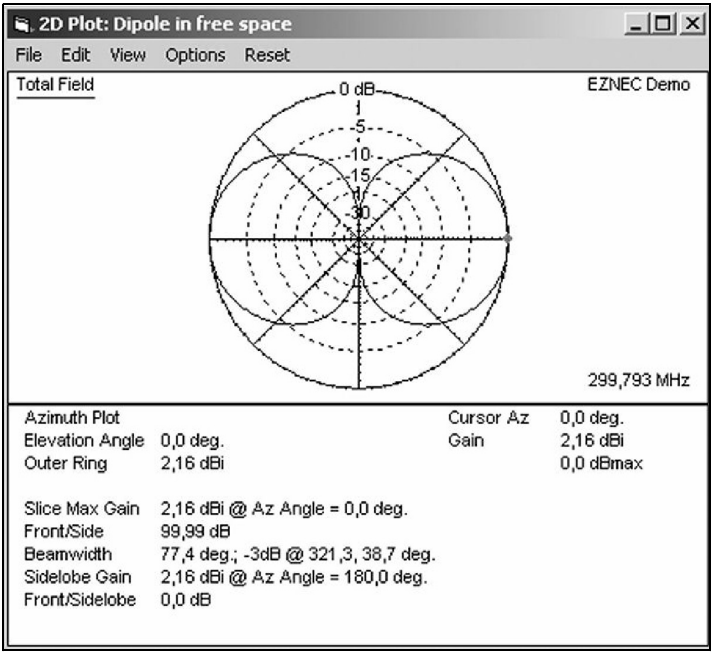

Fig 13: The simulated horizontal Far Field diagram for the antenna.

6.0 Other antenna simulators

On the Internet there are antenna simulation programs that work with approximation formulae. The program "patch16" has already been mentioned in the SONNET chapter, but there is a DOS program on the Web that can show the dimensions of many types of antenna (wire and patch antennas, log periodic antennas, antennas with parabolic reflectors and horn antennas) on the screen by entering the requested data. This free "Jack of all trades" is called "pcaad21.zip" and even as a DOS version does not loose its attraction.

Now have much fun and success with the design of your own projects and simulations. But also use the programs for error analysis, fine tuning and finishing development work.

7.0 References

[1] Linear Technologies, homepage for LTSpice IV, www.linear.com

[2] Gunthard Kraus, DG8GB, homepage under Tettnang Electronic School, www.elektronikschule.de/~krausg

[3] Reader service UKW Berichte, www.ukw-berichte.de

[4] Homepage of ANSOFT, www.ansoft.com

[5] www.sonnetsoftware.com

[8] www.eznec.com

[9] VHF Communications Magazine web site, www.vhfconn.co.uk