*Gunthard Kraus, DG8GB*

# Practice project: Durable and reproducible patch antenna for the 2.45GHz WLAN band, Part 1

**It has always been the desire every radio amateur to increase the range of his or her station as simply and inexpensively as possible. Nothing changes, therefore in this article the same applies to the computer age WLAN range.**

## 1. Preface

Connection of computers into a network and to The Internet using a wireless "Access Point" usually uses around 2.45GHz. Radio amateurs use these techniques to make contact and increase the range by increasing the power to its legal limit. An alternative is to use directional antennas that concentrate the radiated power in a certain direction. They can be used for both the transmit and receive side with interesting possibilities:

- Higher receiver antenna signal over the same distance and thus better transmission quality (e.g. expressed as SINAD improvement) or a greater range for the same received signal.
- No more all round radiation thus reducing unwanted propagation and/or directions of the signal.
- Mechanism for practical point-to-point connections, (e.g. if line of site path available)

At 2.45GHz the wavelength is small (122.45mm in air) and the half wavelength, important for many antenna designs is six centimetres. Therefore quite useful antennas can be designed and the patch antenna was chosen for this design.

This antenna is:

- Extremely simply and easily developed, because it consists of just a double sided printed circuit board.
- It can be designed in such a way that it is matched to 50Ω therefore it does not need an additional matching network.
- It shields against backward radiation therefore offers an excellent front to back ratio which is half the radiation pattern of a dipole with the associated gain.
- It has a maximum gain of 6.5dBi which doubles the receive signal in the forward direction compared to an isotropic antenna. Compared to a dipole or vertical antenna there is about 4.5dBi less gain.

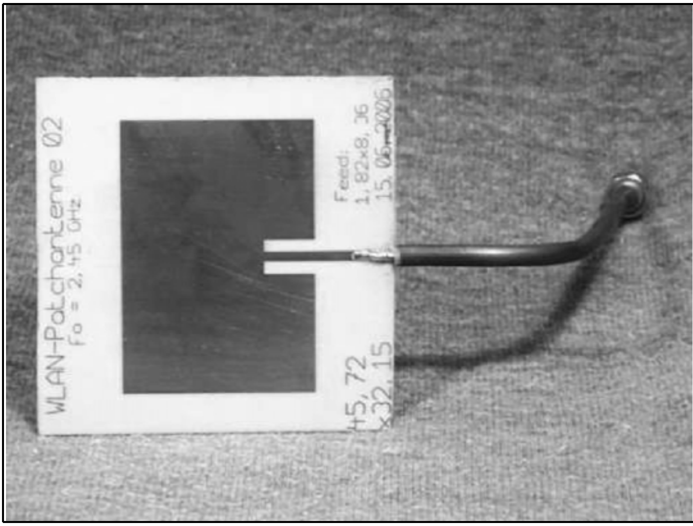


**Fig 1: A rectangular patch antenna with a 50Ω feed.**

If the transmitter uses a similar antenna the overall gain of the connection is approximately 9dB over the original antennas and somewhat more than the "three-way" signal. (WLAN range measurements are often quoted for a three-way connection i.e. three connected PCs – ed.). If this thought process is extended for the far field, the received signal decreases with distance therefore the "three-way" range can be found.

Naturally there are some points to be considered:

The patch antenna is narrow band with a radiation pattern in form of half an eight for a single patch and is ideally operated on a single frequency. Therefore the performance at the band edges must be considered. During the design thicker material and larger width patches can be used to give a wider band. In practice this will probably reduce the three-way range.

The requirements of the printed circuit board are very high. Apart from absolute mechanical stability and small losses at the operating frequency, small temperature and humidity dependence of all electrical parameters are demanded. Additionally the manufacturing accuracy must be kept within 0.01mm. On the positive side, if everything is under control, the finished antenna is very durable and can withstand falling to the ground without damage. The antenna can be reproduced easily, even for large production volumes.

Some articles have already appeared about the function and design of patch antennas for different frequency ranges, e.g. [1], [2] or [3]. Therefore the introduction is to be kept very short and aimed directly at the practical project.

Not only will the design procedure and the necessary steps up to production stage be considered but an important criteria is not to use expensive full versions of the EM Simulator programs. This discourages anyone from using pirate copies of these applications. Therefore one tries to get along with programs that can be obtained free of charge from the Internet, these have things missing that are replaced with an imaginative approach.


**Fig 2: On the left is a rectangular patch antenna with an SMA connector for the feed. On the right is a circular patch antenna with an SMA connector for the feed.**

## 2.

## Defaults and product specifications

Three different forms of single patch antennas are to be designed, built and tested.

The size of the printed circuit boards is to be about 80mm x 60mm, fed by an SMA socket.

The resonant frequency is 2450MHz, with a bandwidth of 60MHz required to incorporate the whole frequency range from 2420 to 2480MHz.

The PCB material used is Rogers R04003 with its excellent mechanical stability and workability. It has a thickness of 1.52mm (= 60 MIL), 35µm copper coating, a dielectric constant of 3.38 as well as a dissipation factor of 0.001 at 2GHz.

A more detailed look at the finished products:

- A rectangular patch with a 50Ω "Insert feed" is shown in Fig 1. The start point of the feeder on the face of the patch is chosen to give an input impedance of 50Ω. A semiridgid cable with an SMA plug is used as a temporary connection, it is better to use a surface mounted SMA plug soldered to the lower surface of the PCB with the centre conductor soldered to the stripline.
- Fig 2 shows the two other antenna designs. These are a rectangular and a round Patch. The feed is made within the patch face by an SMA socket, which is soldered on the lower surface. Both versions are handy and durable but the round version might prove better in hard everyday life because of the missing corners and edges. This is at the cost of a narrower bandwidth. In addition it must be connected to the transmitter and receiver with more care in order to align the polarisation correctly. Even for specialist this is a bit unusual.

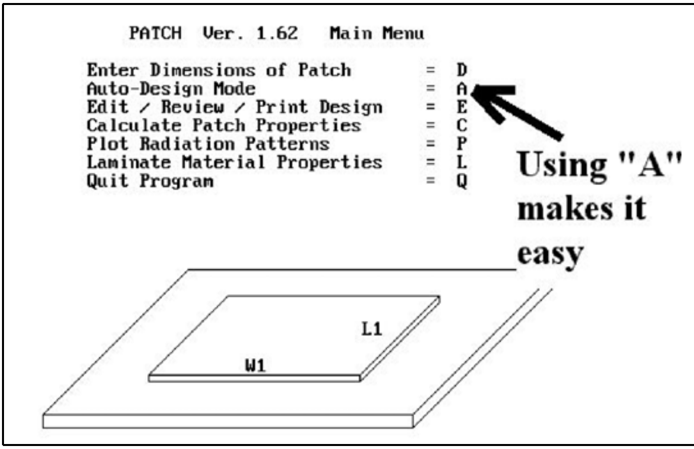


**Fig 3: The introduction screen for "patch16". Select option "A" for Auto-Design Mode.**

## 3. Development of the rectangular patch antenna

To start, the simple rectangular antenna is chosen, fed from the lower surface. There is a free design program available on The Internet called “patch16”. It will be interesting to test how it compares with a genuine high speed EM simulator and to manufacture the first PCB after the simulation. This will give an idea of the differences that can be expected.

### 3.1. Design from the program “patch16”

This program is loaded, installed and started from the Web onto your computer, it runs perfectly under Windows XP. When greeted with the message “Do you need Instructions” answer “no”. The following steps are shown in Fig 3, choose “Auto design mode –A” and use the following values for the questions:

- Resonant frequency: 2.45GHz
- Range: 60MHz
- Dielectric constant: 3.38
- Dissipation factor: 0.001 (Loss tangent)

After these values have been entered, the screen shown in Fig 3 is displayed, but there is a malicious trap to consider (Fig 4). The program has chosen a PCB thickness of 88 MIL (because of the square patch format), in order to give the desired range, however a thickness of 60 MIL was required. As long the square format is chosen, the program cannot change the calculated PCB thickness therefore another way must be found.

Consider the following:

- A larger range for a patch antenna means a thicker PCB or a bigger patch width with the same patch length (additionally a high dielectric constant helps).
- The patch width can be increased with a PCB thickness of 60 MIL until the required range of 60MHz is reached. If necessary the patch length must be changed in order to give the desired resonant frequency of 2450MHz. That can be done as follows:

Note the suggested values for the length (1.24 inch), the width (also 1.24 inch) as well as the feed point (0.45 inch) and press the letter “D” on the keyboard.

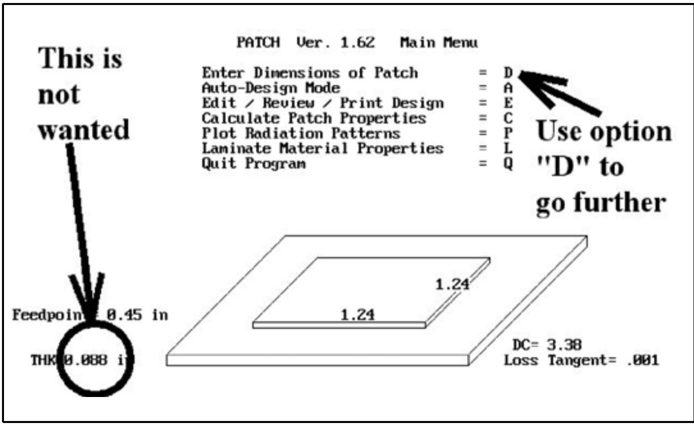


**Fig 4: The Auto-Design mode gives a PCB that is too thick. Use option "D" to change this.**

Enter these values including the data of the PCB material R04003. Once this is done the screen will be like Fig 4. Pressing "C" will calculate the patch parameters including the transformation line if required.

Now increase the patch width to get the required bandwidth of 60MHz. After optimisation the final parameters are shown in Fig 5 with the required resonant frequency and bandwidth. The final state after successful optimisation is to be seen in Fig 5, now tuned for both the resonant frequency and the range. In addition the radiation resistance for the patch edges has been computed. Looking at the main screen shown in Fig 6 the PCB thickness is now 60 MIL. The secret for doing this is revealed.

If "E" is pressed all of the design parameters are available to be edited as shown in Fig 7 and can be changed using the small menu. The patch length, patch width and feed point can be changed, always return to the main menu and press "C" to calculate the results. Finally the results shown in Fig 5 can be achieved. This does not take as long as it sounds.

All important data is summarised in following table:

- Resonant frequency: 2.450GHz
- Bandwidth: 60MHz approximately
- Gain: 6.5dBi
- Radiation resistance: At patch edge 116.88Ω
- Patch length: 1.2595 inches = 31.99mm
- Patch width: 2.2 inches = 55.88mm

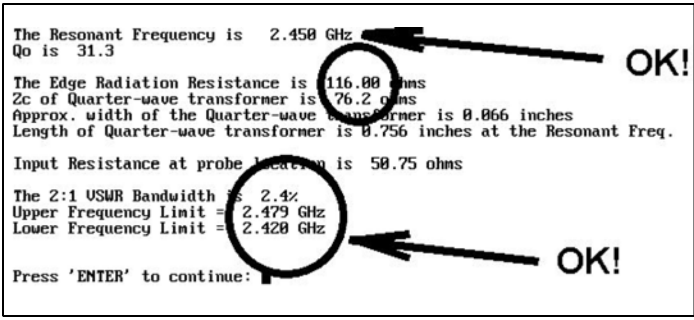


**Fig 5: The results of the "C" calculate option shows the results for the patch antenna.**

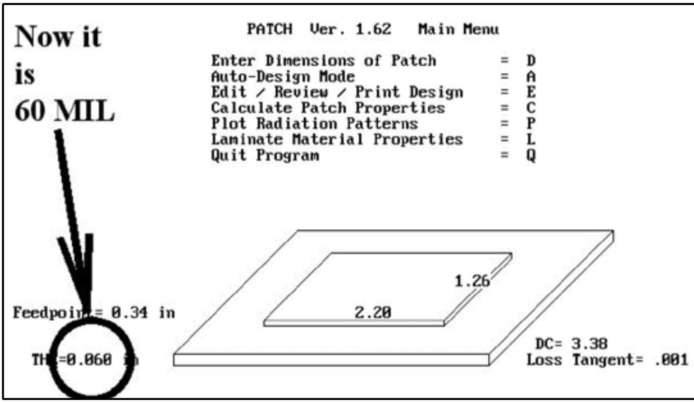


**Fig 6: The PCB thickness is now correct.**

- Distance of the feed point from the front patch edge: 0.34inches = 8.64mm
- Wavelength in air at 2450MHz: 122.45mm

This basic information is enough to be able to analyse the design with a good EM simulator. Currently the best free of charge simulator is "Sonnet Lite".

### 3.2. Preparation for the antenna design with Sonnet Lite

The software can be downloaded and installed from the Sonnet homepage (www.sonnetusa.com). When the software is started only six buttons appear on the task bar. Unfortunately the Farfield viewer is missing which is a shame. To overcome this select the Admin menu and choose the "Register Sonnet Lite" option as shown in Fig 8. Follow the prompts and increase the available memory from 1 to 16Mb which will increase the number of available tasks significantly.

Now the simulation can be started by selecting "Edit Project" then "New Geometry". After the editor is loaded, first store the project in a suitable place with a suitable name. Then carefully carry out the following steps:

***Step 1:*** In the "Circuit" menu, choose the "units" option. Ensure that millimetres and Gigahertz are selected for changes to physical values as shown in Fig 9.

***Step 2:*** Again under "Circuit" select the "Dielectric Layers" option shown in Fig 10. The PCB for the antenna is on the lower line and above it is "Layer 0" which is used by Sonnet as an air layer with a thickness of half a wavelength

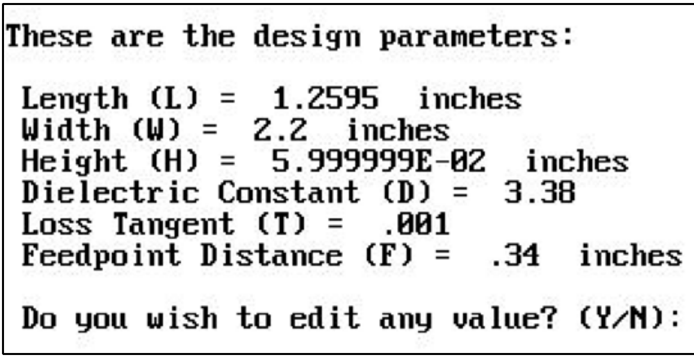


**Fig 7: The "E" (Edit) option goes to this menu where many parameters can be changed.**

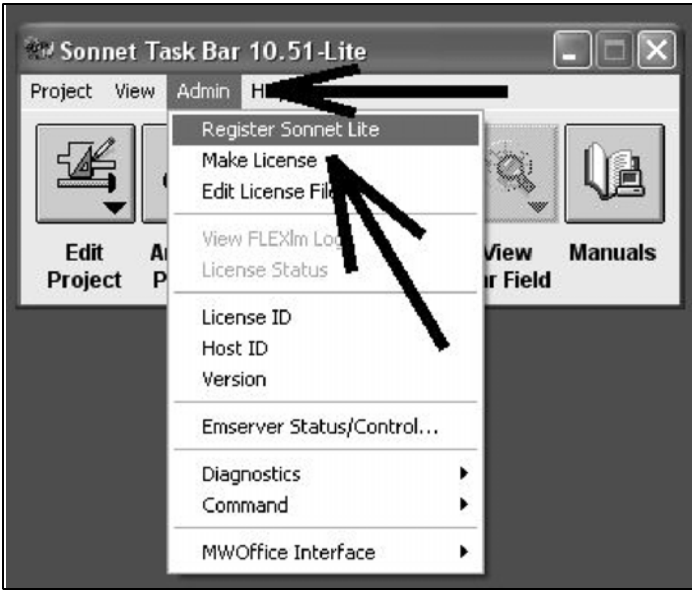


**Fig 8: Use this menu to register Sonnet Lite and increase the available memory from 1 to 16Mb.**

(61.22mm). First click on the lower line to highlight the line. Use "Edit" to change the data for the RO4003 material as shown in Fig 11. Then select the top line and edit the air layer with the following data (Thickness = 61.22mm, Erel = 1.0, Dielectric loss tangent = 0.0, Dielectric Conductivity = 0.0). The screen should then look like Fig 10. The magnetic characteristics (Mrel = 1.0, Magnetic loss tan = 0.0) take over!

***Step 3:*** Again under "Circuit " select "Metal Type", at the start there is only one entry for "lossless". Therefore click on "ADD" to open the metal editor. Click on "SELECT metal from LIBRARY" and then "global LIBRARY" which displays a metal selection menu. Choose "copper" to get the screen shown in Fig 12 and ensure that the thickness is adjusted to 0.035mm, then click "OK" to get to the menu shown in Fig 13. Change

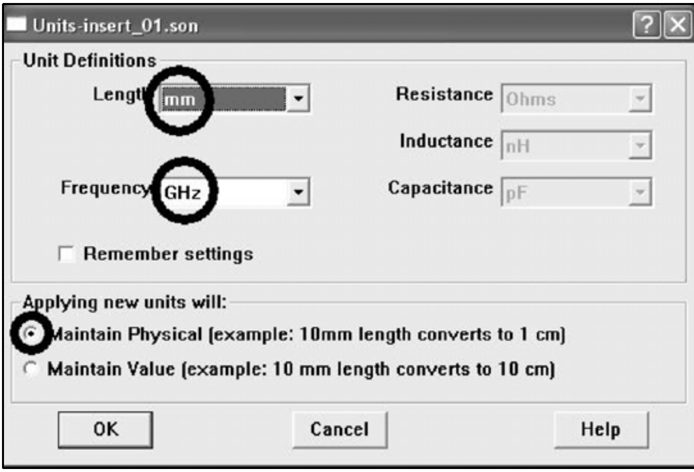


**Fig 9: Use this menu to change the dimensions to mm and frequency to GHz.**

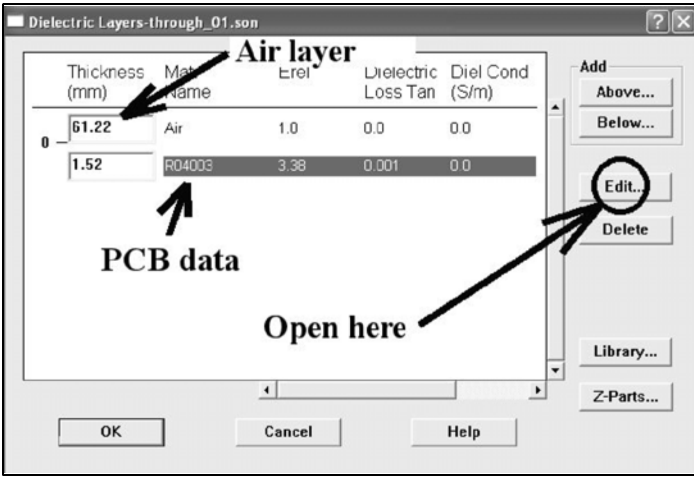


**Fig 10: Set the correct values for the PCB material (R04003) and the air layer.**

the “Metal for New Polygons” to copper and confirm with “OK”. All of the basic adjustments are now made and the antenna design can begin.

### 3.3. Antenna design with Sonnet Lite

You should be familiar with the representation of the antenna outline by coordinates related to the lower left hand corner as shown in Fig 14 because this is the convention that the Sonnet editor uses. The length (31.99mm) and width (55.88mm) are taken from 3.1. The feed point is on the centre line and 8.64mm from the lower patch edge.

Now the most important work starts:

The antenna design must be broken down in small elements for the simulation, this is limited to the point where the memory limit of 16Mb is reached.

Experience must be used to decide on the best compromise, this indicates that the length and width of the smallest cell ($\Delta x$ and $\Delta y$) are approximately 1% to 2% of

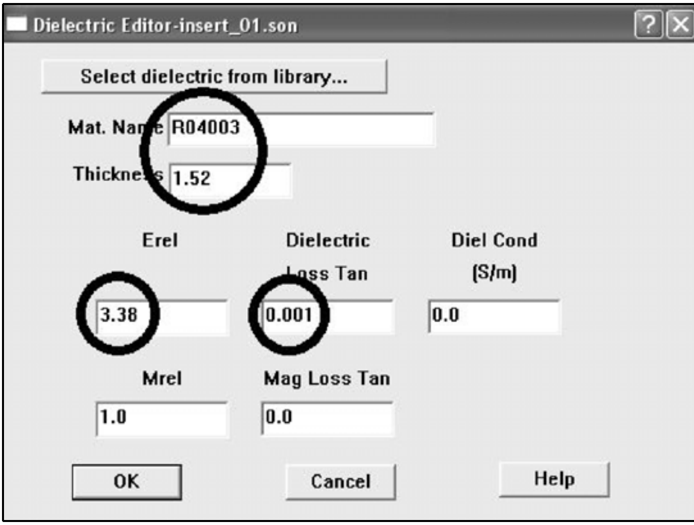


**Fig 11: Use the edit menu to set the parameters for the R04003 PCB material.**

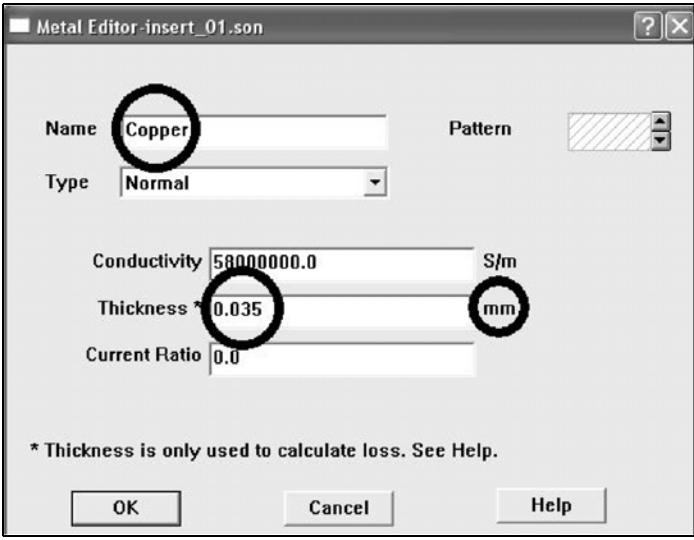


**Fig 12: Use this menu to set the correct values for the metal thickness.**

the wavelength. The pain threshold for rapidly rising simulation inaccuracies is up to approximately 5% and down to approximately 0.4%. In addition the manual suggests that the antenna edges should be at least one wavelength from the housing wall (Sonnet simulates everything in a rectangular metal box), in order to keep disturbing influences of the box wall small. But, the larger the better however the 16Mb memory limit sets the ultimate limit.

The air layer above the PCB is half a wavelength, the cover of the box forms the free space area where the radiation begins. That was taken into account in the Dielectric Layers setup.

The definition of $\Delta y$ is for the patch length. It corresponds about the half a wavelength (50% of $\lambda$) and therefore only needs the length to be divided by 50 to be 1% of the wavelength (in the PCB material, not in air!):

$$\Delta y = 31.99mm/50 = 0.6398mm$$

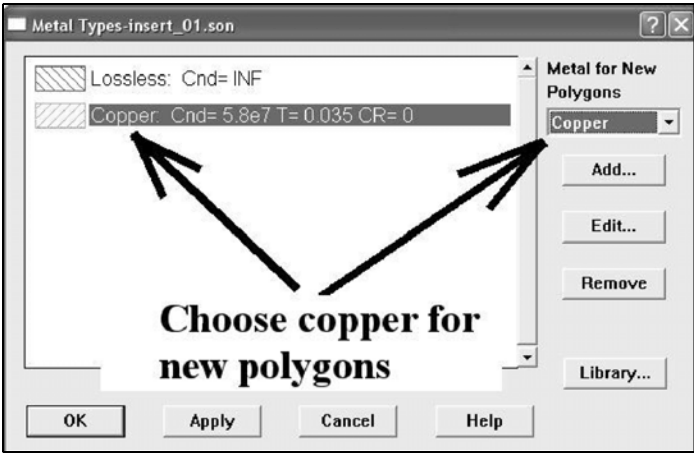


**Fig 13: Use this menu to set copper to the "Metal for New Polygons".**

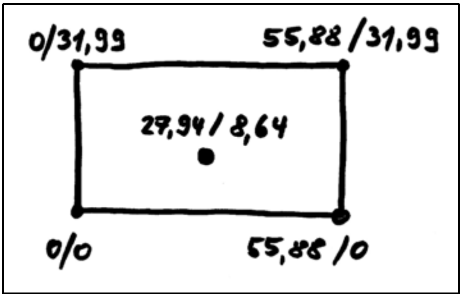


**Fig 14: You should be familiar with the way that Sonnet Lite uses coordinates by making a sketch showing magnitude and coordinates.**

If a wavelength in air of 122.45mm is added to the patch length on both sides the calculation becomes:

122.45mm + 122.45mm + 31.99mm = 276.89mm

And then:

276.89 mm/0.6398 mm = 433 cells in the y direction.

The patch width can use the same cell size, Δx, as Δy. However further thought is needed because the next step in the antenna design is to add the feed This is placed on the patch face to match to 50Ω The width of this is determined using a suitable microstrip calculator or microwave CAD program (PUFF, ANSOFT designer sports SV, Appcad etc.) using the available PCB data. All programs agree a value of 3.48mm, since no housing with covers is required PUFF or Appcad are always used.

Therefore strip line width is used to set the cell size and get the best from the antenna simulation

Thus simply divide the width value of 3.48mm into four equal parts to obtain:

Δx = 3.48mm/4 = 0.87mm

That is about 1.36% of the PCB wavelength and thus everything is in the green range for the Sonnet simulation. To determine the number of cells in the x direction:

The patch width is 55.88mm. On each side, add a wavelength of 122.45mm to get:

122.45mm + 122.45mm + 55.88mm = 300.7 mm

That results in:

300.78mm/0.87mm = 346 Cells in the x direction.

The associated “box” parameters for the

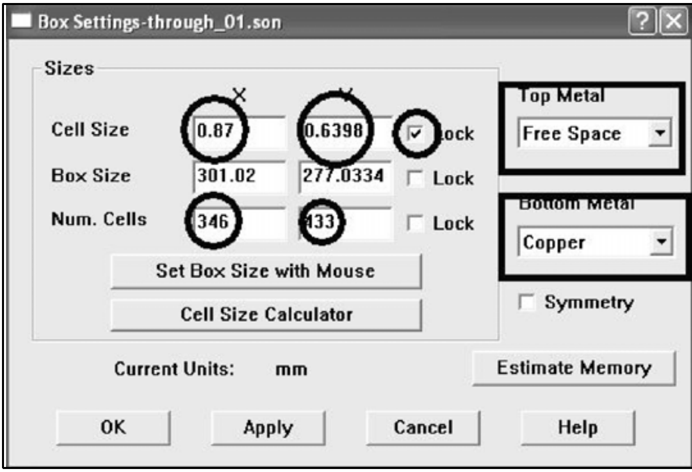


**Fig 15: These are the "Box Settings" for Sonnet Lite to operate with. The x and y cell details are shown as well as the "Top Metal" set to "Free Space"and the "Bottom Metal" set to "Copper".**

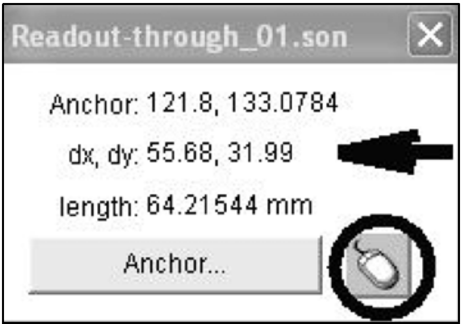


**Fig16: The easiest way to set the zero point is to us the mouse (see text).**

Sonnet circuit menu are shown in Fig 15.

- The line “Cell Size” is set to Δx and Δy with the “Lock” box checked
- The line “Num. Cells” is set to the correct number of cells for x and y
- “Top Metal” is set to “Free Space”
- “Bottom Metal” is set to “Copper”

If the cell size Δx = 0.87mm is used there is a problem because the patch width of 55.88mm is not accurately calculated. This is not so tragic because a difference in the patch width of 1mm does not make much difference but a difference of 0.1mm in the feeder width makes a big difference in the feeder impedance.

The simulation can be started by drawing the antenna but before that select “Measuring Tool” on the “View” menu and click the mouse symbol as shown in Fig 16. The cursor pointer changes to a small cross which is the zero point or lower left corner of the patch, it can, for example, be set on the centre of the display surface. The patch can now be drawn by clicking on “Draw Rectangle” and pulling with the mouse. The sizes of the two values Δx and Δy can be seen in “Measuring Tool”, these should be set to;

Δx = 55.68mm and Δy = 31.99mm.

The display surface can be zoomed to make this easier. The patch edge should be green, which means copper metallisation. If this was not set in “Metal Type” and “Material for New Polygons” it can be set by right clicking he mouse on the patch edge and using “Metal Properties” as shown in Fig 17.

Set a “Via” (plated through hole inside diameter = 1.27mm) at the point (27.84mm, 8.9572mm). The “Via” should be copper and will show in the same green colour as the patch edge.

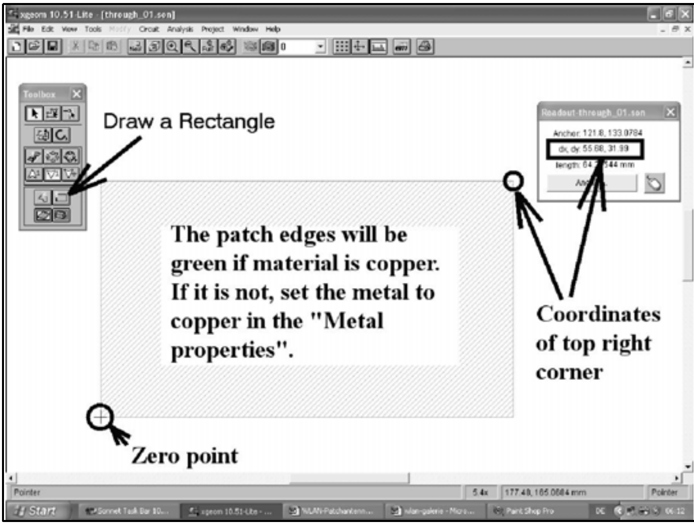


**Fig 17: All of the information about the patch is shown here.**

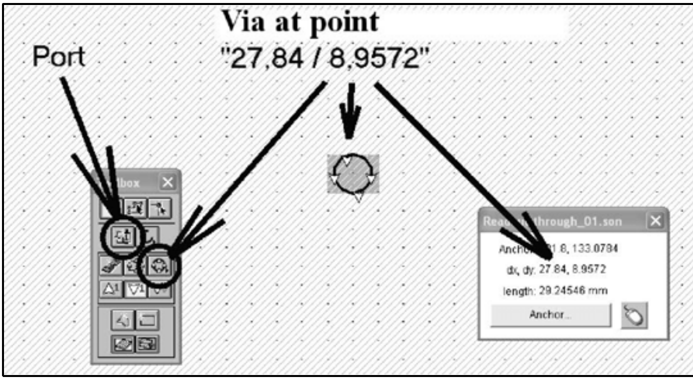


**Fig 18: Setting a Via, it must be set to Copper.**

Sometimes, that is not the case so it must be set to "Copper" with the right mouse button in the properties menu, see Fig 18. A port connected to "Ground level" must be set exactly in the centre of the Via, double click on the port and select the "Standard" attribute. The completed work should be like Fig 19.

Things are fast from now on, in the "analysis" menu there is a "Setup", Fig 20 shows the attributes necessary for "Adaptive Bandsweeps" to be simulated over the frequency range from 2.4 to 2.5GHz. Fig 21 shows the results from clicking "Project" then "Analyze" after waiting for the computing time (if the project has been correctly stored). The key to show the result is shown in Fig 22. The resonant frequency of 2449MHz

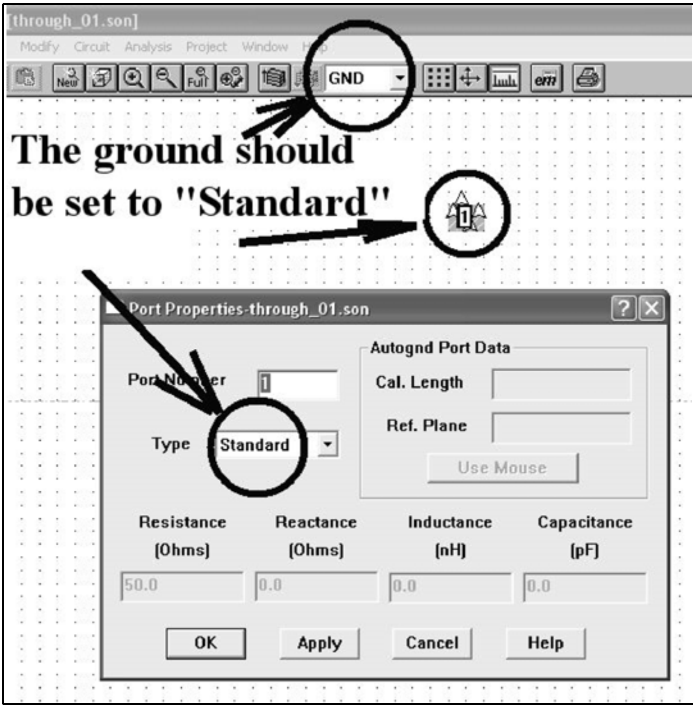


**Fig 19: Setting the ground to a Standard.**

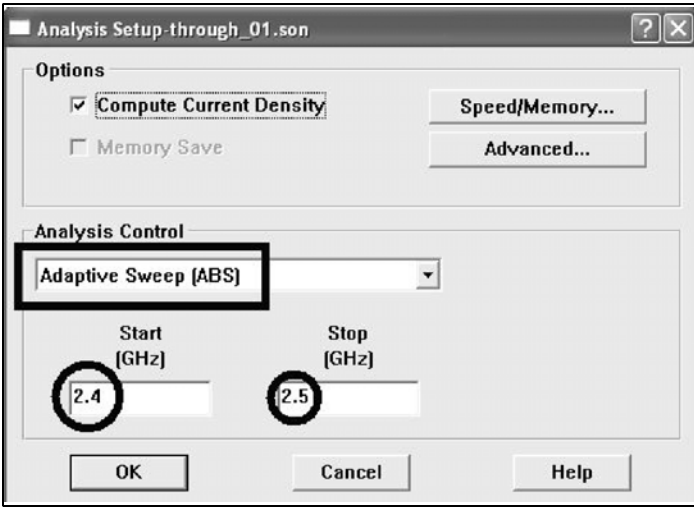


**Fig 20: Setting the analysis parameters.**

could not be better, but the S11 curve is more worrying. A Smithchart can be chosen from Fig 22 which is much more interesting and shown in Fig 23. S11 of approximately –20dB is a reflection factor of approximately 0.1, BUT the feed point is too far from the lower patch edge because the loop in the S11 curve does

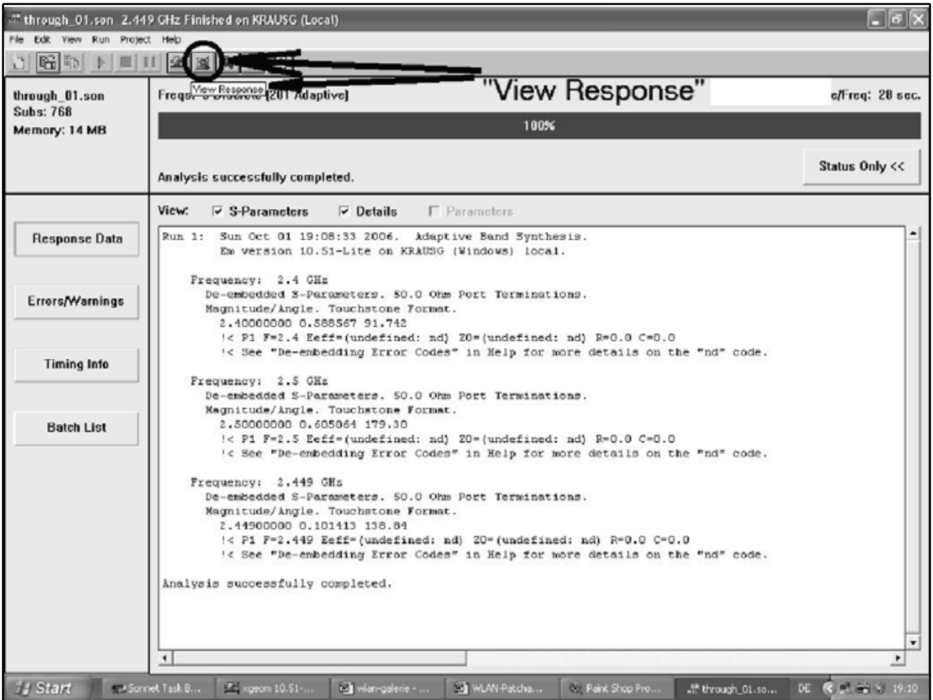


**Fig 21: The results of an analysis.**

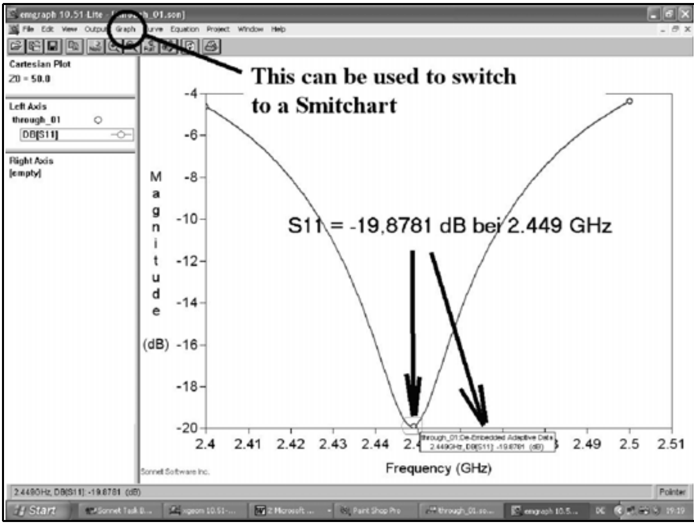


**Fig 22: The results of the simulation of the design from "patch 16".**

not enclose the centre of the Smithchart and that means that the impedance value is under 50Ω! The simulation is repeated after moving the Via to the next smaller distance of 8.3174mm using the mouse. This gives a good result with S11 = -30dB, a perfect matching point at 50Ω (Fig 24) with an acceptable increase of the resonant frequency of 2505MHz. Unfortunately the feed point can only be moved in steps of y = 0.6398mm (because we have to stay within the 16Mb memory limit) and that is simply too rough for a finer approximation. So we have to be content with this simulation result, a PCB with the external dimensions of 80mm x 60mm with the feed point as shown in Fig 25.

Because of many reasons e.g. PCB external dimensions many smaller than assumed in the simulation, manufacturing inaccuracies and edge under etching, influence of the solder track on the patch

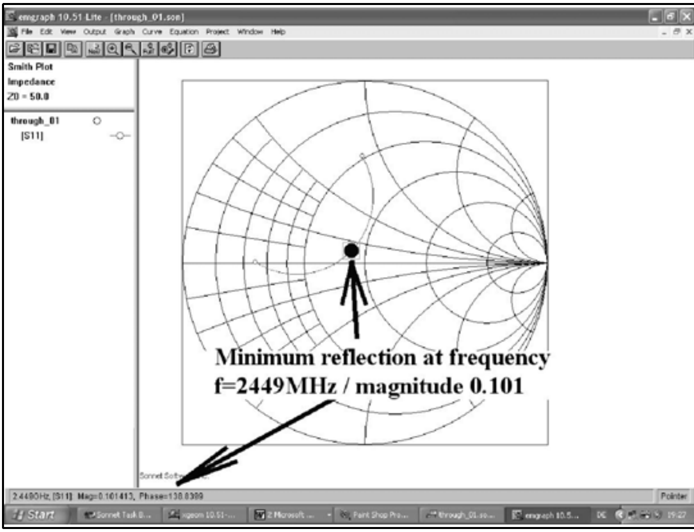


**Fig 23: The results shown as a Smithchart.**

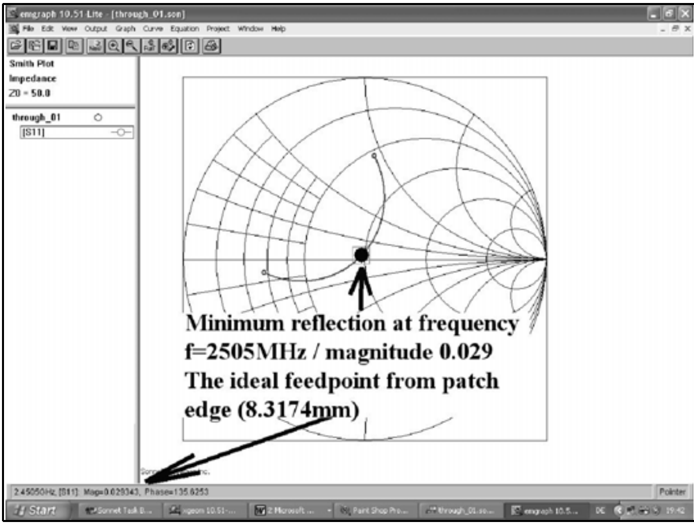


**Fig 24: Smithchart of the results after moving the feed point.**

edges, differences in the material data etc., the result of measurement using a network analysers will surely deviate from the simulation. Then a second PCB is necessary with some changes!

To be continued

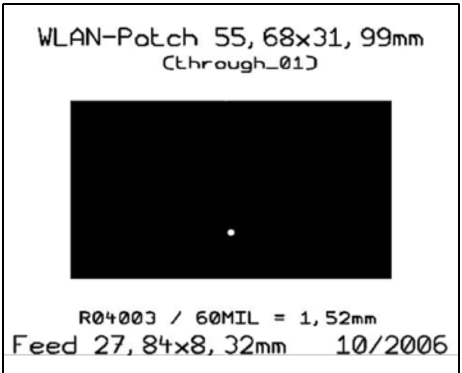


**Fig 25: Finally the finished PCB drawn using TARGET3001. Remember to add enough information to uniquely identify the PCB.**

## X.

## References for Part 1:

[1] Practical Project: A patch antenna for 5.8GH, Gunthard Kraus, VHF Communications Magazine, 1/2004, pp 20 - 29

[2] An interesting program, SonnetLite 9.51, Gunthard Kraus, VHF Communications Magazine, 3/2004, pp 156 - 178

[3] Modern Patch Antenna Design Part I and Part II, Gunthard Kraus, VHF Communications Magazine, 1/2001 and 2/2001, pp 49 – 63 and 66 - 86