Alexandria University
Faculty of Engineering
Department of Electrical Engineering
Fourth Year Communications Section

Microwave Measurements

Lab Manual

Fall 2017

Experiment 1
Scattering Parameters of a Microwave Bandpass Filter

Objective

Getting familiar with the Network Analyzer HP8720 and how to use it to find the scattering parameters of microwave BPF.

Discussion

The scattering matrix is used to quantify how RF energy propagates through multi-port network. The S matrix is what allows us to accurately describe the properties of complicated networks in a simple form.

Consider the normalized incident and reflected waves a_n and b_n where:

$a_n = \frac{V_{n+}}{\sqrt{Z_{0n}}}$ represents the incident wave and $b_n = \frac{V_{n-}}{\sqrt{Z_{0n}}}$ represents the incident wave at port n. The reflected and incident waves are related by the scattering parameters as follows:

$$b_1 = S_{11}a_1 + S_{12}a_2$$
$$b_2 = S_{21}a_1 + S_{22}a_2$$

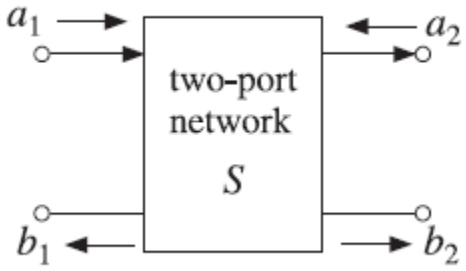

Figure 1 S parameters for a two port network.

In matrix form:

$$\begin{bmatrix} b_1 \\ b_2 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \end{bmatrix}$$

If the output port is matched, i.e. $a_2 = 0$, then

$S_{11} = \frac{b_1}{a_1}$ is the input reflection coefficient when the output is matched.

$S_{21} = \frac{b_2}{a_1}$ is the transmission coefficient from port 1 to port 2 (forward gain).

If the input port is matched, i.e. $a_1 = 0$, then

$S_{22} = \frac{b_2}{a_2}$ is the output reflection coefficient when the input is matched.

$S_{12} = \frac{b_2}{a_1}$ is the transmission coefficient from port 2 to port 1 (reverse gain).

The voltage SWR at port n can be expressed in terms of S parameters as $SWR_n = \frac{1+|S_{nn}|}{1-|S_{nn}|}$.

Procedure

Port Reflection Calibration:

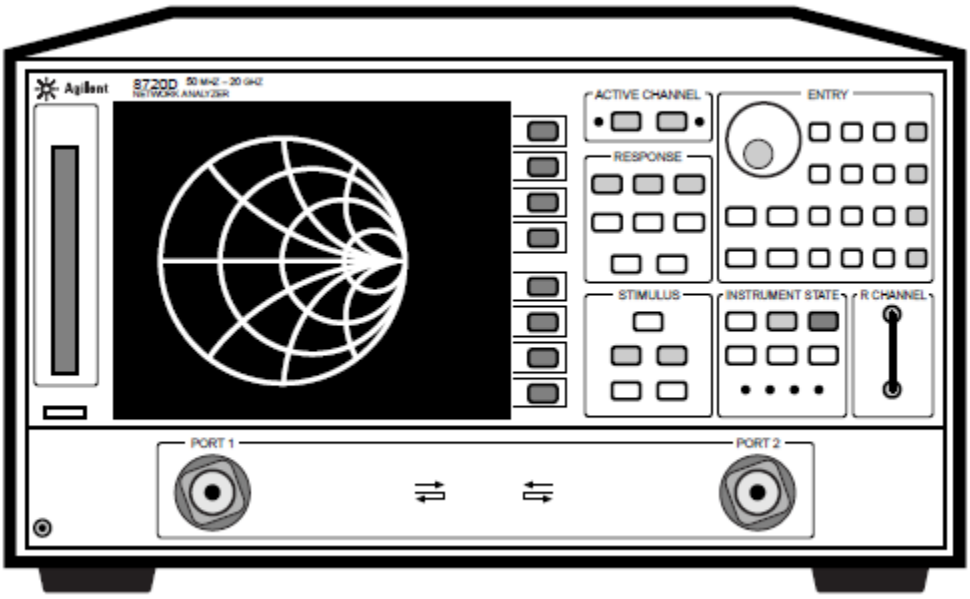

Figure 2 HP 8720D Network Analyzer.

1- Press the CAL key in the RESPONSE panel to the right of the display screen to get the calibration menu.
2- Press the CALIBRATE MENU soft key in the CAL menu to get the CALIBRATE menu.
3- Select RESPONSE, the menu will display three choices OPEN, SHORT and THRU.

4- Connect thru cable from port 1 to port 2 and press THRU, the 8720 will show THRU underlined which means that response has been calculated.
5- Mount both open standards in both terminals and press OPEN and the 8720 will display OPEN underlined.
6- Repeat step 5 for the short load standard, the 8720 will display SHORT underlined.
7- Press DONE RESPONSE to complete the calibration sequence.

Measurement of S Parameters:

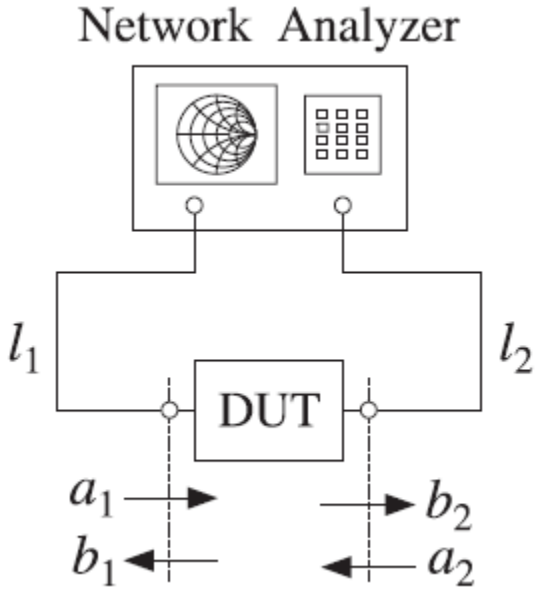

Figure 3 Connection of device under test.

8- Connect the bandpass filter to the 8720.
9- Press the MEASUREMENT key in the RESPONSE panel.
10- Press the "Transmission S12" soft key to view S_{12} as a function of frequency.
11- Press the MKR key from the RESPONSE panel, use the knob to move the marker and obtain the readings at point on the plot.
12- Repeat step 11 for the other S parameters.

Experiment 2
Time Interval Measurement of an FM Signal

Objective

Getting familiar with the HP5372A Frequency and Time Interval Analyzer and how to use it to find the modulation parameters of an FM signal.

Discussion

The HP5372A frequency and time interval analyzer makes both frequency and time interval measurements at rates up to 13.3 x 10^6 measurements per second. This is achieved by dividing the signal under study into arbitrary length frames and analyzing each frame, thus enabling the study the frequency or period variation as a function of time.

Experiment

Objective

To measure

- The carrier frequency.
- Frequency deviation, peak-to-peak deviation.
- Period of the message.

Equipment

- Fluke 6060B signal generator.
- HP5372A Frequency and Time Interval Analyzer.

Procedure:

1- Make sure that all power switches are in the **OFF** position.
2- Connect the equipment as shown in Figure 4.

3- Make the following adjustments to the Fluke 6060B signal generator:
    a. Generate the message (baseband signal) by choosing **internal FM** and choose **1000Hz** sinewave message. The buttons are located on the far left of the generator.
    b. Press the **Frequency** button and type **10MHz**, this is the frequency of the carrier signal.
    c. Press the **Amplitude** button and type **100mV**, this is the amplitude of the carrier frequency.
    d. Press **FM** and adjust the frequency deviation to **2KHz**.
    e. Press **RF ON**.

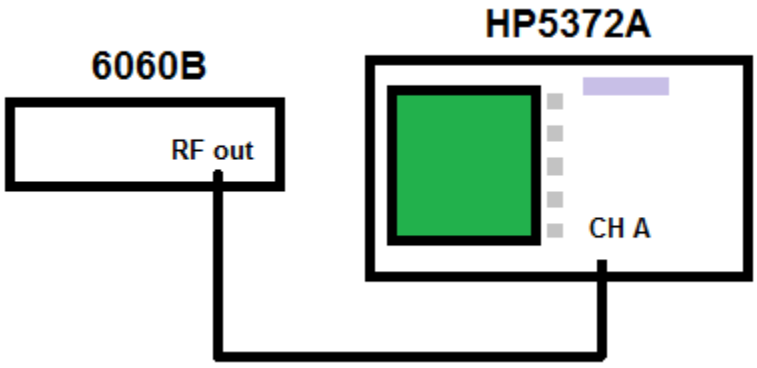

Figure 4 Experiment set up.

4- Make the following adjustment on HP5372A Frequency and Time Interval Analyzer.
    a. Press **PRESET** key.
    b. From the softkey choices select **FREQUENCY**.
    c. Using the knob, move the menu cursor to the **AUTOMATIC** field, then press the softkey to select **SAMPLE**.
    d. Press the **INTERVAL SAMPLING** softkey.
    e. Using the knob, move the menu cursor to the **SAMPLE ARM** field and type 45 and press the **μs** softkey. This will specify the size of each frame analyzed.

5- To display the results:
    a. Press **GRAPHIC** key at the top next to the display.
    b. Press the **SINGLE/REPEAT** key (next to PRESET key) to freeze the display.

c. Press the **TIME VAR** softkey, this will display the instantaneous frequency readings versus time as a discrete graph.
d. To connect the data points, press **CONNECT DATA ON**.
e. Press the **MENU** softkey (top one) to select **MRKR,** this displays the marker options.
f. Using the knob, move the horizontal markers to measure the maximum frequency and minimum frequency.
g. Press **Marker Delta** softkey to measure peak-to-peak deviation.
h. Change the **MARKER ORIENT** softkey to vertical and move the vertical markers to measure the period of the message.

Experiment 3
Measurement of Settling Time and Overshoot of a VCO Signal

Objective

Getting familiar with the HP5372A Frequency and Time Interval Analyzer and how to use it to find the settling time and overshoot of VCO signals. Analyzing the performance of the direct method if FM generation.

Discussion

The voltage controlled oscillator (VCO) consists of LC circuit. This LC circuit will enter a second order transient state when a sudden change in amplitude is applied. The transient response will vary according to the LC values. In this experiment it is required to measure:

- Settling time: time elapsed from the application of an ideal instantaneous step input to the steady state time (almost 5τ).
- Overshoot: refers to the transient value of any parameter that exceeds its steady state value during its transition from one value to another.

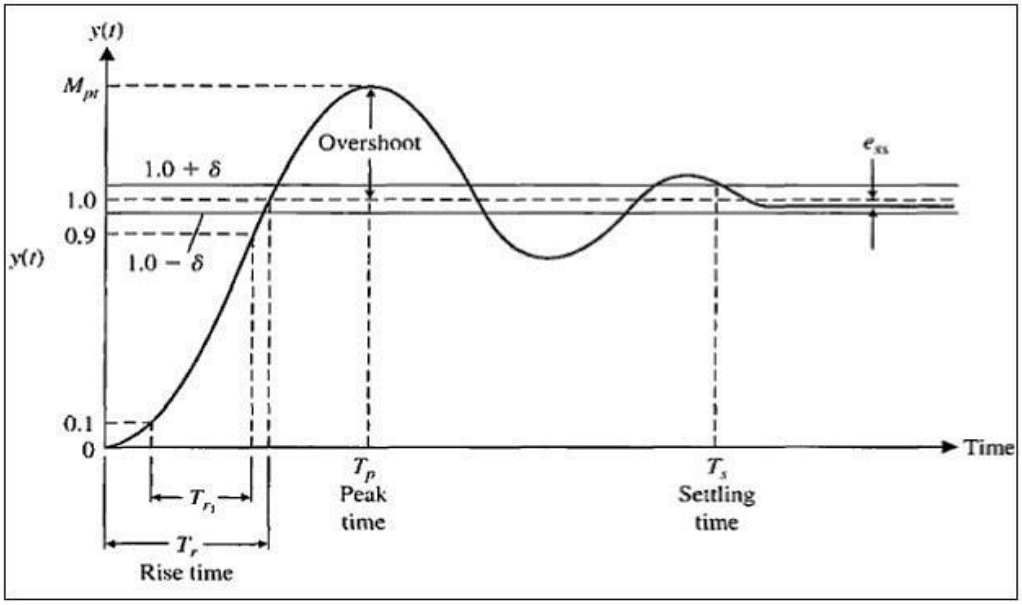

Figure 5 Second order underdamped response specifications.

Equipment

- HP8116A pulse function generator: to produce the square wave to be modulated (baseband signal).
- HP3314A function generator: to produce the carrier signal and perform the frequency modulation.
- TEK2235A oscilloscope.
- HP5372A Frequency and Time Interval Analyzer.

Procedure

1- Make sure that all power switches are in the **OFF** position.
2- Connect the equipment as shown in Figure 6.
3- Adjust the HP8116A pulse function generator to generate a square wave with:
   a. Amplitude 800mV.
   b. Frequency 100KHz.
   c. Duty cycle 50%.
   d. Offset 0V.

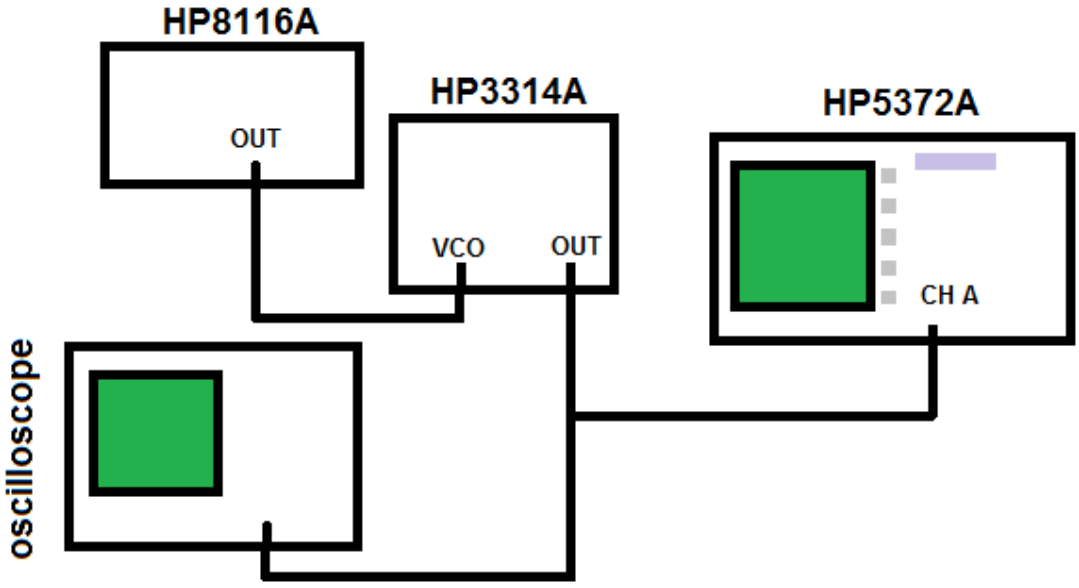

Figure 6 Experiment set up.

4- Make the following adjustment on the HP3314A generator:
   a. Adjust the FM output to VCO.
   b. Waveform sinewave.
   c. Carrier frequency 10MHz.

d. Amplitude 1V.
e. Offset 0V.

5- Make the following adjustment on 2235A oscilloscope :
   a. Time base 0.2 us.
   b. Vertical sensitivity 0.5V.

6- Make the following adjustment on HP5372A Frequency and Time Interval Analyzer.
   a. Press **PRESET** key.
   b. From the softkey choices select **FREQUENCY**.
   c. Using the knob, move the menu cursor to the **AUTOMATIC** field, then press the softkey to select **SAMPLE**.
   d. Press the **INTERVAL SAMPLING** softkey.
   e. Using the knob, move the menu cursor to the **SAMPLE ARM** field and type 100 and press the **ns** softkey. This will specify the size of each frame analyzed.

7- To display the results:
   a. Press **GRAPHIC** key at the top next to the display.
   b. Press the **SINGLE/REPEAT** key (next to PRESET key) to freeze the display.
   c. Press the **TIME VAR** softkey, this will display the instantaneous frequency readings versus time as a discrete graph.
   d. To connect the data points, press **CONNECT DATA ON**.
   e. To add a grid, press **GRID ON**.
   f. Press the **MENU** softkey (top one) to select **MRKR,** this displays the marker options.
   g. Using the knob, move the horizontal markers to measure the positive steady state value and the positive transient peak.
   h. Press the **MENU** softkey (top one) to select **ZOOM,** use this to zoom on the area of interest and adjust the markers.
   i. Press **Marker Delta** softkey to measure the maximum overshoot.
   j. Change the **MARKER ORIENT** softkey to vertical and move the vertical markers to measure the settling time.

Experiment 4
Spectrum Analyzer

Objective

Getting familiar with the HP8592B spectrum analyzer and the HP8362A sweeper by observing a frequency varying signal generated by the sweeper on the spectrum analyzer.

Equipment

- HP8362A sweeper.
- HP8592B spectrum analyzer.

Overview of the spectrum analyzer

- **Frequency** key is used to set the center frequency displayed on screen.
- **Span** is used to set the range of frequency displayed. Note that the resolution and video bandwidth are coupled to the frequency span and are automatically adjusted to appropriate values for a given span.
- **Amplitude** is used to set the reference level which is the maximum amplitude displayed. Generally, placing the signal peak at the reference level provides the best measurement accuracy.

Overview of the sweeper

- **Preset** initializes the front panel settings.
- **Power level** is used to set the signal power in dBm.
- **CW** operation generates a single frequency low noise sinusoid.
- **Sweep Mode** generates a sinusoid with a time varying frequency. The start and the end frequencies are set using the **start** and **stop** keys.
- **Sweep time** sets the time needed to change the frequency generated from the start frequency to the stop frequency.
- **Single** will generate a sinusoid with a single frequency equal to the start frequency.
- **CONT** will start the sweeping mode.