

Microwave Engineering and Antenna Design

Lab Manual

Department of Electronic Science and Technology
College of Electronics and Information Engineering
Tongji University

Instructors: Dr. Huanlei Chen and Prof. Meisong Tong

June 1, 2018

《微波工程与天线设计》实验题目

电子科学与技术系

陈欢磊

利用 CST 或 HFSS 完成下列设计，题目难度按序号递增，所选题目难度与最终成绩挂钩

练习题（熟悉软件使用，必做）

- 1、设计一个半波长偶极子天线 (dipole antenna), 工作频率 10 MHz, 效率>90%, 材料理想金属。
- 2、设计一个喇叭天线 (horn antenna), 工作频率 8 GHz, 天线增益>15 dB, 材料铜 (电导率 $5.7E7$ S/m)。
- 3、设计一个螺旋天线 (helix antenna), 工作频率 100 MHz, 材料铜 (电导率 $5.7E7$ S/m) 辐射效率>80%

文献重现题（通过文献初步感受天线设计，选一完成）

- 4、根据文献 1 设计一个高增益微带线八木天线，将设计结果与文献 1 做比较。
- 5、根据文献 2 设计一个扁平化双波段贴片天线，将设计结果与文献 2 做比较。

6、根据文献 3 设计一个多波段 RFID 天线，将设计结果与文献 3 做比较。

简单天线设计（尝试设计简单天线，选一完成）

7、设计一个贴片天线（patch antenna），工作频率 24 GHz-25 GHz，天线增益 2 dBi，50 欧姆回波反射小于 -10 dB，铜的电导率 $5.7E7$ S/m，厚度为 0.03 mm。介质材料 $\epsilon_r=4.0$ ， $\tan\delta=0.1$ ，采用两种不同的厚度 $h=1$ mm 和 $h=0.3$ mm。比较两种厚度带来的不同

8、设计一个贴片天线（patch antenna），工作频率 76 GHz-81 GHz，天线增益 2 dBi，50 欧姆回波反射小于 -10 dB，铜的电导率 $5.7E7$ S/m，厚度为 0.03 mm。介质材料 $\epsilon_r=3.2$ ， $\tan\delta=0.008$ ，厚度 $h=0.125$ mm。最小线宽 0.15 mm。

面向应用的天线设计（自主选择合适的天线，选一完成）

9、设计一个电波暗室中使用的测试天线，工作频率 18 GHz，带宽 >1 GHz，天线增益 >20 dB，50 欧姆回波反射小于 -15 dB。

10、设计一个 wifi 天线，工作频率 2400 MHz-2483 MHz，天线增益 >3 dBi。

11、设计一个车载天线阵列，工作频段 76 GHz-81 GHz，天线增益 >10

dBi, 旁瓣低于主瓣 16 dB 以上。介质材料 $\epsilon_r=3.2$, $\tan\delta=0.008$, 厚度 $h=0.125$ mm。最小线宽 0.15 mm。

滤波器设计 (必做)

12、设计一个巴特沃思低通滤波器, 其通带为 0 到 3 GHz, 在 5 GHz 的衰减为 20 dB, 其特征阻抗为 75 Ω 。用 ADS 画出插入损耗随频率变化图。

13、设计一个 5 级切比雪夫低通滤波器, 其等波纹响应为 3 dB, 截止频率为 1 GHz, 阻抗为 50 Ω 。用 ADS 画出插入损耗随频率变化图, 并找出 0.6 GHz 所对应的损耗。

结题要求

1、课程设计报告: 包含设计目标, 天线模型, 天线增益或效率图表, 远场辐射图, 回波损耗图等各项反映天线性能指标的图表。滤波器电路图, 插入损耗随频率变化图。

包含滤波器电路图

2、CST 或 HFSS 项目文档, ADS 项目文档。

3、设计答辩: 准备 ppt 演示文稿, 现场解说, 若分组则小组成员需分别介绍自己的工作, 回答问题。

EXPERIMENT #1

OBJECTIVE:

To examine transient response in a transmission line, with different load terminations, for narrow and wide voltage pulses.

INTRODUCTION:

The transient analysis or the time domain reflectometry (TDR) becomes necessary when the incident and the reflected waves have to be resolved and studied separately in a coaxial cable. This can be achieved by applying a pulse at the input port and observing the pulses at the input and/or the output port. It can also be used to find out the rise time of a pulse which can be used to determine the maximum data rate that can be transmitted through the cable.

In this experiment the pulse amplitude should always be kept equal to 5V. The pulse width for the "narrow pulse" and the "wide pulse" will be 1 μ s and 5 μ s respectively. In order to get a narrow or a wide pulse, first remove the cable from the connector of the source and then use the oscilloscope to make proper adjustments in the pulse amplitude and the width.

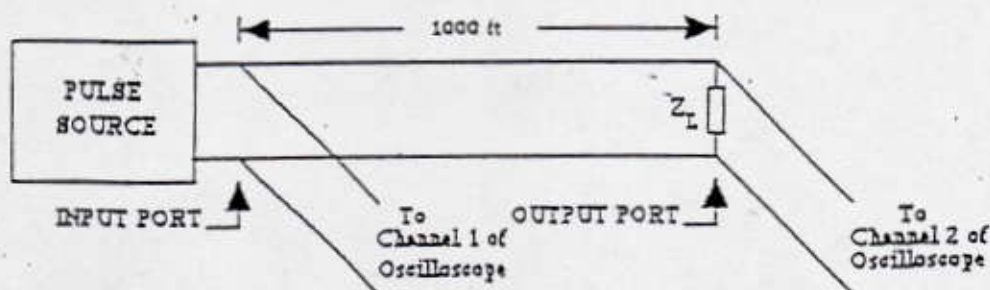
PRE-LAB WORK:

Calculate and tabulate $\Gamma = \frac{Z_L - Z_0}{Z_L + Z_0}$ for the following loads;

- i) Open circuit ii) Short circuit iii) 50 Ω iv) 25 Ω v) 100 Ω

Given that $Z_0 = 50 \Omega$.

SET UP: Set up the following



PROCEDURE & REPORT:

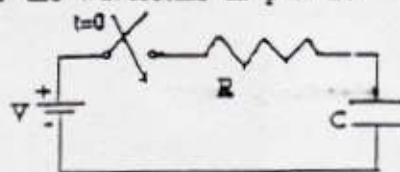
- 1) Note down the output impedance of the pulse generator. It is written on the pulse generator.
- 2) Apply the narrow pulse to one end (the input port) of a 1000 ft long coaxial cable. Simultaneously view the waveforms at the input and output ports. Sketch and dimension (show the pulse amplitudes, pulse widths and delays)

the waveforms at the input and the output ports for the following load (Z_L) resistors:

i) Open circuit ii) Short circuit iii) 50Ω iv) 25Ω v) 100Ω

Following each figure leave enough space to justify the shape of the waveforms observed at the input and the output port. Identify the incident and the reflected pulses. Explain any change in the amplitude and/or width of the pulse by using the appropriate value of the reflection coefficient as calculated in the pre-lab work.

- 3) With the 100Ω resistor at the output gradually increase the pulse width while observing the waveform of channel 1. Keep on increasing the pulse width till the incident and the reflected pulses well overlap each other. Now remove the 100Ω resistor and put a 50Ω resistor. Is there any overlap of the pulses? If not, why?
- 4) Repeat part 2 for the wide pulse.
- 5) Apply the narrow pulse to the coaxial cable. Put the provided capacitor at the output. Draw the waveforms at input and output.
- 6) Using one of the recorded waveforms to calculate the velocity of pulse propagation in the coaxial cable. Compare this with the speed of light. What is the relative permittivity of the dielectric used in the cable?
- 7) A reflected wave can again be reflected when it reaches back at the input port. This will cause multiple reflections. Were any multiple reflections observed in this experiment? If not, why?
- 8) Reflection diagram is a theoretical tool to predict the type of waveforms observed in this experiment. See section 9-5 of the text book (example 9-11) for more detail. Using reflection diagrams draw the input and output waveforms (to the scale) in order to justify the waveforms observed in parts 2 & 4 for 25Ω and 100Ω resistors only.
- 9) Assuming that the switch, in the following circuit, is closed at $t=0$ sec. Plot (v_0/V) vs (t/RC) , where v_0 is the voltage across the capacitor. Based on this plot, what are the distinguishing features of the waveforms observed in part 5 as compared to the waveforms in part 2?



QUESTION

An underground coaxial cable is being used for communication between stations A and B. The stations are 10 km apart. The cable develops a short circuit somewhere in between A and B. How can you precisely determine the position of the short circuit? Write down a step by step procedure and the relevant equations. How can you justify that the problem is a short circuit and not an open circuit?

EXPERIMENT #2

OBJECTIVE:

Study the reflection of plane waves & measure the reflection coefficient.

INTRODUCTION:

When a plane wave traveling in one medium (e.g. air) impinges on the surface of a second medium (e.g. water), the wave is partly transmitted into the second medium and partly reflected back in the first medium. The amount of reflection depends upon the angle of incidence and the polarization of the wave in addition to the material parameters of the two media. The level of reflection is represented by the ratio E_r/E_i , where E_r is the Electric field of the reflected wave and E_i is the electric field of the incident wave. This ratio is called the reflection coefficient Γ .

At a given position, the total electric field of an unpolarized plane wave varies randomly in time, while the total electric field of a linearly polarized wave traces a straight line. A wave can be polarized by letting it pass through a polarizing glass. The circular polarizing glass used in this lab has a notch on its edge. This notch is used to find out the direction of the electric field of the polarized wave as follows. The E-field of the polarized wave is in a direction normal to a line passing through the notch and the center of the circular glass. The wave is said to be "parallel polarized" if its E-field is in the plane of incidence only. It is "perpendicularly polarized" if its E-field is normal to the plane of incidence only.

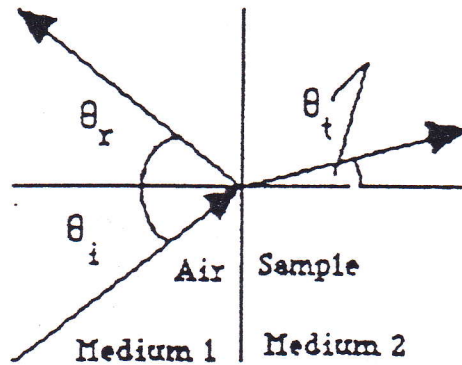
The plane of incidence is a plane, which is normal to the surface of the sample and contains the wave propagation vector.

PRE-LAB WORK:

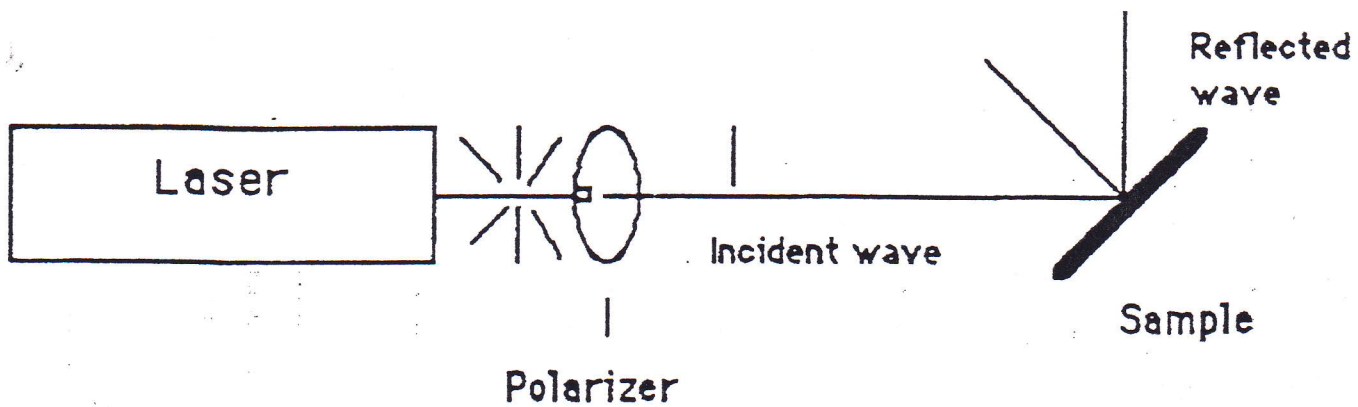
What do the following symbols stand for?

- | | | | | | |
|------------------|---------------|-----------------|-----------------|---------------|---------------------|
| a) θ_i | b) θ_r | c) θ_t | d) η | e) n | f) μ_0 |
| g) μ_r | h) μ | i) ϵ_0 | j) ϵ_r | k) ϵ | l) Γ_{\perp} |
| m) $\Gamma_{//}$ | | | | | |

- 2) A laser beam, traveling in air, is incident on the surface of a sample as shown in the figure below. Plot $|\Gamma_{//}|$ vs. θ_i and $|\Gamma_{\perp}|$ vs. θ_i for $0 \leq \theta_i < 90$. Take enough points to produce a smooth graph. Use the following parameters for the sample (notice that medium 1 is air):
i) $\epsilon_r=2.3, \mu_r=1$ (plastic) ii) $\epsilon_r=12, \mu_r=1$ (silicon).



Set up:



MEASURING THE REFLECTION COEFFICIENT:

In the lab, electric or magnetic fields cannot be measured directly. However, the incident and reflected powers can be measured and then used in the following relations to calculate the experimental value of Γ .

$$P_r = |\Gamma_{//}|^2 * P_i \quad (\text{for parallel polarized wave})$$

$$P_r = |\Gamma_{\perp}|^2 * P_i \quad (\text{for perpendicular polarized wave})$$

The expected difficulties in this experiment are in the measurement of accurate angles and powers (both incident and reflected). The polarization of the light from the lasers changes slowly with time and thus the power of the polarized beam changes also. Therefore, every time the reflected power is measured the incident power should be measured. Also, try to reduce the time elapsed between measuring P_i & P_r . This will help to normalize the effect of any change in power.

PROCEDURE & REPORT:

- 1) Learn to use the power meter. The sensitivity of the meter will have to be changed many times during the experiment, therefore different scale or multiplier should be used. To get consistent results, decide between you and your partner on how to read power from the deflection of the meter. Also learn to measure the angle of incidence using the protractor and the plastic disc. The disc has lines on it which are 5 degrees apart. Please remember that the angles are always measured from the normal on the surface of the sample.
- 2) Learn to parallel polarize and perpendicularly polarize the laser beam by using a polarizer. In the following steps the laser beam must be polarized.
- 3) Using the procedure explained above, measure the experimental value of $\Gamma_{//}$ and Γ_{\perp} for the provided sample to verify each graph of the pre-lab work at least at four points (the points will be specified on each graph by the TA). Put down the observations and results in a table as follows:

θ_i	P_i	P_r	$\text{Exp } \Gamma = (P_r/P_i)^{1/2}$	$\Gamma_{th.} \Gamma$ (from graph)	Error
------------	-------	-------	--	------------------------------------	-------

- 4) Plot the experimental values of $\Gamma_{//}$ and Γ_{\perp} on the corresponding graphs of the pre-lab work. Adjust the values of ϵ_r slightly in the equations of the pre-lab work to produce new graphs for $\Gamma_{//}$ and Γ_{\perp} which would better fit the experimental results. What value of ϵ_r produces the best fit? In the report, include the graphs produced with the new estimate of ϵ_r .

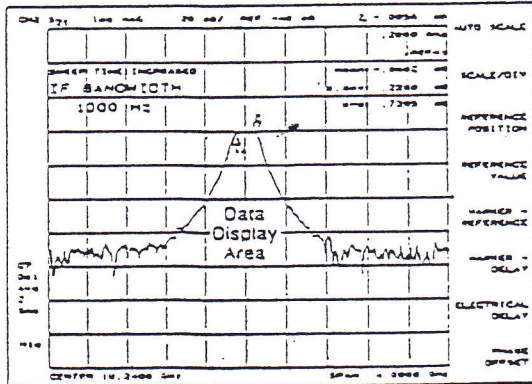
QUESTIONS

- Q1) Is the parallel polarized wave a special case of linear polarization?
- Q2) A laser beam, which is either parallel polarized or perpendicular polarized, is traveling in the air. How can you determine the polarization of the beam by using a Plexiglass sample?
- Q3) A beam from a laser, operating at 10^{13} Hz, is normally incident on the surface of distilled water. The relative permittivity of water is 80, the relative permeability is 1 and the conductivity is 2×10^4 S/m.
 - a) Is the water sample a good dielectric or a good conductor in this case? Why?
 - b) Calculate the magnitude of the reflection coefficient.

LAB 3: Network Analyser

Operating the Network Analyser

Getting Acquainted



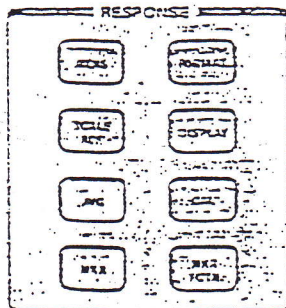
One of the more noticeable characteristics of the analyser is the simplicity of its front panel. Rather than individual keys for each of the many instrument functions, the analyser uses CRT displayed "menus" for operator input. These menus list the possible choices for a particular function, with each choice corresponding to one of the eight "softkeys" located to the right of the CRT. The "hardkeys" on the front panel provide access to the various menus, and are grouped by function.

Active Channel



The analyser has dual trace capability, with many of the measurement and display functions independently selectable for each trace. To modify the parameters of a particular trace, first select either [CH 1] or [CH 2], and then make the desired measurement choices. Note that the LED adjacent to the selected (active) channel is lit.

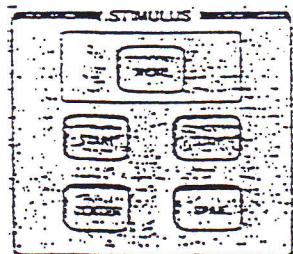
Response



The network analyser's receiver section is controlled with these keys. The [MEAS], [FORMAT], and [SCALE REF] keys are used to choose a measurement configuration, presentation format (amplitude and phase versus frequency, Smith chart, polar coordinates, etc.) and scale and reference values for a full screen display.

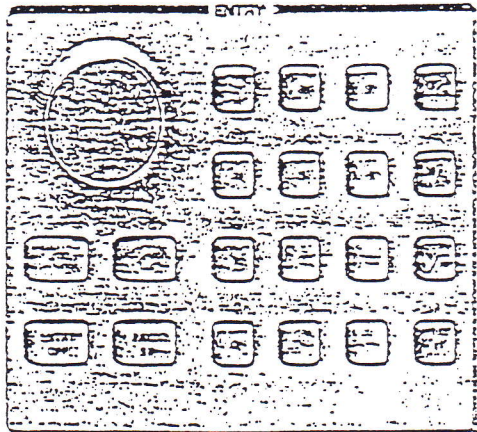
The other keys in the RESPONSE group enhance the usability of the measured data. The displayed traces may be overlaid, manipulated with math function keys, averaged, normalized, or read out at specific points along the trace with up to four independent markers per channel.

Stimulus



These keys are used to define the source output signal. Source frequency may be swept over any portion of the 130 MHz to 20 GHz range, at power levels between -10 and -65 dBm. The maximum frequency of the HP 8719A is 13.5 GHz. The stimulus keys also control the start and stop times in the optional time domain mode. The choices for sweep time, resolution, and sweep type are also selected here.

Data Entry



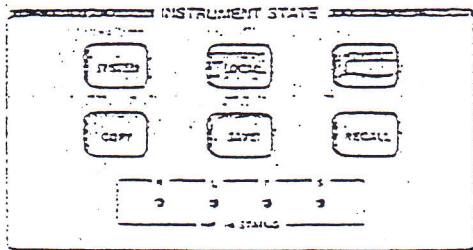
In some cases it is necessary to supply numeric values for a chosen parameter, such as frequency or amplitude. The ten digit keypad is used to supply these values. The keys to the right of the digits terminate the data entry with the appropriate units. The table below lists the data entry terminators and provides their significance for each parameter. In addition to entering data with the keypad, the knob can be used to make continuous adjustments, while the [←] and [→] keys allow values to be changed in steps.

Data entry terminators

Key Name	Frequency	Power	Time
G/n	GHz	—	ns
M/u	MHz	—	us
k/m	kHz	—	ms
x1	Hz	dB(m)	s

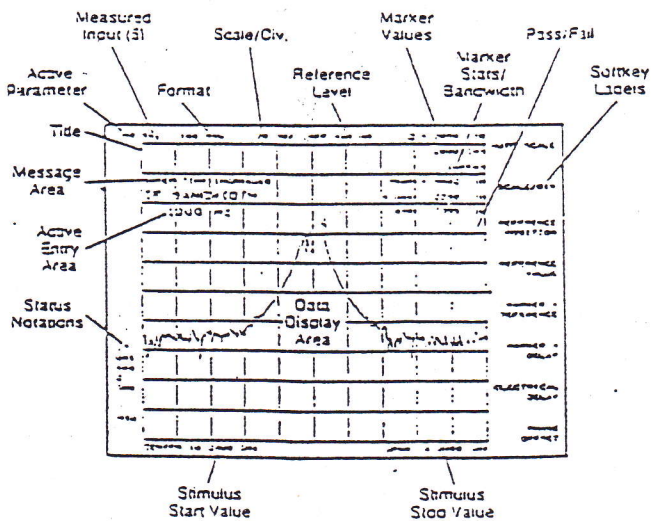
[x1] Always Basic Units for Current Active Function

Instrument State



Several utility functions are implemented with these keys, including instrument preset, front panel save/recall memory, HP-IB plotter and printer control, time domain transform (optional), and built-in diagnostic tests.

CRT Display



With the selection of each hardkey or softkey, the CRT is updated to display the current measurement configuration and status information. For those parameters not continually displayed on the CRT, select the appropriate key to display the parameter in the active entry area.

Status Notations

The status notations area of the CRT is used to show the current status of various functions for the active channel. The table below lists each notation and its meaning.

Notation	Definition
.	Measurement parameters changed: measured data in doubt until a fresh sweep has been taken.
Cor	Measurement calibration (correction) is on.
C?	Measurement calibration is on, but may not be valid.
C2	2-port measurement calibration is on.
C2?	2-port measurement calibration is on but may not be valid.
Hld	Hold sweep.
f	Fast sweep indicator.
Ext	Waiting for external trigger.
Avg	Sweep-to-sweep averaging is on. The averaging count is shown below the indicator.
Smo	Trace smoothing is on.
Del	Electrical delay has been added or subtracted.
Gat	Gating is on (time domain option).

General Measurement Sequence

Even with its wide range of capabilities, the analyzer is easy to operate. Common measurements can be set up with only a few front panel selections. This section describes a general approach to performing network measurements.

The following sequence is used throughout this User's Guide to illustrate the use of the analyzer in its various operating modes. The individual steps are discussed in detail in the sections that follow.

Preset

Return to a known state.

Connections

Set up the test configuration for the device-under-test (DUT).

Controls

Set up the instrument in three steps:

1. Measurement
2. Format
3. Stimulus

Calibrate

Characterize the systematic errors and remove their effect from the displayed data.

Save

Save the instrument configuration and calibration to save time and effort when recalling or resetting measurement states.

Measurement

Measure the performance of the DUT. Utilize the markers and marker functions to extract key measurement information.

Output Results

Create a permanent record by outputting the test results to a plotter or printer.

Step One: Preset

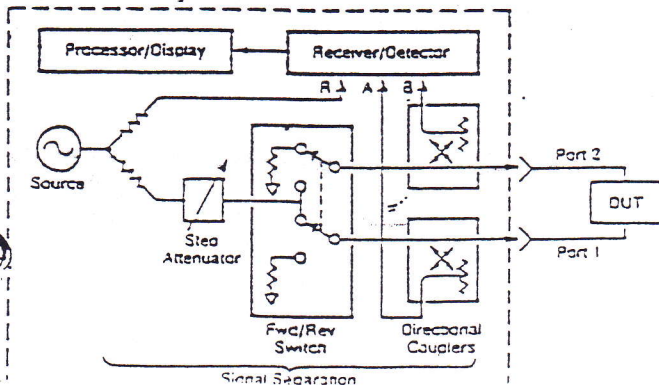
[PRESET] returns the instrument to a well-defined state. When [PRESET] is pressed, the analyzer performs a self test. Following successful completion of the self test, the instrument functions are set to the following preset condition:

Major default conditions at PRESET

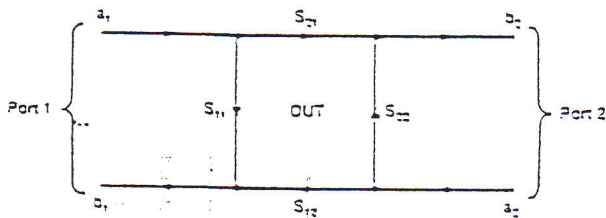
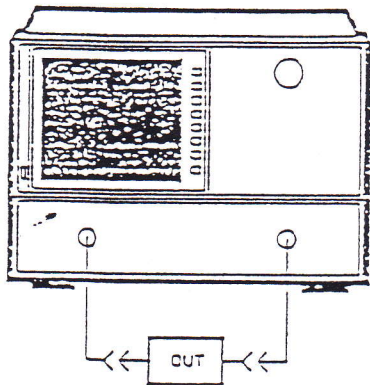
Display Channel:	Channel 1 On Channel 2 Off
Measurement:	S11 on Channel 1 S21 on Channel 2
Format:	Log Magnitude
Display mode:	Dual Channel Off
Scale:	10 dB/div
Reference:	0 dB, center of CRT
Stimulus Frequency:	Start 130 MHz Stop 20 (13.5) GHz
Sweep Type:	Linear Frequency
Number of Points:	201
Power:	-10 dBm
Sweep Time:	Auto, 575 ms
Receiver IF Bandwidth:	3 kHz
Averaging:	Off
Smoothing:	Off
Calibration:	Off

Step Two: Connections

Simplified Block Diagram



The analyzer contains all the hardware required to make simultaneous transmission and reflection measurements in both the forward and reverse directions. The only setup required is to connect the DUT to the two measurement ports. The network analyzer controls the switching functions, so that even reverse measurements can be made without changing device connections. The internal switch simplifies full 2-port measurement calibrations.



The [MEAS] key provides a selection of transmission and reflection measurements, with each choice corresponding to a specific S-parameter. The table below shows the measurement options and their associated S-parameters. The adjacent diagram indicates the signal flow represented by the S-parameters and how it relates to reflection and transmission measurements. When performing these measurements, all of the DUT's ports must be properly terminated.

The test direction column in the table indicates which measurement port provides the internal source signal output. For forward (FWD) measurements, the source signal is provided at port 1, and for reverse (REV) measurements, port 2. The signal path column in the table indicates the ratio measurement made. A, B, and R are the internal receiver/detector inputs shown in the above block diagram.

Measurement	S-Parameter	Test Direction	Signal Path
Input reflection (port 1)	S_{11}	FWD	A/R
Forward gain/loss (Transmission)	S_{21}	FWD	B/R
Reverse gain/loss (Transmission)	S_{12}	REV	A/R
Output reflection (port 2)	S_{22}	REV	B/R

Step Three: Controls

1. Measurement
2. Format
3. Stimulus

Measurement

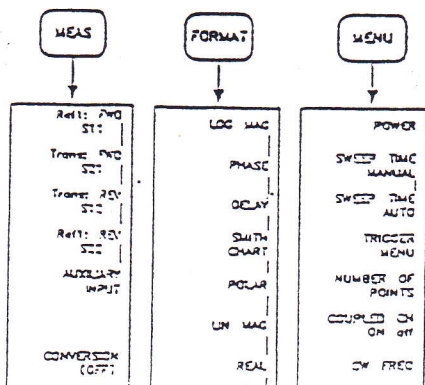
Based on the DUT connections, use the [MEAS] key to choose the appropriate measurement. Select one of four S-parameters or select the [AUXILIARY INPUT] softkey to measure the signal at the AUX INPUT (located on the rear panel).

The [CONVERSION] softkey provided in the [MEAS] menu allows you to convert the measurement data to impedance or admittance parameters. Use the [MEASURE RESTART] softkey to restart the measurement following a 2-port measurement calibration and connection of the DUT. Also use this softkey to restart a measurement's averaging count.

Use the [FORMAT] key to define the display format; logarithmic magnitude, phase, group delay, etc.

Next, specify the measurement frequency or frequencies using the stimulus group's [CENTER] and [SPAN] keys or [START] and [STOP] keys. For a narrowband measurement such as a bandpass filter, it is usually easiest to set a center frequency and total span; for measurements over a broader frequency range it may be easier to choose individual start and stop frequencies.

The stimulus [MENU] key also allows the additional choices of measurement port power level, sweep time, trigger type, number of measurement points, coupled or uncoupled operation of channels 1 and 2, CW (continuous wave)



Format

Stimulus

Step Four: Calibrate

Accuracy in network analysis is greatly influenced by the measurement system. Parts of the measurement setup such as interconnecting cables and adapters (as well as the instrument itself) all introduce variations in magnitude and phase that can mask the actual performance of the DUT.

The calibration step effectively characterizes and removes the effects of repeatable measurement variations in the test system. The "systematic errors" with which the calibration step is concerned are:

- Frequency Response (Tracking)
- Leakages (Directivity and Crosstalk)
- Mismatches (Source Match and Load Match)

The analyzer has several methods of measuring and compensating for these test system imperfections. Each method utilizes the measurement of standard devices and specific equations (error models) to remove one or more of the systematic errors mentioned above. By applying the data obtained from the standards to a specific error model, the calibration step is able to characterize the measurement system and thus increase the accuracy (by reducing the uncertainty) of the DUT's measurement results. The accuracy of the calibration step is dependent on the quality of the standards used for calibrating. Since calibration standards are very precise, excellent accuracy enhancement is achieved.

Normally, data taken during calibration is valid only for the instrument settings in place when the calibration was started. Therefore, after calibration, if a key measurement parameter (frequency range, number of measurement points, power level, etc.) is changed, the calibration becomes invalid, causing the analyzer to turn calibration off.

However, the HP 3719A and 3720B offer a "frequency subset" feature. If turned on, it allows the sweep to be narrowed to any subset within the frequency range used during calibration. The calibration will remain valid, and the measurement will be fully accurate without the need to re-calibrate. Only cardinal calibration frequencies are used (no interpolation), so the start and stop frequencies will be set to the nearest points used during calibration, and the number of points will be set to the number of exact calibration points left in-between.

Other items like sweep time and IF bandwidth may impact the calibration if changed, but will not cause the analyzer to turn calibration off. If either parameter is changed, the analyzer will only display "C?" or "C2?", indicating the calibration is in question and leaving the operator the decision of whether the calibration is valid or invalid.

Select the Calibration Kit

Prior to performing a calibration, ensure the analyzer is set for the calibration kit being used. When calibrating, the analyzer will use the mathematical model that applies to the devices contained in the kit to calculate the error correction coefficients.

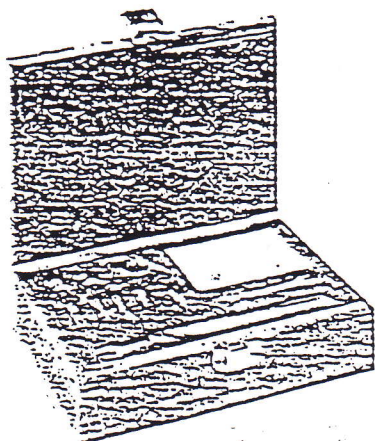
[CAL] [CAL KIT]

Access the calibration kit menu.

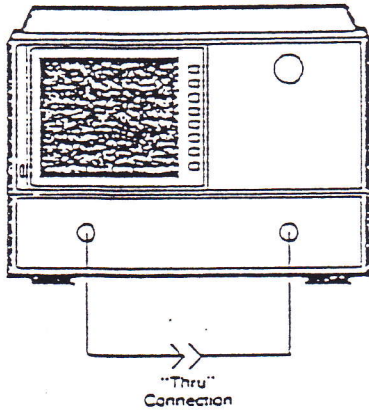
Select the softkey that corresponds to the kit being used.

[RETURN]

Exit the calibration kit menu. The selected calibration kit is displayed directly beneath the [CAL KIT] softkey.



Frequency Response Calibration



- More accurate than no calibration
- Establishes magnitude and phase reference for transmission or reflection measurements
- Requires 1 connection, 1 standard, 1 measurement

A frequency response calibration removes the repeatable frequency response and insertion loss variations of the test system. For transmission measurements substitute a "thru" connection for the DUT to establish a 0 dB loss (or gain) and a 0° phase reference. For reflection measurements substitute either an open or a short circuit for the DUT to establish a total reflection reference (0 dB return loss at either a 0° or 180° phase shift).

[CAL] [CALIBRATE MENU]
[RESPONSE]

To perform a frequency response calibration:

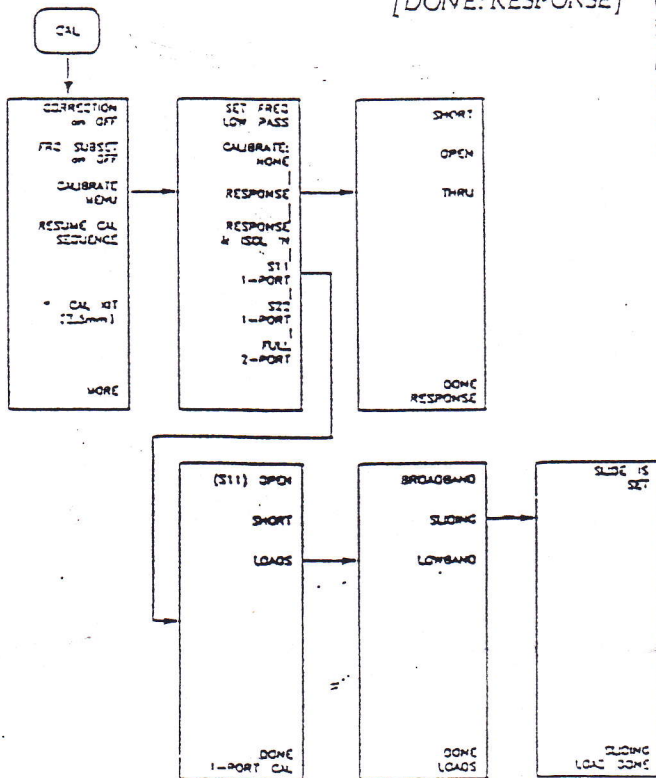
Access the calibration menu and select the response calibration.

Connect the standard to be measured. For this example, connect a "thru" (i.e., remove the DUT and directly connect the measurement port cables). When calibrating the measurement system, perform the calibration at the point of DUT connection (measurement plane).

Select the softkey corresponding to the connected standard (either [OPEN], [SHORT], or [THRU]). Once the standard is measured, the analyzer underlines the selected softkey. [DONE: RESPONSE] completes the calibration sequence.

Only one standard, the last one measured, is used to correct the frequency response variations of the test system. No open/short averaging is performed.

[THRU]
[DONE: RESPONSE]



1-Port Reflection Calibration

- More accurate than the frequency response calibration.
- Establishes magnitude and phase reference for reflection measurements.
- Requires 3 connections, 3 standards, 3 measurements.
- Good choice for reflection measurements of high insertion loss DUTs.

This calibration routine removes all three of the systematic error terms for a reflection measurement of a one-port device. It is best applied to a one-port device since it does not remove the mismatch effects seen from the second port of a two-port DUT. The output mismatch effects of the second port are negligible when the DUT's insertion loss is >6 dB.

To perform the 1-port reflection calibration:

[CAL] [CALIBRATE MENU]
[S11 1-PORT]

Access the calibration menu and select the [S11 1-PORT] (test port 1 calibration) or the [S22 1-PORT] (test port 2 calibration) softkey. The S11 calibration is used for this example. For an S22 calibration, all connections are made to port 2.

In turn, connect an open and short to port 1 and press the appropriate softkey after each connection. The selected softkey is underlined when the measurement is complete.

[LOADS]

The [LOADS] menu provides the following selection of loads: [BROADBAND], [SLIDING], or [LOWBAND]. Connect the load(s) you are measuring to port 1 and then select the appropriate softkey. For stop frequencies <3 GHz, use a fixed load (lowband or broadband). For start frequencies >3 GHz, use either a broadband or a sliding load. Finally, for calibrations covering frequencies above and below 3 GHz, use either a broadband load or a lowband and a sliding load. When choosing between the use of a broadband or a sliding load, you should determine the level of measurement accuracy required. Although more complex, a sliding load typically provides >40 dB effective directivity (best accuracy). A broadband load is simpler to use and provides approximately 35 dB effective directivity.

[DONE: LOADS]
[DONE 1-PORT CAL]

After measurement of the load(s), return to the 1-port calibration menu. The [DONE 1-PORT CAL] softkey completes the calibration sequence.

Upon completion of the 1-port calibration, the analyzer determines the contribution of all three error terms and removes their effect from the measured data. Note that the [CORRECTION ON off] softkey is turned on when the calibration procedure is complete and "Cor" is displayed in the status notations area of the CRT.

2-Port Measurement Calibration

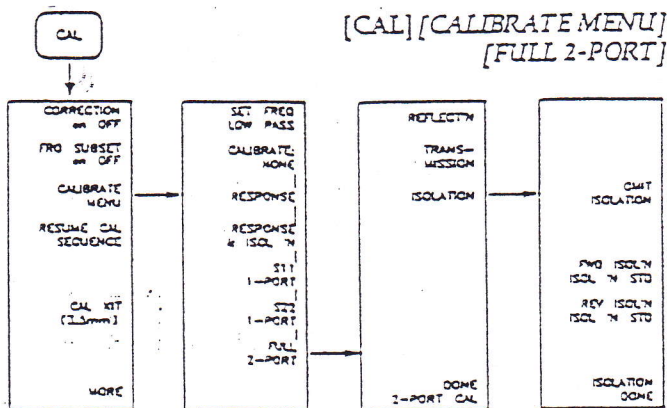
- Most accurate calibration procedure for multi-port devices.
- Establishes magnitude and phase reference for transmission and reflection measurements.
- Requires 7 connections, 4 standards, 12 measurements for full 2-port.
- Measures twelve error terms.

This is the most complete calibration. It measures the three systematic errors (frequency response, leakage and mismatch) in both the forward and reverse directions and removes their effects from the measured data. Full 2-port is the most accurate calibration for transmission and reflection measurements of multi-port devices.

To implement a full 2-port measurement calibration:

Access the calibration menu and select the full 2-port calibration.

At this point a menu is presented with the following options: *[REFLECT'N]*, *[TRANSMISSION]*, and *[ISOLATION]*. One at a time, select the *[REFLECT'N]* and *[TRANSMISSION]* softkeys and follow the prompts. The implementation of this portion of the 2-port calibration is very similar to the frequency response and 1-port calibration described above. The only difference is that both forward and reverse transmission measurements are made as well as reflection measurements on both test ports.



LAB 4

Antenna Near-field Measurements

1. INTRODUCTION

This manual describes the operation of the Nearfield Systems Lite Software. Near-field measurements provide a fast and accurate method of determining the antenna gain, pattern, polarization and provides the capability for computer controlled multi-frequency, multi-beam antenna measurements in a single scan.

Antenna measurements by near-field methods can provide significant advantages over competing techniques. These advantages include:

- Very high accuracy.
- High throughput.
- Complete characterization of the antenna performance.
- Control of zero G effects.
- Minimal real estate requirements.
- Elimination of delay due to weather.
- Measurements can be made in the antenna assembly area.
- Compatible with special project security requirements.

Planar near-field measurements provide a convenient method for determining the performance of medium to high gain antennas. Information available from near-field measurements includes:

1. Far-field pattern
 2. Antenna gain
 3. Antenna directivity
 4. Cross polarization level
 5. Beam width
 6. Beam pointing
 7. Phase center position
 8. Defocusing
 9. Autotrack (monopulse) bias, scale factor and linearity
 10. Phased array element excitation
 11. Reflector surface distortion measurements
-

1.1 SYSTEM OVERVIEW

Figure 1.1-1, shows a typical nearfield measurement system. This system includes three elements: a robotics scanner including an optics system for precise measurement of probe position, an RF subsystem, and a computer subsystem. The RF subsystem consists of a microwave receiver connected to a field probing antenna. The probe antenna is moved under computer control over a planar surface in front of the Antenna Under Test (AUT). The near-field range determines the equivalent far-field antenna performance through two basic steps:

1. It measures the AUT's phase and amplitude at known positions.
2. It applies a Fourier transform to the measured data to evaluate its performance in the far-field.

The NSI program has two main program sections; data acquisition and data processing. In addition, there are several other sections which help analyze and process the data.



Figure 1.1-1 Near-field Measurement System

1.1.1 Data Acquisition

The data acquisition program section consists of three major submodules:

1. Test description menus: - These menus define the type of test and the basic properties of the object under test. This information is also used by the system to automatically setup the test.
2. Scan pattern menus: - These menus let you define the desired scanning pattern. The setup can be automatic or manual.
3. Scan timing menus: - These menus control the timing of the measurement sequence. This menu is needed primarily for complex multibeam measurements.

1.1.2 Data Processing

The data processing section consists of four program modules:

1. Preprocessor: - Provides optional modifications to the raw measurements.
2. Far-field transform: - Converts the preprocessed near-field measurements to the far-field by a Fourier transform. Probe correction, polarization and coordinate systems are specified here.
3. Holographic transform: - Forms a planar field image from the far-field transformed data which can be translated to the antenna aperture for aperture diagnostics.
4. Graphics: - Displays the graphical output of the near-field, far-field and holographic data in a variety of formats such as: Grey scale, Contour, 3D, radiation distribution plots (RDP), and one dimensional cuts and listings. In addition, graphics files can be created for importing into word processing programs.

1.1.3 Calculators

This section includes a series of calculators used for general purpose computations. These calculators are for convenience only and do not affect program results.

2. NSI MENU SYSTEM

The NSI program communicates with you by using a series of nested menus. The Main Menu lets you call various submenus which in turn can call other options and menus.

Menu options are selected by typing the number or letter next to the desired option. At the top of each menu is a menu title and menu code. The menu is divided into two basic parts. The upper part contains numbered options and the lower part contains letter options. The upper part changes from menu to menu. The lower part, which always remains the same, contains general options. These options are:

<F1>	HELP
M	Main menu
<ESC>	Done
<F7>	Exit to interpreter

2.1 F1 - Help and Fault help keys

Most menus have context sensitive help which displays help based on the menu items selected. This help is displayed by pressing the <F1> function key. It gives you summary information that is associated with the current menu and options. During faults, the F1 text line is replaced with a blinking yellow F. Pressing F at this time will provide an explanation of the fault condition, it may also make recommendations to eliminate the fault condition. The <F1> key still works at this time but is not displayed.

2.2 M - Main menu key

The M key always returns you to the Main (highest level) Menu. In this documentation, we use M codes to tell you how to find a given menu. An M code is simply a key stroke sequence that will place you in a given menu. The first symbol is an M because that will place you at a known starting point (the Main Menu). Other numbers or letters navigate you through the menu system. As an example, <M01> will place you in the scan pattern menu. Pressing M will return you back to the Main Menu.

2.3 <ESCAPE> Key

The <ESCAPE> key, which is normally located at the top left side of the keyboard, will take you to a previous menu. If it is pressed enough times, you will return to the Main Menu.

2.4 Additional Real-Time Command Keys

Additional real-time commands are available in the menu system. A list of these is shown by hitting the H key. A list is shown in Table 2.1.4-1:

Table 2.3.4-1 Real-Time key list

Key	Function	Comment
F	Fault analysis	Displays cause of fault
G	Graphic display	Displays last graph selected
H	This help display	
I	Forth Interpreter	Type uppercase M to return to menus
M	Main menu	
P	Plot nth format	Enter selection 0-9,a-z or -
R	Receiver data plot	Plot RF data in current format
S	Measure List Edit	Same As M049
U	Units (English/Metric) toggle	
W	Wavelength mode toggle	
X	DOS shell	'exit' to return to program
^C	Abort transform or plot	May have to reset printer
^H	Hard copy mode	Plots output and switches back to CRT mode
^R	Restore state vector to nth	Enter selection 0-9,a-z or -
^S	Save state vector to nth	Enter selection 0-9,a-z or -

The ^ in the hot key means you hold the <Ctrl> key down while pressing the desired letter. For example, ^S means you hold the <Ctrl> key while pressing the S key.

2.5 Number Entry

Numbers are entered in some of the menu options. Numbers are entered using a combination of the following symbols:

0 1 2 3 4 5 6 7 8 9 . - e E

No other symbols should be used. Examples of valid numeric inputs include:

1 1.1 -1 1.2e5 -e4 -1E-14

3. MEASUREMENTS

3.1 How to make a single-pol near-field measurements

This section will serve as a tutorial on how to make a single-pol, single-frequency, planar near-field measurement. It assumes that you have already assembled the scanner, receiver and computer per the system interconnect diagram provided for your configuration. It also assumes that you are familiar with the menu operation and special key functions described in Section 1. The steps are outlined Table 3.2.1-1 and described in detail in the paragraphs that follow.

Table 3.2.1-1 Steps to making a single-pol near-field measurement

1.) Test setup Index scanner - <M016> Mount AUT Connect RF	4.) Begin measurement Enter comments - <M054> Select filename - <M052> Acquire data - <M050>
2.) Configure test Test title - <M000> Set AUT size - <M006> Set probe size - <M008> (M009) Set probe polarization - <M0086> (M0096) Set probe distance - <M008> (M009) Set max far-field angle - <M007> Auto scan setup - <M00A> (M00B) Set # probe pols to 1 - <M003>	5.) Transform data to far field Set pol type to E principal - <M1380> Select output to CRT - <M114> Select plot type - <M113> Plot pattern - <M115>
3.) Verify RF Operation Check receiver S/N - <M055>	() Menu numbers in parenthesis are for cylindrical & spherical tests.

Step 1 - Test setup:

The first step is to set up the test.

- 1.) Index the scanner. The first time the system is turned on for the day you should index the scanner. This makes the scanner's position known to the software. Since there is no direct encoder feedback and all measurements are relative, the scanner must be indexed whenever the software is restarted.
- 2.) Mount the AUT. In general, the AUT needs to be mounted from 3 to 5 wavelengths from the probe. Most of the time this distance can be set using a precision scale, though precise measurement is not important as long as a mechanical reference measurement is not required. The face of the AUT should be approximately parallel to the scan plane of the probe and in the center of the scanner.

- 3.) Connect the RF subsystem. Particular attention should be given to RF power levels at the probe, AUT and receiving system. Linearity measurements should be made prior to test if the configuration (i.e. AUT, probe, cables, power levels) has changed in the range.

WARNING: Often high near-field power densities can be created from amplifiers and low-gain probes. Always place amplifiers in standby mode before entering the near-field test area. In addition, do not look directly into the probe or AUT when the RF power is on.

Step 2 - Configure the test:

The next step is to configure the test, and specify how the data will be stored and annotated. Steps that are shown as "Not mandatory" are used in the automatic scan pattern set up mode. If you are not going to use the automatic scan setup you may skip these steps.

- 1.) Set the Test title <M000>. The Test title will show in bold italics at the top of each plot. Often the test title will include the name of the antenna and port being tested.
- 2.) Enter the AUT size in menu <M006> (Not mandatory). Enter the size of the antenna-under-test. This menu will help automatically determine the near-field scan size.
- 3.) Set the probe size using menu <M008> (M009 for cylindrical & spherical). Enter the probe's size or using <M0082> (or M0092) select from one of the standard open-ended waveguide (OEWG) options. This menu will help automatically determine the near-field scan size. This option is required in the probe correction model when an OEWG probe is used.
- 4.) Enter the probe polarization using menu <M008> (or M009). Enter the polarization axis, typically this axis is a polarization rotator or RF switch. Only linearly polarized probes are valid for this software option. Enter the Tau angle <M0087> (or M0097): This value is the angle between the probe's principle axis and the scanner X axis. The number in the left column corresponds to the first polarization and the right column is the second polarization. If <M002> is set to single-pol, only the left column will be displayed. Enter the probe correction model <M0085> (or M0095). This is the probe correction model to be used during processing. It is set before the measurement to remind you what conditions the near-field data was taken.
- 5.) Enter the probe distance using menu <M009> (or M00A), (Not mandatory). This is the distance between the tip of the probe and the AUT aperture. Note that for front-fed reflectors the aperture will be significantly further away from the probe because of the interference between the probe and feed or subreflector. If using cylindrical or spherical mode, set the maximum radial extent of the antenna using <M008>.
- 6.) Enter the Max far-field angle using menu <M007> (Not mandatory). This should be set to a value somewhat larger than the maximum angle which applies to far-field data. In general, these values should not be greater than 70 degrees. The greater the number, the larger the automatic scan size. A fault warning will display if the entered requirements exceed scanner size limitations.
- 7.) Use the Auto scan setup feature in menu <M00A> (M00B) (Not mandatory), it automatically calculates the scan size and sample spacing for menu M04. See section 3.1.2.4.

- 8.) Set the measurement to single-pol <M002>. If the probe's polarization axis is a rotator, setting this to 1 will make a single scan and cut the scan time in half. If the probe's polarization axis is a switch and it is switching while scanning, then setting the #pols to one will not affect the scan time.

Step 3 - Verify RF Operation

The next step is to verify the Signal source, receiver, and associated RF equipment are connected and operating properly.

- 1.) Verify that the receiver S/N is high enough for the intended far-field measurement. A good rule is that the principal near-field S/N will be about equal to the principal far-field S/N level for a -30 db sidelobe. The far-field S/N level due to receiver S/N can be computed with menu <M24>.

To increase the near-field S/N try one of the following:

- Minimize cable lengths.
- Increase receiver averaging. <M055>
- Verify that there are good connections to all components.

Step 4 - Begin Measurement

You are now ready to label a file and start the scan.

- 1.) Enter comments about the test in menu item <M054>. These are comments specifically about this test. They may include special test conditions or something else notable about the test. The first comment line will be displayed when you do a directory listing <M053> so choose a descriptive title. All three lines will be displayed on all plots.
- 2.) Select a filename using menu item <M052>. This is the name of the file for this scan. You have eight alphanumeric characters followed by a three character extension. You should choose a filename that is descriptive of the test and also allows for a family of tests. For example XSGH01.dat might be the first in a series of tests made on an X-band standard gain horn. Always end the filename with the .dat extension. This will make it easier to use the NSI file directory in the future. To set the default data file directory and see what filenames have already been used hit option <M053> and enter the path followed by *.dat. For example if the data file path were C:\DATA you would enter C:\DATA*.DAT. This then gives you a listing of all the data files in the C:\DATA subdirectory and sets the default data file directory to C:\DATA.
- 3.) Acquire the data using menu <M050>. This starts the scan. While scanning, the screen will display a "grey scale" plot that is updated (based on the deinterleaver setting) at the end of each ray. You can observe the amplitude data on the left half of the screen, and the phase data on the right.

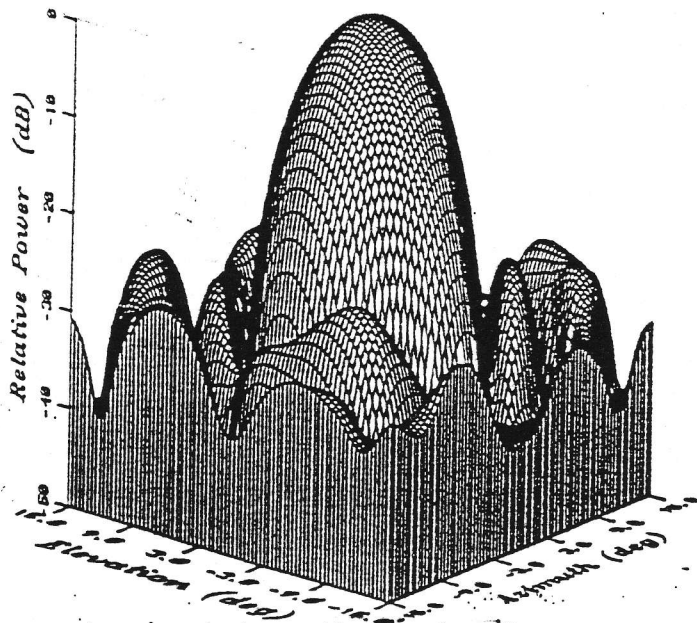
NOTE: If at any time you need to abort or stop the scanner, press the <esc> key.

Step 5 - Transform data to far field

The final step is to transform the measured data into a far-field pattern.

- 1.) Set the polarization to E principal using menu <M1380>. Only E-principal is valid for single-pol scans. E-cross is available for dual-pol scans. See Menu <M138> to set up the Far-field reference polarization options which defines E-principal polarization.
- 2.) Set the Data processing menu to far-field using menu <M110> which cycles between near-field, far-field and hologram.
- 3.) Select output to CRT using menu <M114>. This sets the output to the screen as opposed to a printer. This is usually preferred when looking at the plot the first time.
- 4.) Select the plot type as Greyscale <M113> or another desired plot type as long as there is enough data to make the plot type valid. Note that if V- or H-size <M135, M136> is set to 1, the contour and 3-D options turn red. Red means that this option is invalid, based on the V- and H-size set up values. If an error occurs that displays the message "Surf bad span", this means that the number of points in V and H size <M135 and M136> is inconsistent for a contour or 3-D plot. Make sure <M135 and M136> are not set to 1.
- 5.) Plot the pattern <M115>. This graphs the plot as set up in the menus. It can also be done using the "G" key from any menu. To make a hard copy, cycle menu option 4 until "draft hard copy" or "high quality hard copy" is displayed, then hit the "G" key. A simpler way to get a hard copy is just choose ^H.

Farfield Amplitude



1.1 Lab 5: Computational techniques: Modes in a waveguide

1.1.1 Prelab: Derivation of analytical formulas

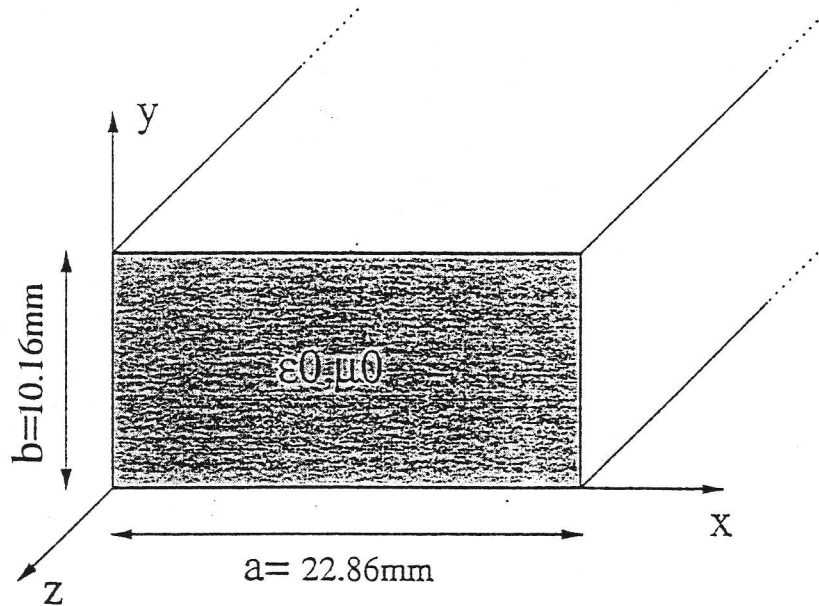


Figure 1.1: Geometry of an air filled X-band rectangular waveguide

In this lab we want to use the Finite Difference Method (FDM) to compute the first four TM^z modes of the standard air-filled X-band rectangular waveguide (see Figure 1.1). Since an exact solution exists to the problem we are solving, your prelab task is to derive the analytical expressions of the electric and magnetic fields, and the cutoff frequencies of the various modes (Figure 1.1). You can use the auxiliary magnetic vector potential for that purpose. The prelab will count for 50 % of this lab report grade.

1.2 Solution to this Eigen Value Problem (EVP) using FDM

1.2.1 Problem's statement

Mathematically the system we have to numerically solve is:

$$-\nabla_t^2 \psi = k_c^2 \psi \quad (1.1)$$

And

$$\psi(x, y) = 0 \begin{cases} \text{at } x=0, a \\ \text{at } y=0, b \end{cases} \quad (1.2)$$

To solve this EVP using the FDM, we convert the $-\nabla_t^2$ operator equation(1.1) to a matrix-based linear equation of the form:

$$[A]\{\psi\} = k_c^2\{\psi\} \quad (1.3)$$

where $[A]$ is the FDM representation of $-\nabla_t^2$, and $\{\psi\}$ is the FDM representation of the eigenvector ψ associated to the eigenvalue k_c^2 . Using Jacobi's FDM scheme $-\nabla_t^2\psi$ can be written as:

$$-\left(\frac{\psi_{i,j+1} - 2\psi_{i,j} + \psi_{i,j-1}}{(\Delta x)^2} + \frac{\psi_{i+1,j} - 2\psi_{i,j} + \psi_{i-1,j}}{(\Delta y)^2} \right)$$

Regrouping all the term on a same denominator,

$$\frac{-1}{(\Delta x)^2(\Delta y)^2} (\psi_{i,j+1}(\Delta y)^2 - 2\psi_{i,j}(\Delta y)^2 + \psi_{i,j-1}(\Delta y)^2 + \psi_{i+1,j}(\Delta x)^2 - 2\psi_{i,j}(\Delta x)^2 + \psi_{i-1,j}(\Delta x)^2)$$

Or,

$$\begin{aligned} & \frac{2((\Delta x)^2 + (\Delta y)^2)}{(\Delta x)^2(\Delta y)^2} (\psi_{i,j} - \psi_{i,j+1} \frac{(\Delta y)^2}{2((\Delta x)^2 + (\Delta y)^2)} \\ & - \psi_{i,j-1} \frac{(\Delta y)^2}{2((\Delta x)^2 + (\Delta y)^2)} - \psi_{i+1,j} \frac{(\Delta x)^2}{2((\Delta x)^2 + (\Delta y)^2)} - \psi_{i-1,j} \frac{(\Delta x)^2}{2((\Delta x)^2 + (\Delta y)^2)}) \end{aligned} \quad (1.4)$$

Let's put

$$\begin{cases} \alpha_0 = \frac{2((\Delta x)^2 + (\Delta y)^2)}{(\Delta x)^2(\Delta y)^2} \\ \alpha_1 = \frac{(\Delta y)^2}{2((\Delta x)^2 + (\Delta y)^2)} \\ \alpha_2 = \frac{(\Delta x)^2}{2((\Delta x)^2 + (\Delta y)^2)} \end{cases} \quad (1.5)$$

we can then write $-\nabla_t^2\psi$ as,

$$\alpha_0(\psi_{i,j} - \alpha_1(\psi_{i,j+1} + \psi_{i,j-1}) - \alpha_2(\psi_{i+1,j} + \psi_{i-1,j})) \quad (1.6)$$

1.2.2 Grid setup

We set up a regular grid in space. All the boundary conditions are Dirichlet, so that the total number of unknowns is $(N_x - 2) * (N_y - 2)$.

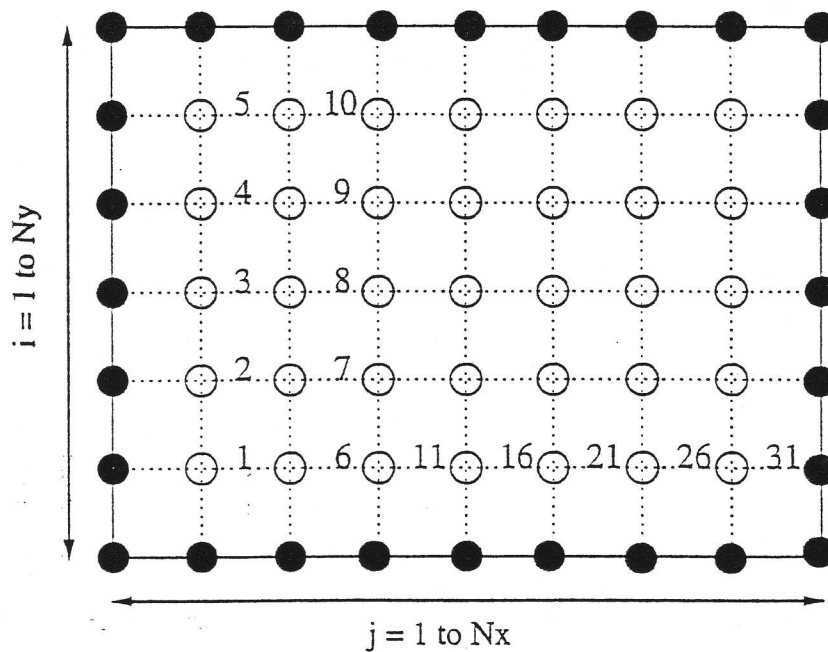
Since $\frac{b}{a} = \frac{22.86}{10.16} = \frac{9}{4}$, we can set up a regular grid using:

$$\begin{cases} N_x - 1 = \# \text{ of grid space in } x = 9q \\ N_y - 1 = \# \text{ of grid space in } y = 4q \end{cases} \quad (1.7)$$

or

$$\begin{cases} N_x = 9q + 1 \\ N_y = 4q + 1 \end{cases} \quad (1.8)$$

The next step is to give the explicit expressions of the FDM representation of ∇_t^2 , at all points of the grid.



● BC: Dirichlet known values

○ Unknowns

Figure 1.2: Grid scheme for $N_x=9$, $N_y=7$

1.2.3 FD representations of $-\nabla_i^2 \psi$

The global expression for $-\nabla_i^2 \psi$, for the Jacobi scheme was derived in equation(1.6). Since α_0 is a common factor for all terms, we need not compute it for each term. We will compute all the terms using the simplified expression below and then multiply [A] by α_0 .

$$-\alpha_1 \psi_{i,j+1} - \alpha_1 \psi_{i,j-1} - \alpha_2 \psi_{i+1,j} - \alpha_2 \psi_{i-1,j} + \psi_{i,j} \quad (1.9)$$

Considering the Dirichlet boundary conditions, we can distinguish nine different cases. In each case, we apply the above formula, but we take into account that $\psi = 0$ along the boundaries.

- for $i=2$ and $j=2$ (bottom left hand corner)

$$-\alpha_1 \psi_{2,3} - \alpha_2 \psi_{3,2} + \psi_{2,2}$$

- for $i=3, N_y-2$ and $j=2$

$$-\alpha_1 \psi_{i,3} - \alpha_2 \psi_{i+1,2} - \alpha_2 \psi_{i-1,2} + \psi_{i,2}$$

- for $i=N_y-1$ and $j=2$

$$-\alpha_1 \psi_{N_y-1,3} - \alpha_2 \psi_{N_y-2,2} + \psi_{N_y-1,2}$$

- for $i=2$ and $j=3, N_x-2$

$$-\alpha_1 \psi_{2,j+1} - \alpha_1 \psi_{2,j-1} - \alpha_2 \psi_{3,j} + \psi_{2,j}$$

- for $i=3, N_y-2$ and $j=3, N_x-2$

$$-\alpha_1 \psi_{i,j+1} - \alpha_1 \psi_{i,j-1} - \alpha_2 \psi_{i+1,j} - \alpha_2 \psi_{i-1,j} + \psi_{i,j}$$

- for $i=N_y-1$ and $j=3, N_x-2$

$$-\alpha_1 \psi_{N_y-1,j+1} - \alpha_1 \psi_{N_y-1,j-1} - \alpha_2 \psi_{N_y-2,j} + \psi_{N_y-1,j}$$

- for $i=2$ and $j=N_x-1$

$$-\alpha_1 \psi_{2,N_x-2} - \alpha_2 \psi_{3,N_x-1} + \psi_{2,N_x-1}$$

- for $i=3, N_y-2$ and $j=N_x-1$

$$-\alpha_1 \psi_{i,N_x-2} - \alpha_2 \psi_{i+1,N_x-1} - \alpha_2 \psi_{i-1,N_x-1} + \psi_{i,N_x-1}$$

- for $i=N_y-1$ and $j=N_x-1$

$$-\alpha_1 \psi_{N_y-1,N_x-2} - \alpha_2 \psi_{N_y-2,N_x-1} + \psi_{N_y-1,N_x-1}$$

1.2.4 Numbering the unknowns

To write the above expressions in a matrix form, it is necessary to assign each node in the grid a unique integer. In other words, we must order the unknowns. Considering Figure(1.2) the following numbering follows:

$$n = (j - 2) * (N_y - 2) + (i - 1)$$

and for the reciproc:

$$\begin{cases} j = \text{ceil}(\frac{n}{N_y - 2}) + 1 \\ i = 1 + n - (j - 2) * (N_y - 2) \end{cases}$$

1.2.5 Filling in the matrix and solving

Once the unknowns are numbered, and the 9 different cases identified, it is easy to fill in the matrix. We can notice that the matrix is sparse, so that it is appropriate to save it in a sparse form. This can be done using the matlab command: "A =sparse([],[],[],N,N,0)" where N is the total number of unknowns. The sparsity of the matrix can be observed using the matlab command "spy". The following figures show this sparsity (Figures (1.3 and 1.4).

We must then diagonalize [A], that is to say find its eigenvalues and the associated eigenvectors. Nevertheless, we need not completely diagonalize [A], since we are only interested in the first four eigenvalues. this can be done using the matlab command "[V,D,flag]=eigs(A,4,0)".

1.2.6 Visualization of the transverse fields

Electric field

The m file used to compute and plot the various electric and magnetic fields will be given to you by the TA. The shape of those graphs is similar to what we got using the analytical formulas.

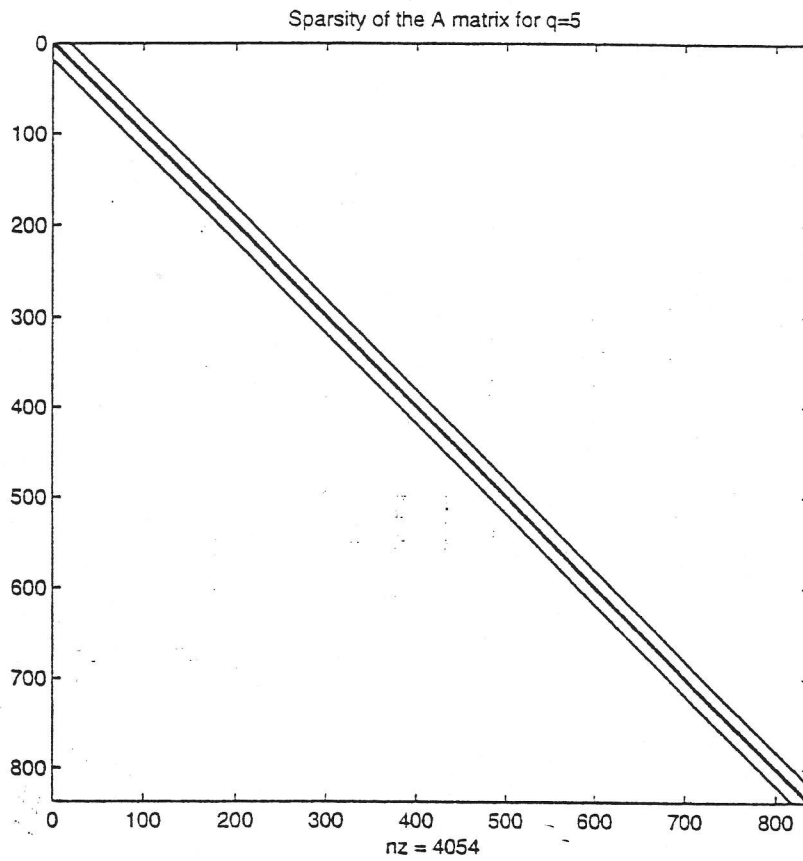


Figure 1.3: Sparsity pattern of the matrix $[A]$ for $q=5$

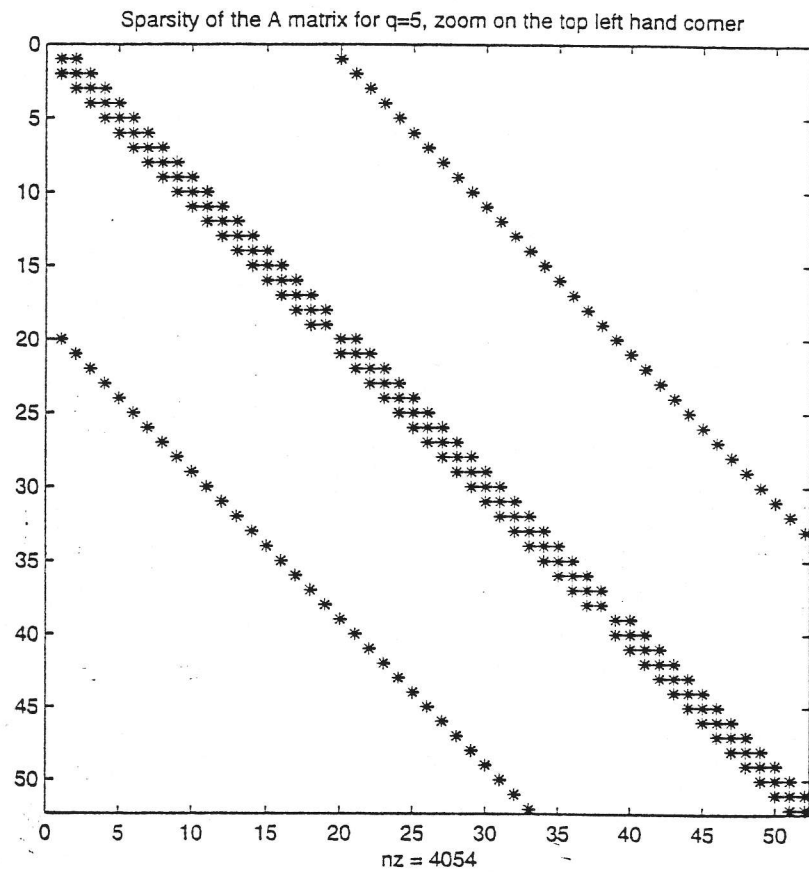


Figure 1.4: Sparsity pattern of the matrix $[A]$ for $q=5$, zoom

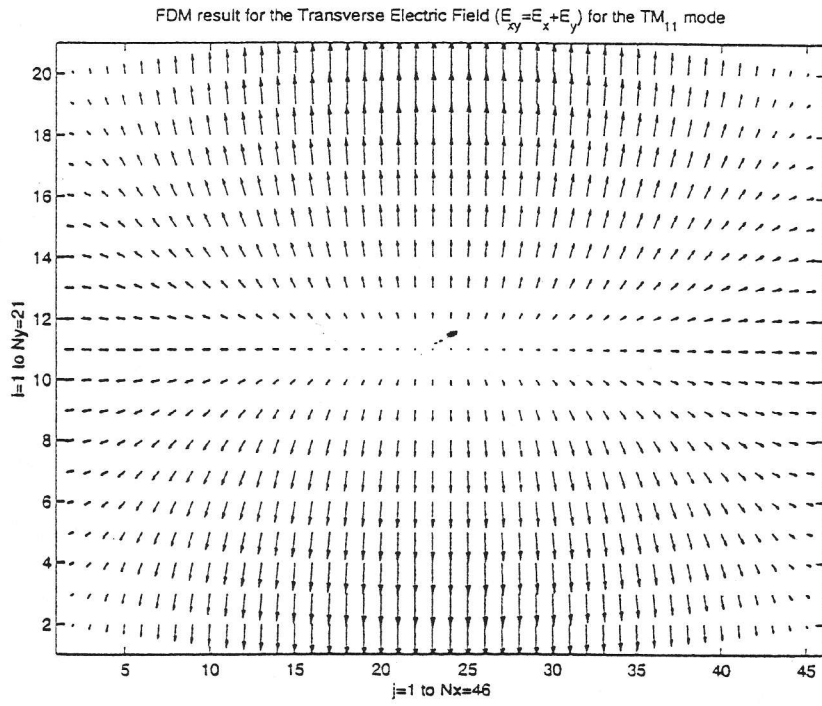


Figure 1.5: TM_{11} : FDM Transverse Electric Field for $q=5$

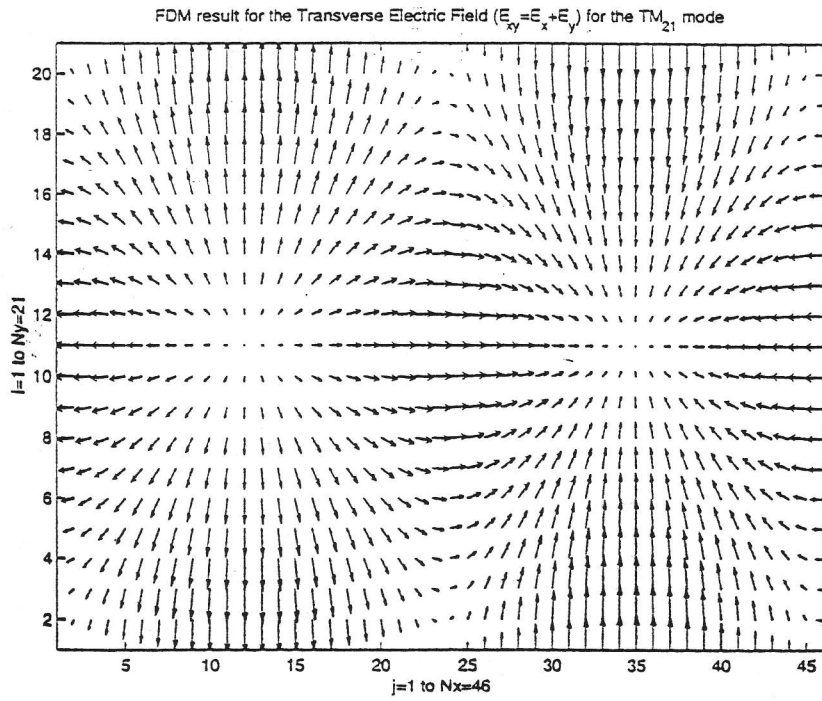


Figure 1.6: TM_{21} : FDM Transverse Electric Field for $q=5$

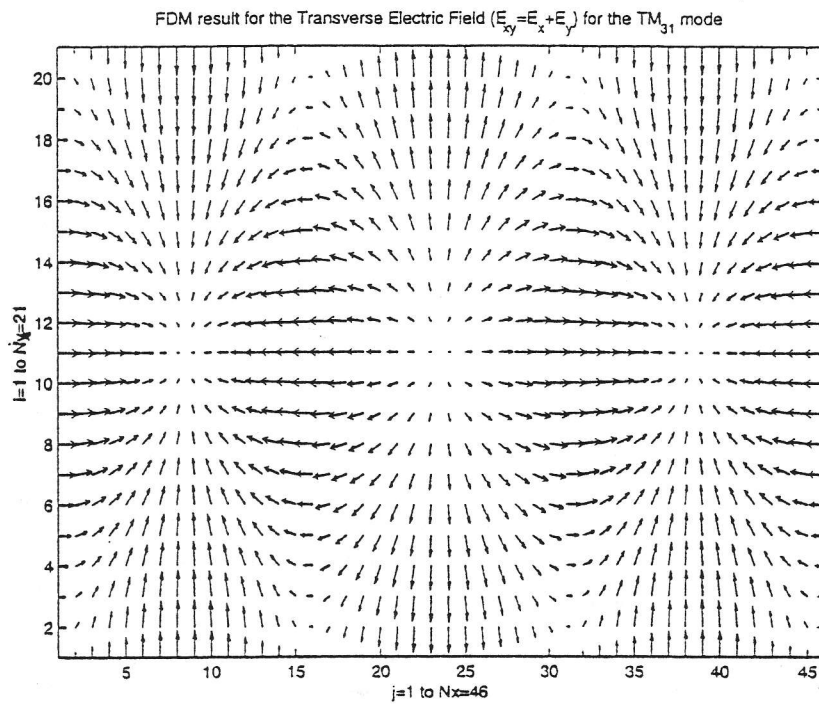


Figure 1.7: TM_{31} : FDM Transverse Electric Field for $q=5$

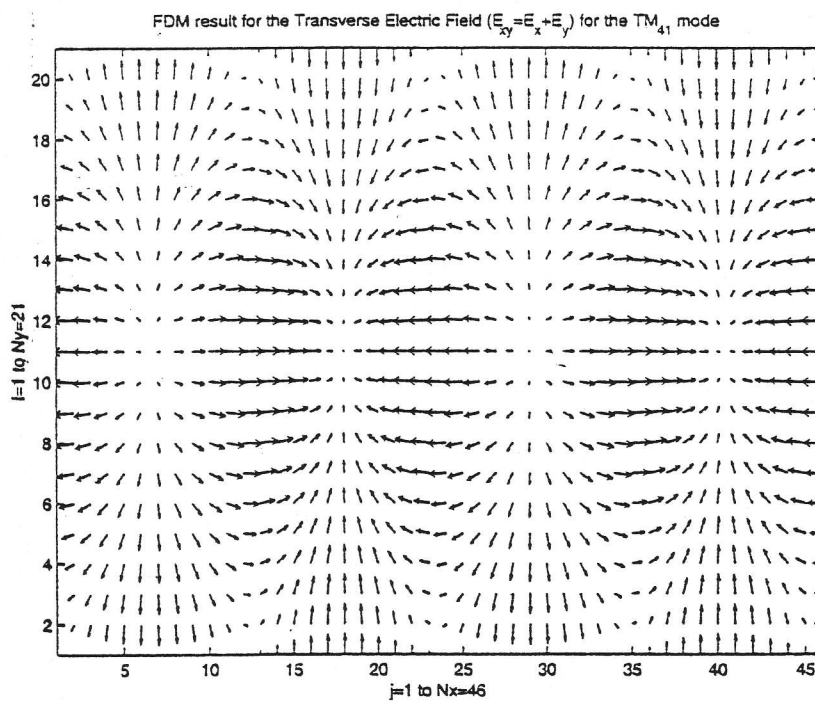


Figure 1.8: TM_{41} : FDM Transverse Electric Field for $q=5$

Magnetic field

The shape of those graphs is similar to , what we got using the analytical formulas.

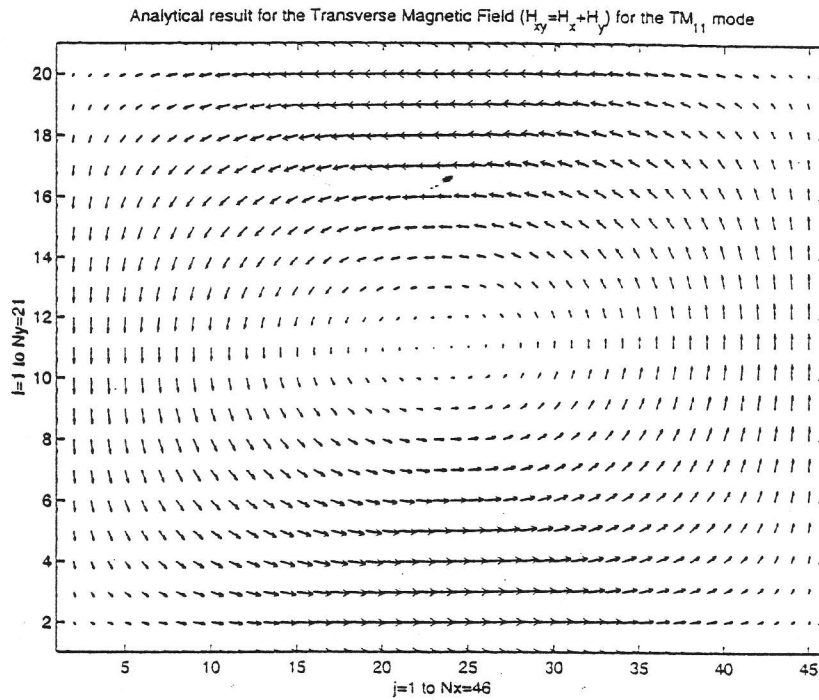


Figure 1.9: TM_{11} :FDM Transverse Magnetic Field for $q=5$

1.3 Report

- Attach the prelab assignment
- Compare the accuracy of the FDM solution with respect to the analytical solution

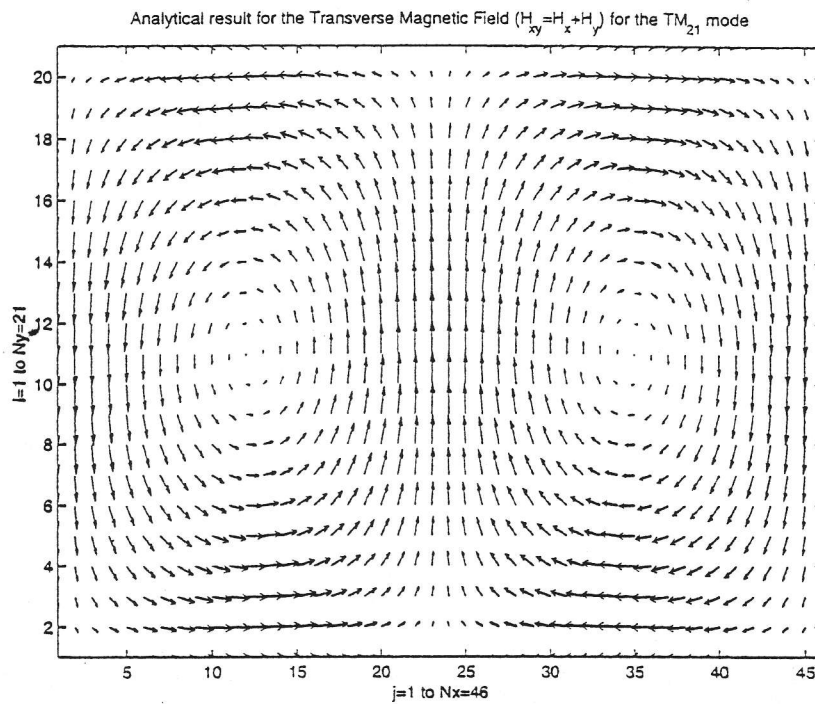


Figure 1.10: TM_{21} : FDM Transverse Magnetic Field for $q=5$

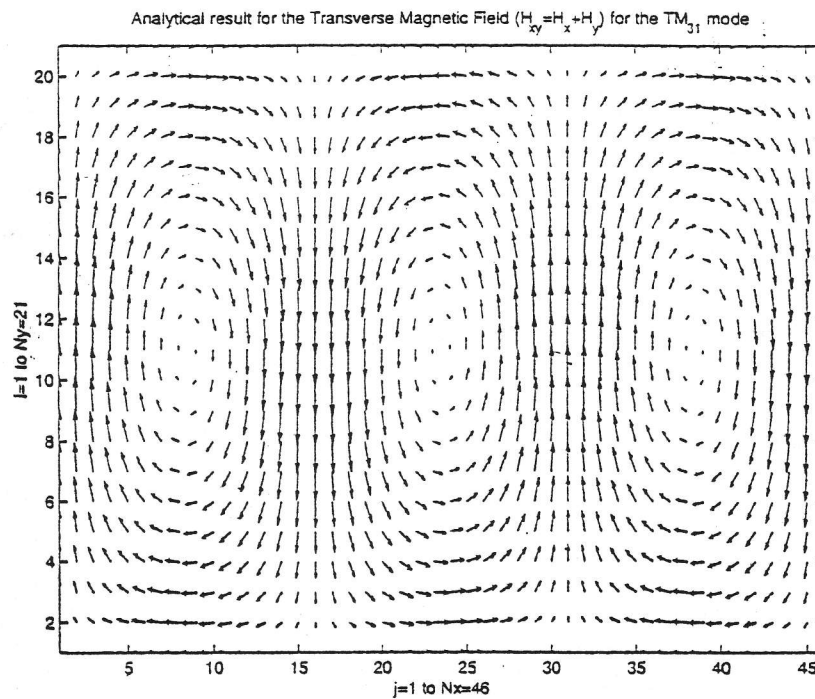


Figure 1.11: TM_{31} : FDM Transverse Magnetic Field for $q=5$

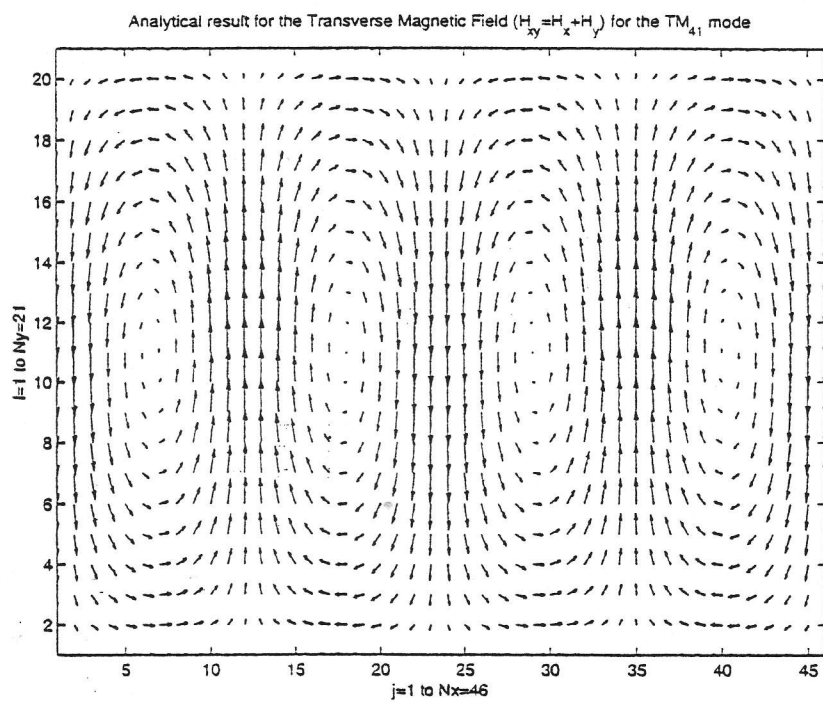


Figure 1.12: TM_{41} : FDM Transverse Magnetic Field for $q=5$

Bibliography

- [1] James T. Aberle, "Class notes for EEE598: Computational Electromagnetics",
<http://trc.eas.asu.edu/aberle/Classes/EEE498/lecture.html>, 1998
- [2] Constantine A. Balanis, "Advanced Engineering Electromagnetics," *John Wiley & Sons*, 1989, pp362-366
- [3] David M. Pozar, "Microwave Engineering," *Addison-Wesley publishing company* , 1993, pp147-153

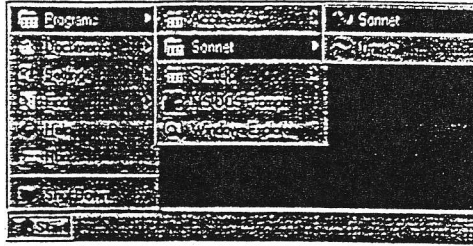
Lab 6: simulation of a microstrip filter using *Sonnet*

This lab was designed to give an overview of a simulation tool currently used in RF design.

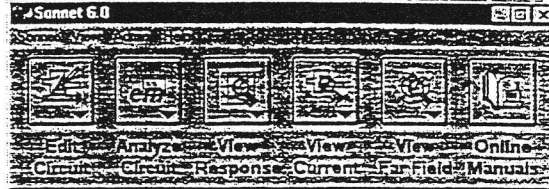
The circuit is a 2 port microstrip filter with a 15 mil Alumina substrate and 100 mils of air above. The surrounding box, which is used to short the resonators, has perfectly conducting sidewalls.

Windows

- 1 Select *Start* ⇒ *Programs* ⇒ *Sonnet* ⇒ *Sonnet* from the Windows desktop Start menu.



The Sonnet task bar appears on your display.



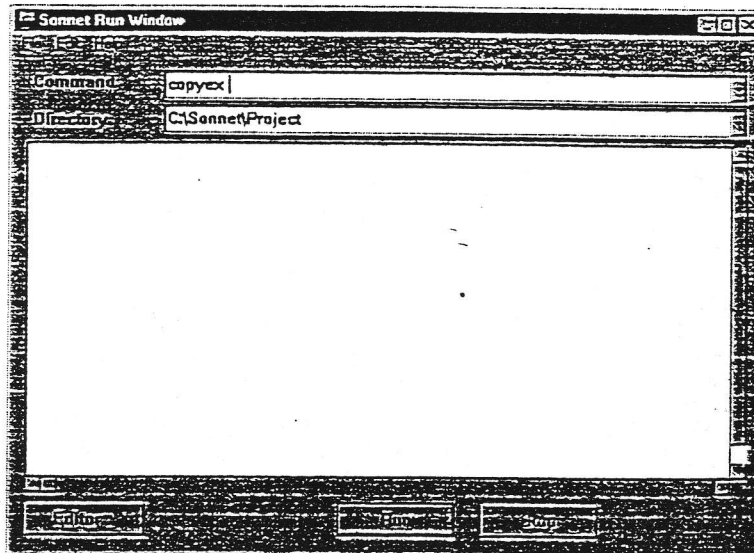
Obtaining the Example Files

Example files are supplied with your software. In order to avoid corrupting the example files, you must copy them into your working directory from the Sonnet examples directory to use them. When Sonnet is installed on a Windows system, a default project directory was made in which you could work. The example files will be copied there by default unless you specify another location.

A utility called *copyex* has been provided with your software which copies an example file into the specified working directory. To copy the first tutorial example file, perform the following:

- 1 Select *Sonnet* ⇒ *Copy Examples* from the main menu of the Sonnet task bar.

A Sonnet Run window will be opened.



- 2 Edit the Command text entry box to say "copyex filtwall.geo"

Be sure that the Directory text entry box is set to the default working directory on your system. For Windows, this is typically "c:\Sonnet\Program Files\Project".

For UNIX, enter your working directory in the Directory text entry box.



TIP

If you enter only "copyex" as the command, a list of all available example files is shown.

Xgeom - The Circuit Geometry Editor

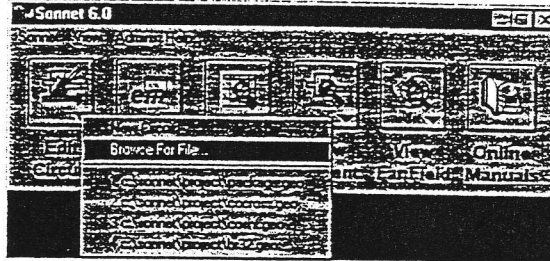
Xgeom provides you with a graphical interface which allows you to specify all the necessary information concerning the circuit you wish to analyze with the electromagnetic simulator, *em*.

Invoking *xgeom*



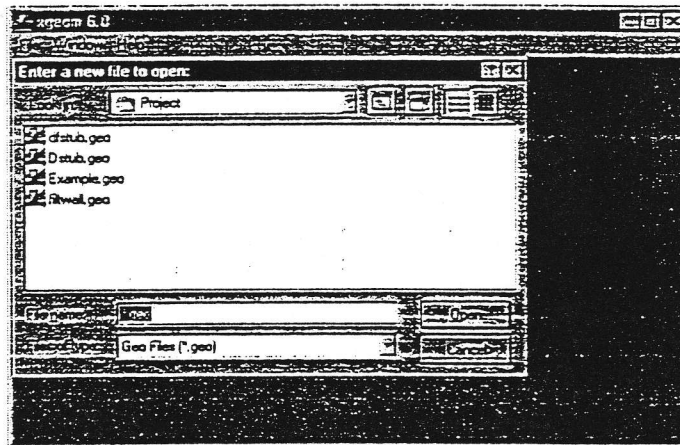
- 1 Click on the Edit Circuit button in the Sonnet task bar.

A pop-up menu appears on the task bar.



- 2 Select *Browse for File* from the pop-up menu.

The *xgeom* program window appears on your display along with the Open File dialog box.



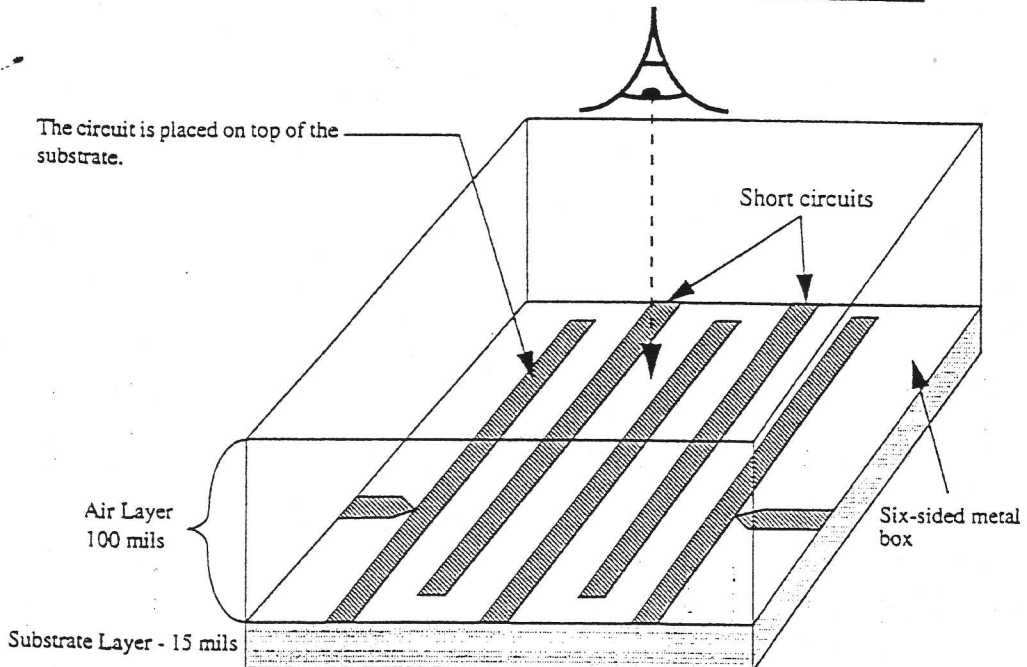
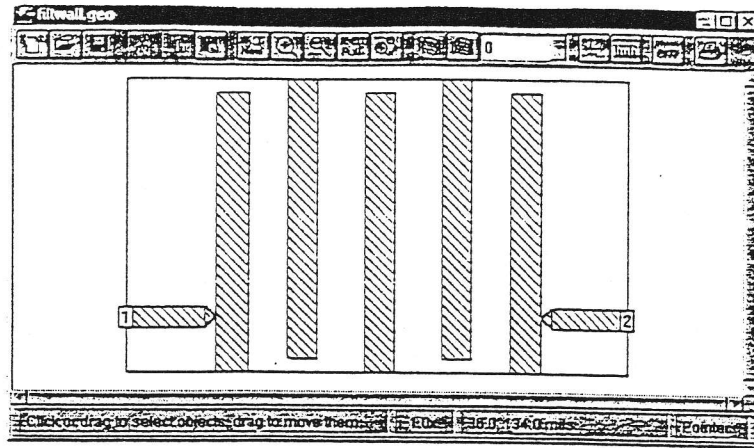
Opening a File in *xgeom*

The Open File dialog box allows you to select a file to edit. If the directory is not set to your working directory, use the controls on the dialog box to go to your working directory.

- 3 Double click on “*filtwall.geo*” to open the file.

An *xgeom* window appears on your display as shown below. The figure below depicts the same circuit in a 3-dimensional view in the six-sided metal box modeled in *xgeom*. The view shown in the *xgeom* window is a two-dimensional view from the top looking directly down on the substrate.

The example circuit, “*filtwall.geo*” is a two port microstrip filter with a 15 mil Alumina substrate and 100 mils of air above. Note that the resonators run to the edge of the substrate, shorting them to the box wall.



A three-dimensional view of the circuit in the six-sided metal box modeled in *xgeom*. The view shown in the *xgeom* window is a two-dimensional view from the top looking directly down on the substrate.

Zoom and Cell Fill

Xgeom has a zoom feature which allows you to take a closer look at any part of your circuit. You will zoom in on a section of the filter, to observe an example of staircase cell fill.

The cell fill represents the actual metalization analyzed by *em*. There are various types of cell fill, one of which is staircase. Staircase uses a "staircase" of cells to approximate any angled edges of polygons. Therefore, the actual metalization analyzed by *em* may differ from that input by you.

- 4 Select **View** ⇒ **Cell Fill** from the *xgeom* main menu to turn off the cell fill.

This command toggles the state of cell fill. Only the outline of the polygons are displayed.

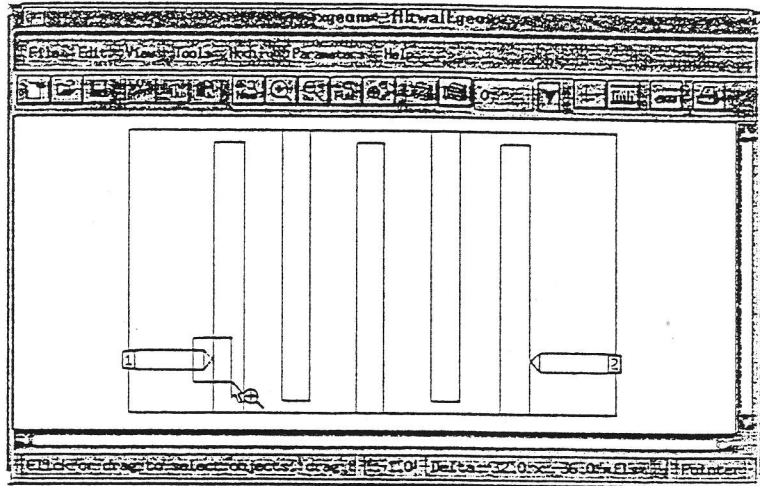


- 5 Click on the **Zoom In** button on the tool bar.

The appearance of the cursor changes. A magnifying glass with a plus sign, the icon on the Zoom In button, appears next to the cursor.

- 6 Select the section of the circuit you wish to zoom in on.

Move the mouse to the upper left of the junction of the feed line and the first resonator, as shown below. Click and drag the cursor down and to the right. A rubber band surrounding the area follows the cursor. When you release the mouse button, *xgeom* zooms in on the selected area.

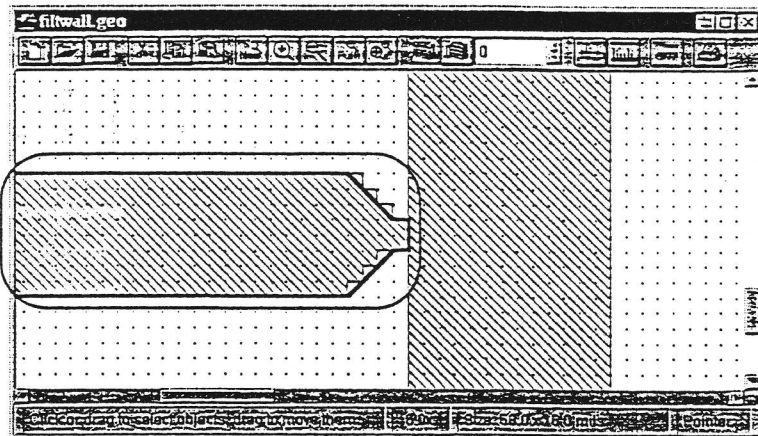


As you can see in the enlargement, one of the metal polygons has a diagonal edge.

- 7 Select *View* ⇒ *Cell Fill* from the *xgeom* main menu to turn on the cell fill.

The metal fill pattern appears in the polygons on your display. Note that the outline of the actual metal has a "staircase" edge which approximates the diagonal edge drawn in *xgeom*. The default setting in *xgeom* is staircase fill.

Note that *em* analyzes the staircase edged metal, not the diagonal polygon.



NOTE:

In some versions of Sonnet, diagonal fill is not available. In that case, skip to step 12. This will not affect simulation results since staircase fill is restored before running the analysis.

- 8 Click on the polygon circled above to select it.

The polygon is highlighted to indicate selection.

- 9 Select *Modify* ⇒ *Attributes* from the *xgeom* main menu.

The Metalization Attributes dialog box appears on your display as shown below.

- 10 Select *Diagonal* from the *Fill Type* drop list.

Diagonal fill type, while more accurately modeling the input edge, requires more processing time.

- 11 Click on the *OK* button to apply the changes and close the dialog box.

The metal fill is updated. As you can see, the metal *em* analyzes is closer to your input than in the case of staircase fill. But the increased accuracy comes at the

In this case, staircase fill provides the required degree of accuracy and will be used for the analysis. The fill type will be changed back to staircase later in the tutorial.



- 12 Click on the Full View button.

The whole circuit appears on your display.

Metal Types

Xgeom allows you to define any number of metal types to be used in your circuit. Multiple metal types may be used on any given level. A metal type specifies the metalization loss used by *em*. Both the DC resistivity and the skin effect surface impedance are accurately modeled in *em*.

Metal types are defined in the Metal Types dialog box where a fill pattern is assigned as part of the definition. For a detailed discussion of metal types and loss, see "Parameters - Metal types," page 138 in the *Xgeom* User's Manual.

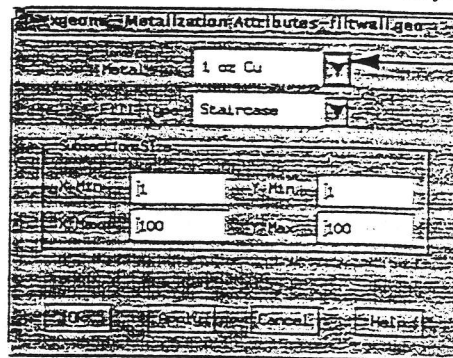
After a polygon is drawn in your circuit, you can change the metal type. In our example, all the polygons are comprised of "Lossless" metal. An example is given below of changing the metal type of one polygon to 1 oz. Copper.

- 13 Click on the any resonator of the filter to select it.

The polygon is highlighted to indicate selection.

- 14 Select *Modify* ⇒ *Attributes* from the *xgeom* main menu.

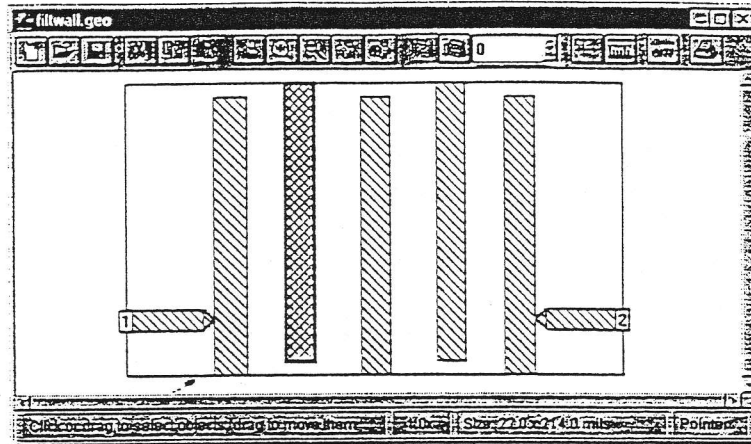
The Metalization Attributes dialog box appears on your display as shown below.



Metal Drop List

- 15 Click on the Metal drop list and select "1 oz Cu" from the list. Click on the OK command button to apply the change and close the dialog box.

The fill pattern changes for the selected polygon as shown below.



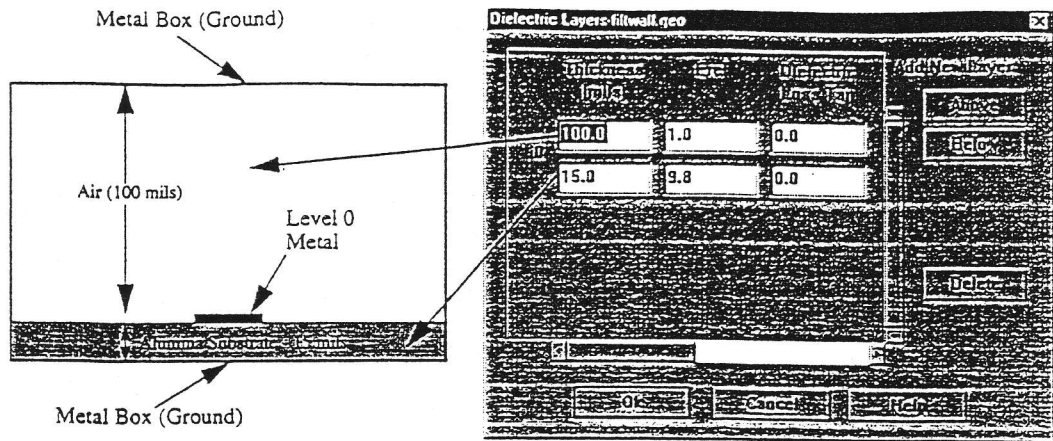
The metal for this polygon now has the loss associated with 1 oz. copper. *Xgeom* allows you to define any number of metal types for use in your circuit. For details on defining metal types, see "Parameters - Metal types," page 138 in the *Xgeom* User's Manual.

Note that 1 oz Cu is not a standard metal type, but was defined in *xgeom*, using *Parameters* \Rightarrow *Metal Types*, for use in this tutorial. You will not be analyzing with copper; the metal is switched back later in the tutorial.

Dielectric Layers

- 16 Select *Parameters* \Rightarrow *Dielectric Layers* from the *xgeom* main menu.

The Dielectric Layers dialog box, which allows specification of the dielectric layers in the box, appears on your display, providing you with an approximate "side view" of your circuit. The *xgeom* "level" number appears on the left. A "level" is defined as the intersection of any two dielectric layers and is where your circuit metal is placed.



Refer to page 11 for a three-dimensional drawing of the dielectric layers. The air layer is required to keep the metal top of the box away from the circuit metal.

- 17 Click on the OK command button to close the Dielectric Parameters dialog box.
- 18 Select *File* ⇒ *Revert to Saved* from the *xgeom* main menu.

This returns "fitwall.geo" to the original form which you copied from the example directory.

To reduce the time required to complete this tutorial, the simulation results are provided. In order to use the pre-computed results, there must be no differences in the source file. This step ensures that you are analyzing the original file and that you are able to successfully launch *em* from *xgeom*.

Em performs electromagnetic analysis for arbitrary 3-D planar geometries, maintaining full accuracy at all frequencies. *Em* is a "full-wave" analysis engine which takes into account all possible coupling mechanisms.

Em Control provides an interactive windows interface to *em*. This interface consists of menus and dialog boxes which allow you to select run options and execute *em* analyses in a Windows or UNIX environment. You may save the settings of the control interface in a job file.

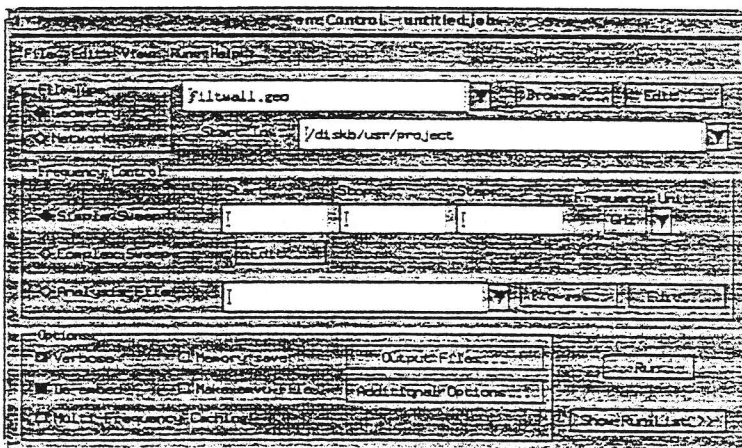
In the next part of this tutorial, you analyze the circuit "filtwall.geo" which you examined in *xgeom*. If you have not already done so, load "filtwall.geo" into *xgeom*.

Invoking the *em* Interface.

The *em* interface, *em Control*, may be invoked directly from *xgeom*.

- 19 Select **File** ⇒ **Analyze** from the *xgeom* main menu.

The *em* main window appears on your display as shown below, with an open job file. The file to be analyzed, "filtwall.geo," is entered in the File text entry box and the file type is already set to Geometry.



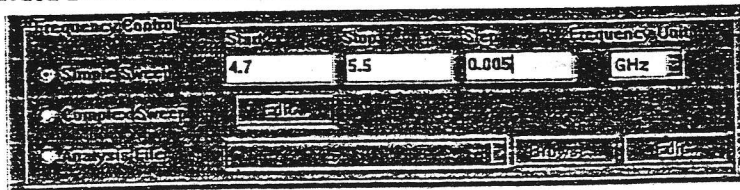
Setting Up Analysis Frequencies

- 20 Click on the Simple Sweep radio button in the Frequency Control section of the job window.

This enables the Start, Stop, and Step text entry boxes and the Frequency Unit drop list.

- 21 Enter the Start, Stop and Step values in the appropriate text entry boxes.

Your analysis starts at 4.7 GHz, increases in steps of 0.005 GHz, and ends at 5.5 GHz. Enter 4.7 in the Start text entry box, 5.5 in the Stop text entry box, and 0.005 in the Step text entry box. The frequency units should already be set to GHz, so no action needs to be taken for this item.



Run Options

Run options for *em* are available in the *em Control* interface. This example uses the Verbose, and De-embed options, which are set by default, as well as the Multi-Frequency Caching option.

The Verbose option causes *em* to display messages which describe the present state of the simulation, in the output window during program execution.

De-embedding is the process by which the port discontinuity is removed from the analysis results. Inaccurate data may result from failing to implement this option, even when you are not using reference planes. For a detailed discussion of de-embedding refer to Chapters 6 and 7 in the *Em User's Manual*.

If this option is on, an output file containing the de-embedded response data is produced. The name defaults to the circuit geometry file basename with a ".d" extension. For this example, since the input file is "filtwall.geo" then the response file is automatically named "filtwall.d".



TIP

The Memory save option (not used in this example) uses less memory and speeds analyses by storing matrix elements as single precision numbers rather than double precision. Its use is recommended in order to execute a simulation that otherwise might not be possible within the bounds of your memory limitations. Its use may result in an imperceptible ripple, or noise, in calculated data, except at low frequency where the effect may be more pronounced.

- 22 Set the Multi-Frequency Caching option by clicking on the Multi-Frequency Caching checkbox in the *em Control* job window.

When analyzing a large circuit over many frequencies, as is the case in this example, the multi-frequency caching option reduces processing time. When this option is selected, *em* pre-computes the frequency independent data, then uses this data during the matrix fill portion of the simulation. This option is not effective for analyses which contain four or fewer frequencies. For a detailed description of Multi-Frequency caching, see "Using Multi-Frequency Caching," page 20 in the *Em User's Manual*.

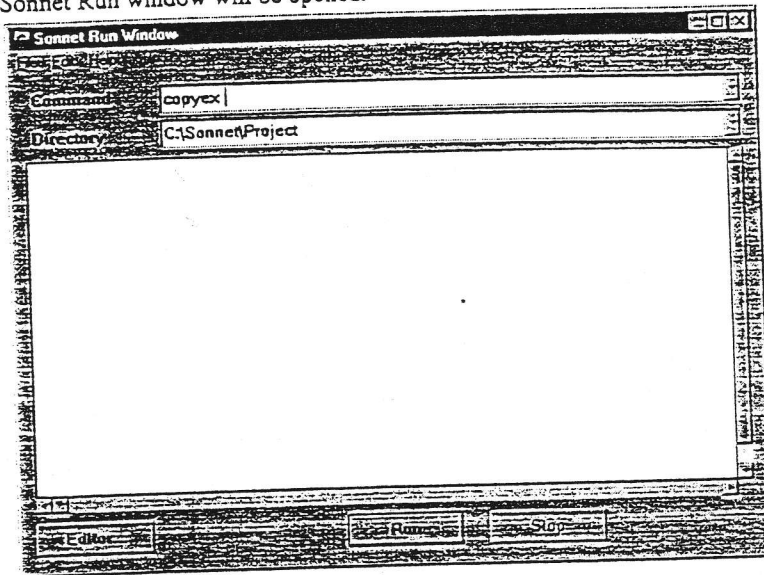
NOTE:

The Multi-Frequency Caching option is only available with a full Sonnet *em* suite.

Executing an Analysis Run

The input geometry file, control frequencies and run options are all specified. This analysis may require over an hour to calculate all the data points depending on the platform on which the analysis is performed. For the sake of expediency, the output files of this analysis are included with your example files.

- 23 Select *Sonnet* ⇒ *Copy Examples* from the main menu of the Sonnet task bar.
A Sonnet Run window will be opened.



- 24 Edit the Command text entry box to say "copyex filtwall.*"

Be sure that the Directory text entry box is set to the default working directory on your system.

This copies the original geometry, S-parameter and current density files.

This returns "filtwall.geo" to the original form which you copied from the example directory. To reduce the time required to complete this tutorial, the simulation results were provided. In order to use the pre-computed results, there must be no differences in the source file. This step ensures that you are analyzing the original file.

- 25 Click on the Run button in the *em Control* job window to execute the analysis.

The *em* output window appears on your display. Messages indicating the progress of the analysis are output since the Verbose option is set. Since the S-parameter data already exists from the previous copy, *em* does not actually need to calculate data. The following messages appear in the output window, as shown below:

Analyzing 4.700000 GHZ.

Using consistent data from pre-existing file filtwall.geo.

Analyzing 4.705000 GHZ.

Using consistent data from pre-existing file filtwall.geo.

Analyzing 4.710000 GHZ.

Using consistent data from pre-existing file filtwall.geo.

Analyzing 4.715000 GHZ.

Using consistent data from pre-existing file filtwall.geo.

Analyzing 4.720000 GHZ.

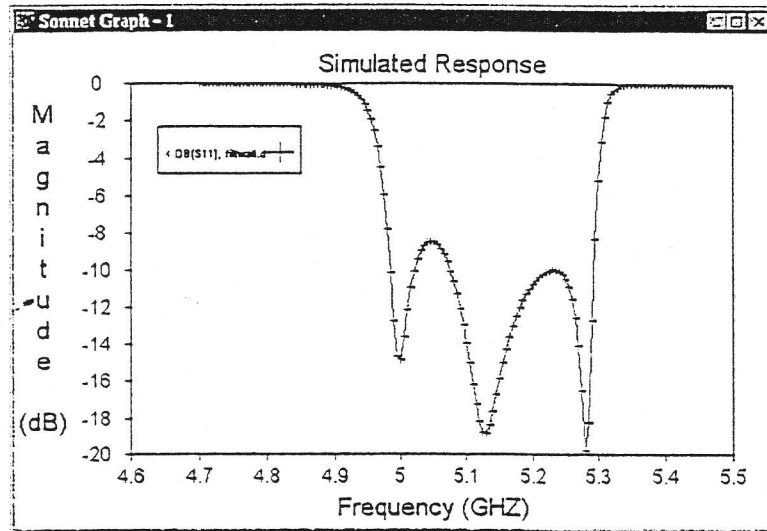
Emgraph - Plotting Your Data

Emgraph is used to display the results from an *em* analysis as either a Cartesian graph or Smith chart. S-, Y- and Z-Parameters can be displayed alone or simultaneously. You can also display multiple files on a single plot or choose to open multiple plots at the same time.

Invoking *emgraph*

- 26 Click on the Open Graph button at the bottom of the *em* output window to invoke *emgraph*.

The *emgraph* window appears on your display with a Cartesian graph of "filtwall.d" open with S11 displayed.



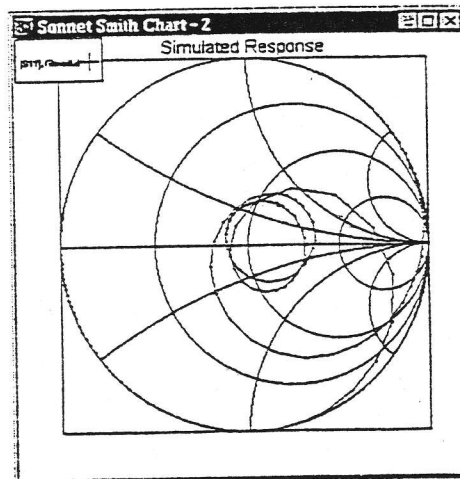
Opening a Smith Chart

- 27 Open a Smith Chart by selecting *File* ⇒ *Open* ⇒ *Smith* from the *emgraph* main menu.

The Open File dialog box appears on your display.

- 28 Double click on "filtwall.d" in the Open File dialog box.

A window containing a Smith chart of "filtwall.d" appears on your display. Again, the S11 parameter is displayed. This parameter is the default curve displayed upon startup.



Selecting Another Curve for Display

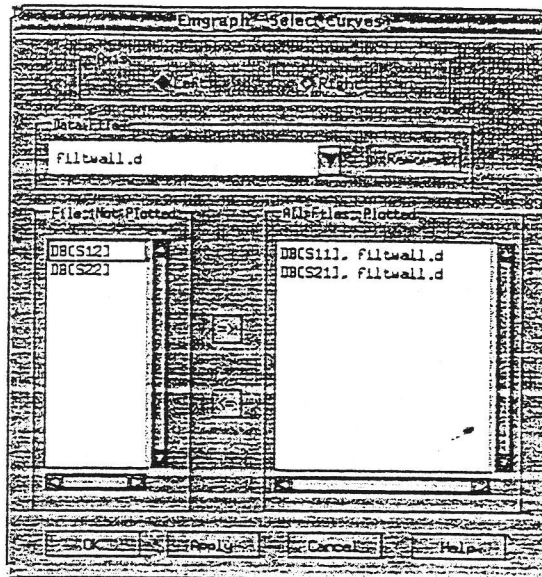
You may display multiple curves from multiple files simultaneously. To select a different curve for display, perform the following.

- 29 Click on the title bar of the Cartesian graph window to make it active.

This is indicated by the highlighting of the title bar.

- 30 Select *Graph* ⇒ *Select Curves* from the *emgraph* main menu.

The Select Curves dialog box appears on your display as shown below.



- 31 Double-click on DB[S21] in the Not Plotted text entry box to move it to the Plotted list.

DB[S21] moves to the Plotted list.

- 32 Click on the OK command button to apply these changes and close the Select Curves dialog box.

The Cartesian graph is updated to display S11 and S21. Since the axes are set to autoscale, the values on the axes are automatically adjusted.

- 33 Select *Graph* ⇒ *Attributes* from the *emgraph* main menu.

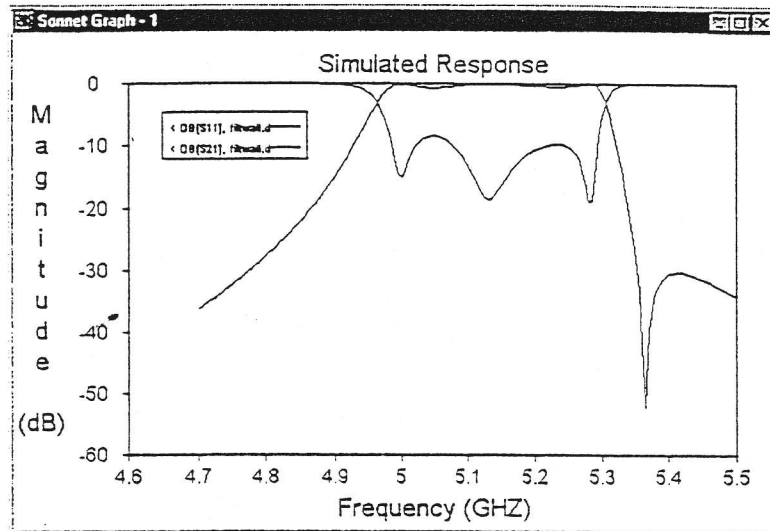
The Graph Attributes dialog box appears on your display.

- 34 Click on the Symbols checkbox to turn off the display of symbols.

Since there are a large number of data points spaced close together, the graph appears cleaner with only the Lines setting on.

- 35 Click on the OK command button to apply these changes and close the dialog box.

The graph should appear as pictured below.



Data Monitor - Smith Chart

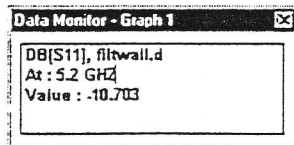
Emgraph has a data monitor that provides a readout on any given data point on your plot. The information given in the readout is dependent upon the type of data point selected. In the next section, you use the data monitor on the Smith chart.

- 36 Click on the title bar of the Smith chart window to make it active.

The title bar is highlighted to indicate that this is the active window.

- 37 Double-click on a data point in the Smith chart to open the Data Monitor.

The Data Monitor appears on the display with information about the data point you selected. To observe data on other points in the chart, use the ← and → keys. The Data Monitor readout is shown below.



Data Monitor - Cartesian Graph

- 38 Double-click on a data point in the Cartesian graph to open the Data Monitor.

The Data Monitor appears on the display. Be aware that a separate data monitor needs to be opened for each plot window in *emgraph*. To move between curves on the display use the ↑ and ↓ keys.

Zooming

At times, it may be desirable to observe a section of your chart in close detail. You can zoom in on any area of a chart you desire.

- 39 Click and drag your mouse in the Smith chart until the rubber band surrounds the area you wish to magnify. Then release the mouse button.

The display is updated with a magnified picture of the selected area. To return to the full graph, simply double-click anywhere in the plot window away from the graph line. Zooming operates in the same manner on a Cartesian graph.

Closing *emgraph*

To close the program, perform the following:

- 40 Select *File* ⇒ *Exit* from the *emgraph* main menu.

The *emgraph* display is closed.

Emvu

Emvu is a visualization tool used to view the results of circuits previously analyzed with *em*. *Em* saves the resulting current density information in a file format ready for input to *emvu*. The file has the extension of ".jxy", where "j" stands for current density. The "xy" refers to the directions of the current (z-directed current is not shown).

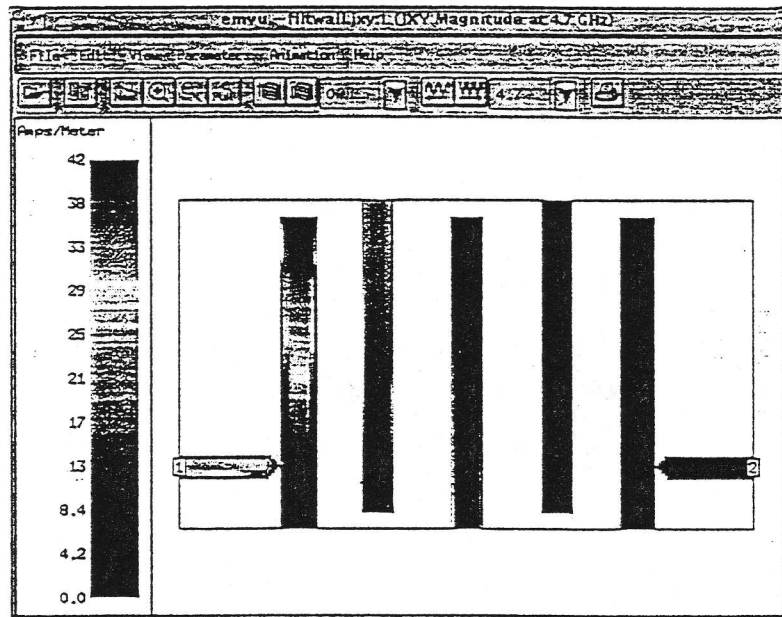
NOTE:

To produce the current density file, you must select the Make *emvu* file option in the *em Control* job window. This was not done in the example above to save on processing time. Instead, the "filtwall.jxy" file was included with the other example files you copied.

Invoking *emvu*

- 41 Click on the Open *Emvu* button at the bottom of the *em* output window to invoke *emvu*.

The *emvu* window appears on your display with the first frequency in "filtwall.jxy" displayed. The color plot should appear similar to the one shown below. You are viewing the magnitude of the current density with a 1 V, 50 ohm source and Port 2 with a 50 ohm load.



Red areas represent high current density, and blue areas represent low current density. A scale is shown on the left of the window which defines the values of each color.

Current Density Values

- 42 Click on any point on your circuit to see a current density value.

The current density value (in amps/meter) at the point that you clicked is shown in the status bar at the bottom of your window along with the coordinates of the location.

Frequency Controls



- A different frame is displayed for each analysis frequency.
- 43 Click on the Next Frequency button on the *emvu* tool bar.

The lowest frequency appears on your display as the default, which in this case is at 4.95 GHz. When you click on the Next Frequency button, the display is updated with the current density at 4.975 GHz.

NOTE:

The scale to the left of the display is also updated.

5.2



- 44 Click on the Frequency Drop list, and select 5.2.

The drop list allows you to go directly to any of the analysis frequencies.

Animation

Emvu provides two types of animation: frequency and time. To do an animation in the frequency domain, *emvu* takes a picture of your circuit at each frequency point and links the pictures together to form a "movie". In the time domain, *emvu* takes a picture of your circuit at instantaneous points in time at a given frequency by changing the excitation phase of the input port(s). Animation allows you to see how your response changes with frequency or time, providing insight into the properties of your circuit.

The animation menu and controls are the same whether animating as a function of frequency or time. *Emvu* accomplishes this by translating the data into frames. The animation menu then allows you to step one frame at a time, or "play" the frames by displaying them consecutively. How the data relates to a frame in either animation mode is discussed below.

Time Animation

For Time Animation, each frame corresponds to an input phase. The type of data plotted is determined by the *Parameters* \Rightarrow *Response* menu. In this case, the data is the JXY Magnitude, the total current density. This is the default setting upon opening an *emvu* window. For more details about response types, see "Parameters - Response," page 20 in the *Emvu User's Manual*.

- 45 Select *Parameters* \Rightarrow *Scale* from the *emvu* main menu.

The Set Scale dialog box appears on your display.

- 46 Click on the User Scale checkbox and enter a value of 100 in the Max Value text entry box.

This allows you to set a fixed scale to avoid automatic updates of the scale during the animation.

- 47 Click on the OK command button to close the dialog box and apply the changes.

The dialog box disappears and the scale to the left of the *emvu* window is now fixed at a maximum value of 100.

Emvu

Emvu is a visualization tool used to view the results of circuits previously analyzed with *em*. *Em* saves the resulting current density information in a file format ready for input to *emvu*. The file has the extension of ".jxy", where "j" stands for current density. The "xy" refers to the directions of the current (z-directed current is not shown).

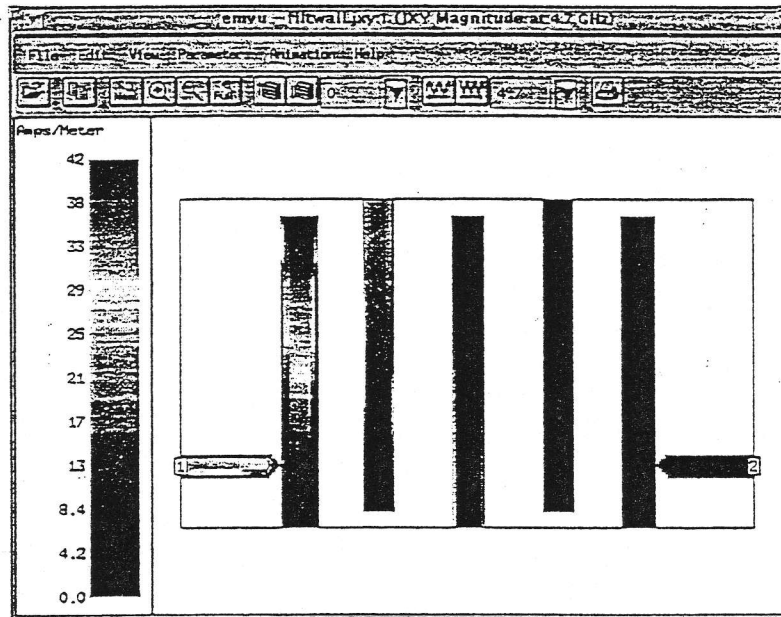
NOTE:

To produce the current density file, you must select the Make *emvu* file option in the *em Control* job window. This was not done in the example above to save on processing time. Instead, the "filtwall.jxy" file was included with the other example files you copied.

Invoking *emvu*

- 41 Click on the Open *Emvu* button at the bottom of the *em* output window to invoke *emvu*.

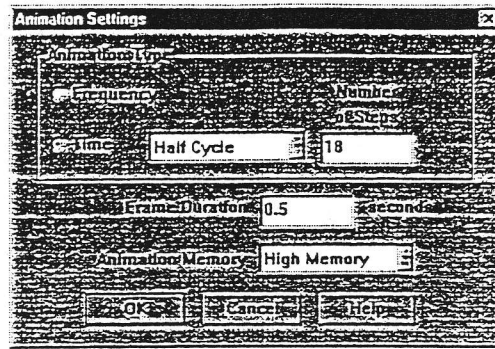
The *emvu* window appears on your display with the first frequency in "filtwall.jxy" displayed. The color plot should appear similar to the one shown below. You are viewing the magnitude of the current density with a 1 V, 50 ohm source and Port 2 with a 50 ohm load.



Red areas represent high current density, and blue areas represent low current density. A scale is shown on the left of the window which defines the values of each color.

- 48 Select *Animation* ⇒ *Settings* from the *emvu* main menu.

The Animation Settings dialog box appears on your display as shown below.



- 49 Click on the **Time** Radio button to select **Time Animation**.

This enables the cycle type drop list and the Number of Steps text entry box.

- 50 Select **Half Cycle** from the cycle type drop list.

Our example uses half cycle, the default, in which the phase ranges from 0° to 180° .

For Full Cycle the phase is ranges from 0° to 360° . For Quarter Cycle, the range is 0° to 90° .

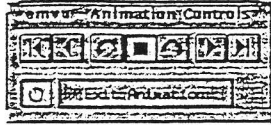
- 51 Enter a value of 36 for the **Number of Steps**.

The number of steps is used to determine how many frames the animation includes. There are 180° in a half cycle; therefore, 36 steps yields a value of 5° for the phase interval.

The lowest phase, 0° , corresponds to the first frame and the highest phase, 175° , corresponds to the last frame. This phase value represents an offset from the Source Phase for any given port set in the Port Parameters dialog box. For details on how to set the Source Phase, see "Parameters - Ports," page 20 in the *Emvu User's Manual*.

For this example, the Source Phase for port 1 is set to the default of 0° . Port 1 is terminated with a 50 ohm load.

The Animation Controls, shown below, also appear on your display.



- 55 Click on the Continuous Play button in the Animation Controls.

This command toggles the *Play* commands in and out of continuous mode. In continuous mode, when the last frame is reached, *emvu* starts over at the first frame and continues to cycle through the frames until a stop command is received. If the play direction is reverse, when the first frame is reached, *emvu* starts over at the last frame and continues to display frames, in reverse order, until you press stop.

Continuous mode is indicated by a change of color of the Continuous Play mode icon.



- 56 Click on the Play button in the Animation Controls.

Selecting *Play* starts the animation at the present frame, 5.2 GHz and 0 degrees, which corresponds to $t = 0$. Subsequent frames are displayed consecutively in 5° intervals. Each frame is displayed for 0.1 seconds, the delay set earlier. When the last frame is reached, the display starts over with the first frame and repeats until you press the Stop button in the Animation Controls.

Be aware that the first time through, *emvu* is calculating the displays for each frame. Subsequently, the displays update at a slower rate. After the initial loading cycle of all the frames is complete, the display updates at the input rate of 0.1 seconds.



- 57 Click on the Stop button in the Animation Controls.

This stops the animation, although the window remains in animation mode.

- 58 Click on the Exit Animation button in the Animation Controls.

This exits the animation mode. The Animation Controls disappear from the display.

Frequency Animation

For Frequency Animation, each frequency will have its own frame. The lowest frequency, 4.95 GHz, corresponds to the first animation frame and the highest frequency, 5.325 GHz, corresponds to the last frame.

- 59 Select *Animation* ⇒ *Settings* from the *emvu* main menu.

The Animation Settings dialog box appears on your display.

- 60 Click on the **Frequency** radio button to select Frequency Animation.

These radio buttons, Time and Frequency, are disabled, when running an animation. Only the Frame Duration may be changed while running an animation.

- 61 Click on the **OK** button to close the Animation Settings dialog box and apply the changes.

- 62 Select *Animation* ⇒ *Animate View* from the *emvu* main menu.

Emvu enters animation mode. The first frame, which displays the JXY Magnitude response for 4.95 GHz, appears in the window. The response type and frequency are determined by the settings when animation mode was launched.

The Animation Controls also appear on your display. Note that Continuous Play mode is still "on" from the previous example.

- 63 Click on the **Play** button in the Animation Controls.

Selecting *Play* starts the animation at the present frame, 4.95 GHz. Subsequent frames, consecutively by frequency, are displayed, each for 0.1 seconds, the delay set previously. When the last frame is reached, the display starts over with the first frame and repeats until you press the Stop button in the Animation Controls.

Note that a description of each frame's contents appears in the title bar of the *emvu* window.

- 64 Select *File* ⇒ *Exit* from the *emvu* main menu.

This command exits the *emvu* program.

- 65 Select *File* ⇒ *Exit* from the *em Control* main menu.

This command exits the *em Control* program.