

## Student manual for lab course in Ferromagnetic Resonance

Departamento de Física de la Materia Condensada, MAGNETRANS group  
Universidad Autónoma de Madrid

# Ferromagnetic Resonance in thin Films

Antonio Lara, Antonia Klein, Juan Francisco Sierra García,  
Ahmad Awad Awad, Diego Caso & Farkhad Aliev

January 27, 2022

### 0.1. Measurement procedure

The idea of the experiment is to sweep the frequency at a fixed external field. This is done for a range of applied fields  $\vec{H}$  ranging from 2200 Oe to  $-2200$  Oe. From the experimental data the uniaxial anisotropy field  $H_K$ , the saturation magnetization  $M_S$  and the Gilbert damping parameter  $\alpha$  can be obtained by fitting the data.

**Activity 1 - Set the VNA parameters - For VNA model E8363C (if you are using VNA model 5071B go to appendix A and read it before continuing).**

Although the measurement settings of the VNA are controlled by a LabVIEW program sometimes there still occur inconveniences. Therefore it is highly recommended to set the VNA directly by hand and select the same settings for the LabVIEW program afterwards.

### ATTENTION!

**Don't touch the experiment during the measurement and while setting the VNA!**



Figure 1: Control panel of the VNA.

- Power on the VNA.
- Choose the  $S_{21}$  scattering matrix parameter:
  - press **Meas** → select  **$S_{21}$**
- Open a 2<sup>nd</sup> window (finally one window for the magnitude and for the phase of  $S_{21}$  is needed):
  - right mouse click → choose **New window....**
  - right mouse click → choose **New trace....** → choose  **$S_{21}$**
- Choose the units of the y-axis:
  - mark upper window (with mouse) → press **Format** → choose **Log Mag**<sup>1</sup>
  - mark lower window (with mouse) → press **Format** → choose **Phase**
- Set the frequency range:
  - press **Freq** → **Start** insert **10 MHz** with the numeric pad
  - press **Freq** → **Stop** insert **12 GHz** with the numeric pad
- Autoscale the spectrum:
  - mark upper window (with mouse) → press **Scale** → choose **Autoscale All**
  - mark lower window (with mouse) → press **Scale** → choose **Autoscale All**
- Set the number of measuring points in the plot (resolution):
  - press **Sweep** → **Number of points** insert **2000** with the numeric pad  
→ confirm with **Enter** (numeric pad)
- Set the IF bandwidth<sup>2</sup> and the averaging:
  - press **Avg** → press **IF bandwidth** → insert **0.5 kHz** or **1 kHz**<sup>3</sup>
  - press **Avg** → press **Averaging on/OFF** till “OFF” is written in capital letters<sup>4</sup>

### Activity 1.1 (optional) - Save the settings

You can save the settings of the VNA for recalling them at a later time. This is advisable if you measure several days.

- Save the settings:
  - press **Save** → **Save As...** **Edit Filename** (bottom left of screen) → keyboard pops up → insert a name, e.g. “*settings\_juan*” → confirm with **OK**

<sup>1</sup>The Lin Mag case would represent  $V_2^R/V_1^S$ , whereas the Log Mag case gives  $20\log(V_2^R/V_1^S) = 10\log(P_2^R/P_1^S)$  keeping in mind that  $P = VI = V^2/R$ .

The Log Mag value is given in dB which is a logarithmic unit for the base 10 logarithm and defined as  $P(\text{dB}) = 10 \cdot \log(P_x/P_y) = 20 \cdot \log(V_x/V_y)$ . For example a power ratio of  $P_2^R/P_1^S = 1$  corresponds to 0 dB and  $P_2^R/P_1^S = 10$  corresponds to 10 dB et cetera.

<sup>2</sup>Some device (synthesizer) inside the VNA generates such a high frequency signal that the VNA cannot detect it. These high radio frequencies are multiplied by a low frequency from a local oscillator. From this frequency mixing a new intermediate frequency (IF) arises. This process is done a few times to convert the radio frequency to a manageable one. The IF bandwidth is connected to the accuracy with which this mixing process is done. A higher accuracy needs more processing time than a lower one.

<sup>3</sup>For higher values for the IF bandwidth the measurement is faster but the result less exact.

<sup>4</sup>In general, taking more averages improves the result. However, for this student lab the accuracy without additional averaging is sufficient.

- Recall the settings:
  - press **Recall** → choose your file, e.g. “2rows.csa” or “settings\_juan.csa” (press **More** to see all files)

**All LabVIEW programs for initiating and evaluating the measurement are found in the folder **Students\_FMR\_programs** on the desktop.**

- create a new folder with the direction **C:\FMR\#** (# is a number)
- open **LabVIEW** → open **S parameters loops in field\_PNA\_L 50GHz\_s12-solenoid6\_2VNAS.vi** (alternatively this program can be found in the folder **Students\_FMR\_programs** on the desktop)
  - insert **start frequency (Hz)** → **10 MHz**
  - insert **stop frequency (Hz)** → **12 GHz**
  - choose **Power dBm** → insert **00** in the green field<sup>5</sup>
  - choose **Save file in** → **C:FMR#**
  - choose **points in trace** → insert **2000**<sup>6</sup>
  - choose **IF Bandwidth (Hz)** → insert **0.5 kHz** or **1 kHz**, respectively
  - choose **averages on/off** → **off**
  - insert **seconds to wait in case of off averaging** → **6** (time between the measurements)
  - choose **RF (ON/OFF)** → **ON**
  - choose **Horizontal Scale** → **LIN**
  - choose **Solenoid:** → **6**<sup>7</sup>
- turn on the power supply<sup>8</sup>
  - press **MODE VOLT/CURR** to change to current source
  - press **STANDBY** to leave the standby mode

<sup>5</sup>In general you could create an array for measuring with different powers. We don't need this for this experiment. 0 dBm corresponding to 1 mV is fine for this lab. The blue box left to the green box indicates the chosen array element and should be **0**.

<sup>6</sup>This gives the precision of the measurement. 3201 is the maximal resolution.

<sup>7</sup>This parameter refers to the conversion of current into field. The conversion might differ for different CPWs. The solenoid with number 6 is used for this lab course.

<sup>8</sup>The power supply is controlled by the LabVIEW program but in general, it can also be regulated by hand.

### Activity 3 - Check the field plan

Once started the measurement, the first thing the LabVIEW program will do is to check the field plan. The field plan gives a list of the measurement fields for which a measurement will be taken (first column). The second column indicates if zero or one measurement shall be taken.

Checking the field plan:

- go to `C:\temp` → open `Plan H.dat`
- recommended range of field plan from 2200 Oe to –2200 Oe in steps of 4 Oe, finish with a line with 0 Oe and 0 measurements
  - If the field plan does not already contain the recommended values you can replace it by a new one using the LabVIEW program `make field plan.vi` which can be found in the folder `Students_FMR_programs` on the desktop.

### Activity 4 - Normalize the spectrum

The spectrum needs to be normalized in order to reduce the background signal. This means a reference spectrum is taken at a certain external field  $\vec{H}$  and is subtracted from all measurement spectra. Instead of a single negative peak this spectra contains although a fake peak, namely the inverse negative peak of the reference spectrum.

When measuring in a range from –2200 Oe to 2200 Oe, we can normalize at the highest field of 2200 Oe. For the chosen frequency range from 10 MHz to 12 GHz the relict of the normalization is then pushed outside the spectrum under consideration.

**We recommend to normalize while setting the current with the LabVIEW program.**

Pause the LabVIEW program `S parameters loops in field_PNA_L 50GHz_s12-solenoid6_2VNAS.vi` for proceeding with the normalization:

- Start the measurement → press the `white arrow` on the top left of the LabVIEW program.
- Immediately press the `pause` sign next to the arrow. It will pause as soon as the highest field is reached (in our case 2200 Oe).
- Normalize the spectrum with the VNA:
  - press `macro local` (This enables you to use the buttons when the system is PC controlled.)
  - mark upper window (with mouse) → press `Memory` → choose `Normalize`
  - mark lower window (with mouse) → press `Memory` → choose `Normalize`
  - **Don't touch the experimental set-up while normalizing!**
- Continue the measurement with pressing the `pause` bottom again.

Alternatively the normalization can be done by hand (not advised because the setting should be touched the least possible).

- Set the current at the power supply by hand:
  - set `1.3 A`<sup>9</sup> by hand
- Normalize the spectrum with the VNA:
  - press `Memory` → choose `Normalize`

Note: For a wider range of frequency values the normalization peak would appear in the spectrum under consideration. In this case it is advisable to normalize at zero field (corresponding to 0 A).

---

<sup>9</sup>This corresponds to an applied field of about 1877.2338 Oe.

## Activity 5 - Run the measurement

After normalizing the measurement and repressing `pause`, running the experiment will take about 6 hours. The program finishes automatically.

During the measurement the LabVIEW creates a file for each field. They contain three columns, giving the frequency, as well as the amplitude and the phase of the  $S_{21}$  parameter. The file name is related to the field, e.g. *V=002dBm\_H=2.16026E+3G\_T=300.0E+0K.DAT*.

## 1. Data analysis & Evaluation

### 1.1. Evaluation of the measurement data

As already mentioned the aim of the experiment is to determine the anisotropy field  $H_K$ , the saturation magnetization  $M_S$  and the Gilbert damping parameter  $\alpha$ . There exist different ways to process the data in order to obtain the above specified parameters. We will present some of them.

For the evaluation only one branch of the symmetric curve is needed. In the following neglect one branch or take the mean value of the positive and negative branch to ameliorate your measurement results.

#### Prepare the data

Start processing you data with the LabVIEW program `measurement_processing.vi`.

- choose `Path` → insert the path referring to the file with highest field, e.g. *V=002dBm\_H=2.16026E+3G\_T=300.0E+0K.DAT* (The program normalizes again and processes all files stored in the same folder.)
- choose `T/H` → insert `H`

Press the white arrow to run the program.

The program converts magnitude and phase of  $S_{21}$  into imaginary part and real part. It presents you a 3D plot of the real part of the  $S_{21}$  parameter and of its derivative with respect to the field. In general the plot of the derivative reproduces a sharper plot. The program creates several new files stored in your folder *C:/FMR#*:

*ImFIT.dat*, *IMmu.dat*, *R(Re+IM).dat*, *Remu.dat*, *fields.dat*, *frequencies.dat*, *2D\_array.dat*, *2D\_array\_derivative.dat*

The most important ones for you in the following are:

- *2D\_array.dat*: gives an array with the real part of  $S_{21}$
- *2D\_array\_derivative.dat*: gives an array with the derivative (with respect to the field) of the real part of  $S_{21}$

#### Possibility A: Proceed data analysis with LabVIEW

For the first task you need to plot the resonance frequencies against the corresponding fields. If your data quality is good<sup>10</sup> you can use the LabVIEW program `resonance_frequencies.vi` to extract them. Insert the path of your *2D\_array.dat*-file and run the LabVIEW program. The program finds the minimum value of the frequency for each field and creates a new file *resonance\_freqs.dat* containing one column with the field and second with the corresponding resonance frequency. Delete bad data<sup>11</sup> points in the following procedure if necessary.

You can use another LabVIEW program `FWHM_students.vi` for the second part. Insert the path to the file *2D\_array\_derivative.dat* and run the program. The output file *FWHM.dat* contains two columns,

<sup>10</sup>If the resonance frequency is only at a local minimum but not at a global minimum the program might not work properly.

<sup>11</sup>Bad data points are data points that clearly don't belong to the curve you obtain.

one for the value of FWHM in Oe and one for the corresponding frequencies. In the final fitting procedure to obtain  $\alpha$  it is very important to choose only the range with good data points. The best way to determine the FWHM of a (negative) peak is to fit the curve – in our case  $Re(S_{21})$  against  $H$  for fixed frequencies – with a Lorentz curve

$$L(x) = y_0 + \frac{2A}{\pi} \frac{\omega}{4(x - x_0)^2 + \omega^2}. \quad (1)$$

The fit parameter  $x_0$  gives the centre of the curve and  $\omega$  the FWHM. The parameter for the area,  $A$ , is one for a perfect Lorentz curve.

However, the LabVIEW program proceeds in a different way to obtain approximately the FWHM. It calculates the difference between minimum  $x_1$  and maximum  $x_2$  of the differentiated  $S_{21}$ -curve (this means the distance between the points where the slope of the original plot is maximal). We can express this distance in terms of the FWHM  $\omega$

$$\Delta x = x_2 - x_1 = \frac{\sqrt{(3)}}{3} \omega \approx 0.58 \omega. \quad (2)$$

Therefore, the LabVIEW program multiplies this value at the end with the reciprocal of 0.58 to obtain a good approximation for the FWHM.

### Possibility B: Proceed data analysis with Origin

Instead of proceeding as described in previous section (Possibility A) the proceeding data analysis can also be done by hand with a suitable program, e.g. OriginPro8 installed on the PC in the lab. (This is for example necessary if the resonance frequency is no global minimum of you data).

#### Evaluate the first part with Origin

- Open a new matrix:
  - File → New... → Matrix
- Import the *2D\_array.dat*-file:
  - File → Import → Single ASCII → choose 2D\_array.dat
- Plot the data in 3D:
  - Plot → Image → Profiles
- Make the plot less blurred:
  - right click on plot → Plot Details... → choose Layer1 → choose Size/Speed → untick Matrix data, maximum points per dimension

Take care which axis denotes field and which frequency. Figure 2 gives an example of how the plot might look like for a single branch. Changing the position of the yellow cursors changes the cross section of  $Re(S_{21})$  against field (upper plot) or field against  $Re(S_{21})$  (right plot), respectively.

- Examine at least for 20 fixed fields the cross section  $Re(S_{21})$  against frequency:
  - right click on plot → Go to Horizontal/Vertical => then click 2 times in plot => plot details=> worksheet>Save data for fits
  - a new table pops up → make a new 2D plot
  - fit the peak with a Lorentz curve to obtain the position of the plot<sup>12</sup> or use the Data Reader for determining  $f_0$  approximately

<sup>12</sup>For making a Lorentzian fit can be done with Origin via Fitting → Nonlinear Curve Fit → choose Open Dialog... → choose Settings → Function Selection → Function → Lorentz. Make sure you restrict the active set of points to a small surrounding around the peak.

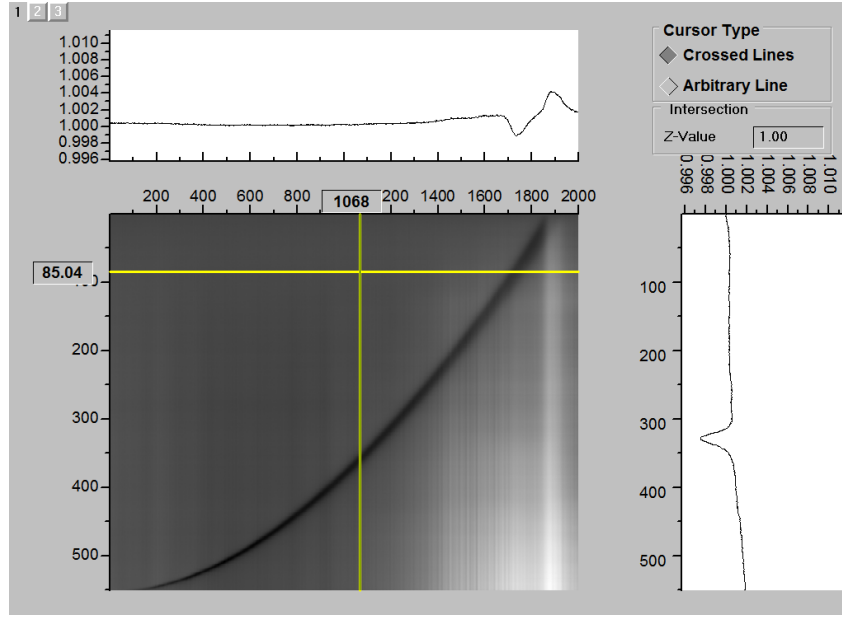


Figure 2: 3D plot of field against frequencies (centre), cross section of  $Re(S_{21})$  against field (upper plot), cross section of field against  $Re(S_{21})$  (right plot).

- Make a new table with a column for resonance frequencies  $f_0$  and a column for the corresponding field  $H$ .

NOTE: besides cross-sections to analyse FMR width (to obtain damping) you need to copy with you the following files to be able analyse materials parameters from FMR data

- (1) "fields.dat" ;
- (2) "frequency.dat" ;
- (3) "2d\_array.dat"

### Evaluate the second part with Origin

The second part can be evaluated in a similar way. At the end you need to create a table with at least 20 values for  $\Delta H$  (FWHM) and their corresponding frequencies.

You can either use the matrix created for the first part with the *2D\_array.dat*-file and fit this time  $Re(S_{21})$  against fields with a Lorentz fit. The parameter  $\omega$  gives you the FWHM.

Or the other possibility is to create a new matrix with the *2D\_array\_derivative.dat*-file and determine the difference between the minima and maxima and multiply this value with  $100/58$ .

### Fit the final data with Origin

At this point of the analysis you should have created two tables, one with resonance frequency  $f_0$  against field  $H$  and one with the FWHM  $\Delta H$  against frequency  $f$ .

Plot both of them. As mentioned above the first one should be fitted with the FMR expression. For doing so, we make a little transposition of the equation

$$f_0 = \frac{\mu_0 \gamma_e}{2\pi} \sqrt{(H + H_K)(H + H_K + M_S)} \quad (3)$$

$$= \frac{\gamma_e}{2\pi} \sqrt{\mu_0(H + H_K)\mu_0(H + H_K + M_S)} \quad (4)$$

$$= \frac{\gamma_e}{2\pi} \sqrt{(B + B_K)(B + B_K + \mu_0 M_S)}, \quad (5)$$

exploiting the fact that in vacuum the magnetic field  $\vec{H}$  and the magnetic induction  $\vec{B}$  are connected by  $\vec{B} = \mu_0 \vec{H}$ . For the final results we want to give  $B_K$  and  $\mu_0 M_K$  in the typical SI units for the magnetic induction, tesla (T). So far we have measured the magnetic field in units of the Gaussian unit system (GGS), namely in Oersted (Oe). The magnetic field in CGS and SI units is related by

$$[H]_{\text{CGS}} = 1\text{Oe} = \frac{1000 \text{ A}}{4\pi \text{ m}} = \frac{1000}{4\pi} [H]_{\text{SI}}. \quad (6)$$

The vacuum permeability in SI units is given as

$$[\mu_0]_{\text{SI}} = 4\pi \cdot 10^{-7} \frac{\text{kgm}}{\text{s}^2 \text{A}^2}. \quad (7)$$

Using the two equations we get for the conversion

$$[B]_{\text{SI}} = [\mu_0 H]_{\text{SI}} = [\mu_0]_{\text{SI}} \cdot \frac{1000}{4\pi} [H]_{\text{CGS}} = 10^{-4} \frac{\text{kg}}{\text{s}^2 \text{A}} = 10^{-4} \text{T}. \quad (8)$$

Thus 1Oe corresponds to  $10^{-4} \text{T}$ .

Having converted everything to SI units you can start with the fitting procedure with origin.

- Fitting → Nonlinear Curve Fit → choose Open Dialog... → choose Settings → Function Selection → Category
  - choose FMR\_1 (under “—User Defined—”)
  - if you work on another computer create your own fit with <New...>
- this should give you a fit with the Code  $y = a \cdot \sqrt{((x+b) \cdot (x+b+c))}$
- $a = \frac{\gamma_e}{2\pi} = \frac{1.76 \cdot 10^{11} \text{ Hz}}{2\pi} \approx 2.8025 \cdot 10^{10} \text{ Hz/T}$  is a fixed parameter
- play with the start values of  $b$  and  $c$  for making the fit converge

Proceed in a similar way for the second part. Fit with equation for the linewidth and again use a fixed prefactor  $\frac{4\pi}{\gamma_e} \approx 7.1400 \cdot 10^{-11} \frac{\text{T}}{\text{Hz}}$ .

For estimating the mistake use standard deviation of the fitted parameters.

## A. Model E5071B

This VNA model is older than the E8363C model and thus why the maximum microwave frequency is limited by 8.5 GHz. The proceeding is the same as in the VNA E8363, with a few exceptions:

- **VNA MODEL:** in the LabVIEW program (*S parameters loops in field\_PNA\_L 50GHz\_s12-solenoid6\_2VNAS.vi*), set the VNA model to E5071B instead of E8363C (which is set by default).
- **Stop frequency:** instead of 12 GHz, it will be set to 8.5 GHz, you will have to introduce it both in the VNA and in the LabVIEW program that controls the experiment.
- **Points in trace:** the maximum resolution is of 1601 points, which will be set both in the VNA and in the LabVIEW program.
- **Normalization:** manual normalization is not possible to perform in the VNA, however, the LabVIEW program for analysis (*measurement\_processing.vi*) does it automatically, skip the normalization step.
- Also, in this model you cannot open two windows at the same time to display the signal in LogMag and Phase format, you have to get two traces in the same window (or visualize one at a time, the LabVIEW program automatically sets it to measure both).

The analysis of the resonance is performed exactly the same way for both VNA models.