

# A network analyzer

To measure the frequency response of a filter (or an amplifier), we can use a sine wave generator and an oscilloscope.

At each frequency, we have to measure the frequency, the amplitude at the input, the amplitude at the output and also the phase.

A lot of time is necessary to do that, and the measurement precision is not very good.

It is also difficult to measure a capacitor, an inductor or a resistor at several frequencies.

One solution is to use a **network analyzer**.

The network analyzer we will describe here, is mainly a sine wave generator driven by a personal computer (PC) and a voltmeter connected to the same PC.

The voltmeter is like a synchronous detector, it measures the amplitude and the phase of the signal at the frequency of the generator.

The frequency range is from 200Hz to 60MHz, the frequency resolution is approximately 0.035 Hz.

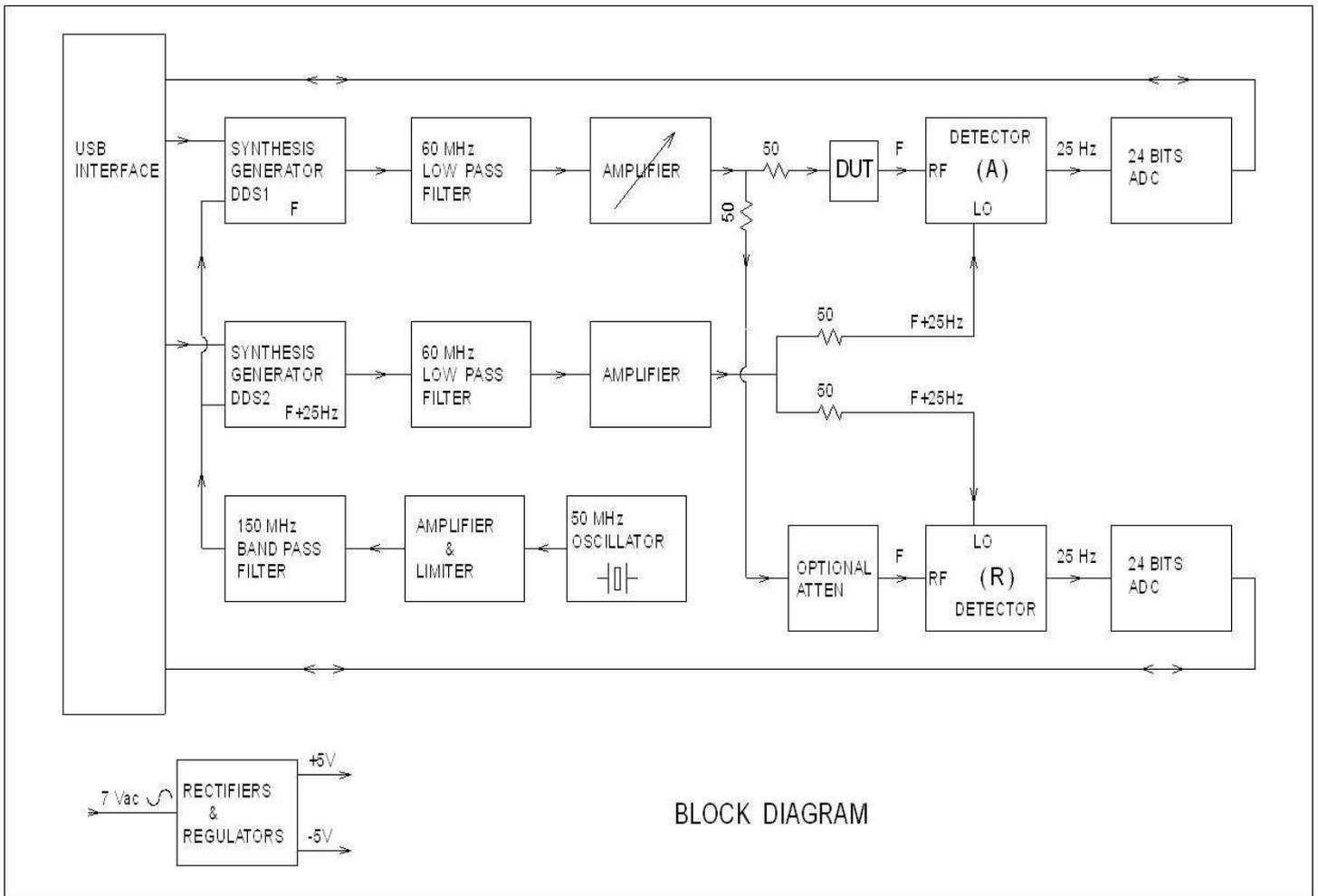
With some restrictions, it is possible to extend the frequency range down to 20 Hz.

The network analyzer measures the **gain and the phase** (or the group delay) of filters and amplifiers (versus frequency). The gain scale is linear or logarithmic (dB scale); the frequency sweep is linear or logarithmic.

A network analyzer which measures at the same time the amplitude and the phase of the gain, is sometimes called: Vector Network Analyzer (VNA).

The network analyzer measures **impedances**, it can display :

- the module and the argument (phase) of the impedance .
- the real part and the imaginary part of the impedance.
- the module and the phase of "S11"
- for a capacitor, the value of the capacitor and the quality factor (Q), with the series model or with the parallel model.
- for an inductor, the value of the inductor and the quality factor (Q), with the series model or with the parallel model.



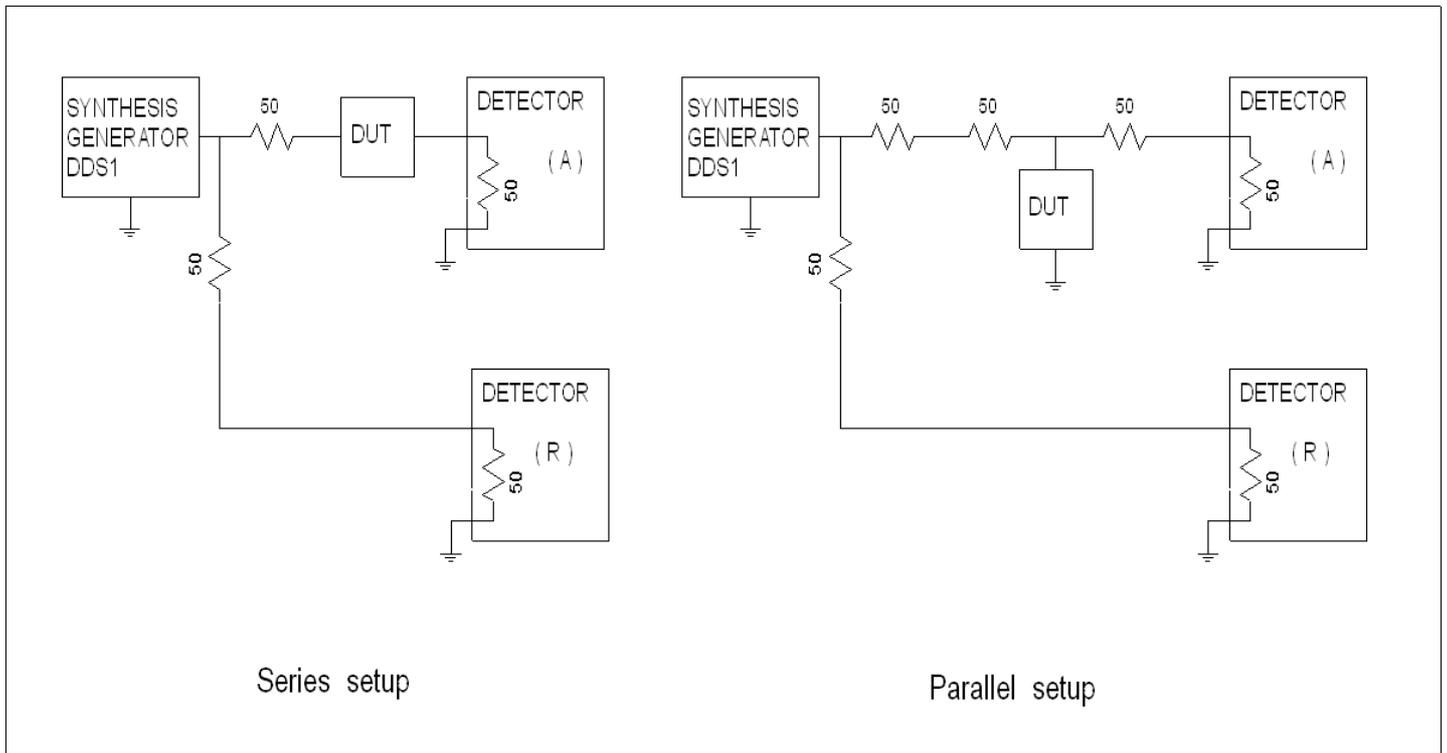
### How it works :

Two direct digital synthesis generators (DDS) , (Analog Devices AD9851), are driven by an USB port of a PC. One generator (DDS1) is at the frequency  $F$  , the second generator (DDS2) is at the frequency  $F+25\text{Hz}$ . DDS1 drives the input of the device under test (DUT); the output of the DUT is the RF input of a detector (Motorola MC1496). This RF input is the "A" input of the network analyzer. DDS1 drives also the RF input of a second detector (through an optional attenuator). This input is the "R" input of the network analyzer.

The second generator DDS2, drives the local oscillator (LO) inputs of the two detectors. At the output of the two detectors, there is a low-pass filter, the signal of interest is at the frequency of 25Hz, (the frequency difference between DDS1 and DDS2). Two 24 bits analog to digital converters (ADC) , measure the signal at the outputs of the two detectors. Then, with the comparison of the two signals at 25 Hz (amplitude and phase), the PC can calculate the attenuation (or the amplification) through the DUT; and also the phase rotation through the DUT.

The generators need a 150 MHz clock. To get this 150 MHz clock, we use a 50 MHz quartz oscillator. The third harmonic (150 MHz) is selected with a 150 MHz band-pass filter. An AC adapter (7Vac) is used to make the +5V and the -5V needed by the electronic circuits.

To measure **R, L, C** components, we have the choice between the "series" setup and the "parallel" setup.



We can put the DUT in **series** between the generator (DDS1) and the input of the detector (A).

The impedance of the DUT reduces the signal at the output of the (A) detector.

If we compare the amplitude and the phase of the signal at the output of the (A) detector with the signal at the output of the (R) detector, we can calculate the DUT impedance.

With the series setup, the measurement precision decreases when the impedance of the DUT becomes very low (compared with 50 ohm).

We can put the DUT in **parallel** with the detector (A) and the generator (DDS1).

Two 50 ohm resistors are added (next to the DUT) (these resistors are optional).

If we compare the amplitude and the phase of the signal at the output of the (A) detector with the signal at the output of the (R) detector, we can calculate the DUT impedance.

To measure very low impedance values (compared with 50 ohms), the precision is better with the parallel setup than with the series setup.

Parallel setup is the only solution to measure an impedance with a ground connection : (antenna, or input/output of an amplifier).

The measurement precision is not good if the impedance of the DUT is high (compared with 50 ohm).

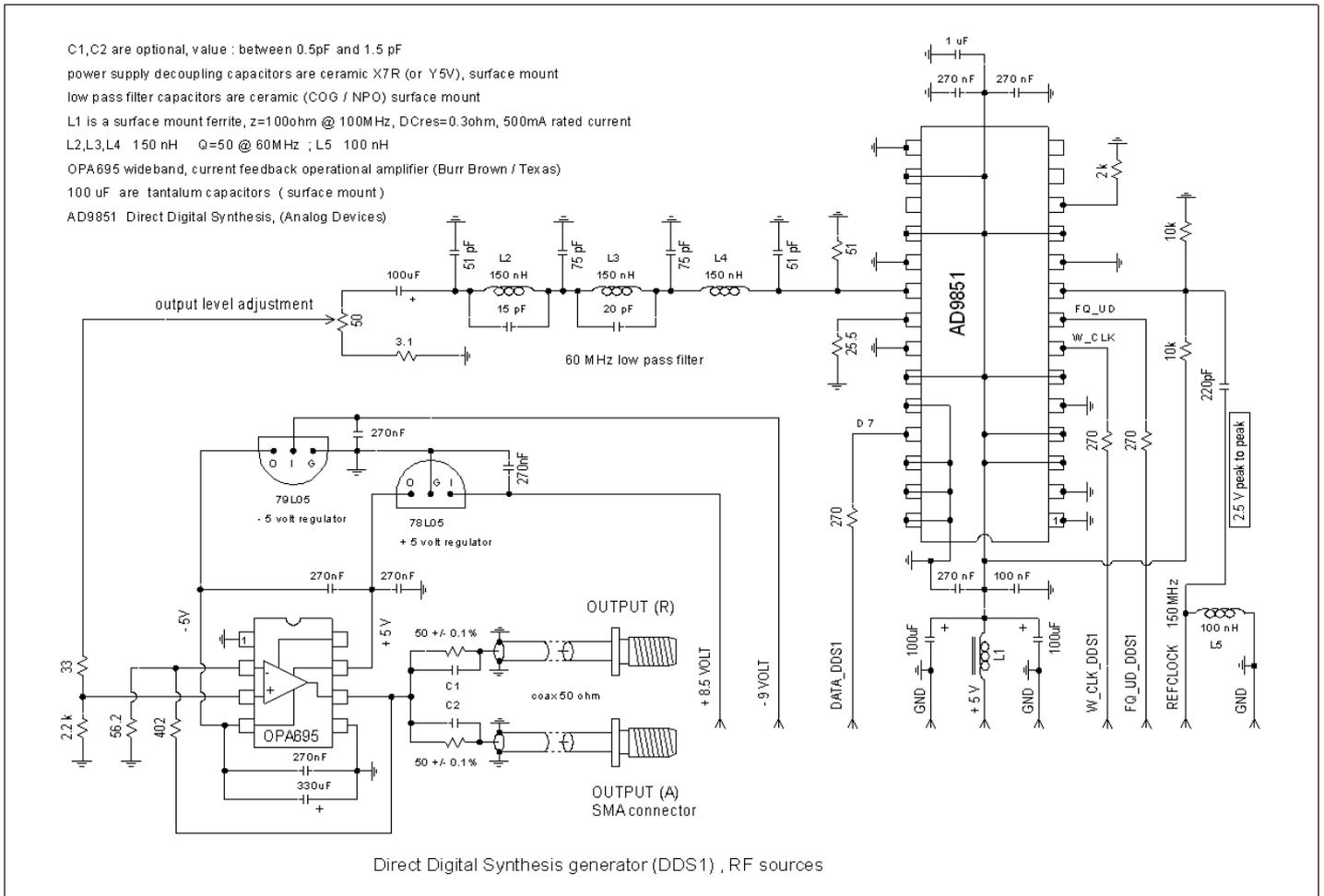
## Electrical schematics

The network analyzer is built with 8 printed circuit boards.

- a DDS1 generator
- a DDS2 generator
- a detector (A)
- a detector (R) , the (A) detector and the (R) detector are identical
- a 150 MHz clock generator
- an USB interface
- a power supply
- a clock generator to get a clock needed by the analog to digital converters

All the PC boards are connected together with wires and coaxial cables.

# The DDS1 generator



The AD9851 is the direct digital synthesis generator (Analog Devices), see the data sheet of this circuit to see how it works.

The AD9851 needs a REFCLK (here 150 MHz), 3 control lines FQ\_UD, W\_CLK, D7 and a +5volt supply. The signal at the output is not a smooth sine wave, but is like stairs.

A 60 MHz low-pass filter is necessary to get a clean sine wave.

This filter is design with a source and a load impedance of 50 ohm.

The goal of this filter is to have a good rejection of the frequencies above 90 MHz.

The capacitors in parallel with L2 and L3 help to get this rejection.

For the Q of the inductances, the higher Q at 60 MHz is the best choice.

At the output of the filter a 100  $\mu$ F tantalum capacitor removes the DC voltage.

Then a 50 ohm potentiometer is used to adjust the output level.

A wideband current feedback operational amplifier (OPA695) increases the output level and reduces the impedance.

Two high precision 50 ohm resistors send the signal to the output (A) and to the output (R).

Capacitors C1,C2 are optional ; with a network analyzer, we can find the value of C1 et C2.

**Output (A) and output (R) are DC coupled, do not put DC voltage on the outputs !**

The +5Volt for the OPA695 comes from a 78L05 voltage regulator. The -5Volt comes from a 79L05 voltage regulator. We use through hole voltage regulators because we had this parts available.

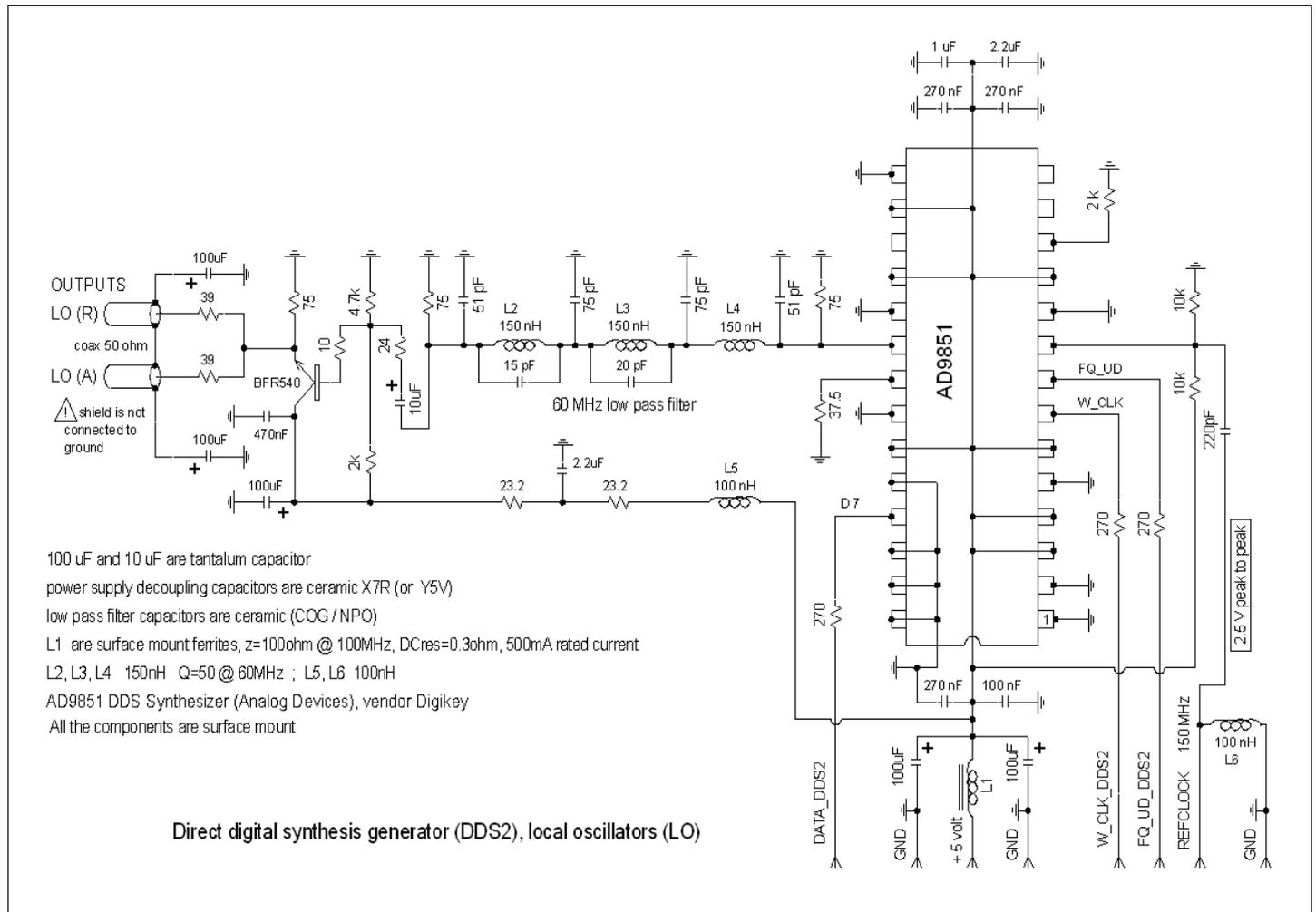
There is a 330  $\mu$ F capacitor at the output of the 79L05 regulator. This capacitor decreases a small defect on the OPA695 output impedance (at about 200KHz).

Is it a weakness of the 79L05 ? (we need to investigate). There is no pad for this capacitor on the printed circuit.

Result of the investigation in connection with the problem with the 79L05: the 79L05 that we used is a Motorola ( TO92 case). The output impedance of this regulator is high and present an instability at a frequency

ranging between 50KHz and 500KHz (the frequency varies according to the output current). This instability cannot be removed by capacitors at the input or at the output. We tested several parts, all have the problem. Solution: we replaced the 79L05 (Motorola) by a 79L05 (National Semiconductor), which is OK. The 79L05 Motorola have they all this problem? Or is it only a bad batch? In the doubt, we will choose National!

## The DDS2 generator

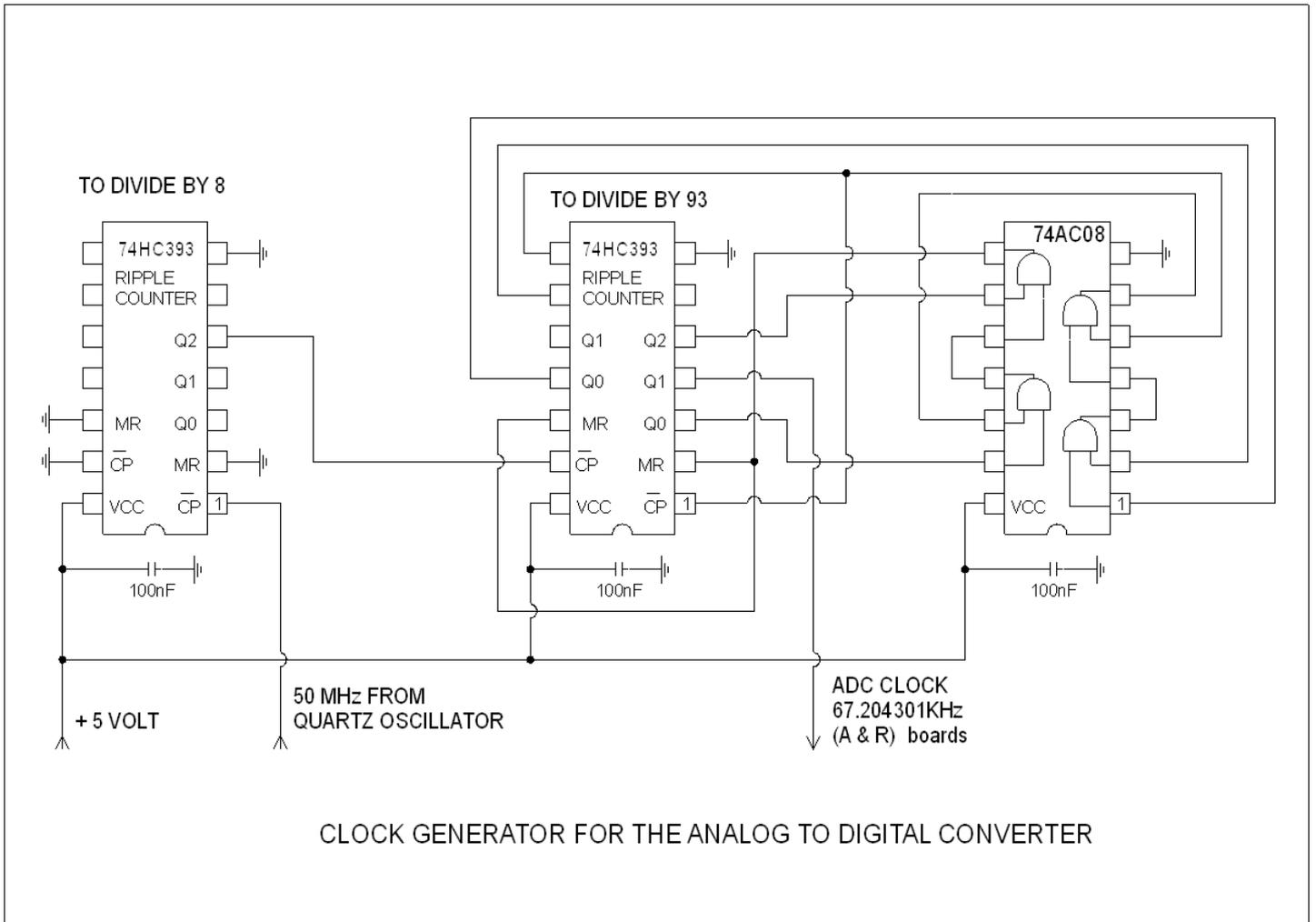


The DDS2 generator and the DDS1 generator are very similar. To get more amplitude, the filter is loaded with 75 ohms impedances. (the frequency response of the filter is not very flat, because with 75 ohms impedances, 150nH is not the best choice, 220 nH would be better, but we have only 150nH available.) At the output of the filter, an emitter follower (BFR540) is used to reduce the impedance. The outputs are connected to the local oscillator inputs of the detector boards, (A) and (R). The shield of the 50 ohm coax cables is not DC connected to ground !





# The ADC CLOCK generator



The analog to digital converters (ADS1251) need a clock.

The frequency of the clock is  $384 * \text{measurement frequency}$ .

We start with the 50MHz oscillator :  $50000000 = 8 * 93 * 384 * 25.0016001 * 7$

$8 * 93 =$  frequency divider

$25.0016001 * 7 =$  number of measurements per second

384 = needed by the ADS1251

The frequency of the ADC CLOCK is  $50000000 / (8 * 93) = 67204.3 \text{ Hz}$

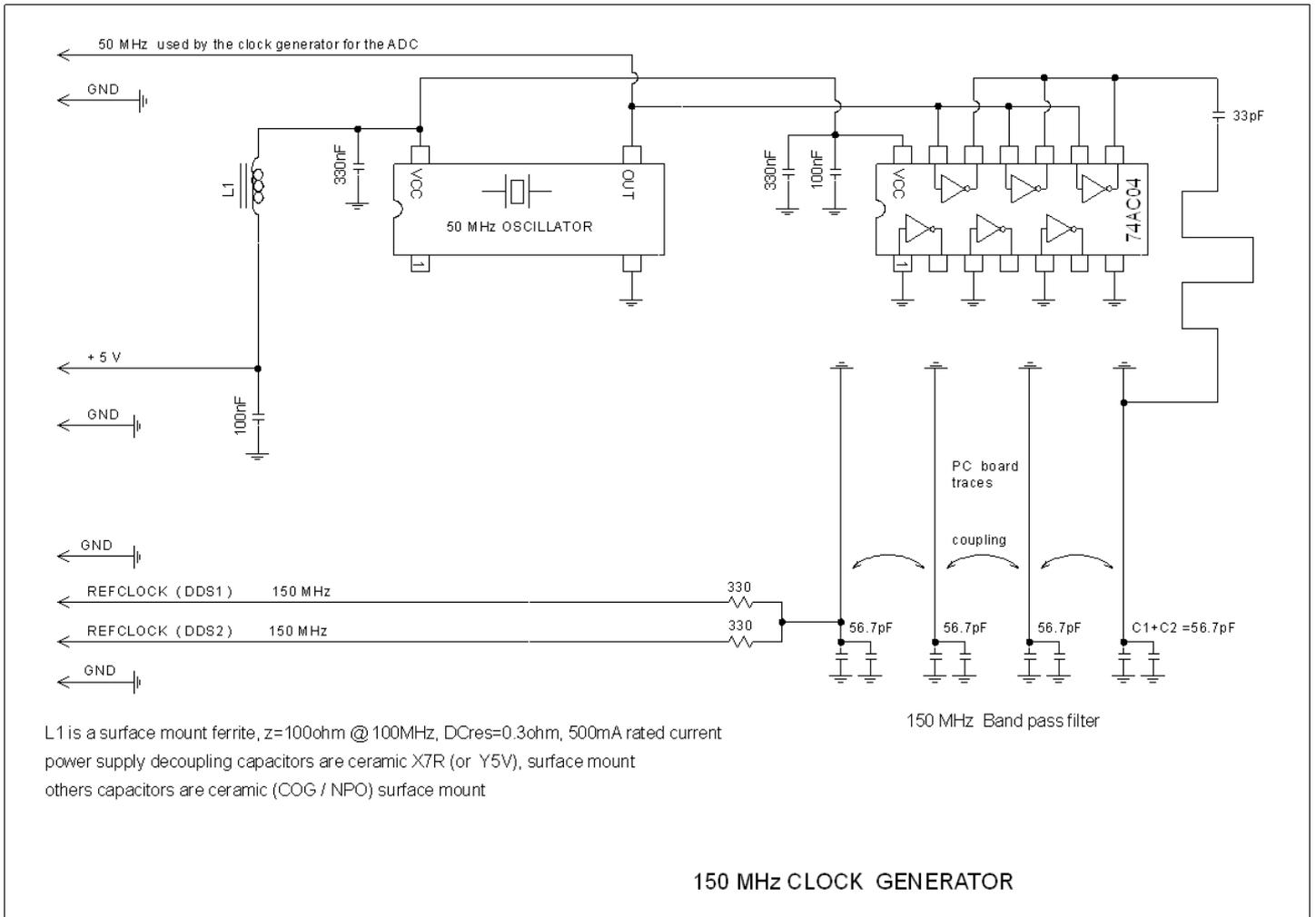
The frequency of 25.0016001Hz was chosen to help to remove the parasitic power line frequency (50 Hz in Europe).

7 is the number of measurements per period.

The first divider is a 74HC393 ripple counter. The frequency of 50 MHz is a little out of specification for the 74HC393, but typically it works; check with an oscilloscope that the divider works correctly. If you prefer, put two 74AC74 in place of the 74HC393.

To divide by 93 we use a 74HC393 counter and a 74AC08 gate to reset the counter at 93.

# The 150 MHz clock generator



The AD9851 DDS generators need a clock at the frequency of 150 MHz.

A 50 MHz quartz oscillator drives a 74AC04 inverter. Three inverters are connected in parallel to get a low output impedance.

The best is to have a duty cycle of 50% at the output of the 74AC04; if the duty cycle is 50%, there is no even harmonic in the signal.

A 150 MHz band pass filter selects the third harmonic.

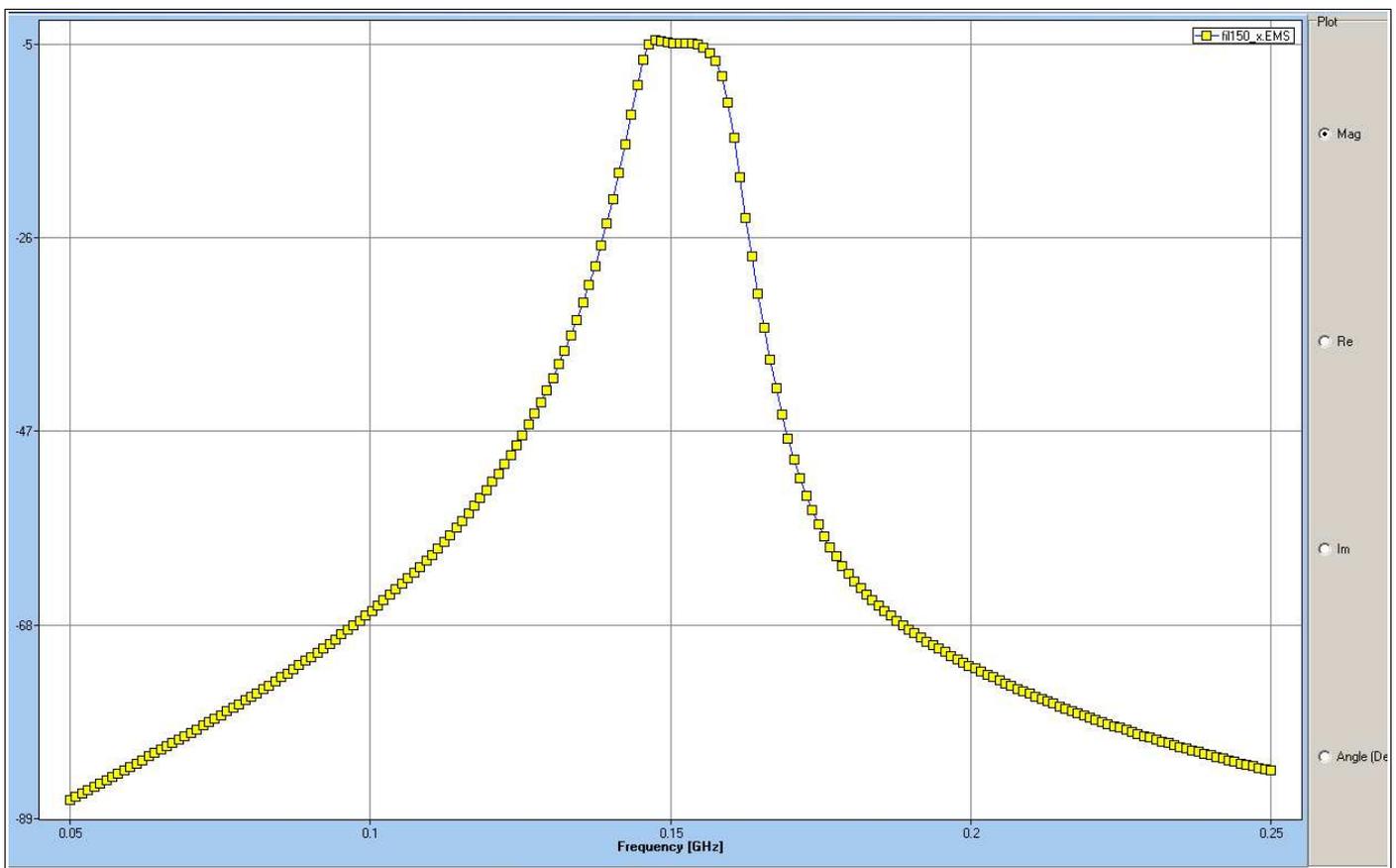
This filter is made with four resonators, these resonators use PC board traces and surface mount capacitors. A capacitor at the output of the 74AC04 is tuned with the inductance of the trace (between the inverters and the first resonator).

This filter was design with the software : « free EM3DS » <http://www.memresearch.com/download.htm>

The bandwidth of the filter is wide then it is not necessary to adjust the capacitors of the resonators .

The 150 MHz signal is connected to the DDS1/DDS2 board with twisted wires.

The amplitude of the 150 MHz signal must be greater than 2 volts peak to peak (measured at the input of the AD9851 with a low capacitance probe (< 1pF)).



Frequency response of the 150 MHz filter (dB) , (simulation).

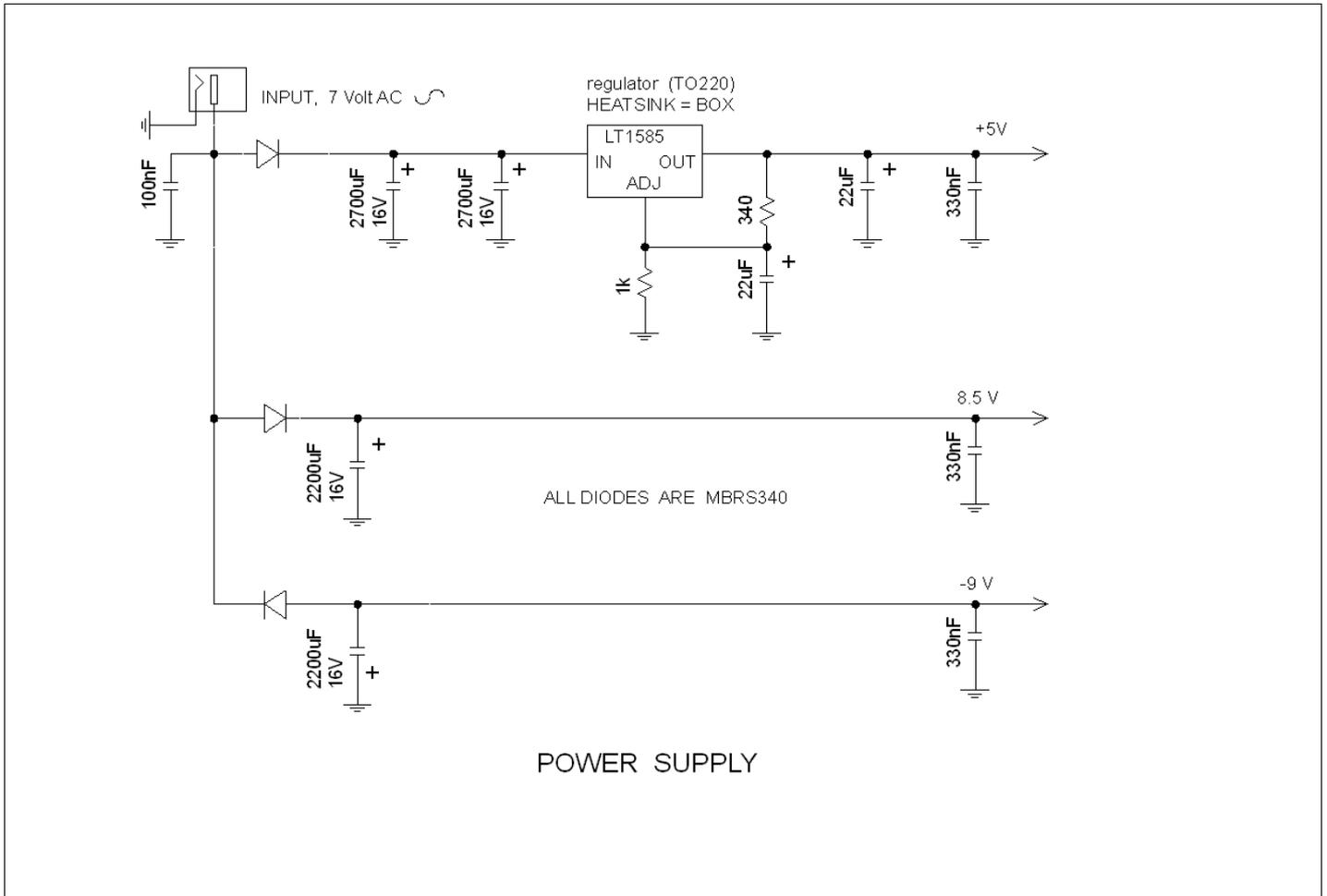
It must be possible to do not use the 150MHz generator (but we have not tried !).

Inside the AD9851, there is a Selectable 6x Reference Clock Multiplier.

We can use a 25MHz crystal quartz oscillator ( $25 \times 6 = 150$ ). Then, we have to modify the ADC clock generator (the first 74HC393 must divide only by 4). And, we have also to do a small modification in the micro-controller program (to select the 6x Reference Clock Multiplier).

The specification of the AD9851 indicates that the phase noise is higher with the Reference Clock Multiplier enabled. But we do not know if this phase noise can modify the dynamic of the analyzer ?

## The power supply



The power supply uses a small external 7 volts (rms) AC adapter (7 volts measured without load).

The diodes are 3 Amperes schottky rectifiers (surface mount).

The +5 volts regulator (LT1585) uses the box as heat-sink (it is necessary to have an electrical insulation between the regulator and the box).

## The software

Do not expect a beautiful program , the capabilities of the programmer are very limited !

All the programs are written in basic language (freebasic). <http://www.freebasic.net/>

A linux version and a windows version are available.

There are four programs :

Zmeasure.exe : to measure the impedances of the resistors, capacitors ..etc

Fmeasure.exe : to measure the insertion gain of the filters or amplifiers.

Zdisplay.exe : to display later on the measurements made with the Zmeasure program  
Fdisplay.exe : to display later on the measurements made with the Fdisplay program.

**Video mode :** The programs use a windows of 1024\*768 pixels.

It is not possible to resize the windows.

If the PC has a 1024\*768 screen, it is better to use the full screen mode (with alt + enter).

## USB FT232R driver installation

**windows :** install the FT232 driver, choose the Virtual Com port (VCP) driver

<http://www.ftdichip.com/Drivers/VCP.htm>

Connect the USB cable and the power supply of the analyzer.

In the windows menus, find the COM port address of the FT232R USB interface (COM3 or COM4 ..etc)  
(start > run > devmgmt.msc )

Configure the COM port : 57600 bits/s , 8 bits, 1 stop bit, no parity .

Run Fmeasure.exe and press the F2 key to write the COM port address.

**Linux :** (with the UBUNTU distribution)

We do not need to install a driver; but it is necessary to remove a braille program : brlty

Type the command : sudo apt-get remove brlty , or with synaptic remove the brlty program.

Connect the USB cable and the power supply of the analyzer.

In the directory /dev/ there is a file named ttyUSB0 or ttyUSB1 or ttyUSB2 ...etc

Run Fmeasure.exe and press the F2 key to write the named of this file (ttyUSB0 or ttyUSB1 or ...).

To get more information : <http://www.arduino.cc/>

The name of the board is diecimila.

The first time we run Fmeasure or Zmeasure , a small file named CONFIGU is automatically created.

This file contains :

- the COM port address (windows) or the name of a file (ttyUSB0 or ttyUSB1 or ...) (linux)
- the value of the reference resistor used for the calibration
- the value of the parasitic series inductance of the reference load
- the value of the parasitic shunt capacitance of the reference load
- the value of the amplitude and phase mismatch between the input (A) and the input (R).
- the exact value of the clock frequency (about 150 MHz)

The CONFIGU file is not a text file, you can write in this file through Fmeasure or Zmeasure.

## A Help for Fmeasure and Zmeasure

- To start or stop a scan, press the space bar or use the «start/stop» button; when the scan is stopped, you can access the menu.
- To move across the menu, use the mouse .
- To change a number, put the mouse cursor on this number then type the new number + “enter”
- To increment/decrement a digit of a number, put the mouse cursor on the digit, then move the mouse scroll wheel.

- To move the markers use the mouse.
- The frequency sweep range can be set with "**freq max**" and "**freq min**" or with "**center**" and "**span**".
- To copy **marker1** to Freq Min and **marker2** to Freq Max, type CTRL+V (it's like a zoom function, but you have to scan again). If marker1=marker2 then CTRL+V copies marker1 to Freq Center.
- **The number of measurements per scan** «meas/scan» can be set at the following values : 51, 101, 201, 401, 801. The default value is 401.  
When you change the «meas/scan» value, the calibration (normalization) is cleared and the measurement memory is cleared.  
On the graph, the measurement points are connected by the mean of straight lines.  
The markers indicate the measurement values of the nearest measurement point (There is no interpolation).
- The **measurement time** (for each frequency point) can be set between 40ms and 5120ms. The possible values are : 40, 80, 160, 320, 640, 1280, 2560, 5120 .  
When the sweep time increases, the noise is reduced, and the dynamic range increases. At each point, before the measurement, there is a stabilization time (40ms).  
The sweep time is function of the number of measurement points per scan and the measurement time at each frequency point.  
With meas/scan = 51 the sweep time can vary between 4 secondes and 263 secondes.

"	= 101	"	8	521
"	= 201	"	16	1037
"	= 401	"	32	2069
"	= 801	"	64	4133
- **The dynamics range** : if we check the dynamics range of the analyzer, we can see that , above 10 MHz, the dynamics range decreases.  
The root cause of this problem comes from the local oscillator and the leakage inside the detectors. To solve this problem, add an external 20db attenuator on the (R) input, then normalize.  
A modification of the DDS1 generator could decrease this problem.
- **Calibration** :with the program Zmeasure, if we use only the "**series set up**" , it is possible to measure components without calibration, but calibration improves the precision.  
With the "parallel setup" it is necessary to calibrate.  
To calibrate , set the "freq max" and the "freq min", select "**calibrate asked**", then use the "start calibrate" button to start a scan, then follow the indications written onto the screen.
- **Normalize** : with the program Fmeasure , it is possible to measure filters without normalization , but normalization improves the precision (some network analyzer imperfections are removed). To normalize : replace the DUT by a through connection, in the setup choose "**normalize : asked**" then use the "start normalize" button to start a scan. When the scan is finished, remove the through connection and put the DUT, now the new measurements will be normalized.  
Normalization is not a full calibration, because it does not correct the errors of the input and output impedances of the analyzer. But full calibration is difficult and take a lot of time.  
The input and output impedances of the analyzer are close to 50 ohm (see the measurements).  
It is recommended to calibrate (normalize) each time we change the start or the stop frequency but it takes a lot of time; if we don't need precision, very often we can measure without calibration (normalization).
- When a scan is finished , with the button «**Copy measure to Memory**», we can put a measurement to a memory . On the screen, we can **display** simultaneously the content of the memory and a new measurement. It is useful to compare two components, or to see the effect of an adjustment.

- length** : it is possible to add an electrical length (positive or negative) in series with the DUT. It is useful only if we want to measure a component without normalization or calibration. "length" allows to compensate the difference of length between the (A) channel and the (R) channel.  
To find the length value, use Fmeasure, replace the DUT by a through connection , when a scan is finished adjust the length value to have the smallest phase (between "fmin" and "fmax") . To do that, put the mouse cursor on a digit of the length value, then use the mouse scroll wheel to adjust the value of "length".
- Save measure to File** : when a scan is finished, we can save the measurement in a file.  
We can write a bmp file (this is a copy of the analyzer screen window).  
We can also write the numerical measurement data in a file with the extension .s1p or .s2p  
With these file formats (Touchstone), we can import our measurements into a S-parameter simulation software (Qucs, RFSIM99 or PORTVIEW for example)  
With our programs Fdisplay (for the filters) and Zdisplay (for the impedances) , we can display our measurements . The files with the extension s1p and s2p must be in the same directory than Fdisplay and Zdisplay.  
To save the measurement of a one port device, we choose the s1p file format. This file is a text file with 3 columns, the first one is the frequency (Hz), the second one is the s11 amplitude (dB), the third one is the s11 phase (degrees).  
To save the measurement of a two ports device , we choose the s2p file format. This file is a text file with 9 columns, nb 1 is the frequency (Hz), nb 2 is the s11 amplitude (dB), nb 3 is the s11 phase (deg), nb 4 is the s21 amplitude (dB), nb 5 is the s21 phase (deg), nb 6 is the s12 amplitude (dB), nb 7 is the s12 phase (deg), nb 8 is the s22 amplitude (dB), nb 9 is the s22 phase (deg).

This network analyzer can only measure one parameter at a time.  
With Fmeasure, we measure s21 (forward insertion gain), and we save the data in a s2p file.  
We swap the input and the output of the DUT to measure s12 (reverse insertion gain), and we save the data in the same s2p file.  
With Zmeasure, we measure s11 (at the input) and we save the data in the same s2p file; then we measure s22 (at the output) and we save the data always in the same s2p file.  
When we measure s11, it is necessary to put a 50 ohms load at the output.  
When we measure s22, it is necessary to put a 50 ohms load at the input.  
If the DUT has only passive components then  $s_{21}=s_{12}$ .  
To put in a same file all the s-parameters of a two ports device, it is necessary to keep the same start/stop frequencies for all the measurements, and to keep the same number of measurements per scan.
- Bar graph indicator** : on the bottom right of the screen , there is a bar graph indicator.



This indicator shows the signal level on the (A) input and on the (R) input. We can adjust the signal level with the potentiometer . Do not exceed 7dBm , to have a good dynamic range set the signal level at +6dBm, to have a good precision reduce the signal level at 0 dBm.

The peak to peak voltage at the output of the OP279 must be less than 4volts

With the Fmeasure program, we can display the signal level on the (A) and (R) inputs versus frequency. To do that, in the "display" possibilities chose : **dB, A,R**.

If we choose "fmin" =1 MHz and "fmax"=99.99 MHz (and a linear sweep), we can see the effect of the 60 MHz low pass filter of the DDS1 generator. The amplitude at the output of a DDS generator varies according to the formula  $a(f)=\frac{\sin(\pi*f/f_{clock})}{(\pi*f/f_{clock})}$  with  $f_{clock}=150\text{MHz}$ .

The amplitude decreases when the frequency increases ; at 60 MHz the decrease is 2.4dB ; this decrease is added to the attenuation of the 60 MHz low pass filter.

Above 60MHz, the amplitude of the local oscillators decreases quickly ; a signal at (150MHz-f(DDS)) is coming within DDS1 and DDS2 signals (see the AD9851 specification).

Above  $f_{clock}/2$  (75MHz), the amplitude increases again, now the main signal is the signal at (150MHz-f(DDS)).

- **The group delay** : with the Fmeasure program we can measure the group delay versus frequency. The group delay ( $\tau(\omega)$ ) measures how much the phase is increasing (or decreasing) when the frequency is increasing,  $\tau(\omega) = -\partial\theta(\omega)/\partial\omega$  with  $\omega = 2\pi f$ ,  $\theta(\omega)$  is the phase.  $\tau(\omega)$  is measured in second. For filters, it is sometimes useful to have a constant group delay (in the pass band of the filter). To calculate the phase change, it is necessary to select two frequency points ( $f_1$  and  $f_2$ ), ( $f_2$  at a small distance of  $f_1$ ) and to measure the phase  $\theta_1$  and  $\theta_2$  at  $f_1$  and  $f_2$ . The group delay is calculated as  $-(\theta_2 - \theta_1)/(2\pi(f_2 - f_1))$ . The distance between  $f_1$  and  $f_2$  is the **delay aperture**, ( delay aper in the menu of Fmeasure). The delay aperture can be adjusted between 0.25% and 16% of the full scan (the lowest limit varies with the number of measurement points per scan). The best is to select the lowest value, but if the trace is too noisy, it is necessary to increase the value of the delay aperture. For the cables, which have a frequency response relatively flat, the group delay represents duration that the signal puts to be propagated along the cable; but for a filter this is not true; the group delay can be positive or negative . Inside the delay aperture, if the phase increase (or decrease) is more than 180 degrees then the measurement is wrong ( it is necessary to reduce the delay aperture or the frequency span).
- **Source and load impedances** : to define the frequency response of an amplifier or a filter, it is necessary to specify what is the source impedance and what is the load impedance. The network analyzer has a 50 ohm source and load impedance, to work with different values of impedances it is necessary to add resistors in series or in parallel. An other possibility is to measure the 4 s-parameters of the 2 ports device (s11, s21, s12, s22) with a 50 ohms source and load impedance; save the measurement in a s2p file. Open this file with an S-parameter simulation software (Qucs, RFSIM99 or PORTVIEW), then adjust the source and load impedances to find the best values.
- When a scan is finished we can stop the scan with the space bar or the stop button. After that , choose the parameter to be measured : gain, impedance, S11 etc ; adjust the scales to your needs. If the frequency scale is modified, press the space bar to start a new scan.
- **To setup the scales** , we can use : k = kHz , m = MHz , k = k $\Omega$  , m=M $\Omega$  , u =  $\mu$  , n = nano , m = milli
- **The linearity correction of the modulators (MC1496)** : when the amplitude of the signal on the inputs (A or R) increases, the gain of the modulator decreases. To see this gain decrease : put a 30dB attenuator on the (R) input and no attenuator on the (A) input; the signal on the (R) input is low, the gain on this input does not change , but on the (A) input we can see that the gain decreases (a little bit) when we increase the amplitude of the signal. To correct this linearity defect : by calcul, we add a linearity correction . This correction is proportional to the amplitude<sup>2</sup> . Today, we can modify this linearity correction only in the source code. When the signal at the input is 6dBm, the correction is 0.0208dB  
When the signal at the input is 0dBm, the correction is 0.00486dB
- **"A/R" gain variation with the temperature.**  
We built two analyzers. The first analyzer is built with two AD9850 DDS generators. The AD9850 is specified at 125MHz (max clock frequency), but we run it at 150MHz. The second analyzer is built with two AD9851 DDS generators. The AD9851 is specified at 180MHz. With the first analyzer, after startup, the "A/R" gain varies with the self-heating of the analyzer.

Four hours later, the temperature is stabilized, and the "A/R" gain variation is about 0.003dB. With the second analyzer, the gain variation is lower.

- **To measure the amplifiers distortions :**

Today, the program to measure the second and the third harmonics is not written.

It seems that the hardware does not need modification to do this measurement.

The harmonics of the 25Hz are removed by the Fourier Transform (the harmonics 2, 3, 4 and 5); the harmonics 6 and higher are removed mainly by the ADS1251 and also by the low pass filter.

The main modification is to change the programming of the DDS2 generator and the menus on the screen.

One thing is needed, that is the courage to do that !

## **The printed circuit boards**

All the PC boards have the traces on the top, and a ground plane on the bottom ; except the power supply board which is a single face PC board.

The thickness of the PC boards is 0.8 mm for the DDS1 and DDS2 generators, but it is possible to use 1.6 mm PC boards without problem .

The thickness of all the other boards is 1.6 mm ; the material is FR4. For the 150 MHz band pass filter, the design need to be modified if we change the thickness of the board or the material.

The connections from the traces (on the top) to the ground plane (on the bottom) are made with some small wires soldered.

To make all the PC boards we used the toner transfer method. We need a laser printer, a copper clad board, a clothes iron and some Epson glossy photo paper for inkjet printers ;

The etchant is a ferric chloride solution. During the etching process we pass a brush on the board to speed up the process and to make uniform the etching. We use acetone to remove the toner.

Most of the time, the traces are 0.635mm wide (or more), and there is no problem with the "iron method" ; around the AD9851 , the wide of the traces is 0.3mm, be careful.

The two detector boards are identical. All the components on these boards are on the traces side ;

1 $\mu$ F and 220nF polyester low pass filter capacitors are through hole components but they are soldered as surface mount components. The 78L05 and the 79L05 regulators are through hole components, but they are soldered as surface mount components.

To help, the layout and the electrical schematic are very similar.

The two generators (DDS1, DDS2) : to solder the AD9851 , we put a lot of solder ; the solder creates short circuits between the pins, then we remove the short circuits with a de-soldering braid.

On the DSS1 board there is a potentiometer, this potentiometer is soldered as a surface mount component. This potentiometer is adjusted with a small screw driver (from the outside of the box, via a hole in the box).

The 150MHz clock board : the 50MHz quartz oscillator is a through hole component, there are some cuts in the ground plane (around the VCC and the output pins) to isolate these pins from the ground plane.

A small wire is used to connect the output of the oscillator to the inputs of the 74AC04.

A small wire is used to connect the VCC of the oscillator to the +5V ; these wires are on the ground plane side. (Perhaps, if you modify the printed circuit board, you can put the quartz oscillator on the other side (ground plane side) of the printed circuit board.)

On the USB interface, the micro-controller, the quartz crystal and the USB connector are loaded on the ground plane side.

To isolate the pins of these components (from the ground plane), we use a milling bit, the shape of the milling bit is a ball.

All the PC board ground planes are electrically connected to the metallic box with the fixing screws.

## The measurements :

The first time that we run the analyzer , connect the output (A) to input (A), and the output (R) to input (R) , (with the short cables).

Run the Fmeasure software.

Choose to measure : gain (U) vs frequency ; scan from 1kHz to 60MHz

The gain must be nearly constant and probably between 0.9 and 1.1

If we disconnect the cable on the (A) input , the gain must be zero.

Now, to correct the bad symmetry between the (A) channel and the (R) channel, press the "F5" key and follow the indications written onto the screen.

By this way, we correct the lack of symmetry " not frequency dependant ".

The " length " correction allows to compensate the difference of length between the (A) channel and the (R) channel.

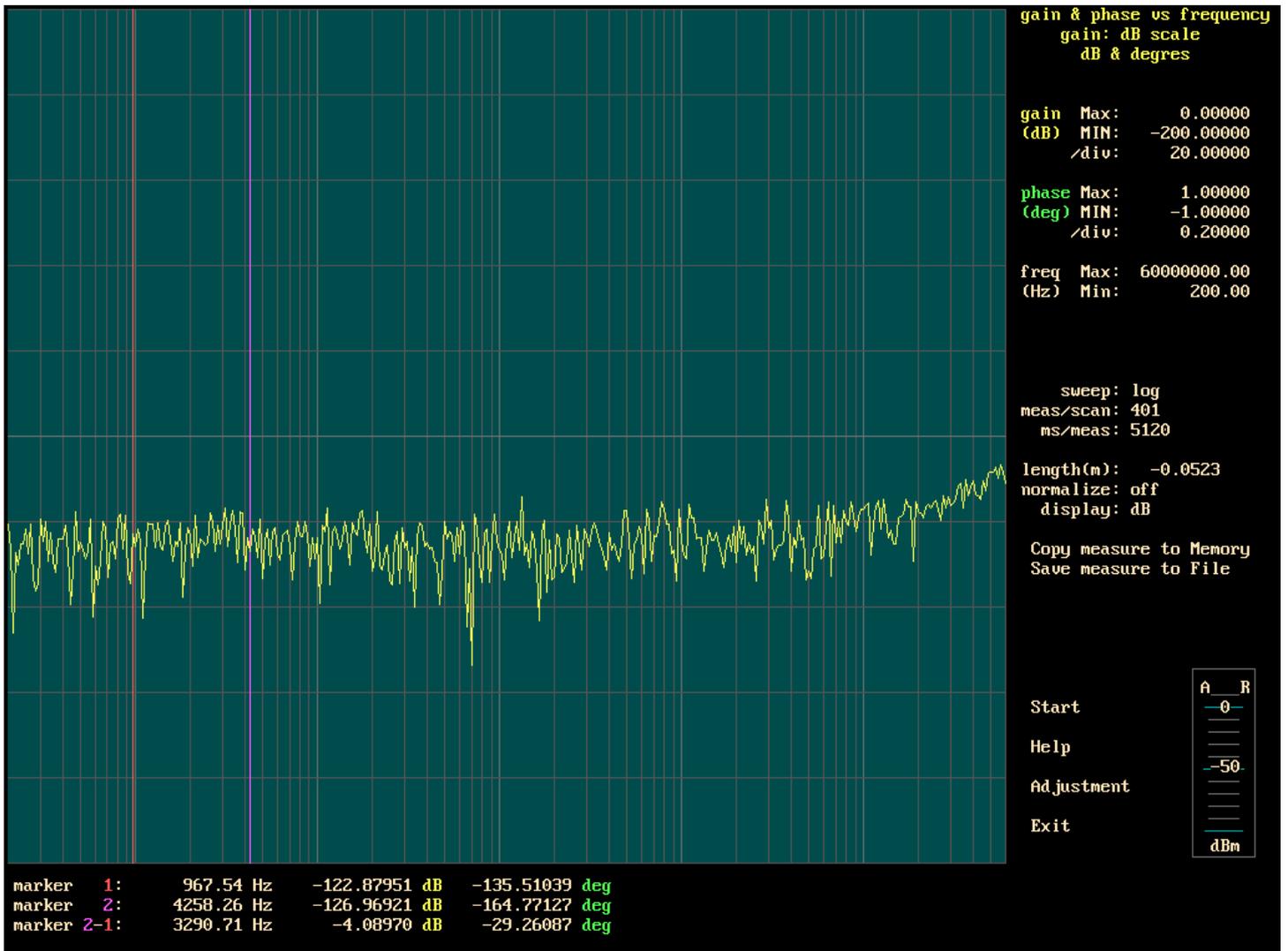
The symmetry correction and the length correction are not necessary if we use the calibration.

To check the precision of our measurements, we used 0,1% resistors and 1% capacitors .

When the frequency is higher than 10 MHz , it is more difficult : we have to find the value of the parasitic elements of the reference load (used for the calibration) to get the best correlation with capacitors and inductances available data.

A capacitor 's manufacturer (AVX) has a program (SPIMIC) that gives the quality factor (Q) of surface mount capacitors at several frequencies.

Some programs calculate the inductance and the quality factor of air coils (at several frequencies).

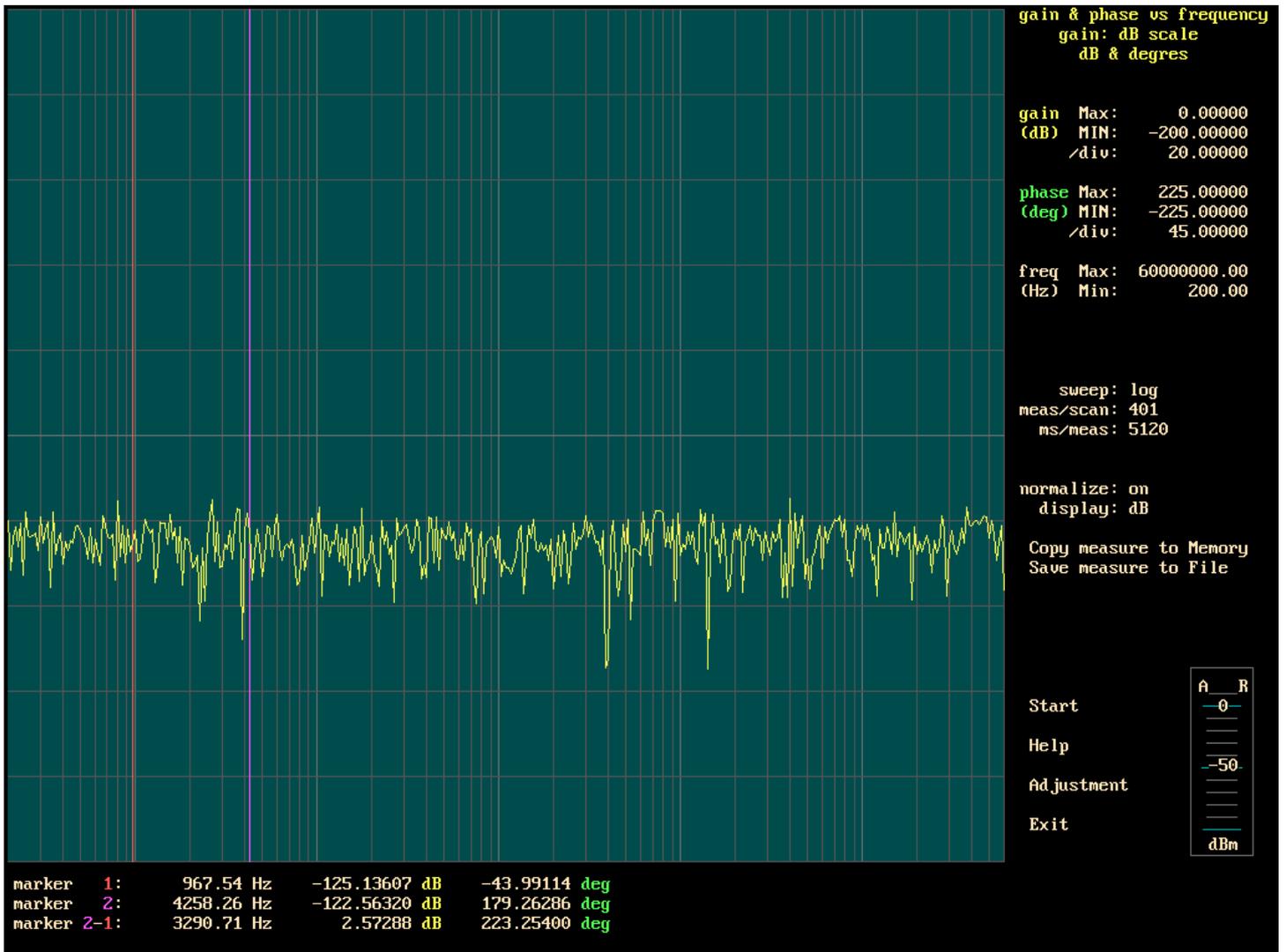


### Dynamics range of the analyzer (measured at the slowest speed).

The signal on the (R) input is 6.5 dBm, the (A) input is open.

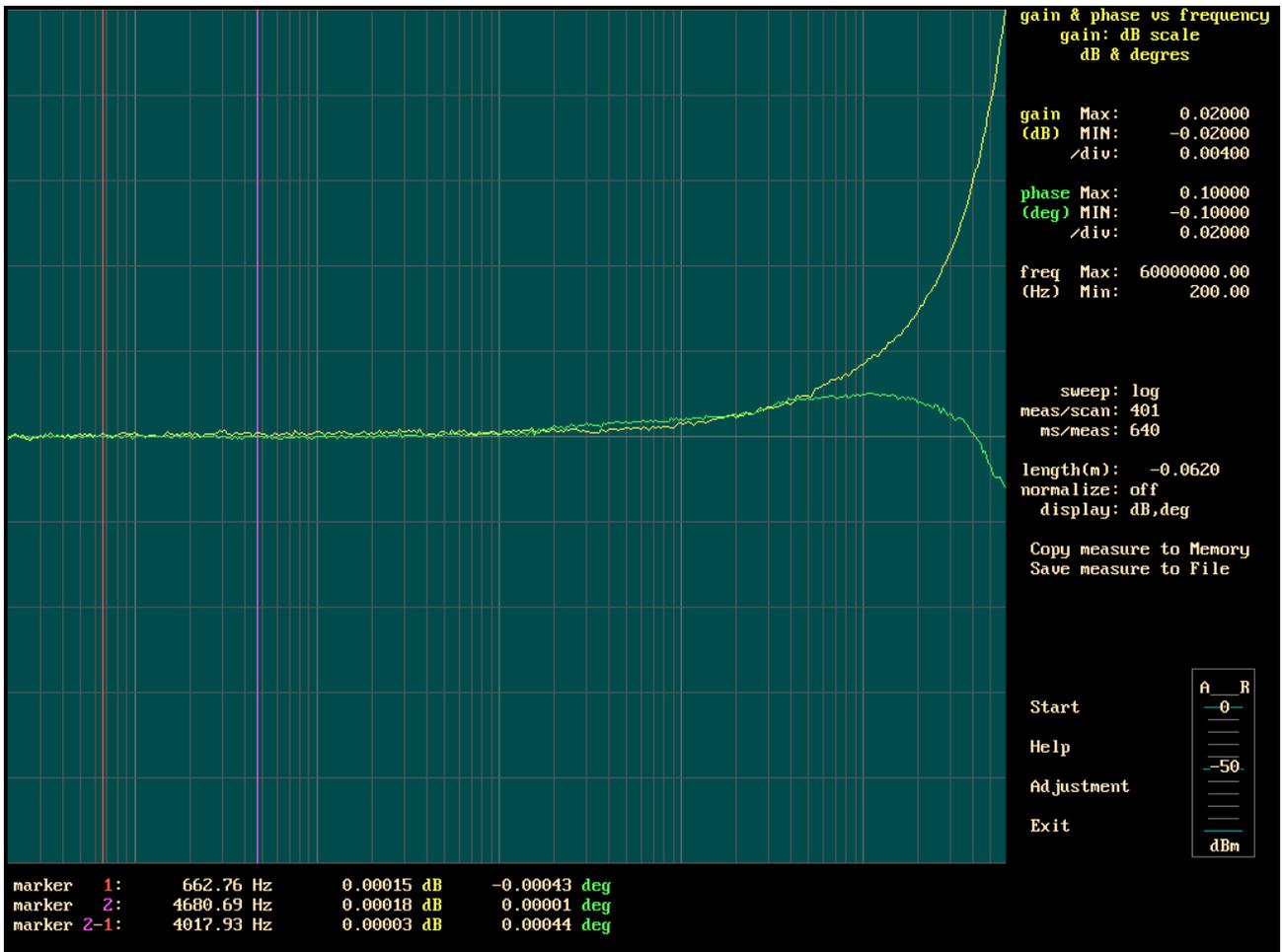
We can see that, above 10MHz, the dynamics range decreases. The root cause of this problem comes from the local oscillator and the leakage inside the detectors. To solve this problem : add an external 20dB attenuator on the (R) input, then normalize.

A modification of the DDS1 generator could decrease this problem.

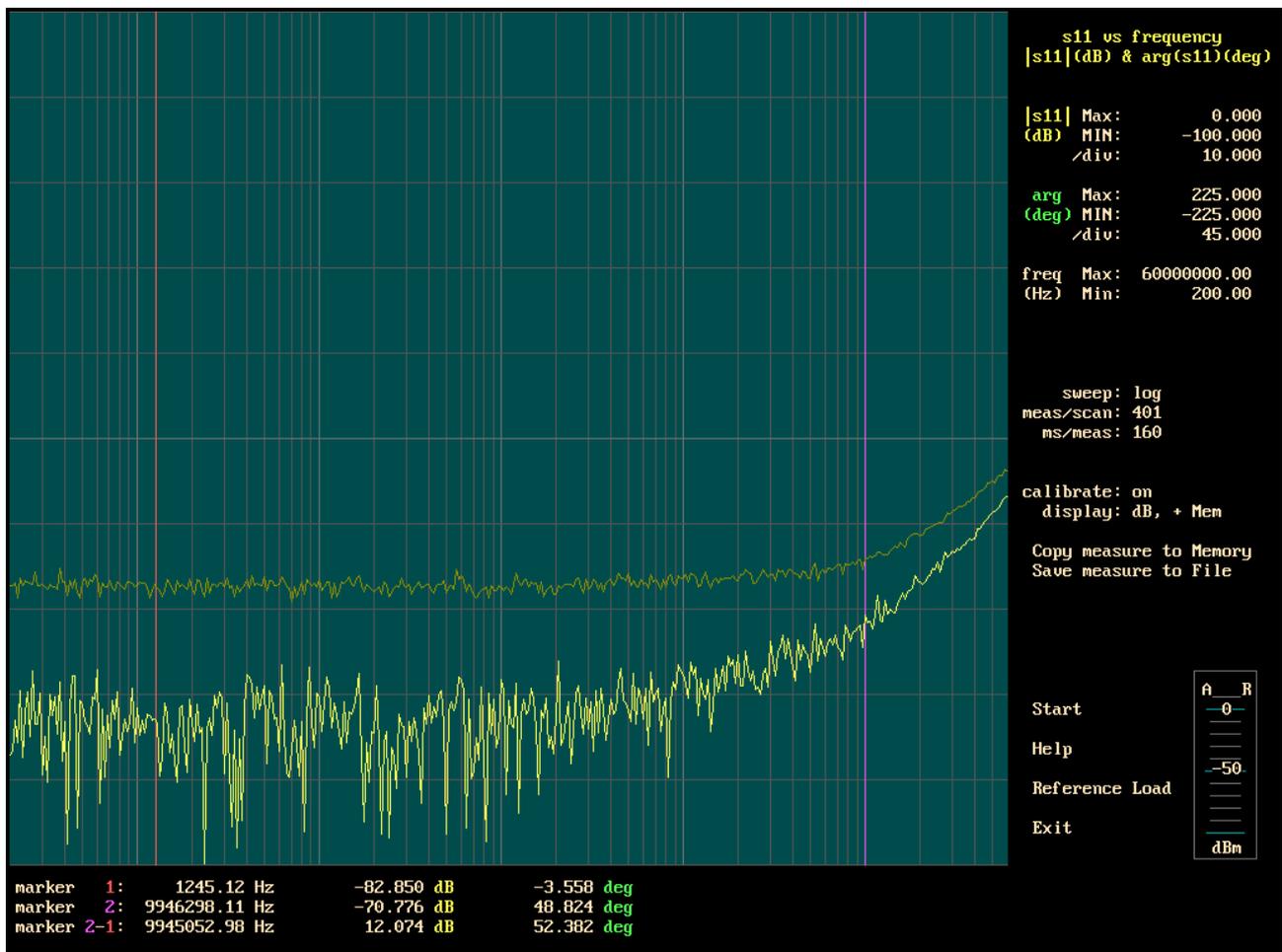


### Dynamics range of the analyzer

With a 30dB attenuator on the (R) channel and a normalization, the dynamics range (above 10MHz) does not vary.

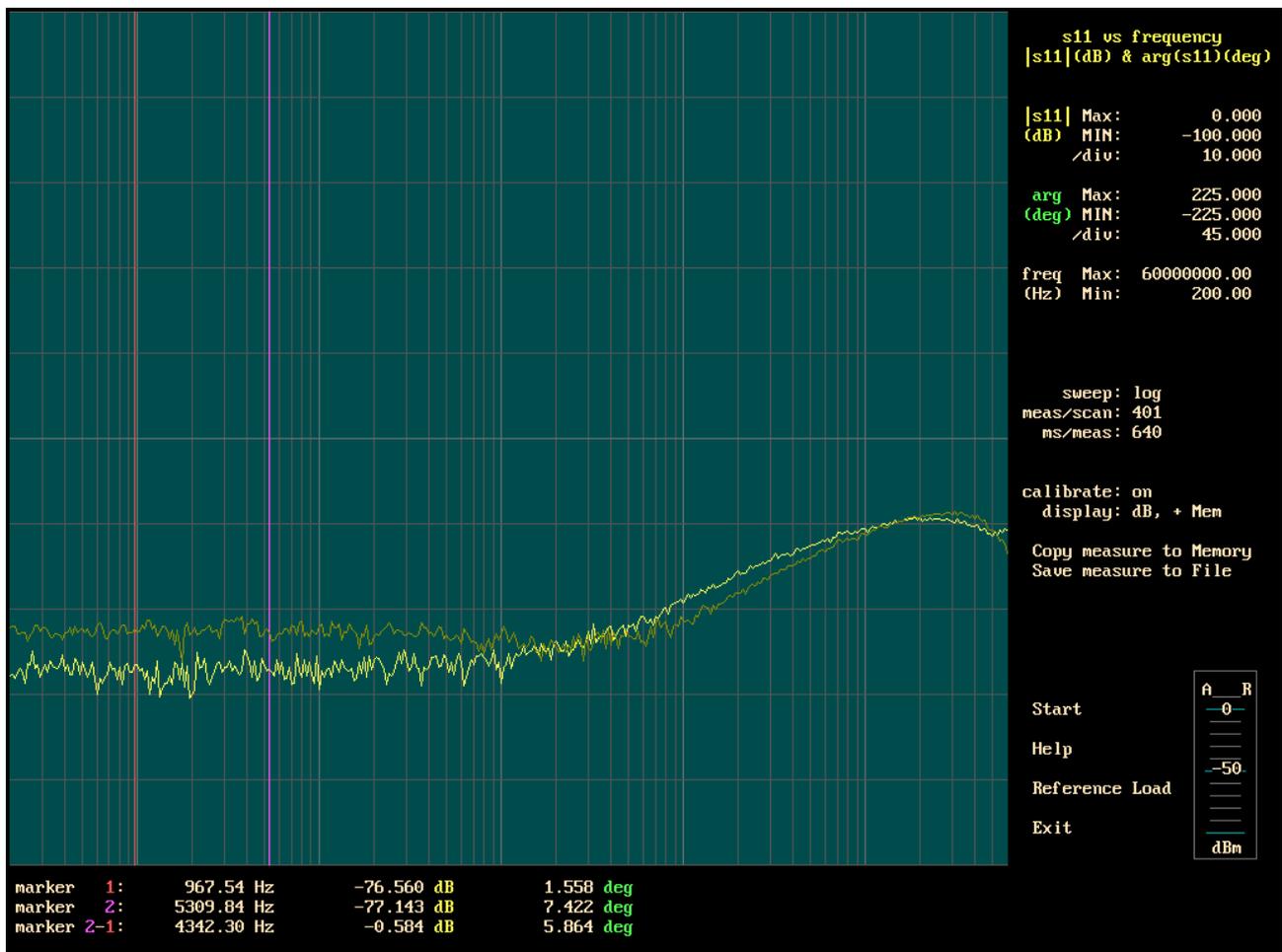


On this picture, we can see the **gain differences between the (A) channel and the (R) channel** ; (without normalization, but with the " length correction " and with the " non frequency dependant " correction ).  
 If, in the frequency range of a measurement, these gain differences are acceptable, then it is not necessary to normalize.



## Impedance measurement of the A and R inputs (in the s11 format)

To measure these impedances, we need two network analyzers ; (we built two network analyzers) .  
 At low frequencies, s11 is defined by the precision of  $R1+R2=50$  ohms.  
 Above 10 MHz, a small capacitor (about 1 pF in parallel with R2) can improve the value of s11.

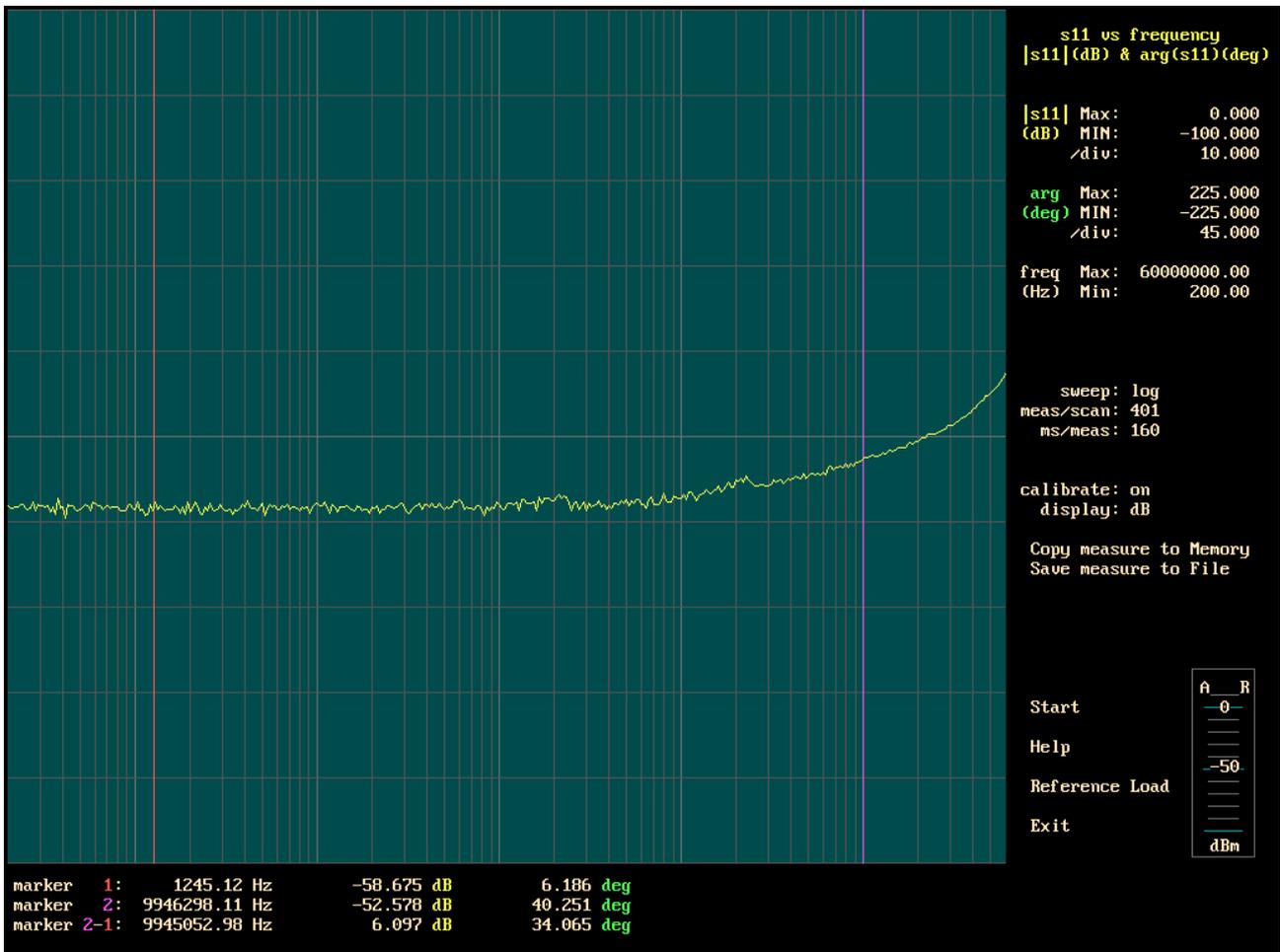


### Impedance measurement of the A and R outputs (in the s11 format)

For the network analyzer application, the output impedance of the OPA695 amplifier does not add error on the measurement of the gain vs frequency.

Then, to do this measurement : remove the power supply of the analyzer to be measured, connect to the ground the output of the amplifier OPA695, then measure , (do not forget to remove the short circuit after the measurement !)

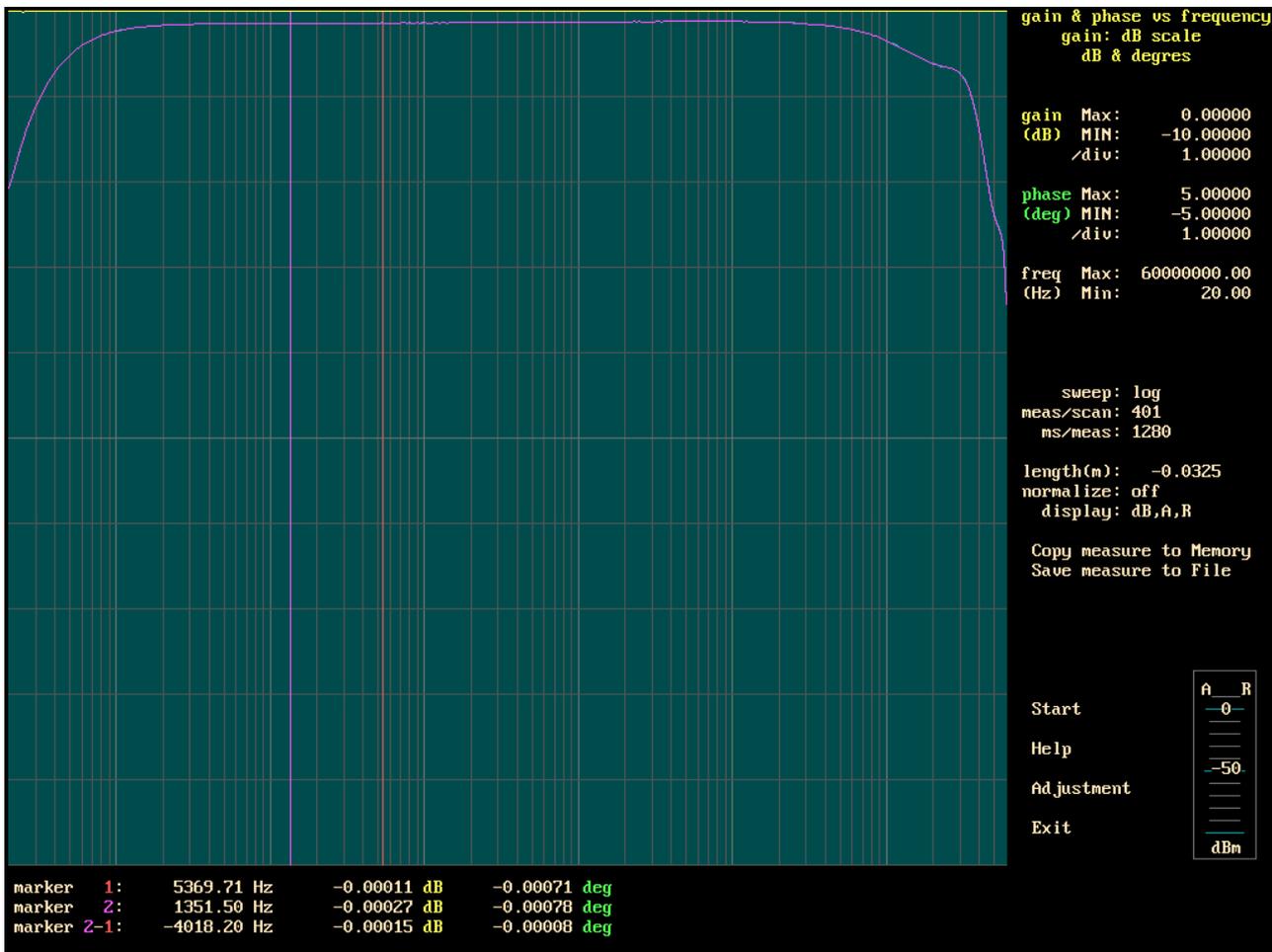
The capacitors in parallel with the 50 ohms output resistors are 1pf (the value of the capacitors can change with the way used to connect the coax cables to the board.) . These capacitors are optional , it is just an improvement.



### Impedance measurement of the (A) or (R) outputs (in the s11 format)

If we used the analyzer outputs as a general purpose generator, then output impedance of the OPA695 amplifier contributes to the output impedance.

To make this measurement, connect the power supply of the analyzer to be measured, put the potentiometer at minima position , and with another analyzer measure the output impedance.



### (A) or (R) generators output signal level.

On this picture, the purple trace shows the variation of the voltage at the output (R) or (A) (measured in dBm). (The scale is identical for dB and dBm)

The -2dBm decrease at 20Hz comes from the 100uF capacitor (between the 60MHz low-pass filter and the potentiometer).

At 60MHz, the decrease comes from the AD9851 and the low-pass filter.

### To measure at a frequency lower than 200 Hz

It is possible to extend the frequency range down to 20Hz, but with some restrictions.

When the frequency becomes lower than 200 Hz, the amplitude on the (A) and (R) inputs must not exceed 0dBm (to avoid over-scale problems on the inputs of the ADC).

When the frequency decreases, we must increase the sweep time ; at 20 Hz to measure each frequency point, we need 1280ms or 2560ms.

At 20Hz the dynamic range is also reduced .

When the frequency becomes lower than 150 Hz, the low-pass filter at the output of the detectors and the ADS1251 low-pass filter function, cannot remove the signal at the frequency of : local oscillator frequency + input frequency.

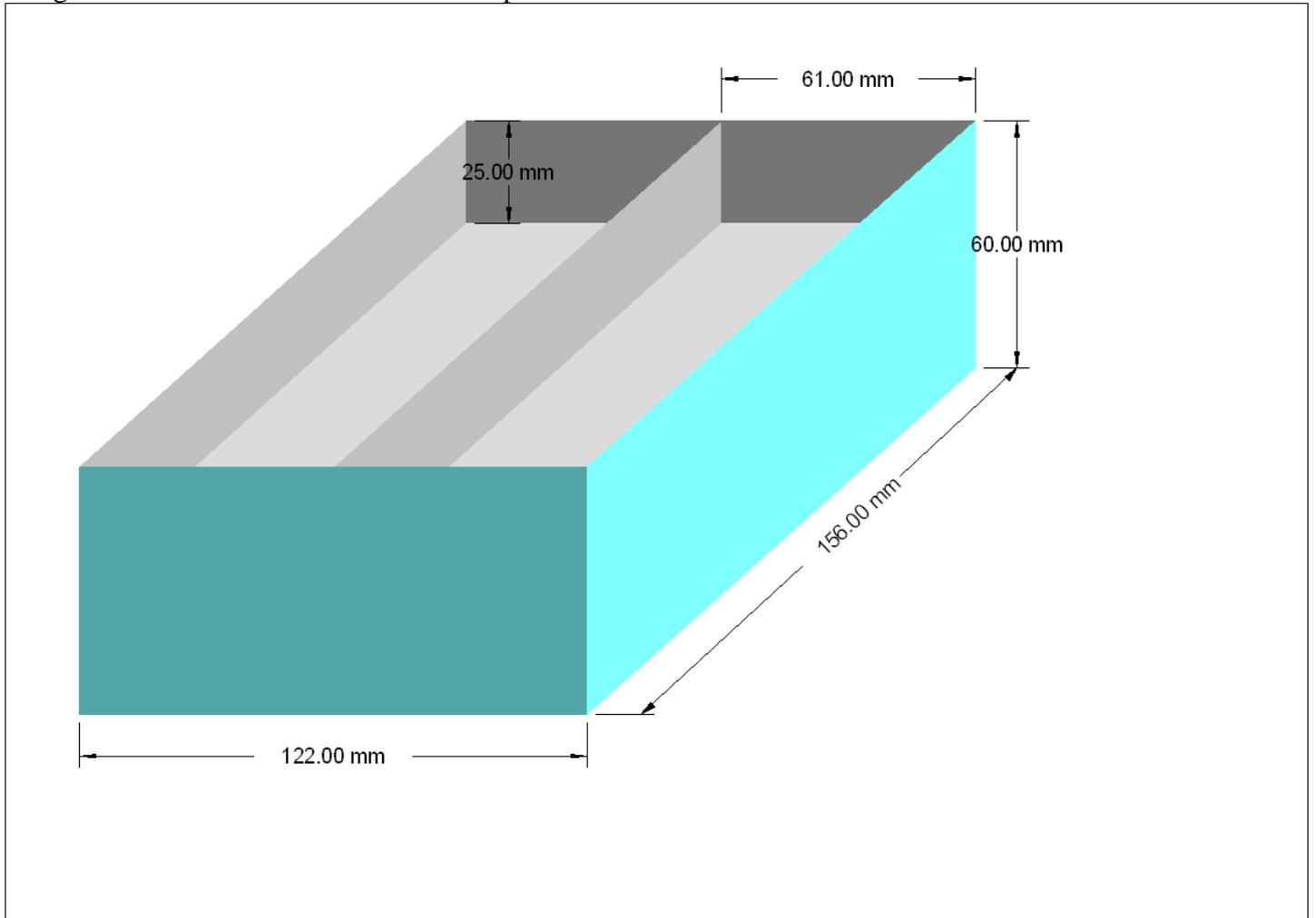
To measure the signal, the program takes the Fourier transform of the signal; if we increase the measurement time at each frequency, we reduce the bandwidth of the " band-pass Fourier transform filter" and it becomes possible to remove the signal at the frequency of : local oscillator frequency + input frequency.

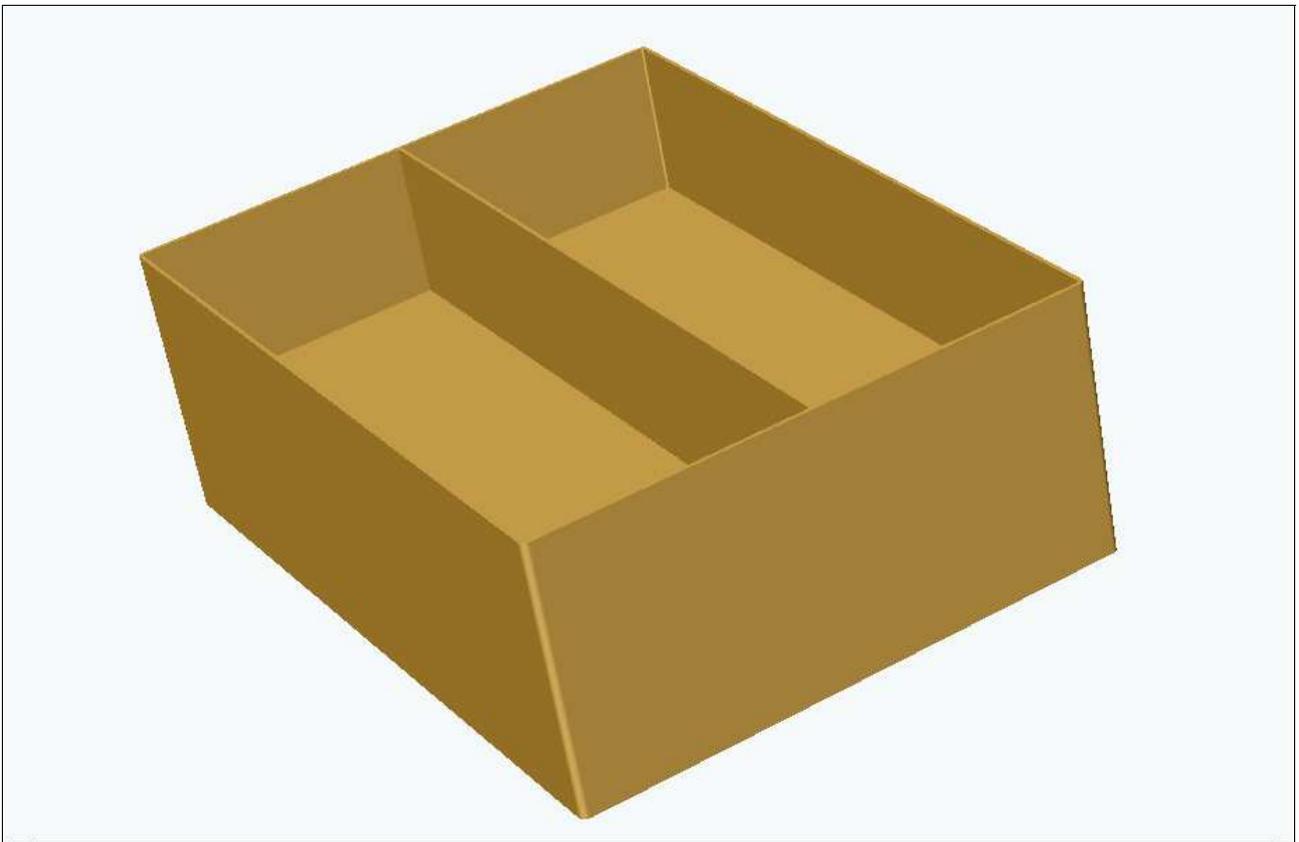
For example : to scan from 20Hz to 20kHz, from 20Hz to 150Hz set the sweep time at 1280ms/measurement; at 150Hz stop the scan, change to 40ms/measurement, then continue the scan.

At the frequency of 25Hz, we can see some small defects ; because 25Hz is also the frequency at the output of the mixer and the isolation of the mixer is not infinite.

## The box

To have more than 100dB of dynamic range, it is necessary to separate the detector (A) and the detector (R). The generator DDS1 and DDS2 are also separated.





The outer wall of the box is made with an iron sheet, dimensions 551mm\*60mm\*0.8mm

There is a bend in each corner.

The exact length of the sheet depend of the bending radius. The junction is in the middle of the back side.

The vertical dividing wall, and the two horizontal dividing walls, are made with 3 iron sheets, dimensions : 154mm\*60mm\*0.8mm.

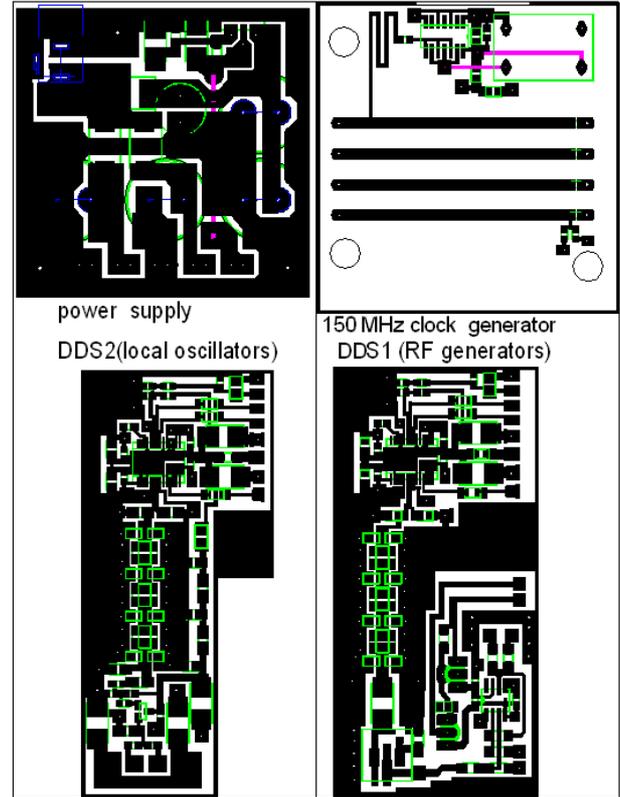
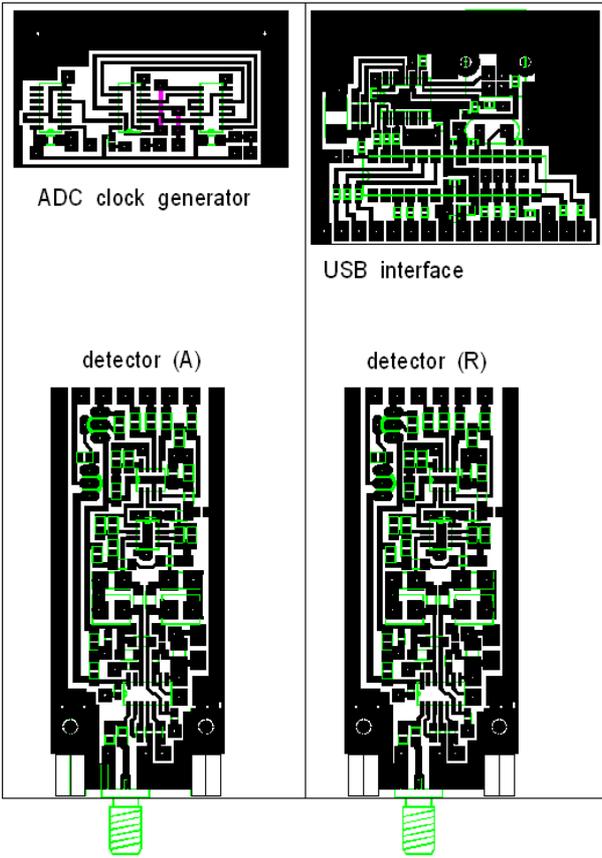
The top and bottom covers are made with 2 aluminium sheets, dimensions : 124mm\*158mm\*2mm

Iron sheets are soldered with a soldering wire used to solder electronic components.

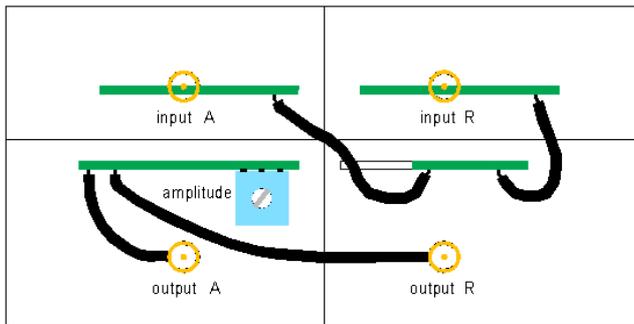
We used (iron + nickel) sheets, because we had this sheets; but tinned iron is less expensive.

Top view

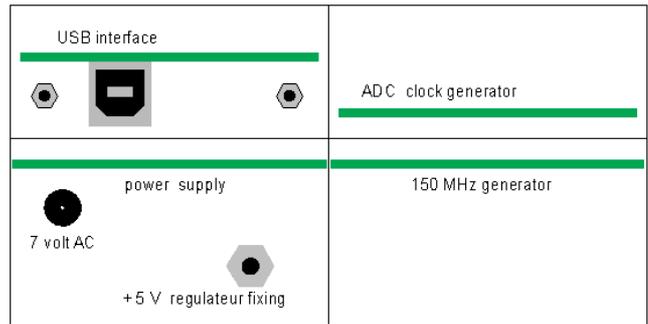
Bottom view



Disposal of the printed circuits inside the box .



Front view



Back view

# Atmega168 programming

To program the micro-controller there are two possibilities :

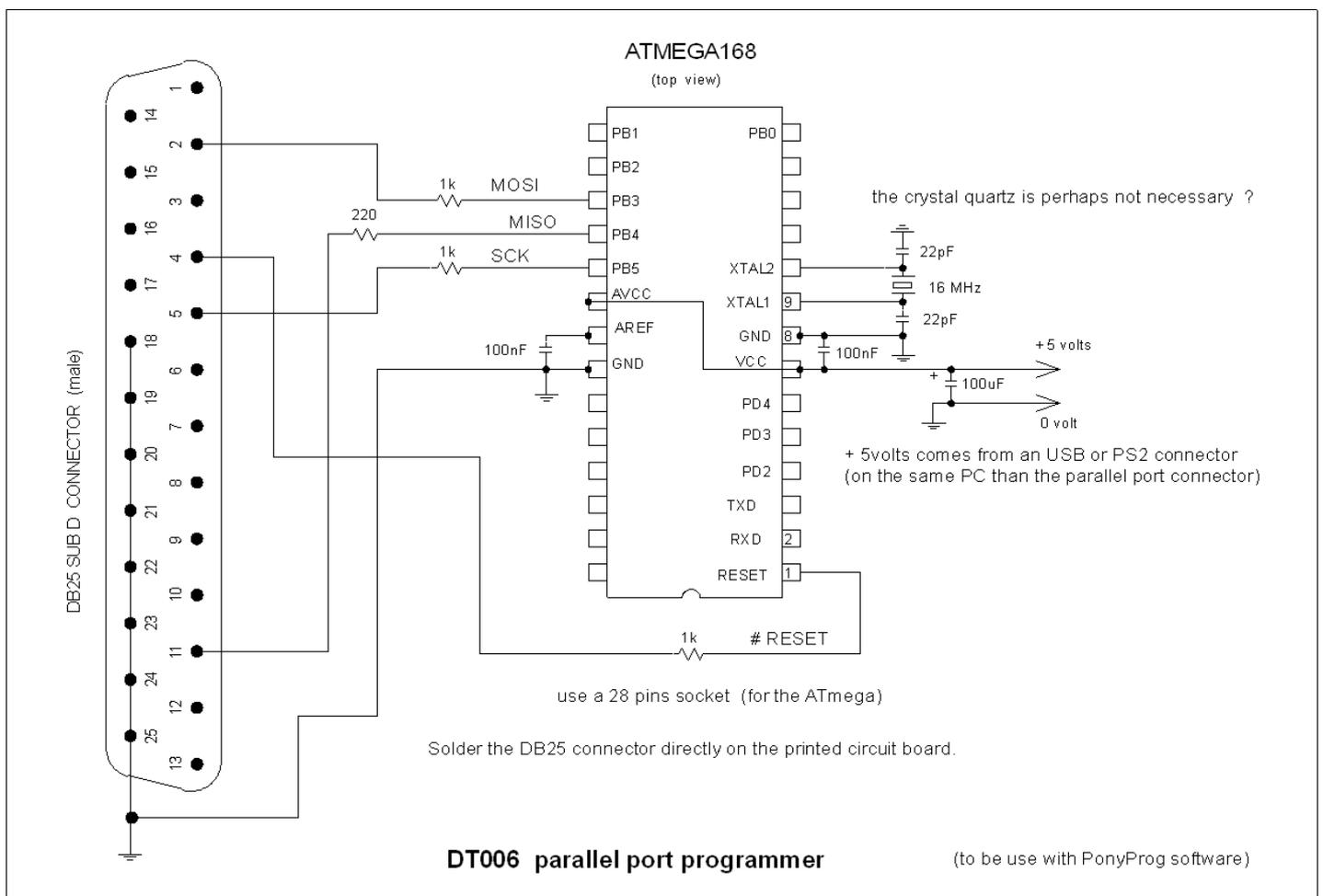
1 - Buy a Diecimila board, and load the network analyzer program with the Arduino software (through the USB port).

The source code to be used is : **arduianajac2**

2 – Buy an ATMEGA168 and program the flash memory with a parallel port programmer. The Arduino project has a parallel port programmer, but we do not succeed with this programmer ?

We have used a parallel port dongle DT006 with the PonyProg software . <http://www.lancos.com/prog.html>

The DT006 dongle is very easy to built : 4 resistors and a parallel port connector.



In the PonyProg menus choose : AVR micro , Atmega168 and DT006 parallel.

If there is nothing in the flash memory, when we read it, there is FF everywhere.

With the PonyProg software, write the file **arduino10.e2p** (in the Atmega168 flash memory).

Then write the **configuration and the security bits** (write the flash memory before the security bit).

**Configuration and Security bits**

7  6  BootLock12  BootLock11  BootLock02  BootLock01  Lock2  Lock1

7  6  5  4  3  BOOTSZ1  BOOTSZ0  BOOTRST

RSTDISBL  DWEN  SPIEN  WDTON  EESAVE  BODLEVEL2  BODLEVEL1  BODLEVEL0

CKDIV8  CKOUT  SUT1  SUT0  CKSEL3  CKSEL2  CKSEL1  CKSELO

Checked items means programmed (bit = 0)  UnChecked items means unprogrammed (bit = 1)

Refer to device datasheet, please

Configuration and security bits