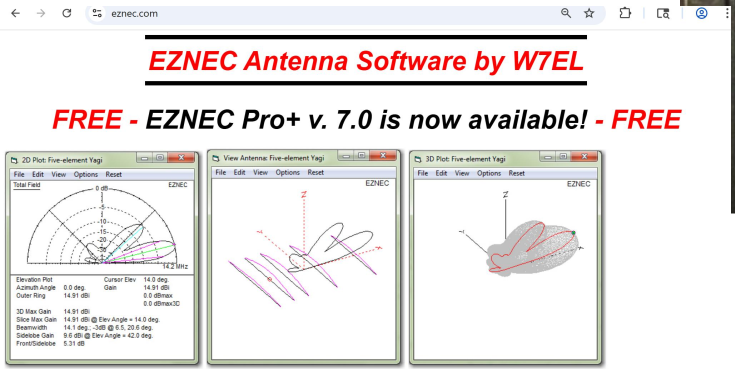
Radio Design 201 Microwave Circuits and Antennas
Episode 4 - Outfitting Microwave Labs on a Budget

Slides downloaded from: https://ecefiles.org/

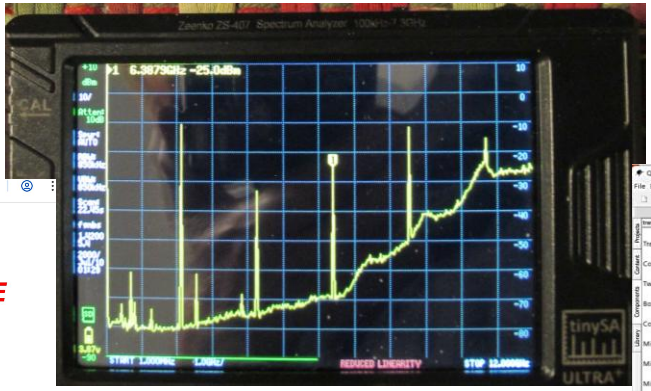
Companion videos at: https://www.youtube.com/playlist?list=PL9Ox3wpnB0kpOb7NdpHAK-jDpsZm3Xedx

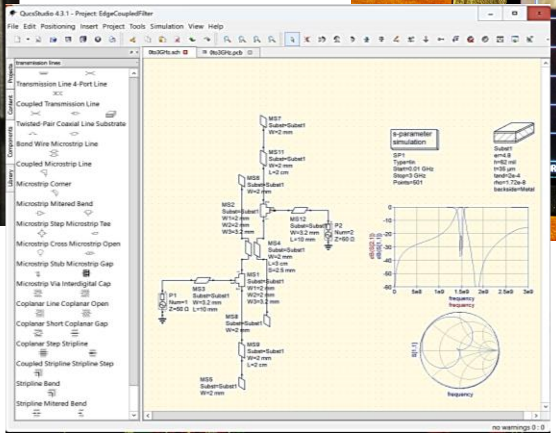
This material is **provided by ecefiles.org for educational use** .

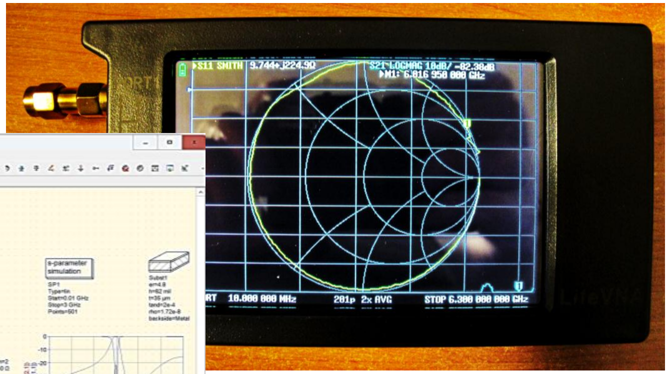
In this episode we show software and hardware options for building a home or university lab on a budget. In particular, we discuss antenna simulation and EM software that can be used for free, and recently introduced TinySA Ultra, NanoVNA, and LiteVNA spectrum and vector network analyzers that can be bought for relatively low prices. For those interested in course/class design, we also discuss lab assignments and projects, including ones we may implement during future episodes.



Above: Screen shots from several ***EZNEC*** displays. **Right:** 3D far field pattern, with 2D elevation "slice" highlighted. Any azimuth or elevation slice can be highlighted. **Center:** View Antenna display, showing the "wires" making up the model of the five-element beam, with currents and 2D slice superimposed to show orientation. Several other items, such as currents and wire numbers, can be added to this display. **Left:** 2D display showing detailed information about the selected slice.

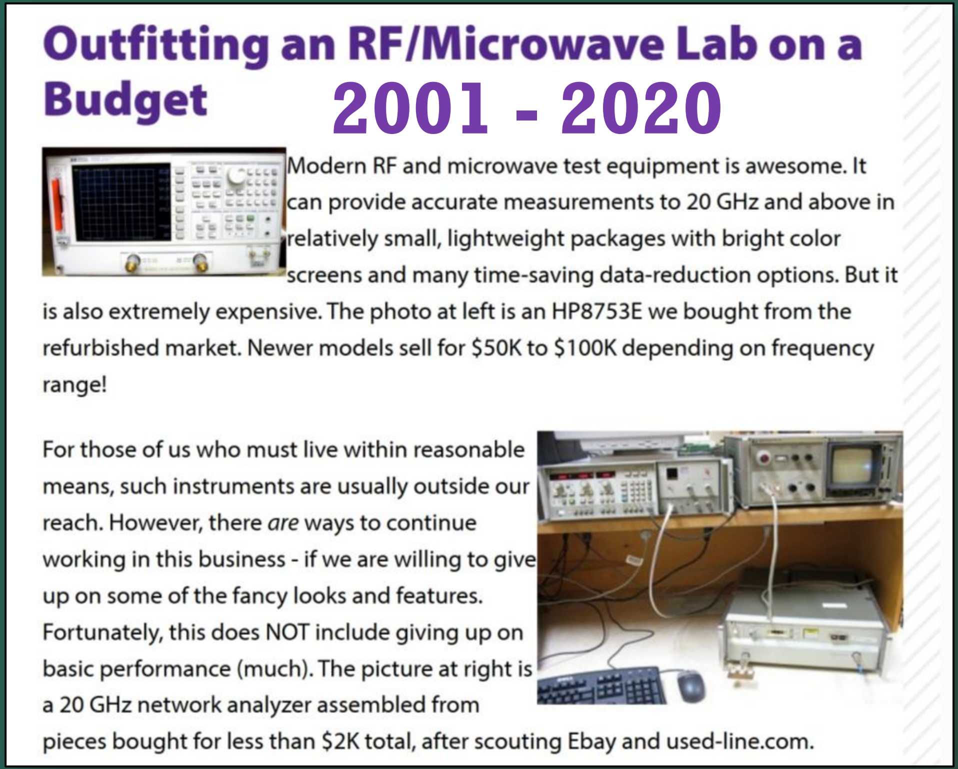






Radio Design 201 #4

Outfitting Microwave Labs
On a Budget

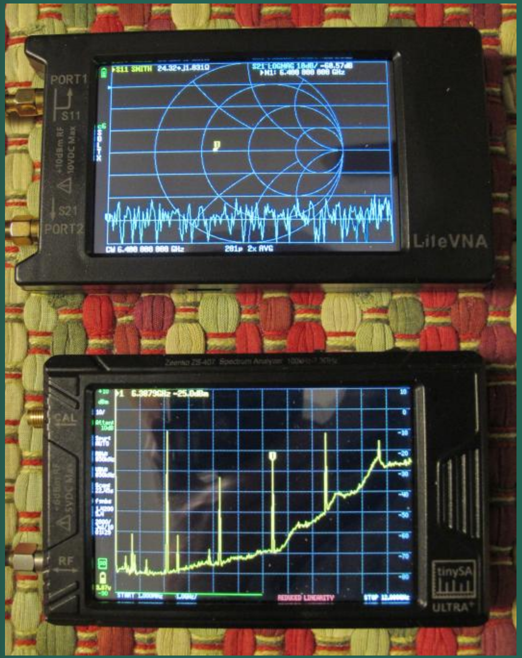


Outfitting an RF/Microwave Lab on a Budget **2001 - 2020**

Modern RF and microwave test equipment is awesome. It can provide accurate measurements to 20 GHz and above in relatively small, lightweight packages with bright color screens and many time-saving data-reduction options. But it is also extremely expensive. The photo at left is an HP8753E we bought from the refurbished market. Newer models sell for $50K to $100K depending on frequency range!

For those of us who must live within reasonable means, such instruments are usually outside our reach. However, there *are* ways to continue working in this business - if we are willing to give up on some of the fancy looks and features. Fortunately, this does NOT include giving up on basic performance (much). The picture at right is a 20 GHz network analyzer assembled from pieces bought for less than $2K total, after scouting Ebay and used-line.com.

2026



Today's "Lecture"

Announcements

Welcome to Episode 4
(Final episode in 4-part introduction)

See Episodes 1 through 3 for

- *Definitions of "microwave",*
- *Example semester projects, and*
- *Day 1 lecture material*

Today's Topics

➡ *Example Project and Lab Assignments*

Radio Design 201 Lab and Project Ideas

Free *Software Examples*

"Low Cost" Test Equipment Examples

Project/Lab-Oriented Learning in Open Laboratories (See Episode 3)

Homework: Work in this course centers around the design, construction, testing, and documention of real-world antennas, circuits, and systems. Therefore, there are no classic homework problems. Rather, "homeworks" consist of lab and project assignments and associated documentation.

Labs/Projects: During the first half of the semester, we will be covering antennas, propagation, and transmission line fundamentals. We will be learning through lectures, lab exercises, and associated writeups. ***The lab exercises will be done in teams of 2 or 3*** and will include:

- Lab 1: Measurement of radio propagation in the field
- Lab 1: Basic antennas and performance measurements.
- Lab 3: Circuit component parasitics, transmission lines, matching networks, and other "microstrip" component design.

During the second half of the semester, we will be designing, building, and testing a microwave system - a Doppler/FMCW radar operating in the 5.8 GHz Industrial, Scientific, and Medical (ISM) frequency band. This will be accomplished in two stages:

- Prototyping of main radar functions (e.g. amplifiers, filters, etc.)
- Full radar product design, construction, and testing.

As with the labs, the radar designs will be done in teams (in this case, of 3 or 4 person "companies"). Each person within your company will be responsible for a separate part of the design, and the whole company must work out interfaces between sections and get the product functional by the end of the semester.

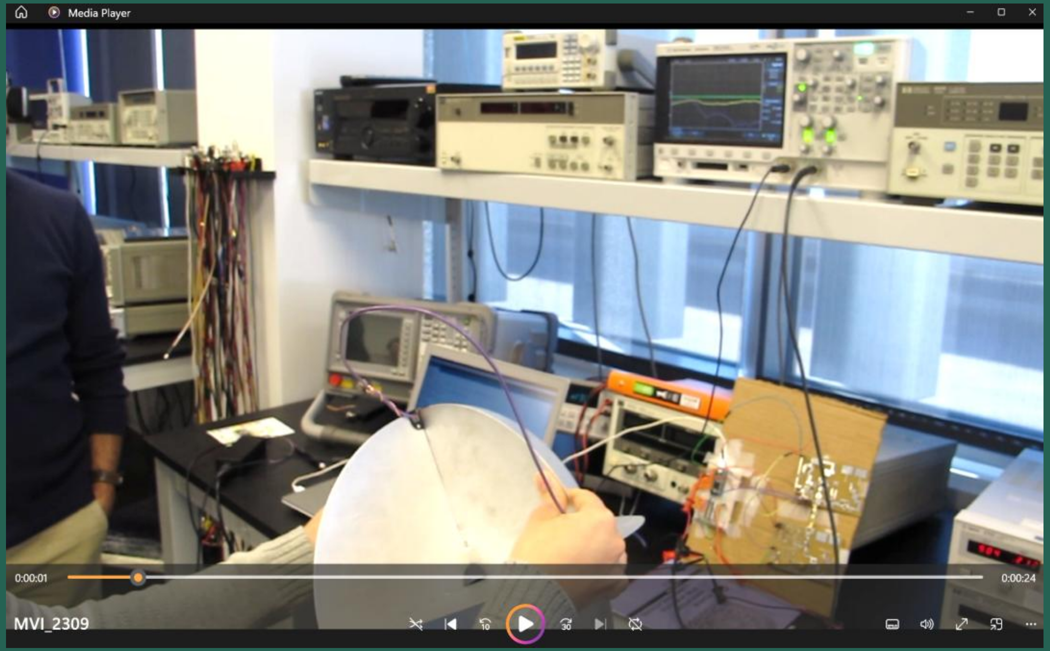
For the final project demos, we will run tests in the lab to prove that you can measure speed and distance. We may also go out to the street and check the speed of some cars driving by if it is not too cold at the end of the semester :-)

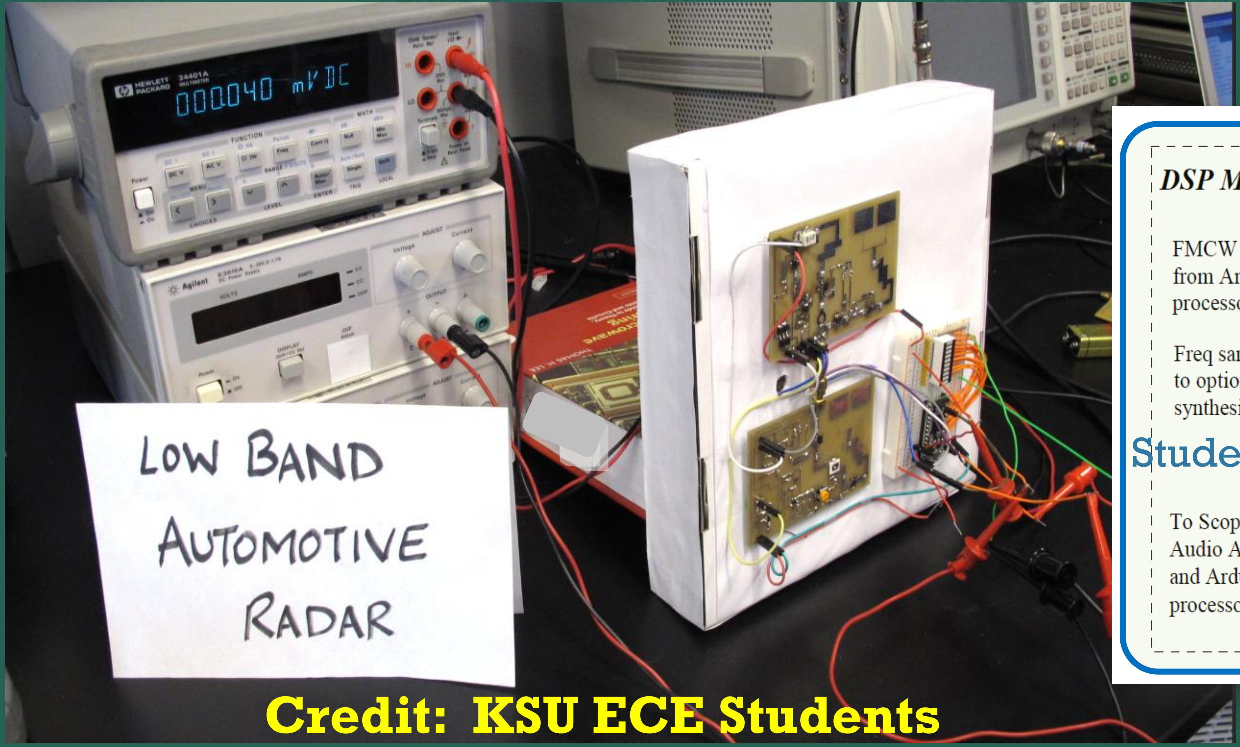
Exams: The final-exam will be this demonstration of your project plus a company report with each student writing his/her section.

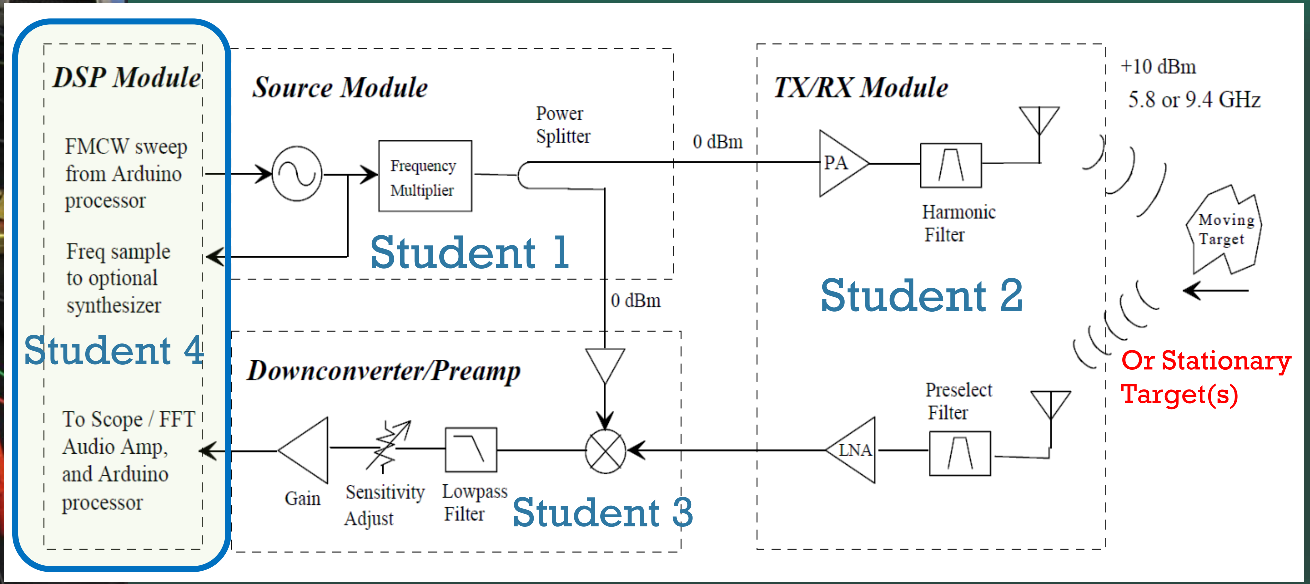
Project Kit: You will need to purchase a project kit that contains the core components required to complete the labs/project. This kit may be obtained from the ECE shop and is in two parts. ***Each student** needs to purchase **one individual kit** and **one group kit***. Some of the larger parts as well as fabrication of PCBs will be available with proof-of-purchase.

Example Class Project at 5.8 GHz:

FMCW Radar
(See Episode 2)







Credit: KSU ECE Students

Example Lab3 Assignment (Page 1)
(Spring 2020)

ECE 764 Lab 3 - Introduction to Microstrip Circuits
Due Friday, March 20, 2020

Purpose

This lab will familiarize you with designing, building, and testing microwave circuits on PC boards, in preparation for the upcoming radar project. You will be introduced to Keysight's ADS software that will be used extensively in the project. You will also learn several important issues with board design including parasitic inductances of vias and capacitances of PC board traces. While often a problem, we will also see how these parasitics can be embraced in microwave design to form bandpass filters. Finally, you will build and test microwave amplifiers and filters to learn about circuit layout, surface-mount construction techniques, S-parameter measurement using vector network analyzers (VNAs), and some important system design issues such as amplifier compression points and stability.

Teams

As before, the assignment is constructed for ***teams of two persons***. You should help each other out. However, in this project **each person is responsible for their own simulations, circuit measurements, and associated writeup sections**. By having each person focus on a different amplifier, frequency of operation, etc., the lab will allow teams to learn about the range of options you may encounter in the upcoming project.

Lab Overview

Circuits to be simulated, designed, constructed, measured, and documented include:

- Simple simulations of PC board vias and traces to learn about inductive and capacitive parasitics.
- Bandpass filters.
- Lambda/4 bias decoupling lines used in amplifier biasing.
- ~~Single-stub and lambda/4 impedance-transformer based matching networks (simulation only).~~
- Building and measuring amplifier boards to learn about their performance.
- Using "cal kits" and identifying associated "reference planes" when measuring S-parameters.
- Measuring the cascaded response of amplifiers.
- Adding bandpass filters and attenuation "pads" to address excess low-frequency gain and potential stability problems in amplifier cascades.

Operating Frequencies

In the project we may have circuits operating at ***2.45 or 5.2 GHz***. ***Each person on a team must select a different frequency to use in their investigations.***

2/24/2020 WBK

https://ecefiles.org

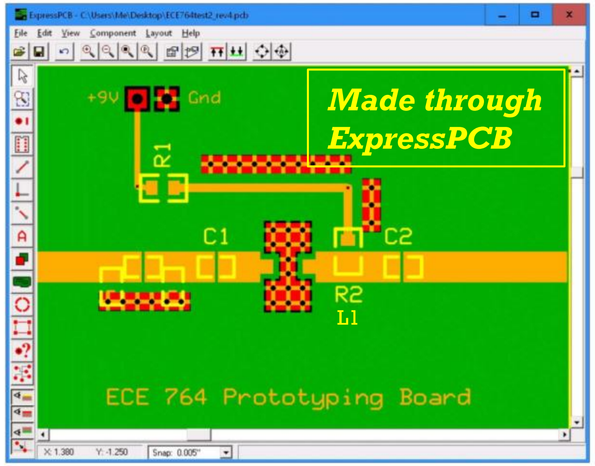
Lab Assignment (Pgs 2 – 4)

(Spring 2020)

PC Boards and Amplifiers

For this lab we will use 0.062” (62mil or 1.6mm) thickness, double-sided “FR4” (fire-retardant fiberglass-epoxy) printed circuit boards. Connections to test equipment will be through female SMA end-launch connectors. You will work with pre-designed amplifier boards and populate them with amplifier ICs and surface-mount biasing components. In addition, you will prototype simple bandpass filters, build them, and combine them with your amplifier(s).

Each person on a team must chose a different amplifier (LEE-39 or Gali-2). Be sure to read the datasheets for the amplifiers before making final selections to be sure you understand their biasing and performance, and can get reasonable gain at the frequency you selected above. A picture of the PC boards on which you will build your amplifier circuit is shown below.



Lab Instructions

The sections below give an outline of what you should do and document in this lab. Additional details for some parts (e.g. using ADS, building circuits, and operating the VNA equipment) will be presented in class. As always, if you are not sure what to do or how to do something, ask someone.

Part I Interconnect, Via, and Component Parasitics at Microwave Frequencies

At high frequencies, wires or PC board traces can no longer be treated as simple nodes. When their length exceeds more than about 1/20th of a wavelength (e.g. 1.5 mm at 5 GHz for vp = 0.5c), the phase shifts from one end to the other are significant enough that they cannot be ignored. In general, the line has distributed inductance and capacitance that must be modeled using the transmission-line theory as we have elaborated in class.

Engineers use transmission-line theory extensively to model PC board interconnect behavior in the GHz frequency regimes. However, if the line is shorter than 1/8th of a wavelength, one can sometimes simplify and talk about the “parasitic” inductance of a grounded trace or via or the parasitic capacitance of an open trace. For longer lines (in particular, ¼ lambda), much more interesting things occur as discussed in class. This part of the lab investigates some of these issues using the ADS simulator.

1. Behavior of ground connections at microwave frequencies

i. Read the handout on operation of ADS and then draw a schematic consisting of a short (400 mil = 0.4 inch) length of trace on a printed circuit board, with one end connected to ground. Use the MLIN element from the ADS TLines-Microstrip components menu, and the ground symbol from the top menu bar.
ii. Enter a width value of 10 mil and a length of 400 mil for the PC board trace modeled by the MLIN element. Then place an “MSUB” item from the TLines-Microstrip menu and configure it with the parameters of our FR4 PC board material: (H=62 mil, Er=4.6, TanD=0.01).
iii. Switch to the Simulation-S_Param menu and place a Term (with ground) on the other end of the MLIN, and an “SP” simulation setup item anywhere on the schematic. The Term item acts as a 50 Ohm source or load and will be used to emulate measurements made with a network analyzer. Edit the simulation setup item to sweep from 1 MHz to 8 GHz, with a step size of 10 MHz.
iv. Print out a readable copy of your resulting schematic to include in your writeup
v. Run the simulation using the Gear icon on the top menu bar and plot the result by placing a Smith Chart item in the resulting display window and selecting “S(1,1)” to plot.
vi. Place markers at the lowest and highest frequencies, and at your particular frequency (2.45 or 5.2 GHz). Print out the result. Notice the curve begins near Z = 0 as one would expect for a grounded trace at 1MHz, where the line is much shorter than a wavelength. At higher frequencies, the impedance is of the form Z = 0 + j X, which is inductive, and it eventually wraps around to an open-circuit on the right and then into the capacitive region of the chart past 5 GHz.
vii. Below the frequency where Z = 0 + j50, the impedance is approximately proportional to frequency, so that this shorted line can be reasonably well modeled as an inductance to ground at this lower frequency range. Place additinoal markers to find the frequencies where the Z value is +j 50, where it is + j 25, and where it is + j 12.5 to verify this. Is the reactance reasonably proportional to frequency?
viii. Use these values to estimate the inductance of the wire. What is this approximate inductance per unit length implied ? Does it agree roughly with the often used estimates of 1 nH/mm or 20 nH / inch ?
ix. Modify the trace to be 100 mils wide and repeat to determine the inductance per unit length. How does width of the trace affect the inductance?
x. At a sufficiently high frequency, the impedance of a grounded trace actually becomes an *open circuit*. Find this frequency by placing a marker at the Z=infinity point on the chart (right side) and use it to estimate the velocity of propagation on the 100 mil x 400 mil line. (Recall that it should become an open circuit when the length of the line is lambda/4). Find the velocity of propagation expressed as a percent of c.

2. Behavior of vias to ground on PC boards

One difficulty we will need to deal with when building our radars if we use 1/16” thick FR4 board is the fact that even if there is no trace length leading to the ground connection, the vias needed to make connection to the back-side ground plane will show inductive behavior themselves. For example, the ground pin on our amplifiers will not really be truely at ground. It will be separated from ground by an inductive connection through the via (as well as its physical pin). In this part, you will estimate how large this inductive impedance may be at your chosen frequency and look at some solutions to the problem.

i. Replace the ground symbol on the right side of your schematic with a VIAGND item from the TLines-Microstrip menu.
ii. Edit the MLIN to make it only 5 mils long (effectively 0), so that we will be simulating only the via’s inductance. Print the resulting schematic.
iii. Simulate and use markers to find the complex impedance and then the inductance at your frequency. Print the resulting Smith Chart and write the inductive reactance in Ohms and the implied inductance on it.
iv. Since this value is pretty high relative to the signal impedances (50 Ohm source and load values) elsewhere in the circuits we will build, microwave engineers often use multiple vias (as well as multiple ground pins on the ICs). Add a second VIAGND in parallel, but connect it to the first one using a short MLIN to model that it is not physically in the same location. Use a 100 mil wide line with a 25 mil length to represent a realistic distance given the hole sizes in the via.
v. Simulate and comment on how much it lowered the inductive parasitics in the ground-via connection. Why is it not half of the previous value?
vi. Examine the amplifier board on page 2 and note that 7 vias are used on each of the two amplifier ground pins. Estimate what the net inductive parasitic impedance would be in this case. (Don’t sim. Just reason through it to arrive at an educated guess. Sims are great - but they don’t provide as much insight as actual thinking !)

3. Behavior of open-circuit “stubs” on PC boards

Just as a shorted line acts as an inductance and can become an open, an open line can act as a capacitance and become a short at sufficiently high frequency. Later, we will use this to make simple LC bandpass filters in our experiments and possibly in our radars. In this part, we will just look at the effects with simulation.

i. Remove the vias from the previous schematic and restore the original trace to its 400 mil length (with 100 mil width).
ii. Simulate the resulting open-circuit line and find the capacitance per unit length and propagation velocity. Explain how you did this and print your schematic and Smith chart results to back-up your estimates.

Part II Designing and simulating microstrip circuits

Most microwave amplifiers are designed to have close to 50 Ohm input and output impedances so that they can use 50 Ohm transmission lines to connect to other circuits without the need to deal with impedance transformations. While such amplifiers purchased commercially are typically “well-matched”, user-designed matching networks can sometimes be useful to achieve somewhat higher gains. To save time, we will go over this topic in class, but not do it here. Instead, we will concentrate on the basics of biasing, decoupling, and DC-blocking in amplifiers, and on their performance in terms of the maximum power levels they can be used at. This will

Lab Assignment (Pgs 5 – 7) (Spring 2020)

be needed when you do your radar designs. We will also look at some simple bandpass filter design techniques.

1. Simulating amplifiers and effects of bias lines and DC blocking capacitors

i. Create a new ADS schematic using an S2P item from the Data Items menu with a 50 Ohm Term on both the input and the output. Download the S-parameter model file from Minicircuits or our class webpage and set the File parameter to point to the S2P model file.
ii. Simulate from 100 MHz to 8 GHz (without any MLIN traces at input or output) to verify the S11 and S21 (input impedance and gain behaviour) in the datasheet. Just plot dB graphs, not Smith charts. Compare with the datasheet.
iii. Modify your schematic by adding a lambda/4 bias decoupling line at the amplifier output pin. Such lines are often used to supply Vdd power to the device as discussed in class. The lambda/4 decoupling line should be a simple 10 mil wide MLIN trace with length of lambda/4 at your operating frequency. Place a 10nF capacitor to ground at the end opposite the amplifier. Do not include any resistors or inductors here, since we want to investigate the frequency selectivity caused by the lambda/4 decoupling line first. Adjust the line's length as needed to account for the via inductance and the length of the 0603 capacitor and pads we will use in the actual construction.
iv. Next, add DC blocks (AC coupling caps) at the input and output. For these DC blocks (AC coupling caps), use the C component from the Lumped Components menu as for the bypass cap, but also add a 40 mil long by 20 mil wide MLIN trace to model the impedance-bump the capacitor creates (since it is not 105 mils wide, and the transmission line in this part will therefore have a higher Zo value than 50 Ohms). Use reasonable standard-value capacitances that will have an impedance of about 0.1 Ohm at your targeted operating frequency.
v. Simulate S21 from 100 MHz to 8 GHz and confirm the gain at your targeted frequency is not significantly changed from the datasheet and simulation values before the bias lines were added. Why is the gain now less than before at low frequencies? (There may be multiple reasons).
vi. Simulate S22 and see if the output return loss is OK at your frequency. What is it at low frequencies and why?
vii. Print your schemaitc and your S21 and S22 plots and include in your writeup to support your answers.

3. Filter Design

i. Using the techniques discussed in class, design a one or two pole bandpass filter ***at your personal selected frequency***, with a fractional bandwidth of 10% to 30% and add it on the input side of your amplifier. *NOTE: Each person on your team should build a different type of filter (in addition to being at a different frequency than that of your teammate)*. For example, one person may make the inductor needed out of an open-ciruit line which is slightly longer than lambda/4, while the other may make the inductor out of an electrically short grounded line. Or one person can do a single-pole while the other does a two pole design.
ii. Create a schematic of the filter in Agilent ADS. Simulate it alone (with Terms on both the input and the output) and refine it until the center frequency and bandwidth are correct to within about 5 percent, the shape is good, the insertion loss is reasonable, and the in-band input and output return loss values are at least 10 dB. Be sure to use appropriate "microstrip T's" to account for the stubs merging into the main line.
iii. Print/plot your final schematic and response plot(s).
iv. Add the filter to the ***input*** side of your amplifier and simulate the composite design. Plot and comment on S21. Does the filter help reduce gain at the frequencies outside of the designed frequency band? In particular, does it help to reduce low-frequency gain and make the gain concentrated reasonably well around f_o ?

Part III Measurement of Amplifier Parameters

In this last part, you will gain experience in biasing and soldering an amplifier and characterizing its performance. The construction of your amp will help you with design/layout decisions when you are called to make your own circuit boards in the project, and the data you collect will help you in making choices on which amplifiers to use in which situations and whether or not to add filters and/or attenuation "pads" to improve stability. More details will be provided in the coming week while the circuit boards are being sent out for fab, but here is a brief outline.

1. Biasing and Construction

i. Decide how you want to configure the bias circuits per discussions in class. Assume a 9V DC supply (since that is what we will design for in the project) and see the device datasheet for your amp for more information on the current and voltage needed at the output pin.
ii. Solder the main components on your board, including the bias resistor and decoupling line capacitor, the DC blocking capacitors, the amp itself, and the SMA connectors.

2. Smoke-(and oscillation)-Test and Initial Scalar Gain Measurements

i. Connect it to a power supply with the supply off and the voltage knob turned all the way down (CCW). Using the signal generator supply an RF input signal and monitor the output power. IMPORTANT: You should input a reasonably low level RF signal (e.g. -20 dBm) since the amplifier will not show the proper gain (and could even be damaged) if the input signal is too high and the output saturates.
ii. Connect the output of the amplifier to a spectrum analyzer.
iii. Slowly increase the DC supply voltage toward 12V while monitoring the current. Verify the amplifier draws a reasonable amount of current. Stop if the current exceeds 50 mA, which could be enough to damage it.
iv. Check to see if the circuit is stable. This is very important at microwave frequencies, as we will come to learn. Verify that the spectrum analyzer shows only the expected output (as well as possible low-level harmonics of course). This should be done on a spectrum analyer that goes to at least 8 GHz or more, since the amplifier still has some gain at such frequencies. If there are other substantial magnitude spectral lines (other than harmonics) - you have stability problems that need to be fixed. If you do not see other spectral lines, try disconnecting the cable on the input and see if you see any outputs. Sometimes amplifiers will oscillate if their ports are not well terminated (in 50 Ohms).
v. Verify that the amplifier has reasonable gain, recording the value and comparing it with the expected gain from the datasheet and your simulations. NOTE: The signal generator and spectrum analyzer may each be off by as much as a dB, and the interconnect cables could have a dB or more of loss, so don't expect exact gain values. Try to setup a good method to measure "insertion gain" to address these error sources. Record/document your procedure and measurement results.

3. Compression Point Measurements

1) The amplifier input was specified as -20 dBm above since the amplifier gain will not be correct if it is "overdriven". In this part, we will find the critical input signal level where the gain begins to fall due to excess input signal power.

i. Slowly increase the signal input power in 1 or 2 dB steps. Record data of output power versus input power in dBm until the gain falls by at least 1 dB from the value at low input levels. Record the input power at which this happens as the *1dB input-referred compression point*. You should ideally adjust for cable loss here. How does the compression point compare to the published value? Which is the data sheet using, input referred or output

referred? (If output referred, divide by the gain to see what the data sheet thinks the input referred compression point is...
ii. Take additional data beyond the compression point until the output power does not increase any further . Typically you should go about 5 or 10 dB higher to see what the maximum output power is, assuming that is in the safe range. (***BUT - don't exceed the max allowed value at the input for the specific amplifier device or the maximum allowed output the test equipment can accept without damage !)***
iii. Plot your results (Pout vs Pin) in dBm and label the 1dB compression point - both input-referred on the horizontal axis and output-referred on the vertical axis. Also label the maximum output power.

4. Vector Network Analyzer (VNA) Measurement of S-Parameters

i. Using the instruction sheet near the VNA, set its output power to -20 dBm to prevent overdriving the amp. This typically involves setting the Port-1 attenuator in the unit to 20 dB.
ii. Set the frequency range to cover 40 MHz to 8 GHz (for the Advantest VNA), or 2 to 12 GHz (for the HP8510 VNA) .
iii. Calibrate the analyzer for measuring S21 using the cal-board's through path. Use the "response-cal" method. Then remove the cal board and put the amp in its place.
iv. Measure and plot/photograph the amplifier gain in a log-magnitude dB format. Set a marker at your frequency, record the gain value, and compare the measured value with the expected gain from the datasheet and from the previous measurement method. Is it reasonable, based on published values in the data sheet? How does it compare to the scalar measurement above? Discuss which may be more accurate and why and if you can't decide, discuss what your gain uncertainties are.
v. Next, calibrate for S11 using the 1-Port cal method and the cal board's SOL standards (as discussed in class). Then measure/plot the amplifier input impedance using Smith Chart format. *(Be sure to terminate port 2 in 50 Ohms during this measurement!)*
vi. Set the markers at various frequencies, print/plot/photograph the screen, and compare the measured values with that in the datasheet and/or S2P file.
vii. Turn your board around so that you are measuring S12 (isolation) and S22 (output reflection coefficient/impedance) and repeat.

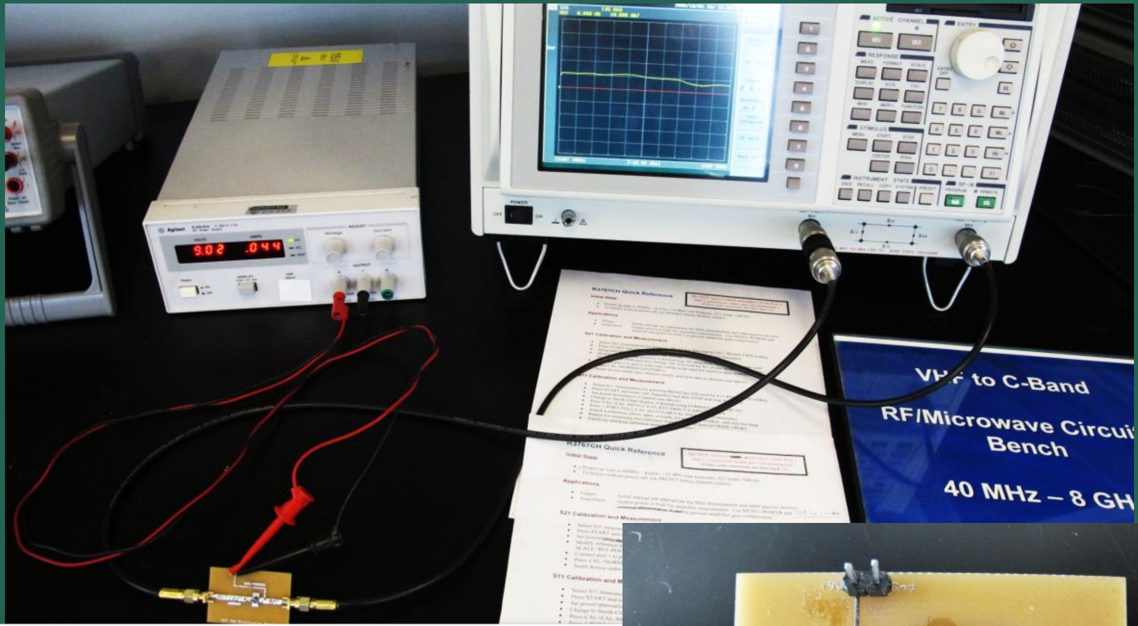
2. Filter Testing

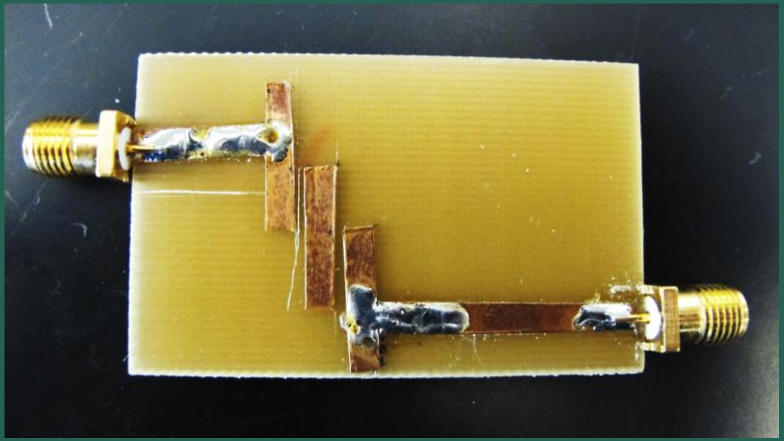
i. Build your filter designed in Part II. Use the one-sided blank board from your kit and copper tape. Solder the tape at the T or cross junction with the mainline. Measure the filter response from at least 100 MHz to 8 GHz. (Measuring to 6 Ghz is OK in this part if your filter is designed for 2.45 GHz.)

3. Cascaded Amplifiers and Filters

i. Try cascading with your partners' amp and measuring the overall gain. WARNING: You may need to use more than 20 dB attenuator setting to keep the input power low enough since the total cascaded gain may be more than 30 dB.
ii. Do the same, but with the filter added at the input of the cascade.
iii. If you experience problems with stability, try adding a 3dB "pad" at the input to the first amp (to address out-of-band reflections from the filter), or between the amps. See class discussion/notes for information on pads.

Construction/Testing (Spring 2020)





Surface Mount

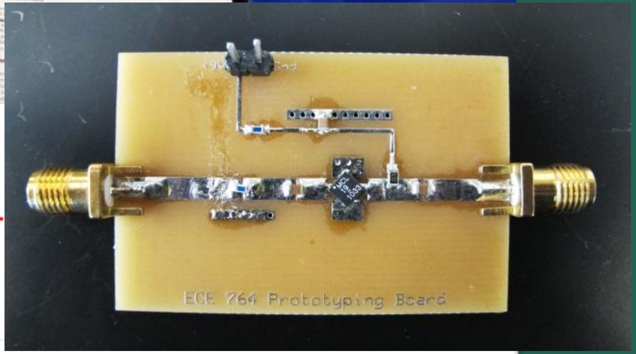
Monolithic Amplifier

DC-8 GHz

Product Features
- DC-8 GHz
- Output power, 10.2 dBm typ.
- Excellent package for heat dissipation, exposed metal bottom
- Flat output power to 10 GHz
- Aqueous washable
- Protected by US Patent 6,943,629

Typical Applications
- Cellular
- PCS
- Communication receivers & transmitters
- Satellite communication, military

LEE-19+



Function	Pin Number	Description
RF IN	1	RF input pin. This pin requires the use of an external DC blocking capacitor chosen for the frequency of operation.
RF-OUT and DC-IN	3	RF output and bias pin. DC voltage is present on this pin; therefore a DC blocking capacitor is necessary for proper operation. An RF choke is needed to feed DC bias without loss of RF signal due to the bias connection, as shown in "Recommended Application Circuit".
GND	2,4	Connections to ground. Use via holes as shown in "Suggested Layout for PCB Design" to reduce ground path inductance for best performance.

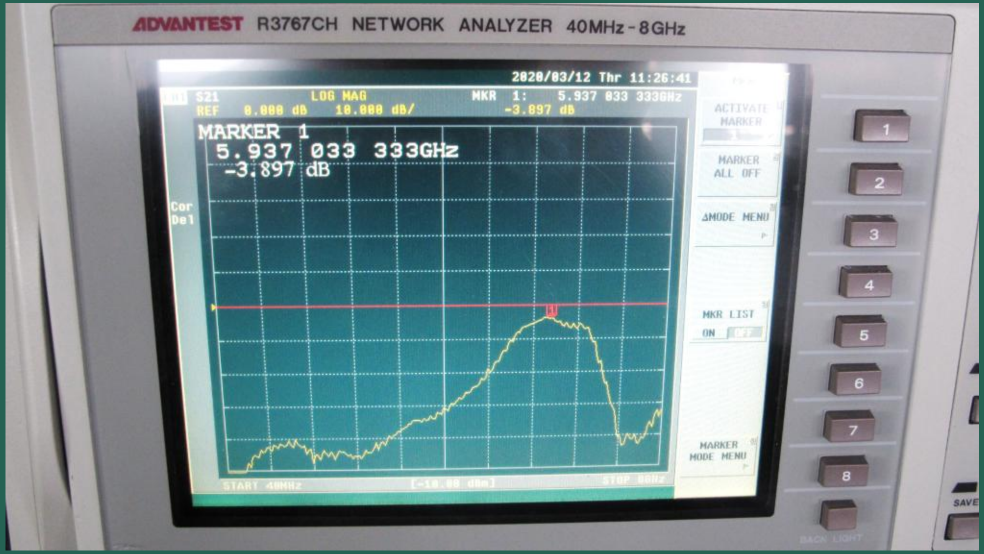
Mini-Circuits® minicircuits.com
ISO 9001 ISO 14001 AS 9100 CERTIFIED

P.O. Box 350166, Brooklyn, New York 11235-0003 (718) 934-4500 Fax (718) 332-4661 For detailed performance specs & shopping online see Mini-Circuits web site

The Design Engineers Search Engine Provides ACTUAL Data Instantly From MINI-CIRCUITS At: www.minicircuits.com

IF/RF MICROWAVE COMPONENTS

REV. H M120653 ED-10757/1 LEE-19+ RS/YB/FL 081215 Page 1 of 4



Today's "Lecture"

Announcements

Welcome to Episode 4
(Final episode in 4-part introduction)

See Episodes 1 through 3 for

- *Definitions of "microwave",*
- *Example semester projects, and*
- *Day 1 lecture material*

Today's Topics

- ✓ *Example Project and Lab Assignments*
- ⇨ *Radio Design 201 Lab and Project Ideas*
- *Free Software Examples*
- *"Low Cost" Test Equipment Examples*

Possible Radio Design 201 Labs/Demos

- Antenna simulations and experiments using EZNEC, TinySA and NanoVNA (See also Antenna Briefs series)
- Simulation, construction, and measurement of microstrip lines, filters, and amplifiers using QucsStudio, and NanoVNA

Radio Design 201
"Project" Ideas

- 2.4 GHz ISM Band Doppler Radar with Dish Antenna
- 2.4 or 5.8 GHz Doppler + FMCW Radar
- 2.4, 3.4*, or 5.8 GHz Transverter (to/from VHF)
- 5 or 10 GHz Moon-Bounce (EME*) -- as "stretch goal" ☺

(* Amateur/Ham Radio License Required for transmit …)

Today's "Lecture"

Announcements

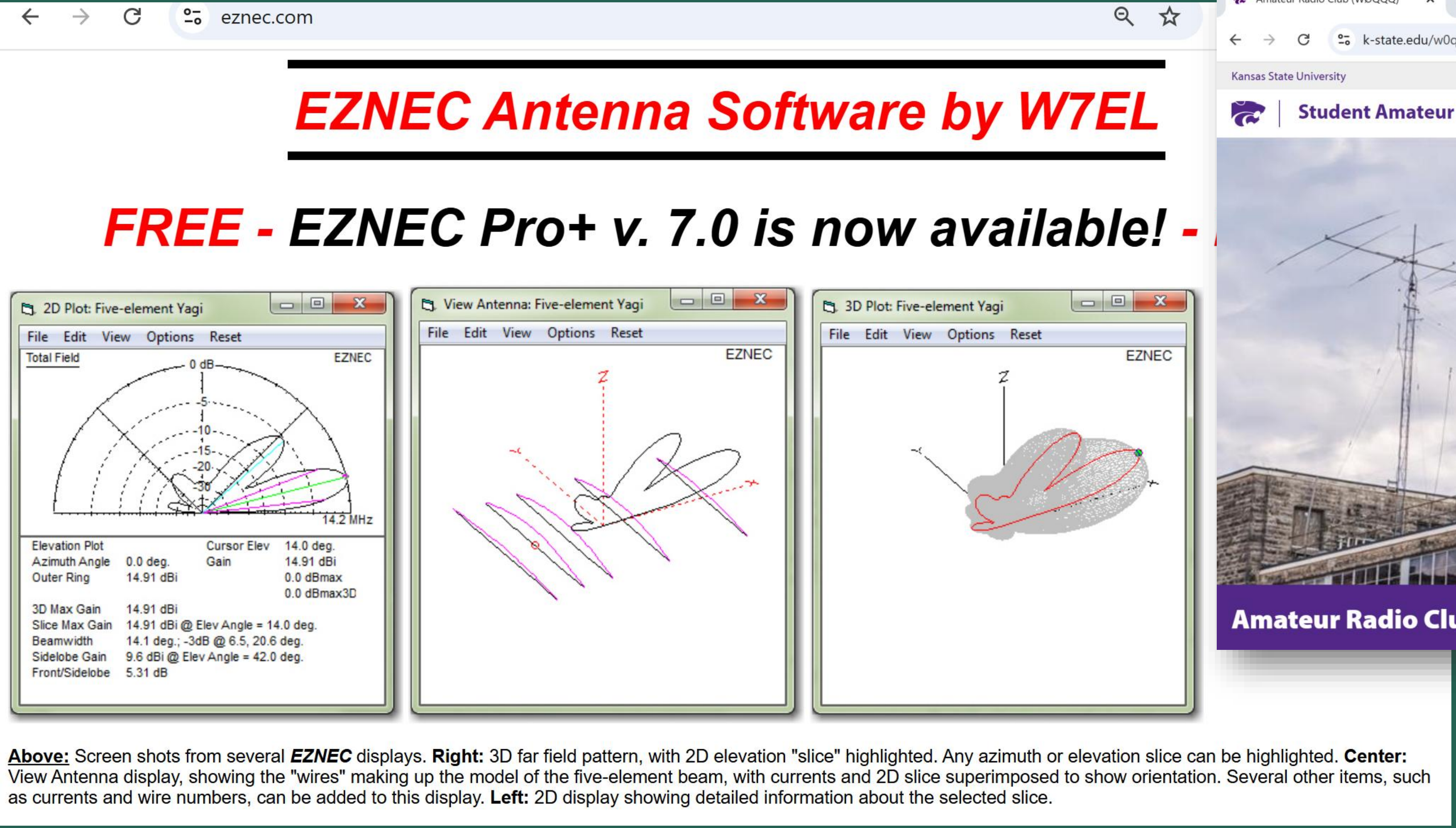
Welcome to Episode 4
(Final episode in 4-part introduction)

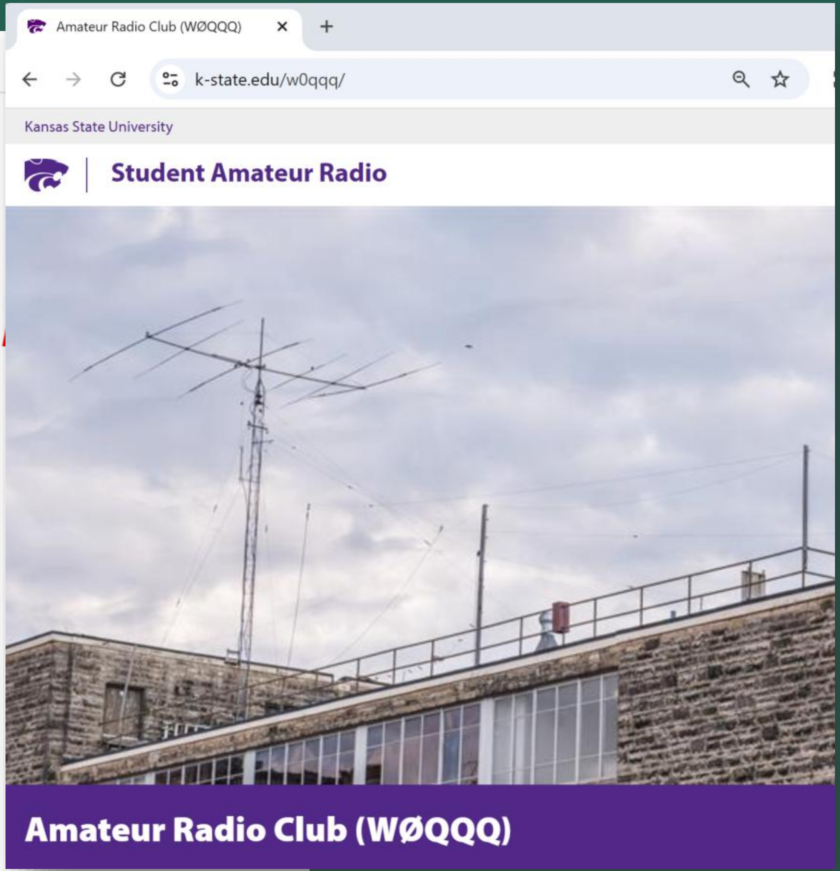
See Episodes 1 through 3 for
- ➢ *Definitions of "microwave",*
- ➢ *Example semester projects, and*
- ➢ *Day 1 lecture material*

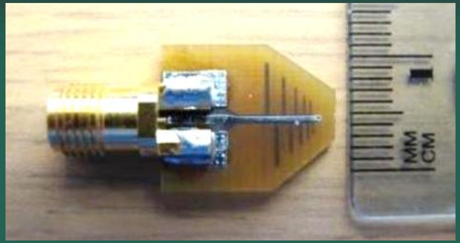
Today's Topics

- ✓ *Example Project and Lab Assignments*
- ✓ *Radio Design 201 Lab and Project Ideas*
- ⇨ *Free Software Examples*
- *"Low Cost" Test Equipment Examples*

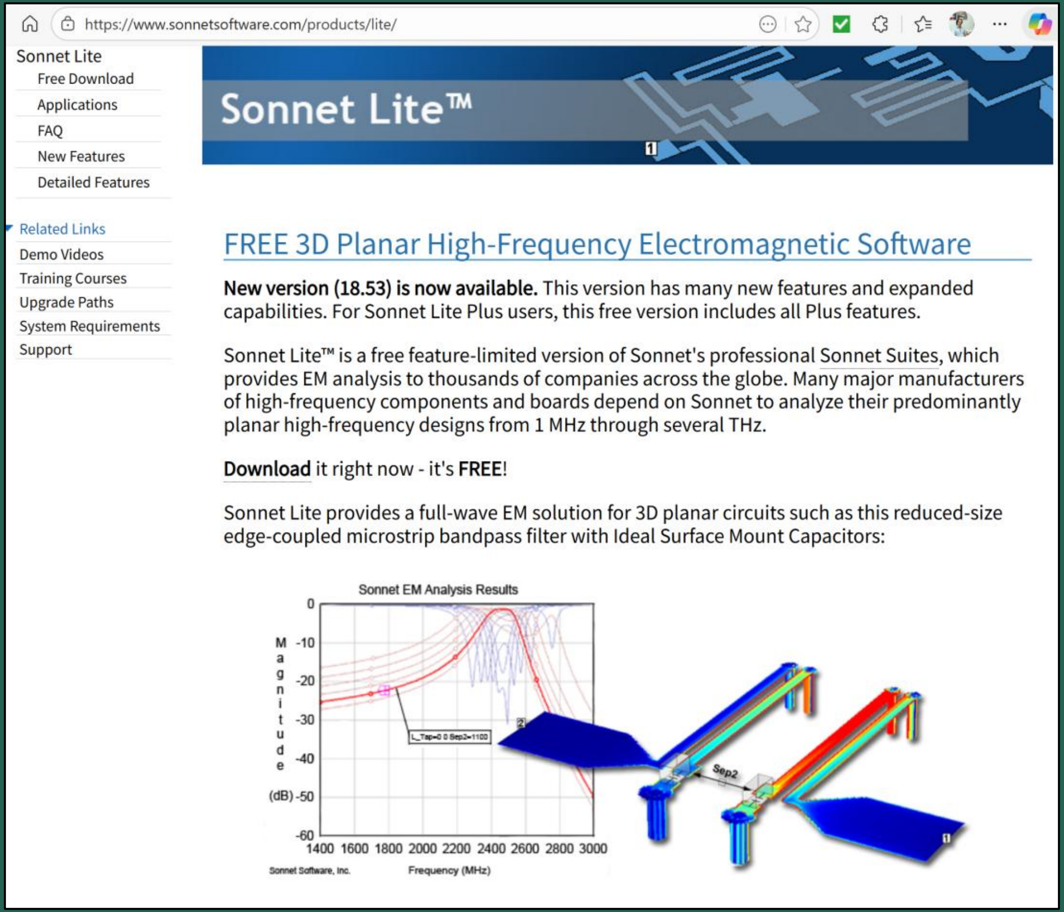
Free Antenna Simulators

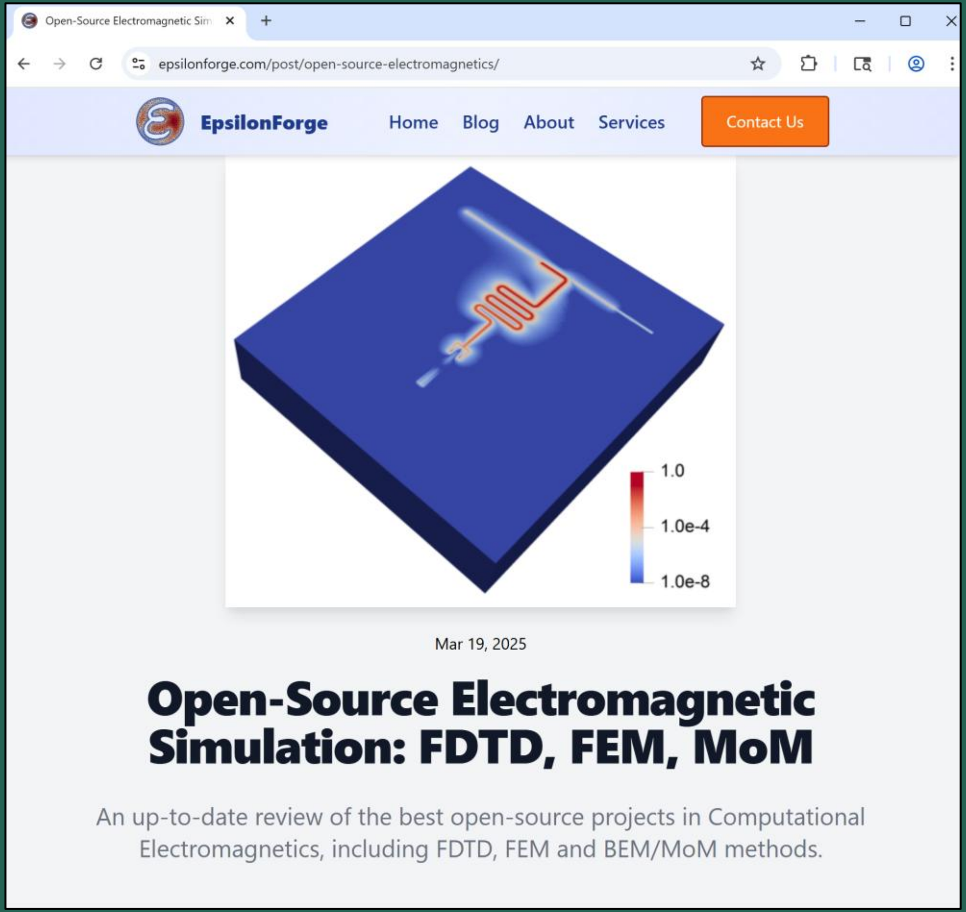






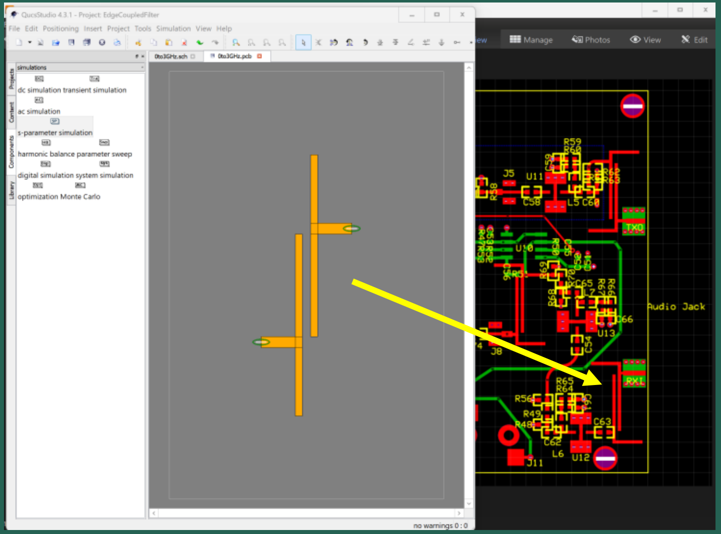
Some Free EM Simulators

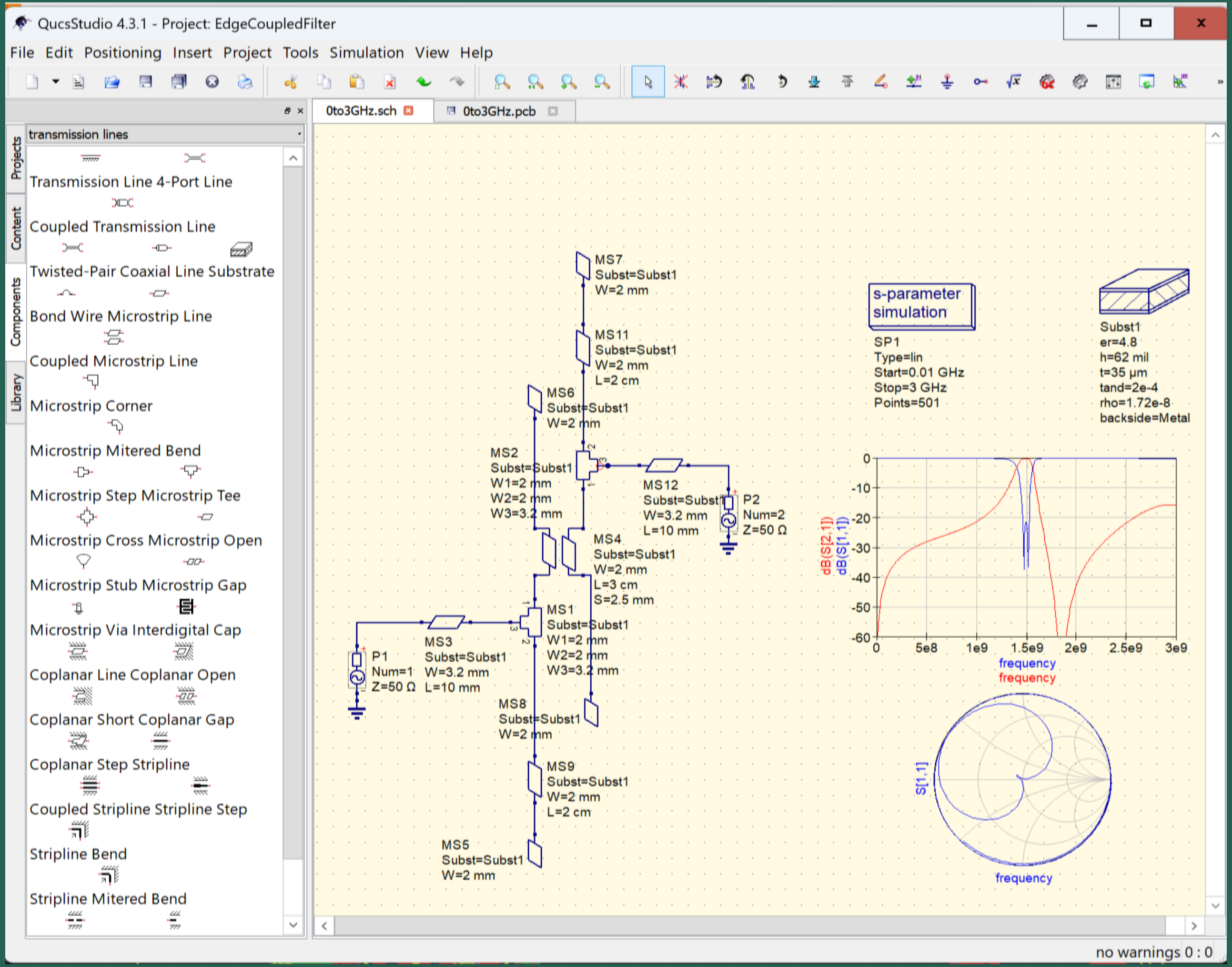




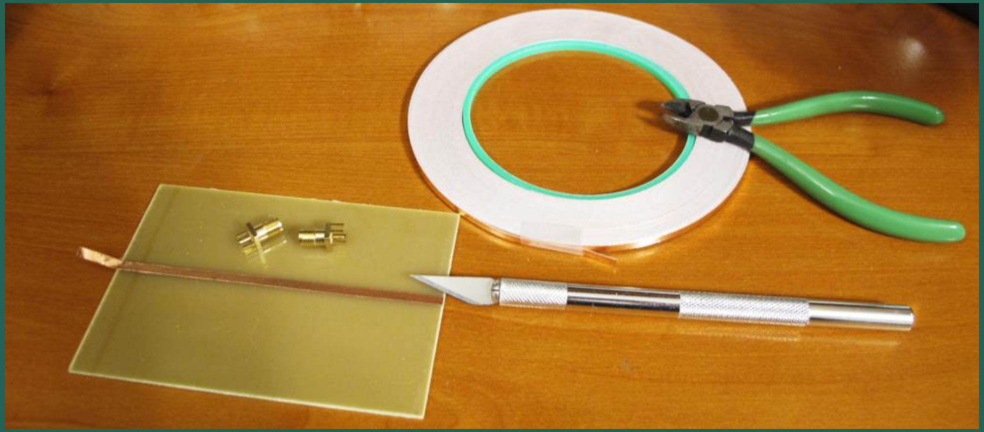
Microstrip Simulation with QucsStudio

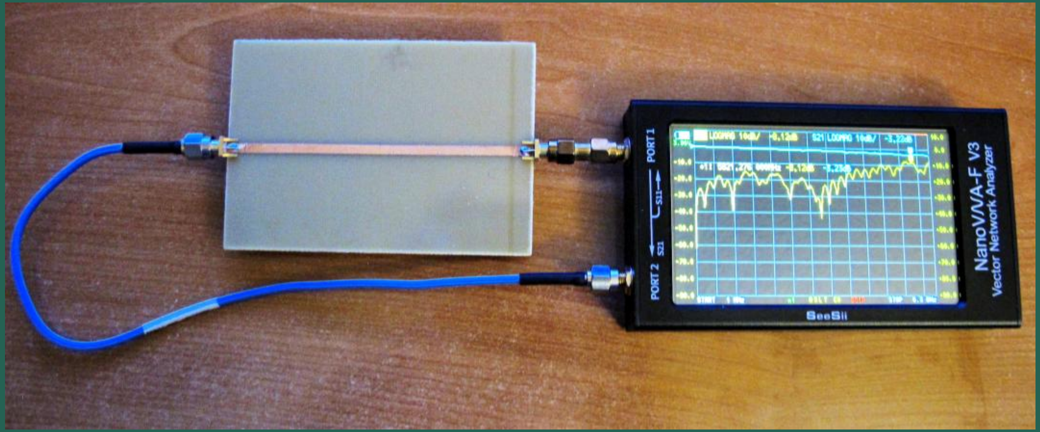
(aka "uSimmics")

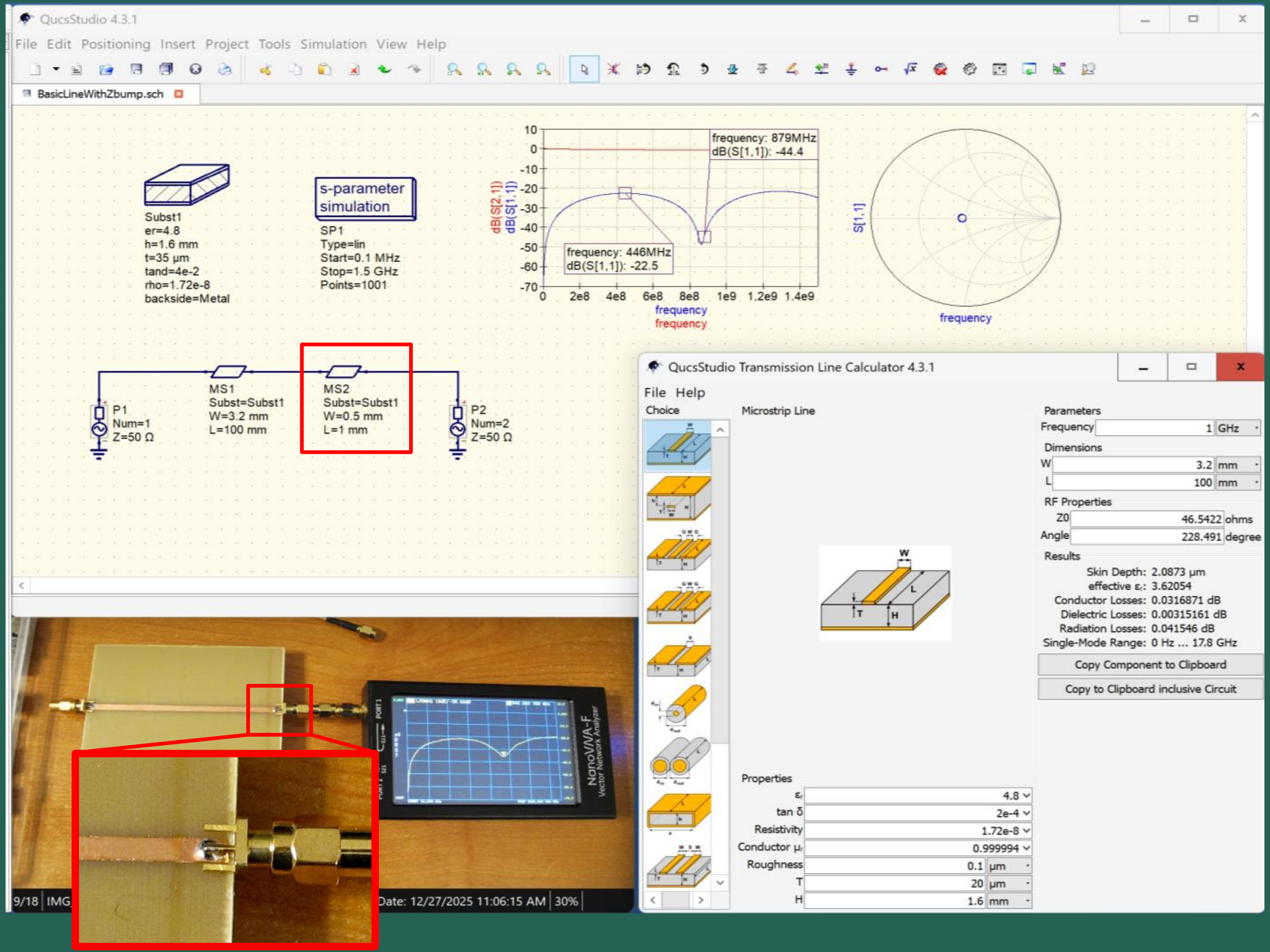




Microstrip Construction and Measurement







Today's "Lecture"

Announcements

Welcome to Episode 4
(Final episode in 4-part introduction)

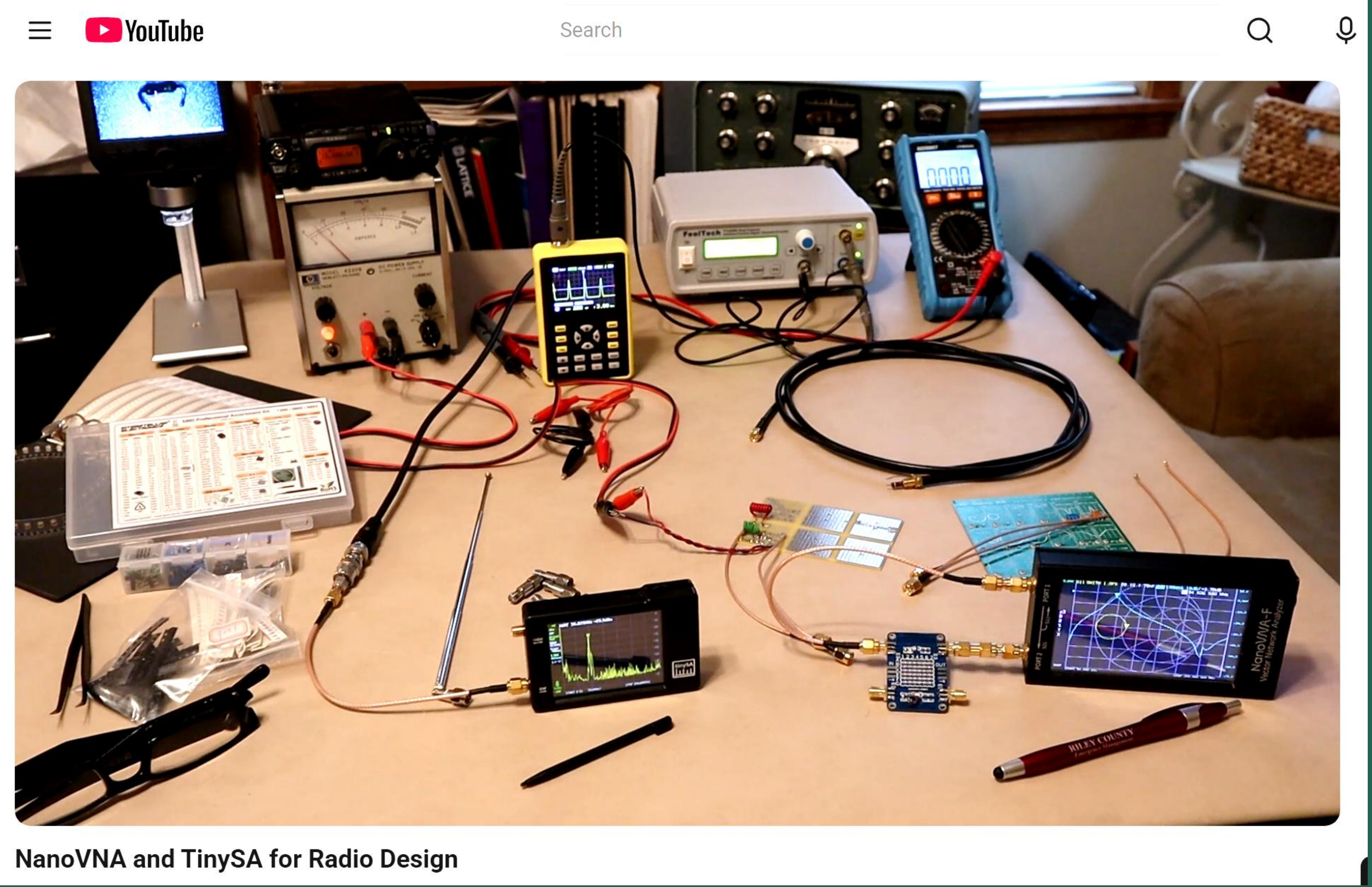
See Episodes 1 through 3 for

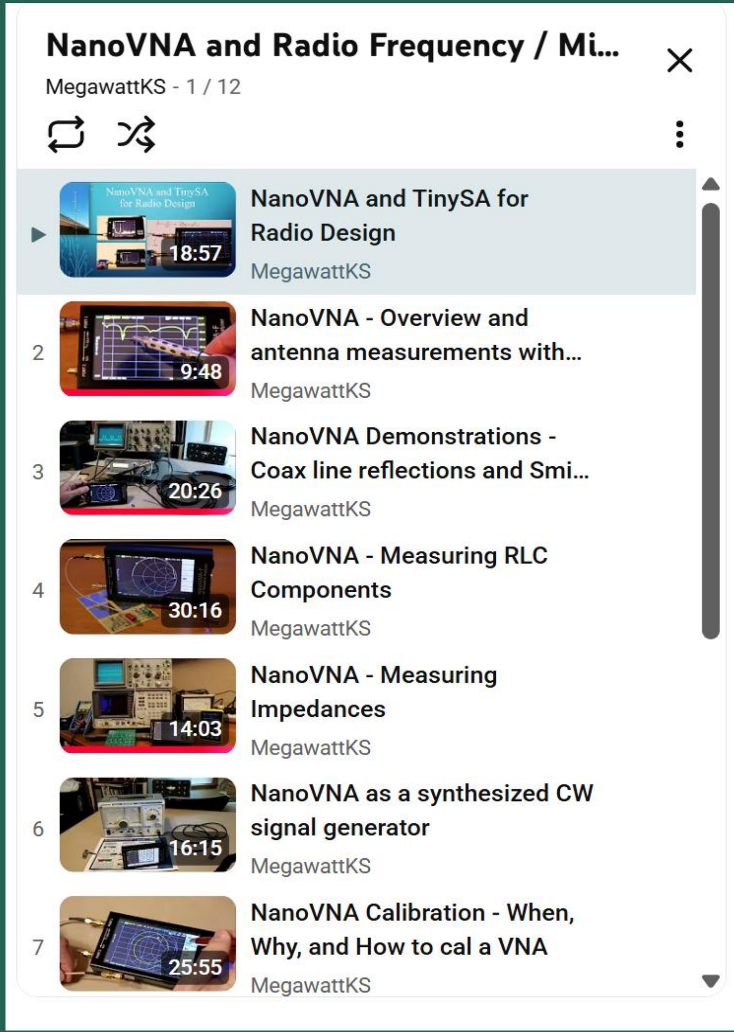
- ➢ *Definitions of "microwave",*
- ➢ *Example semester projects, and*
- ➢ *Day 1 lecture material*

Today's Topics

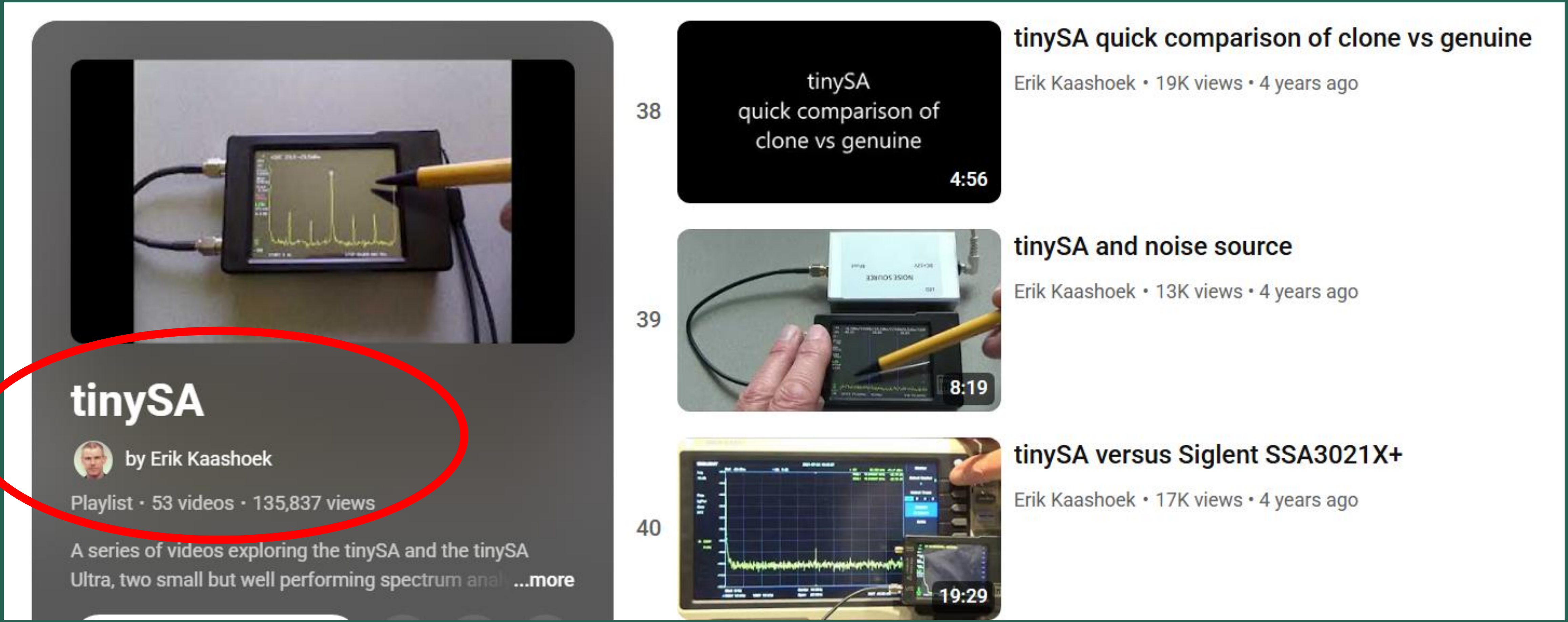
- ✓ *Example Project and Lab Assignments*
- ✓ *Radio Design 201 Lab and Project Ideas*
- ✓ *Free Software Examples*
- ⇨ *"Low Cost" Test Equipment Examples*

Home-Office RF/Microwave Lab (2021)

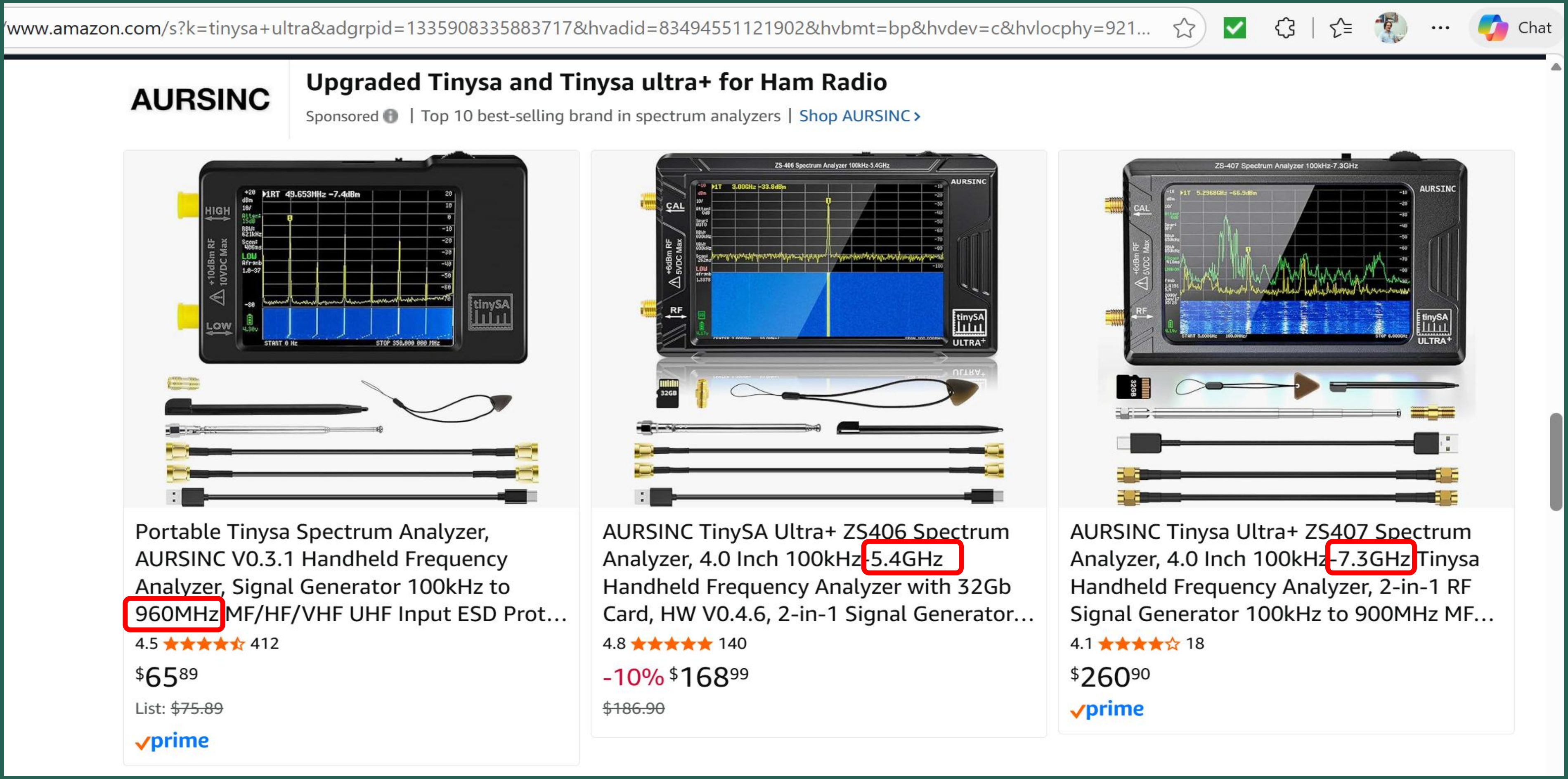




TinySA Spectrum Analyzers

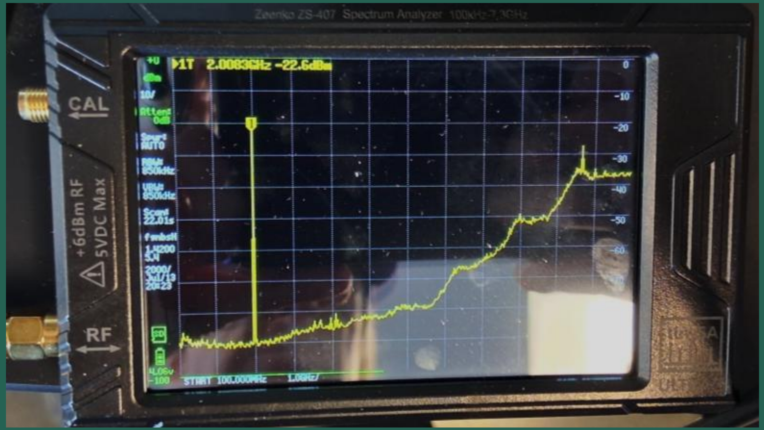


TinySA and TinySA-Ultra Spectrum Analyzers

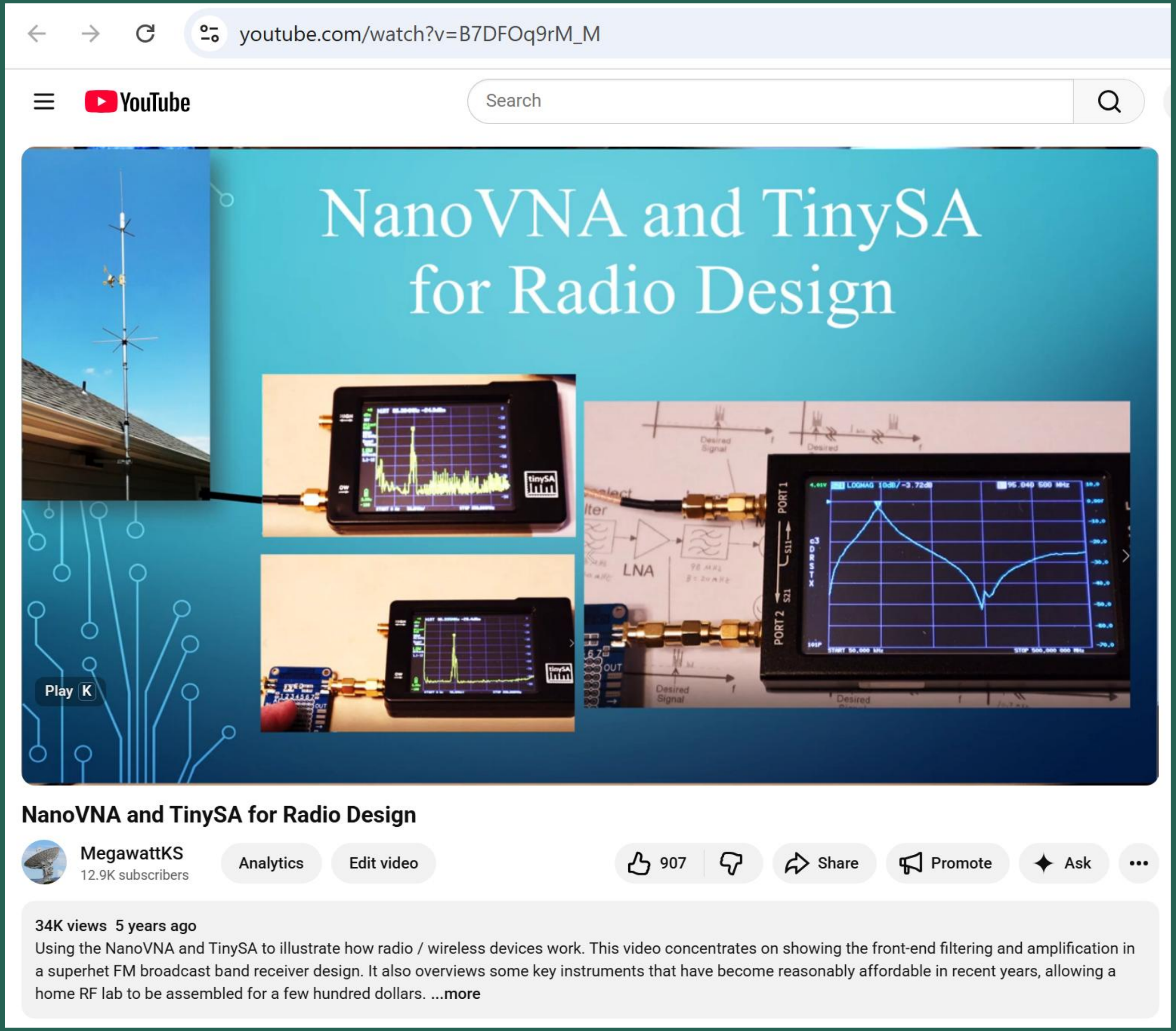


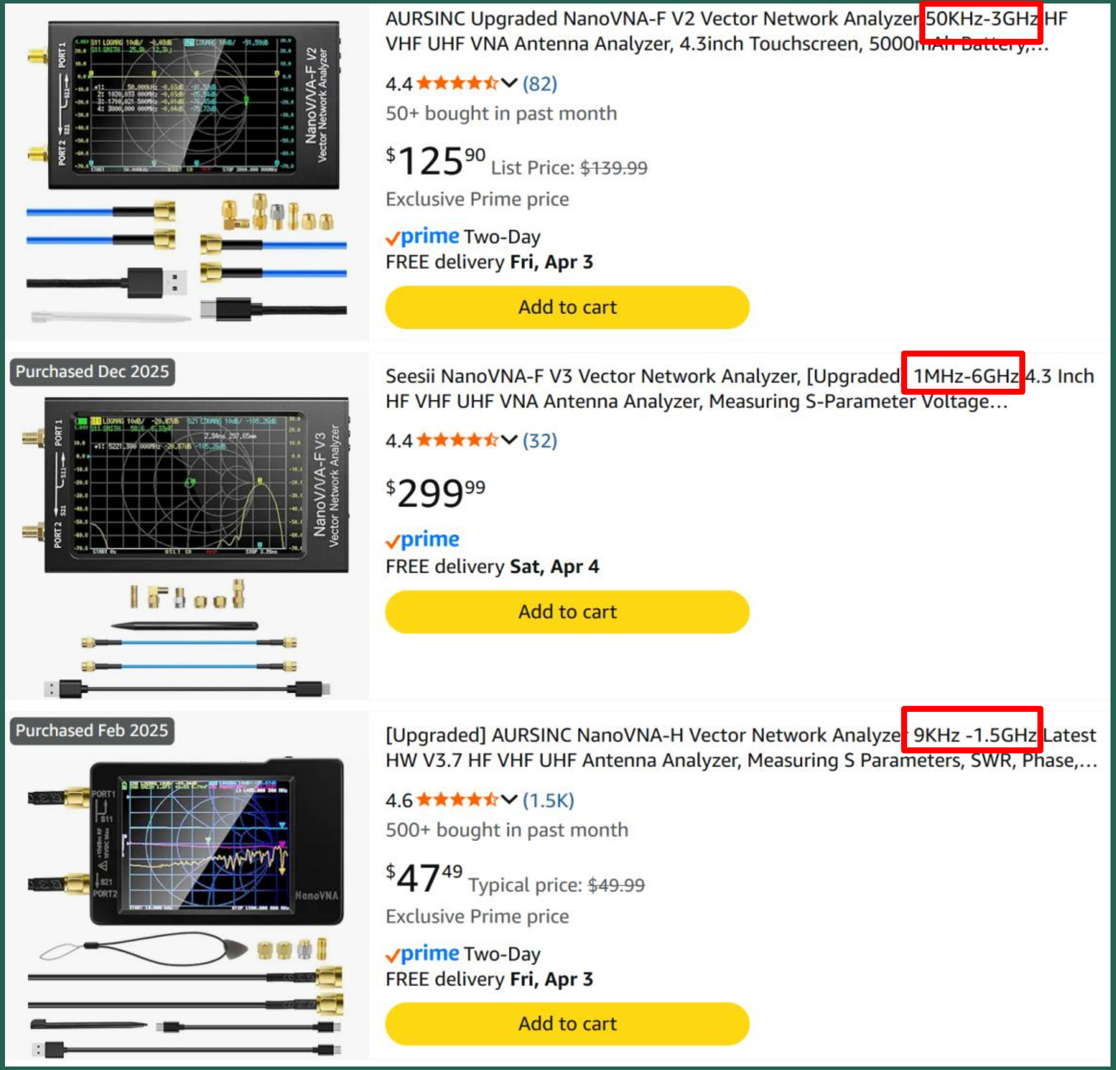
TinySA Ultra+ ZS407

2 to 10 GHz (over-range test)



NanoVNA Instruments





Lower-Cost 6 GHz "NanoVNA"

amazon prime | Deliver to William Manhattan 66503 | Industrial & Scientific | Search Amazon | EN | Hello, Acco

Click to see full view

Ask Rufus

- Can it measure antenna performance?
- Does it come with a carrying case?
- Is it compatible with Windows?
- Ask something else

LiteVNA-64 VNA Analyzer 50KHz-6.3GHz Portable Vector Network Analyzer Antenna Analyzer 4" Display

Brand: TZT

4.1 ★★★★☆ (27) | Search this page

Amazon's Choice

$179.00

Or $44.75 /2 weeks (x4). Select from 2 plans

prime Two-Day

FREE Returns

Usable to 8+ GHz !

Brand	TZT
Power Source	Battery Powered
Style	Digital
Color	Black
Item Weight	8.27 Grams

About this item

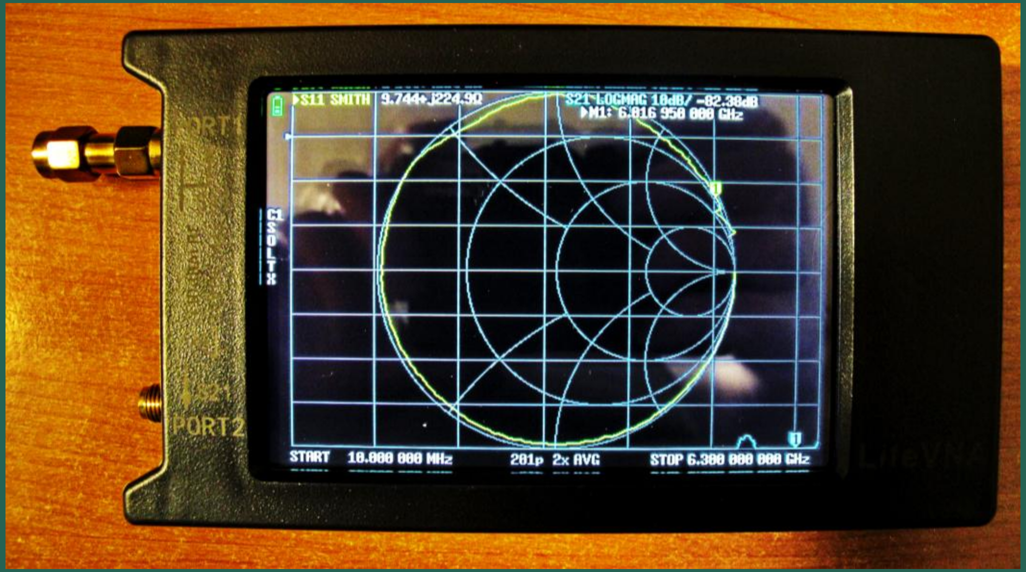
- with multiple internal RF switches for S11 and S21 measurements, and IFFT calculations for TDR and DTF measurements.
- the LiteVNA-64 now brings faster scanning speeds and more scan points in addition to a wider measurement range
- Combined with an easy-to-use interface consistent with the NanoVNA, the LiteVNA-64 can now be easily used as a field test tool.
- With a measurement range of up to 6.3GHz, the LiteVNA-64 is capable of meeting common amateur radio and IoT applications, as well as emerging 5GHz testing, enabling the application of the latest 5.8GHz wifi and 5.8GHz image transmission

› See more product details

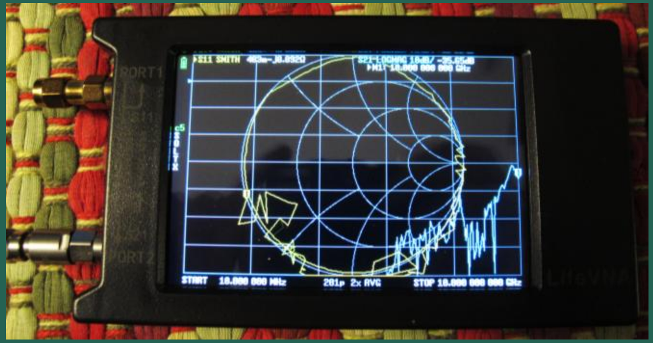
LiteVNA-64 Cal Verifications

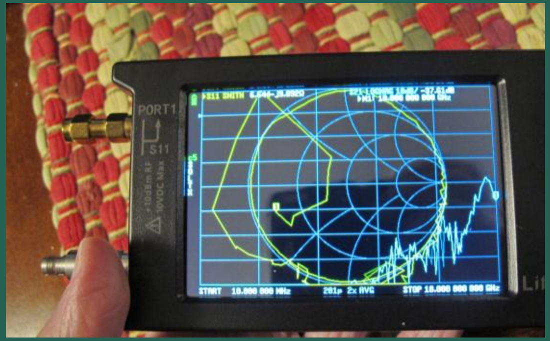
Including over-range Testing

Calibration check/verification:
Looking at 2 cm open line after calibration thru rated 6.3 GHz

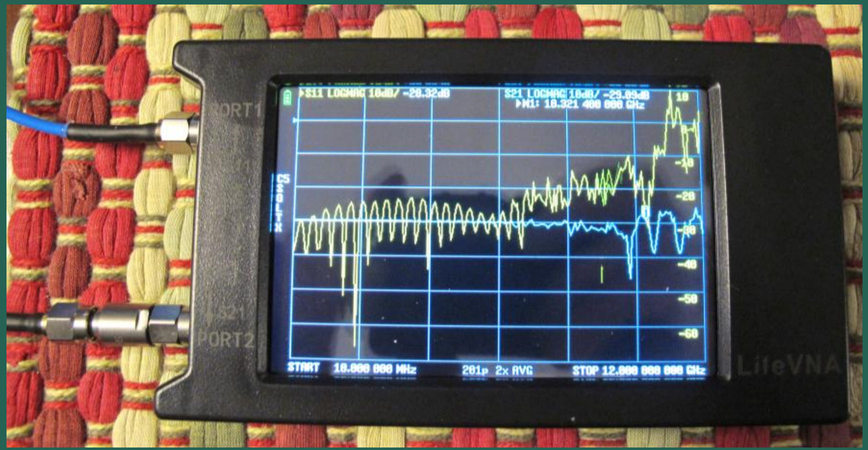


S11 Cal-verification thru 10 GHz

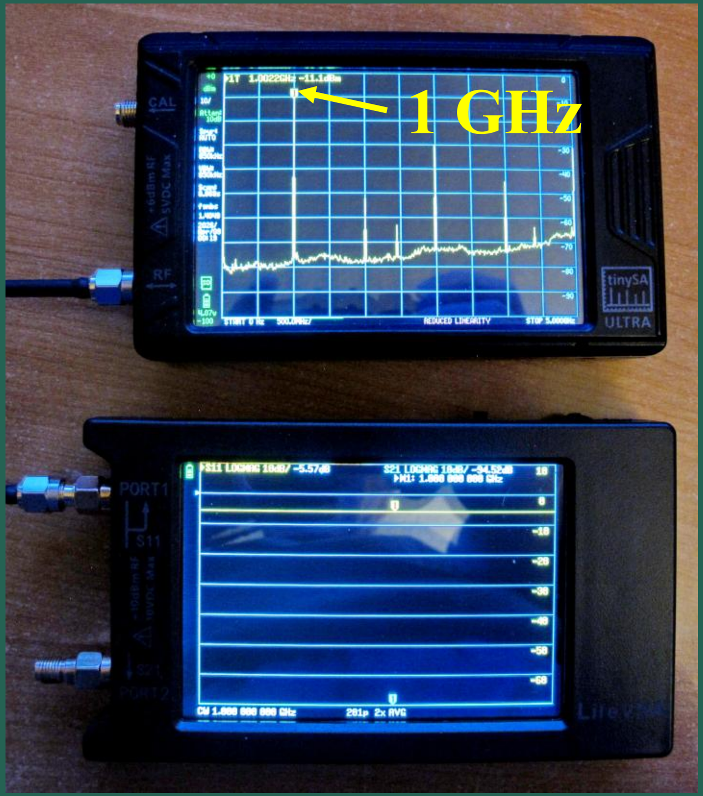




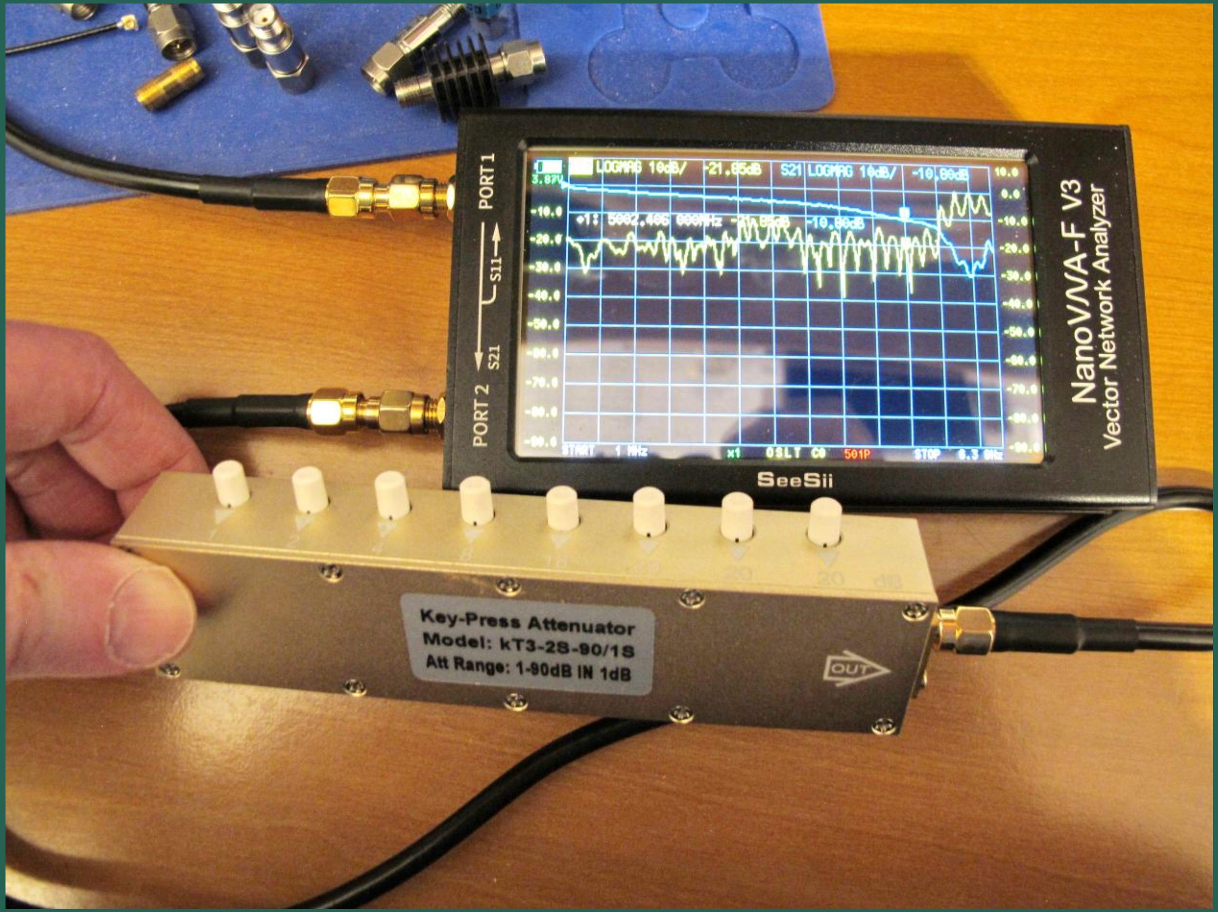
Looking at 30 dB attenuator thru 12 GHz…

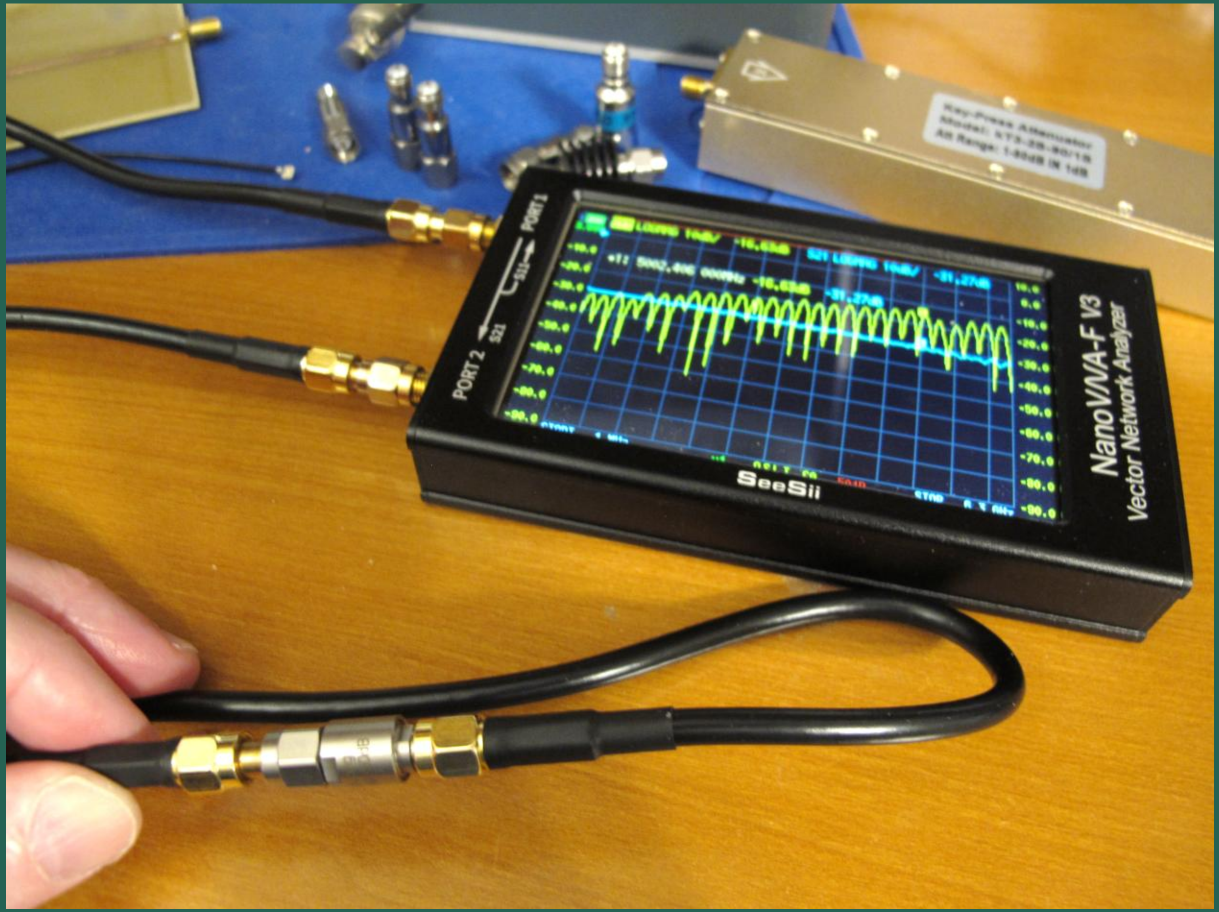


TinySA, LiteVNA, and NanoVNA as
Signal Generators

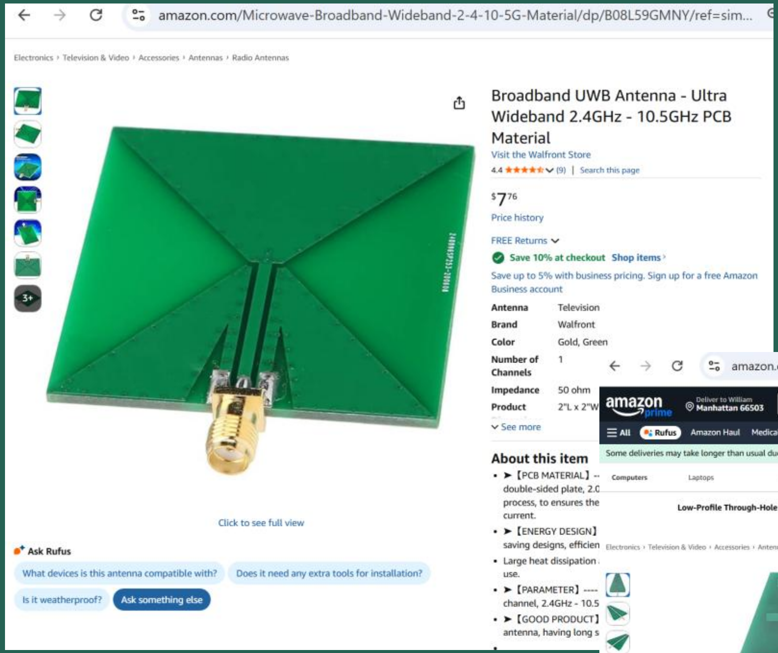


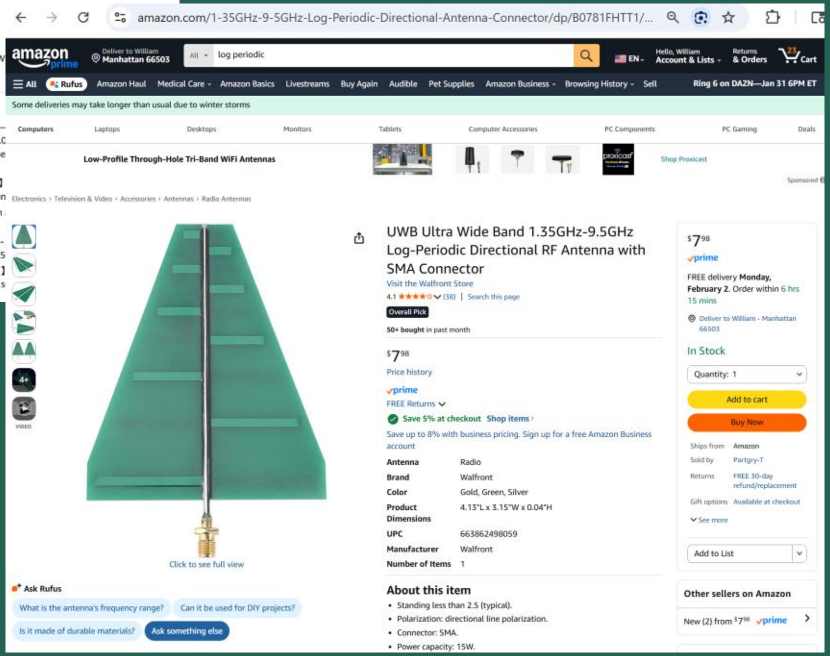
Attenuators

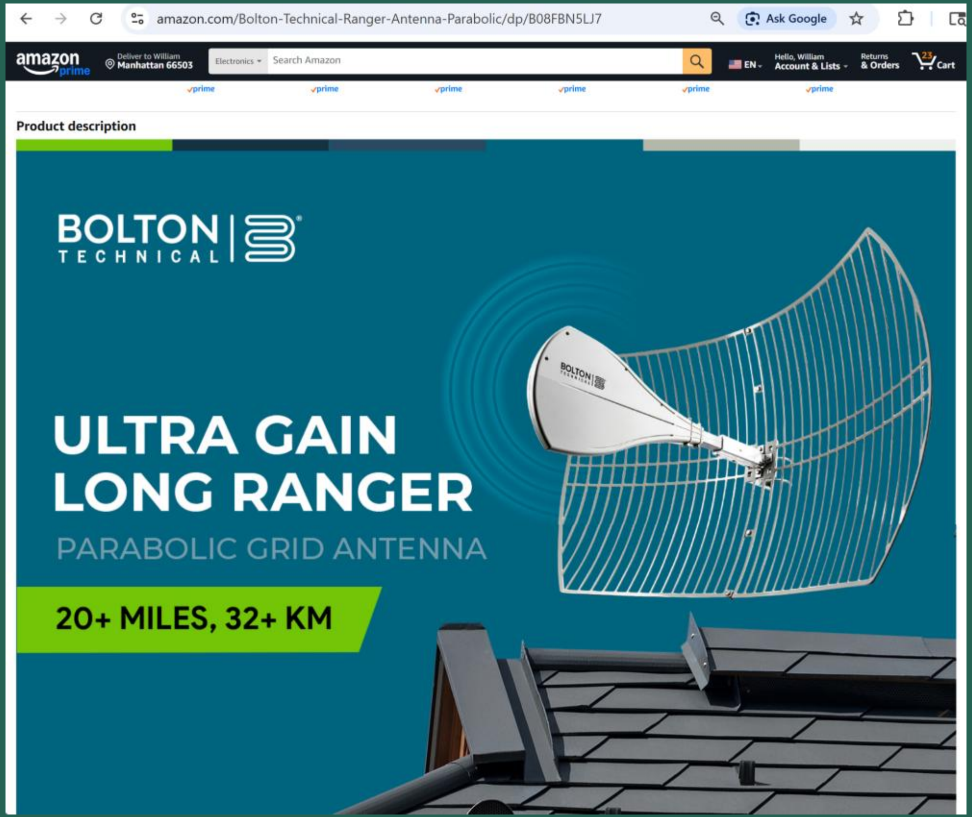




Low Cost Microwave Antennas



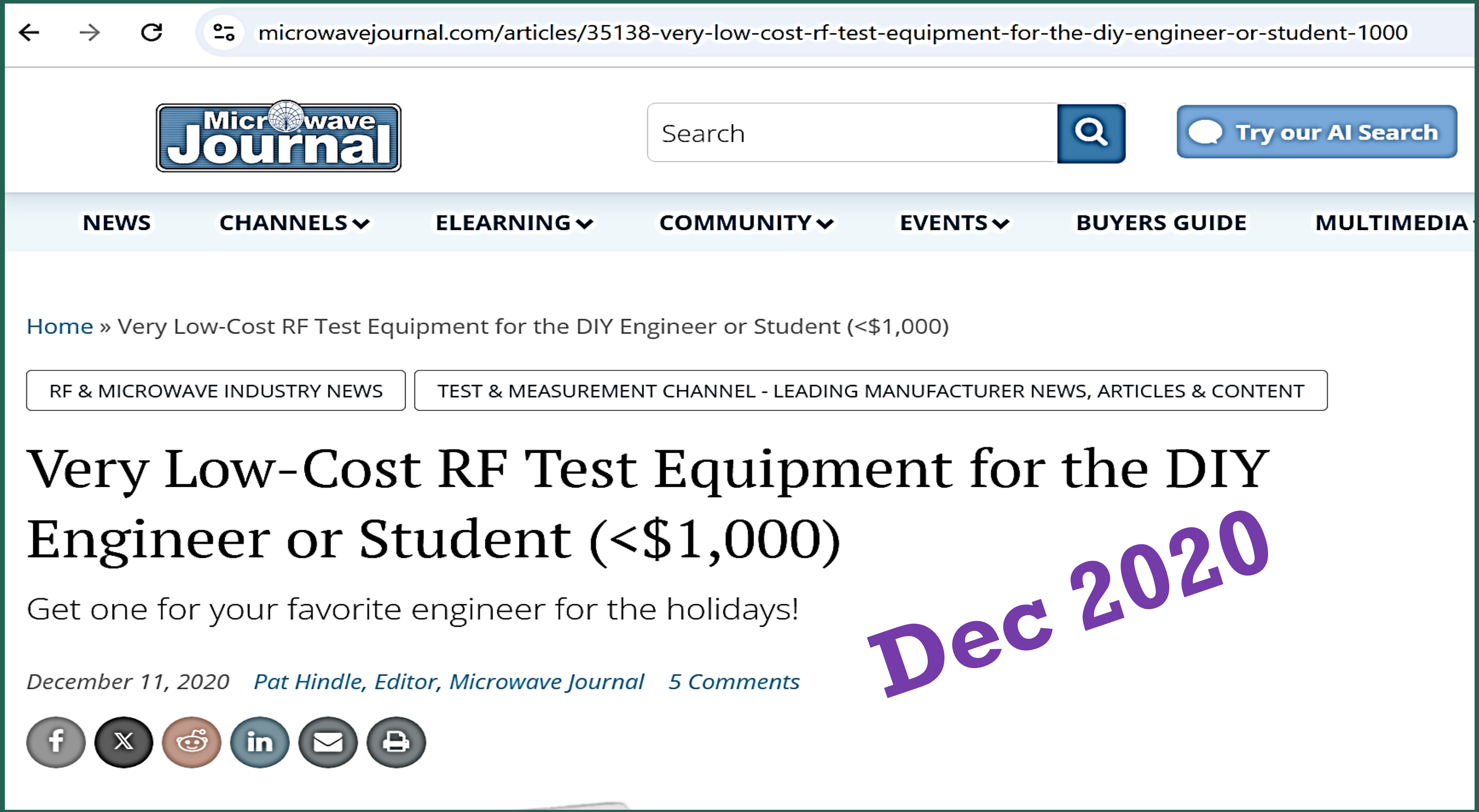




Thanks For Watching !

[MegawattKS - YouTube](https://www.youtube.com) https://ecefiles.org

Related Microwave and Lab-on-a Budget Info


microwavejournal.com/articles/35138-very-low-cost-rf-test-equipment-for-the-diy-engineer-or-student-1000

Microwave Journal

Search

Try our AI Search

NEWS | CHANNELS | ELEARNING | COMMUNITY | EVENTS | BUYERS GUIDE | MULTIMEDIA

Home » Very Low-Cost RF Test Equipment for the DIY Engineer or Student (<$1,000)

RF & MICROWAVE INDUSTRY NEWS | TEST & MEASUREMENT CHANNEL - LEADING MANUFACTURER NEWS, ARTICLES & CONTENT

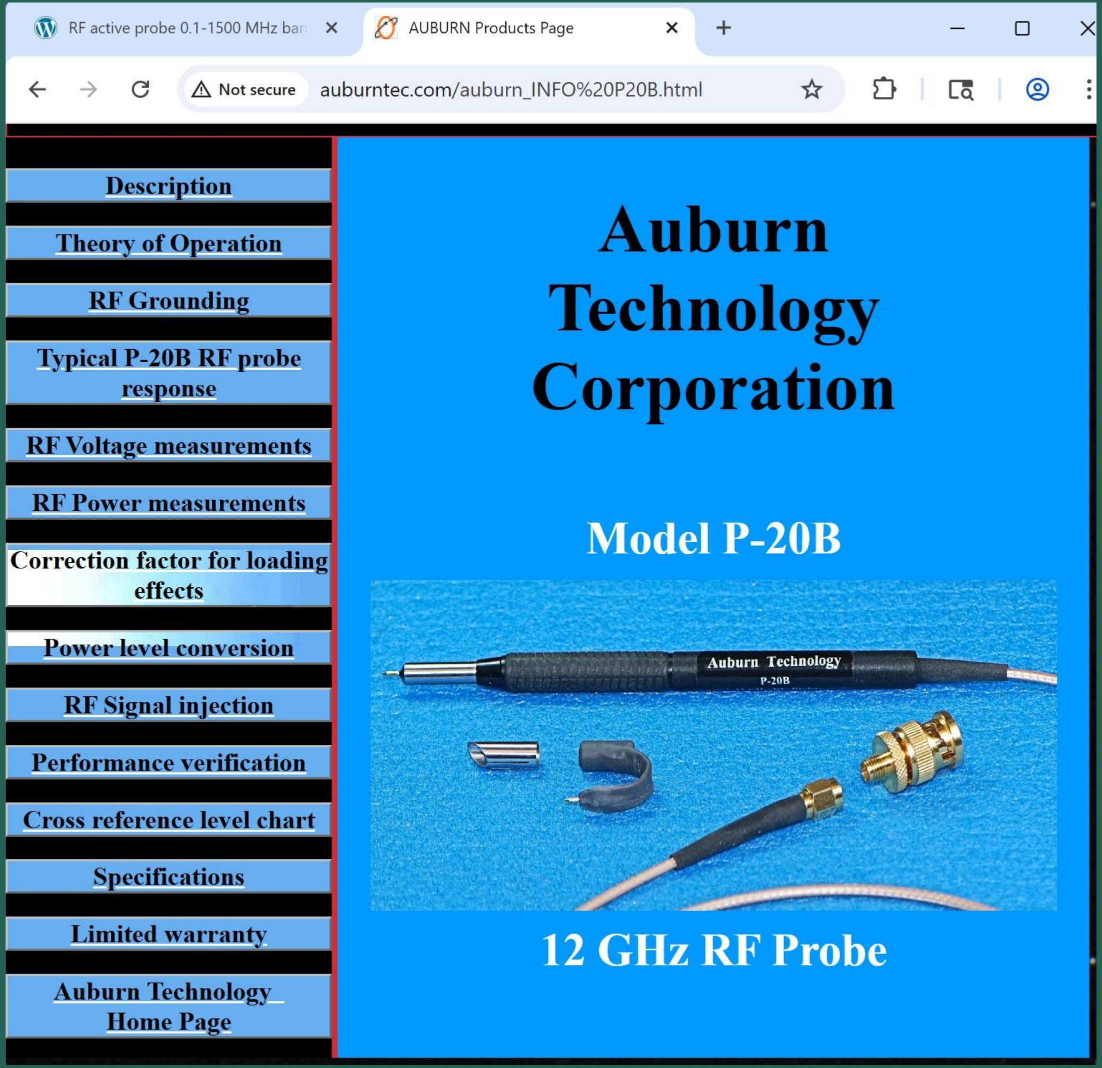
Very Low-Cost RF Test Equipment for the DIY Engineer or Student (<$1,000)

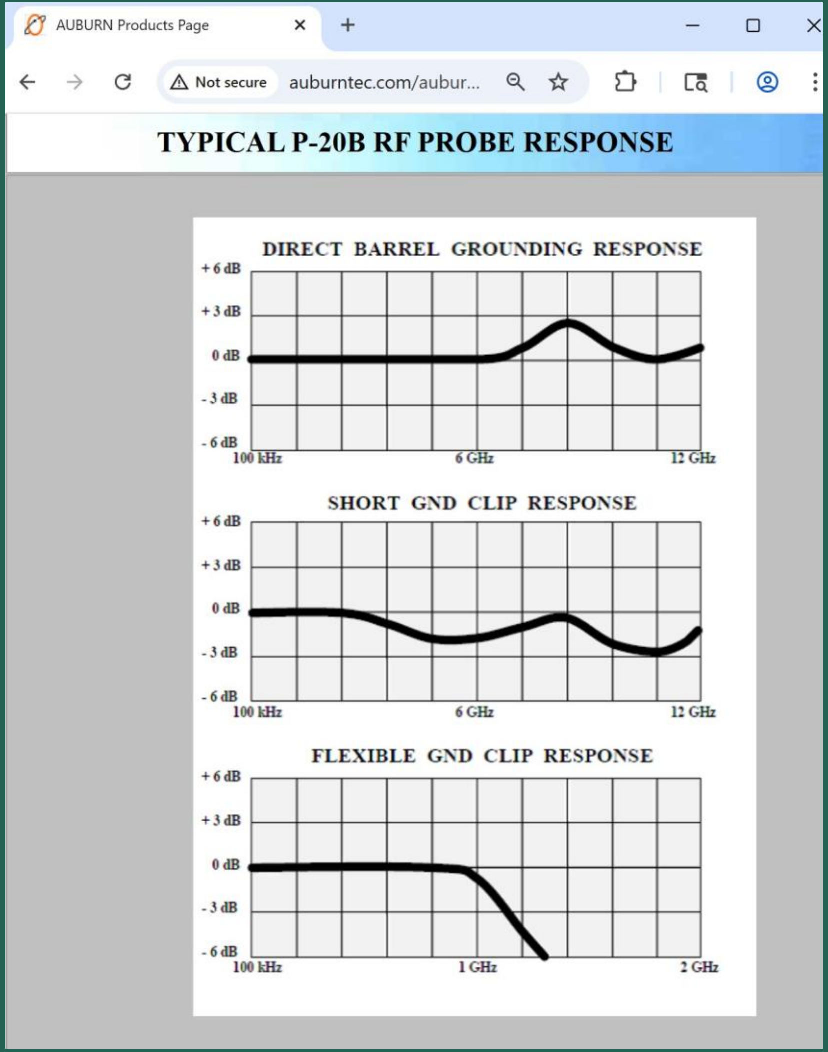
Get one for your favorite engineer for the holidays!

December 11, 2020 *Pat Hindle, Editor, Microwave Journal* *5 Comments*

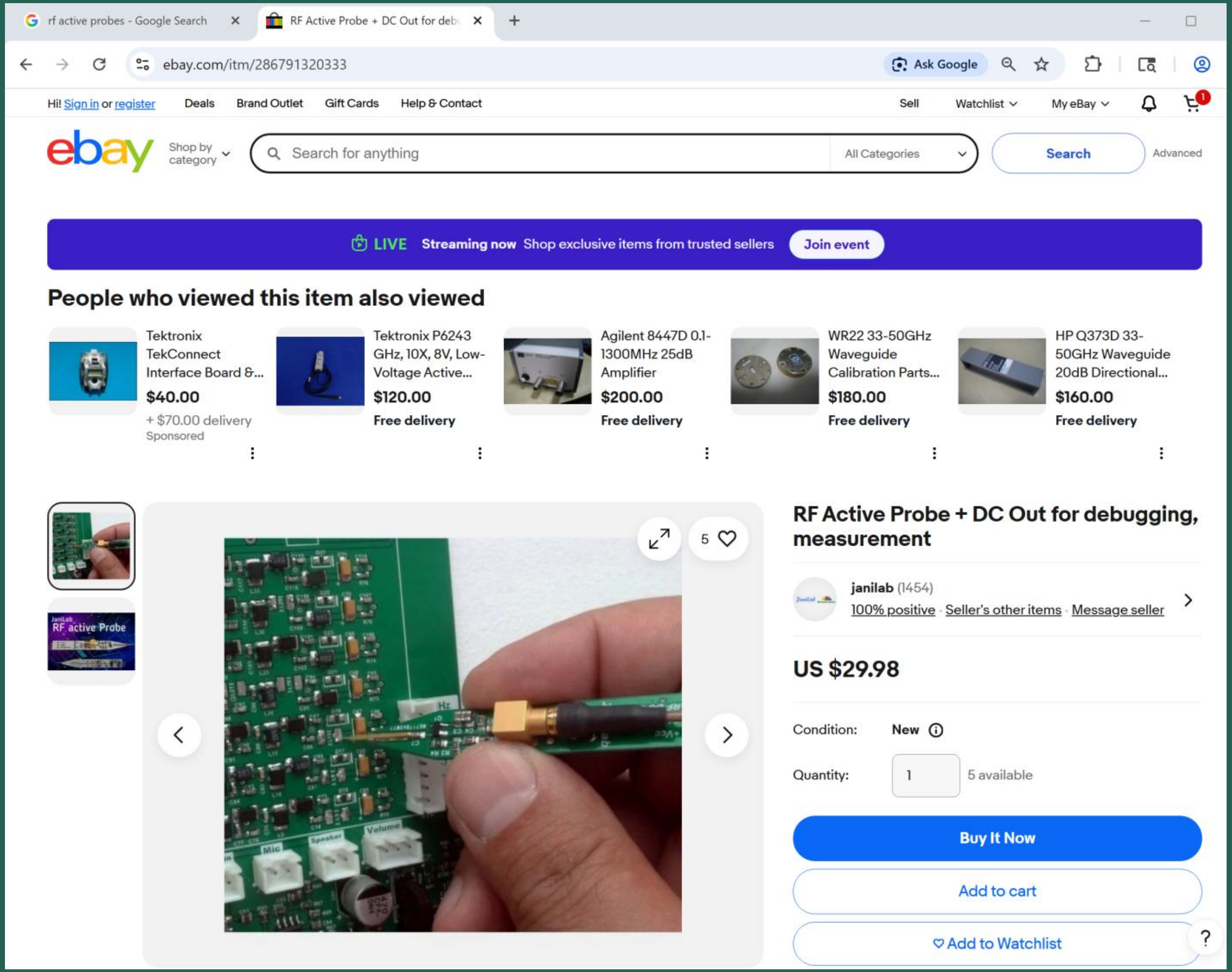
Dec 2020

RF/Microwave Passive Probes

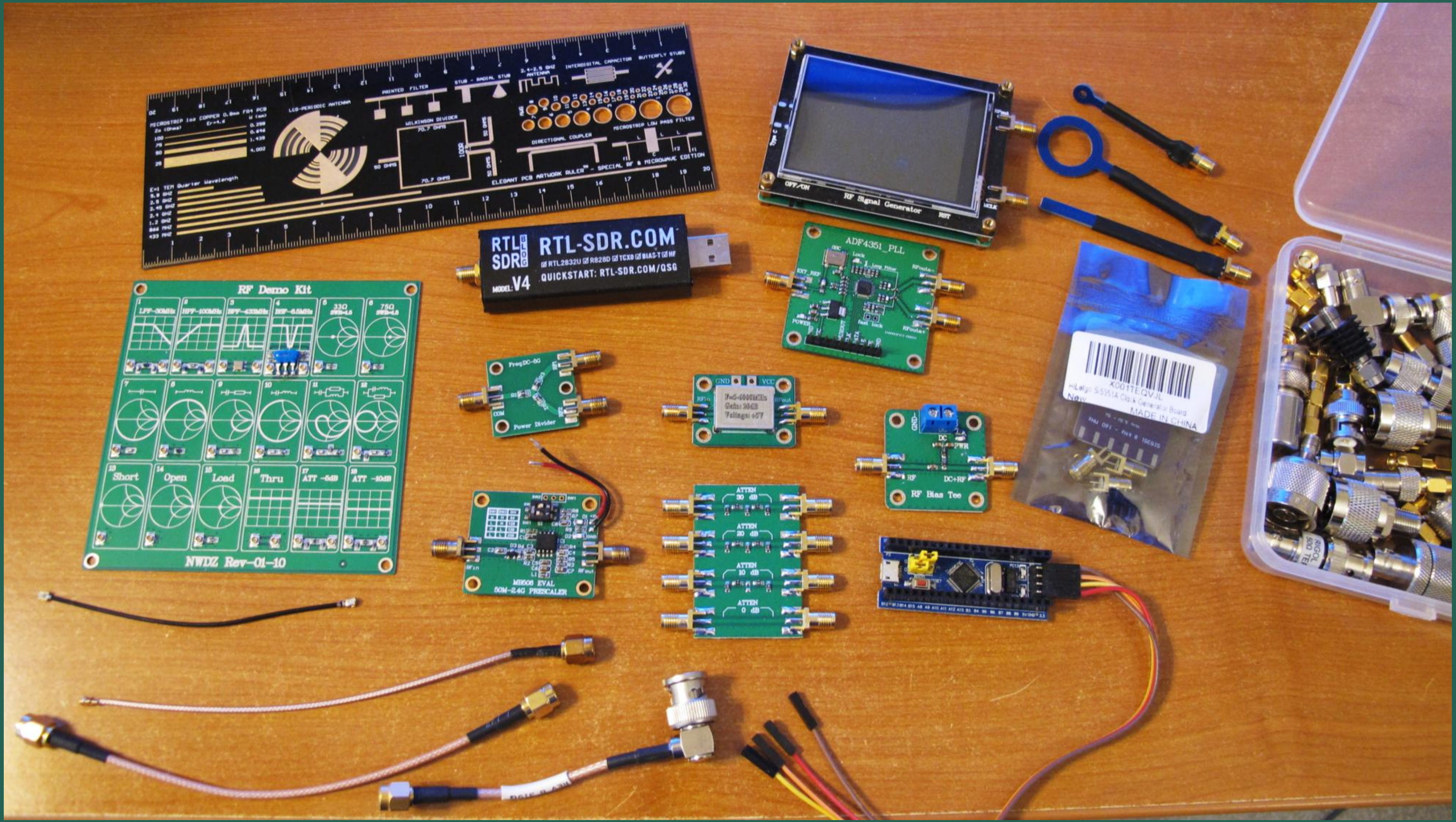




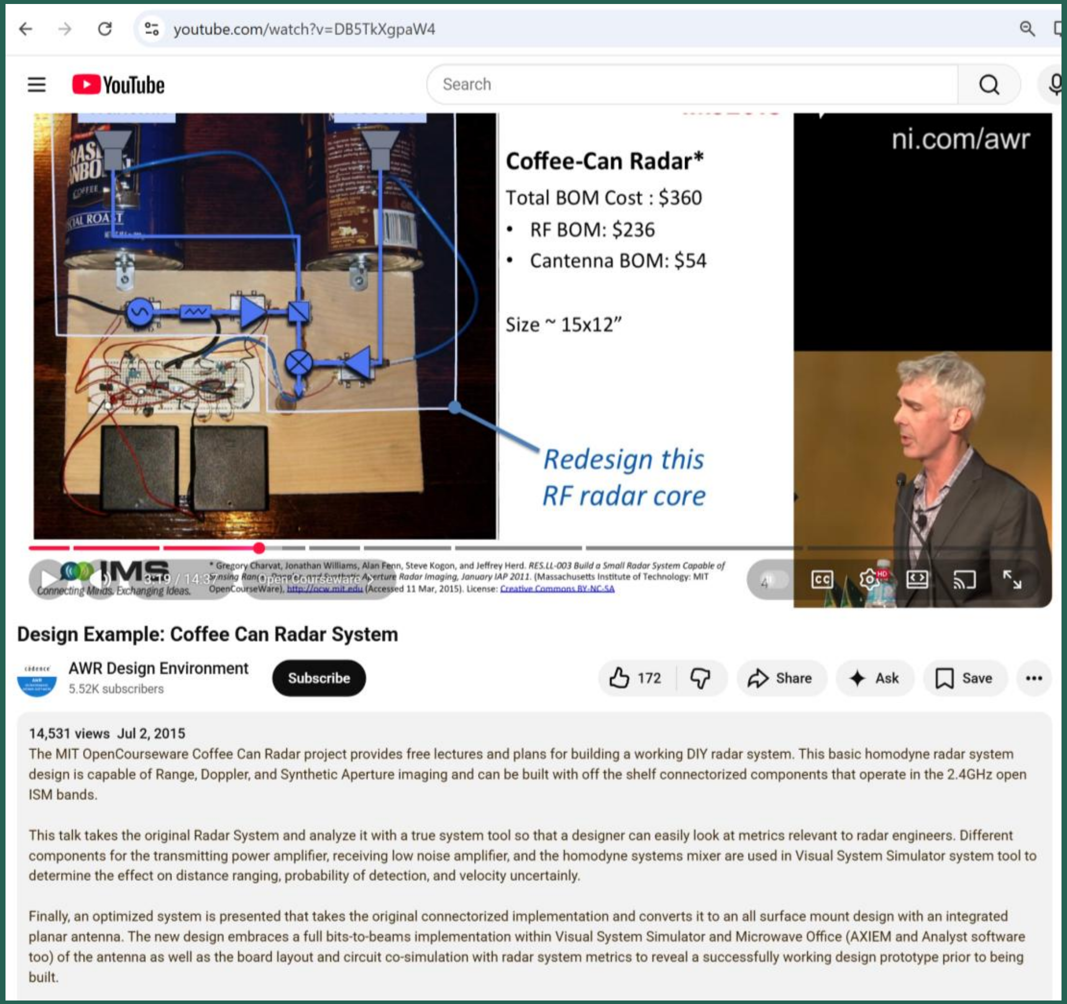
RF/Microwave Active Probes

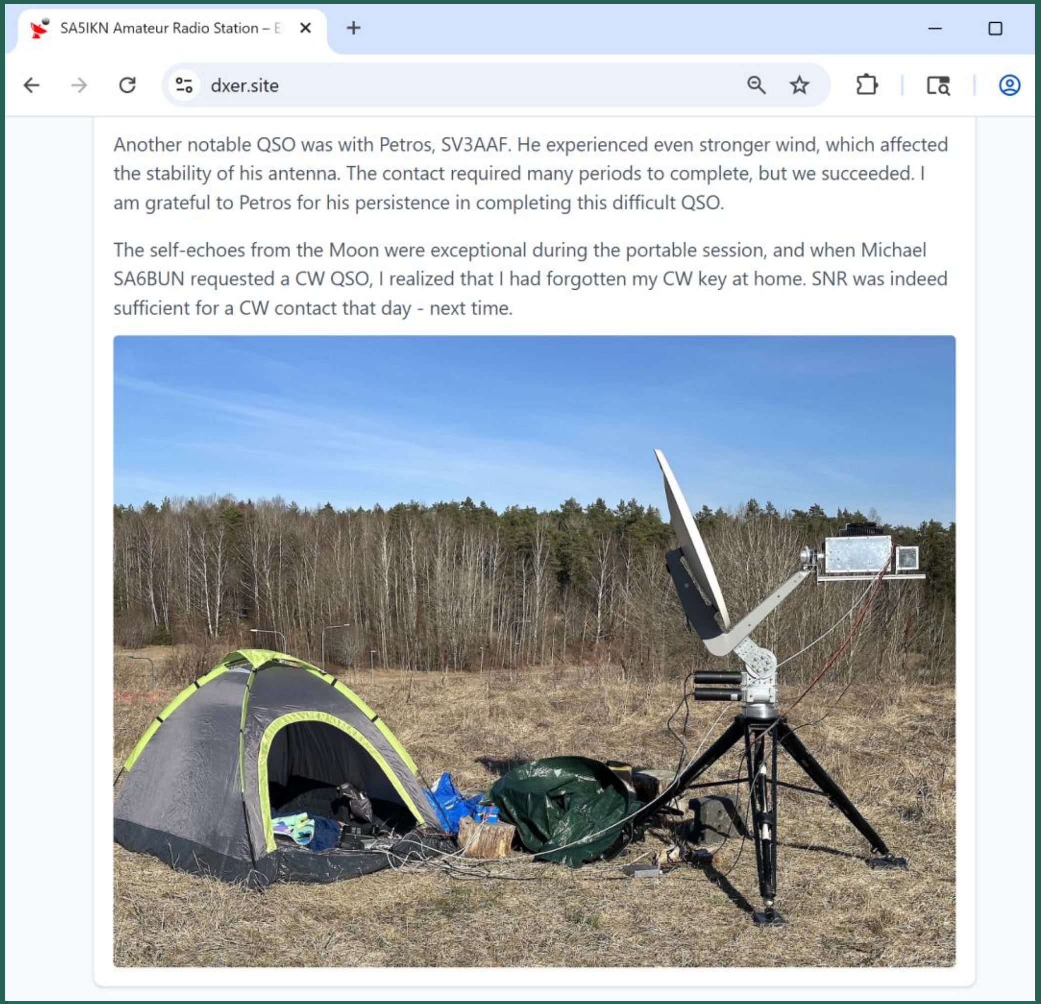


Miscellaneous Stuff ☺

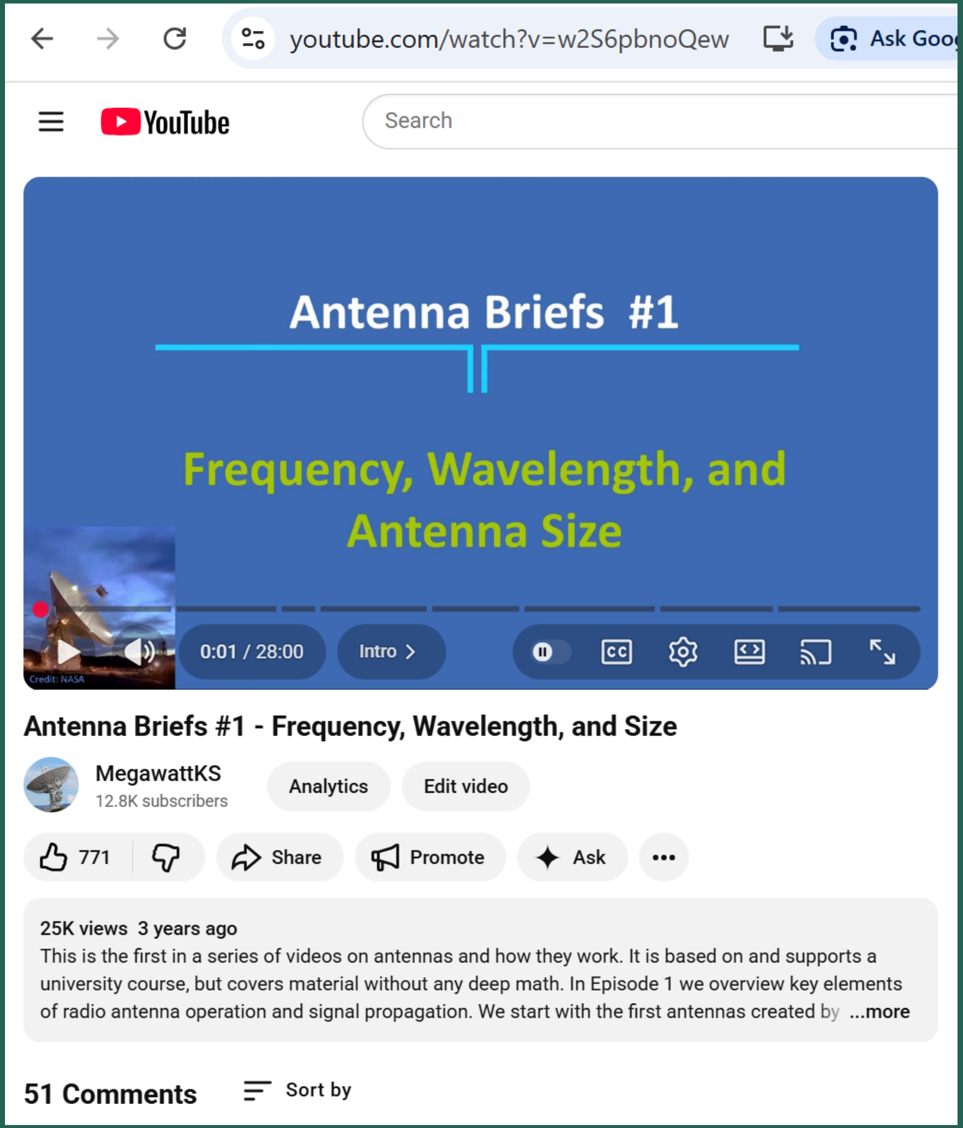


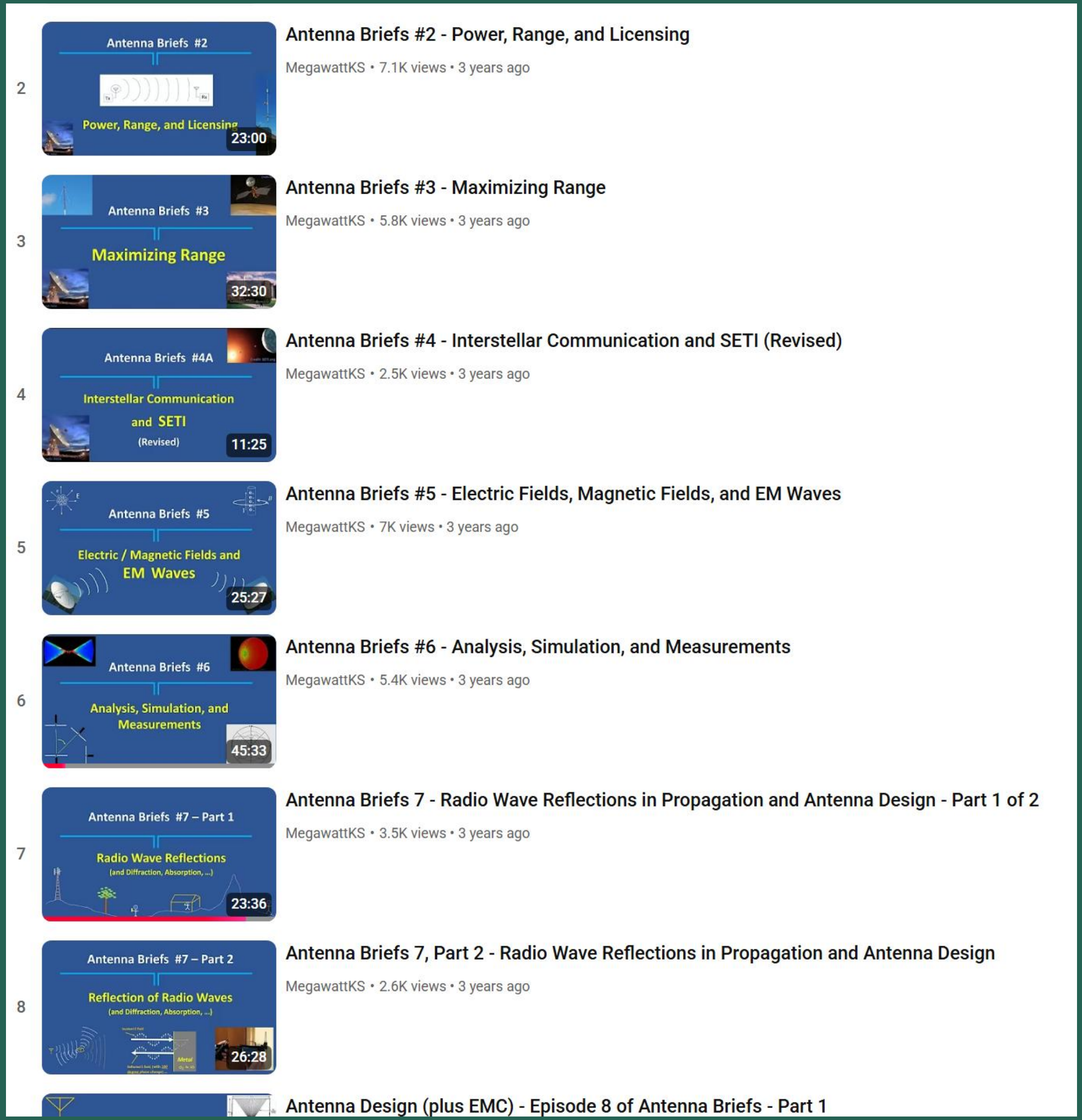
Some Related Videos/Sites



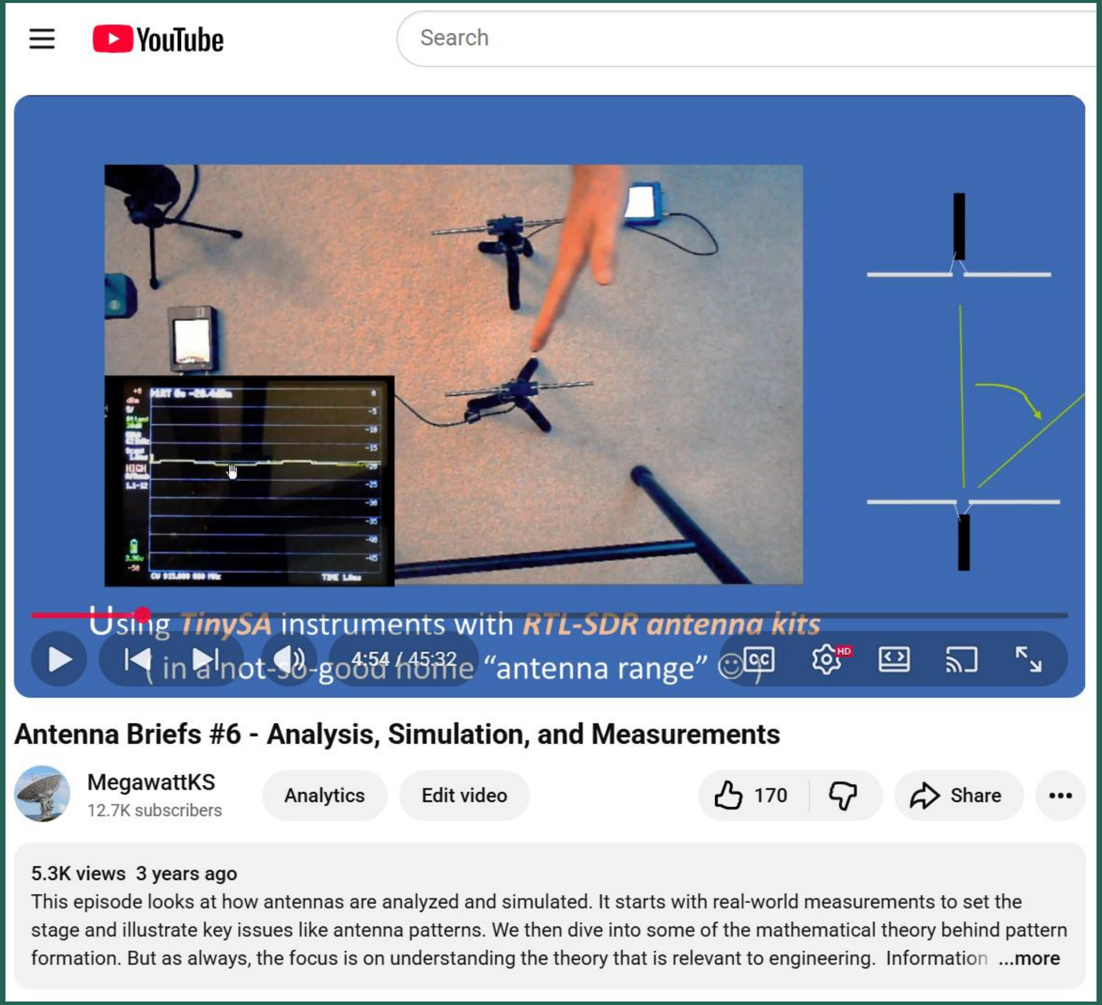


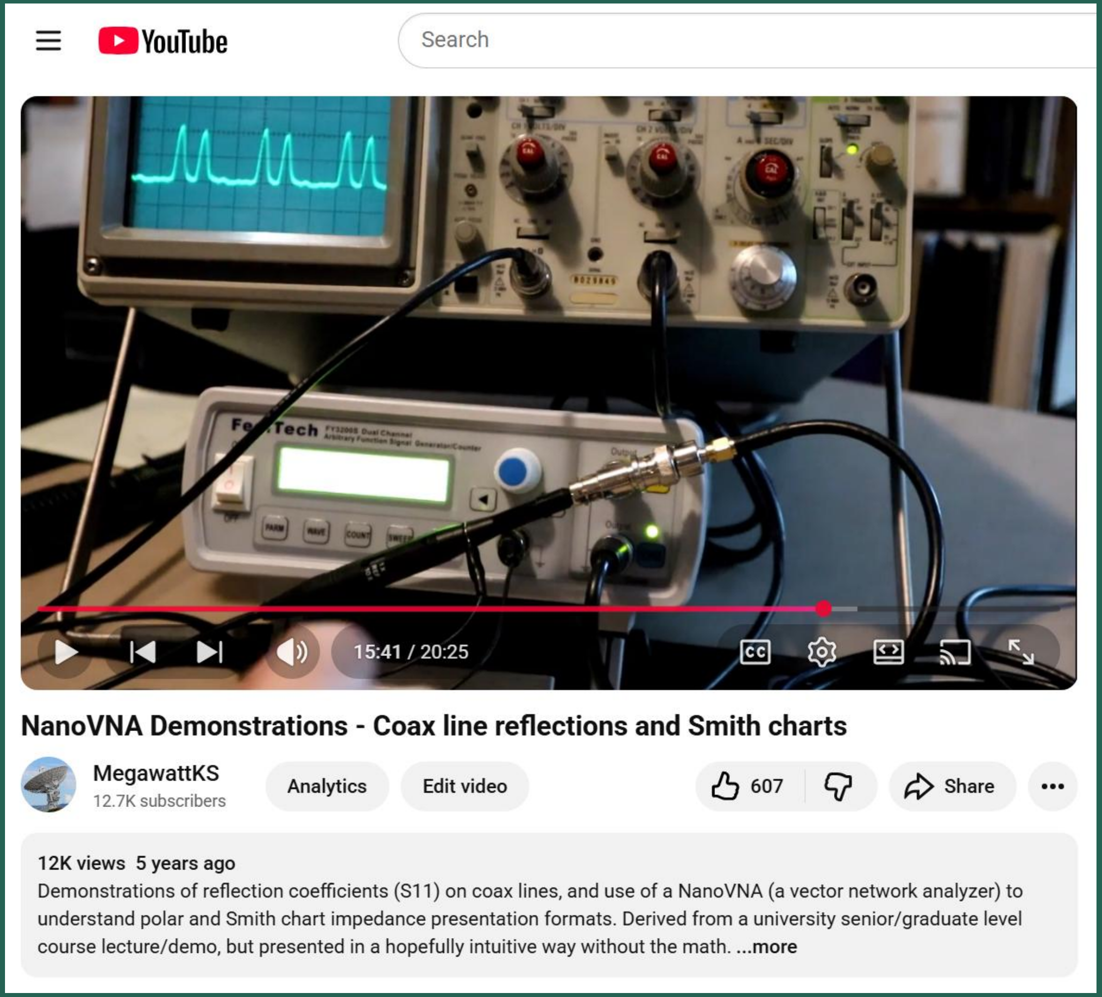
"Antenna Briefs" Playlist



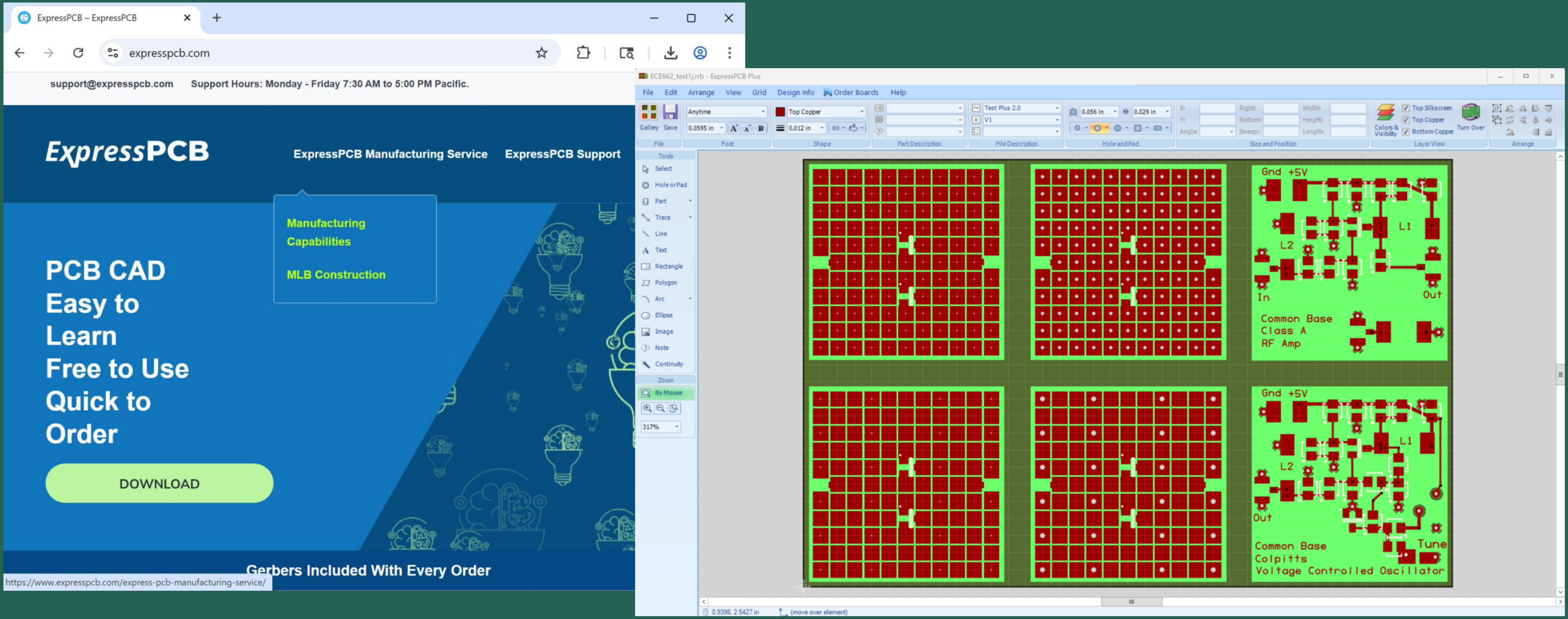


Demos in Home Lab





Fast and Easy PCB Layout Software and Fab (US-based)



KiCad Layout Software
and typical Fab vendor (non-US-based)