LTspice for HAM's

- Getting started
- Selected educational examples
- DRAKE TR4-C Ring-Modulator
- Collins 618S-1 Ratio-Detector

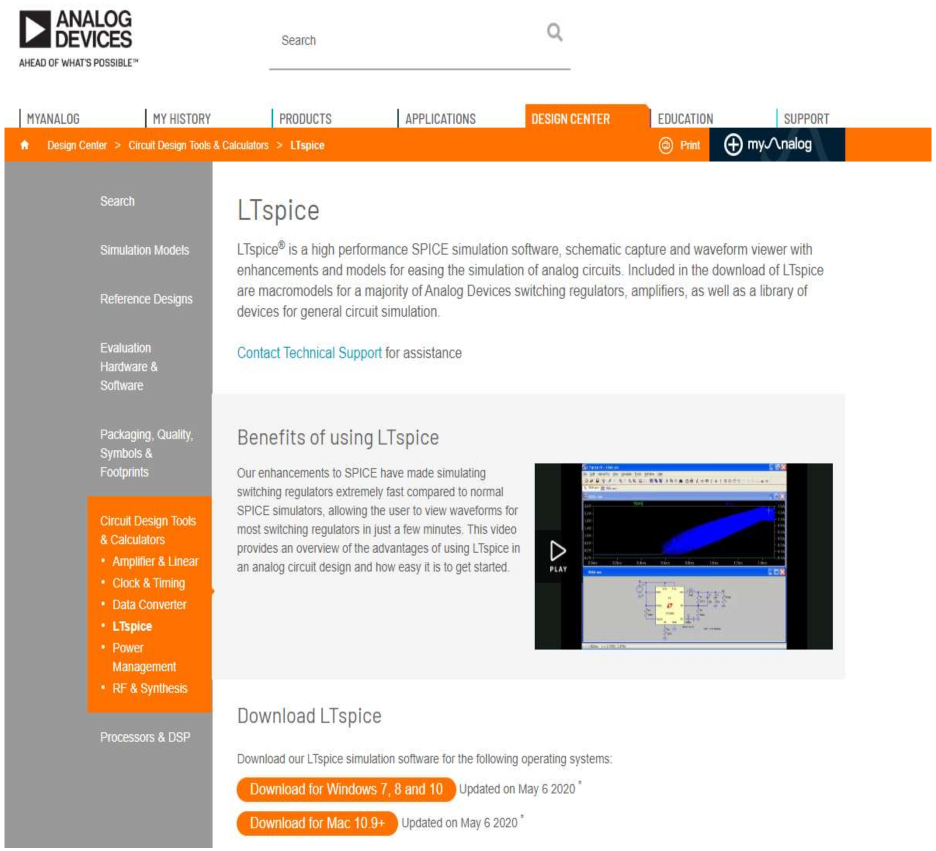
RF Webinar, May 10th 2020

Robert Plat, PA0PLT

rmplat@yahoo.com

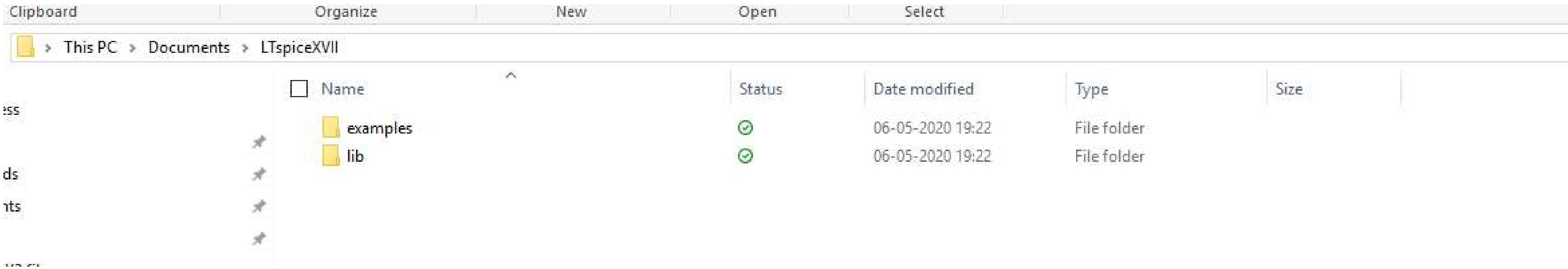
Download and installation

- Visit: https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html
- Download and install a Windows or Mac version.
- And here you go…



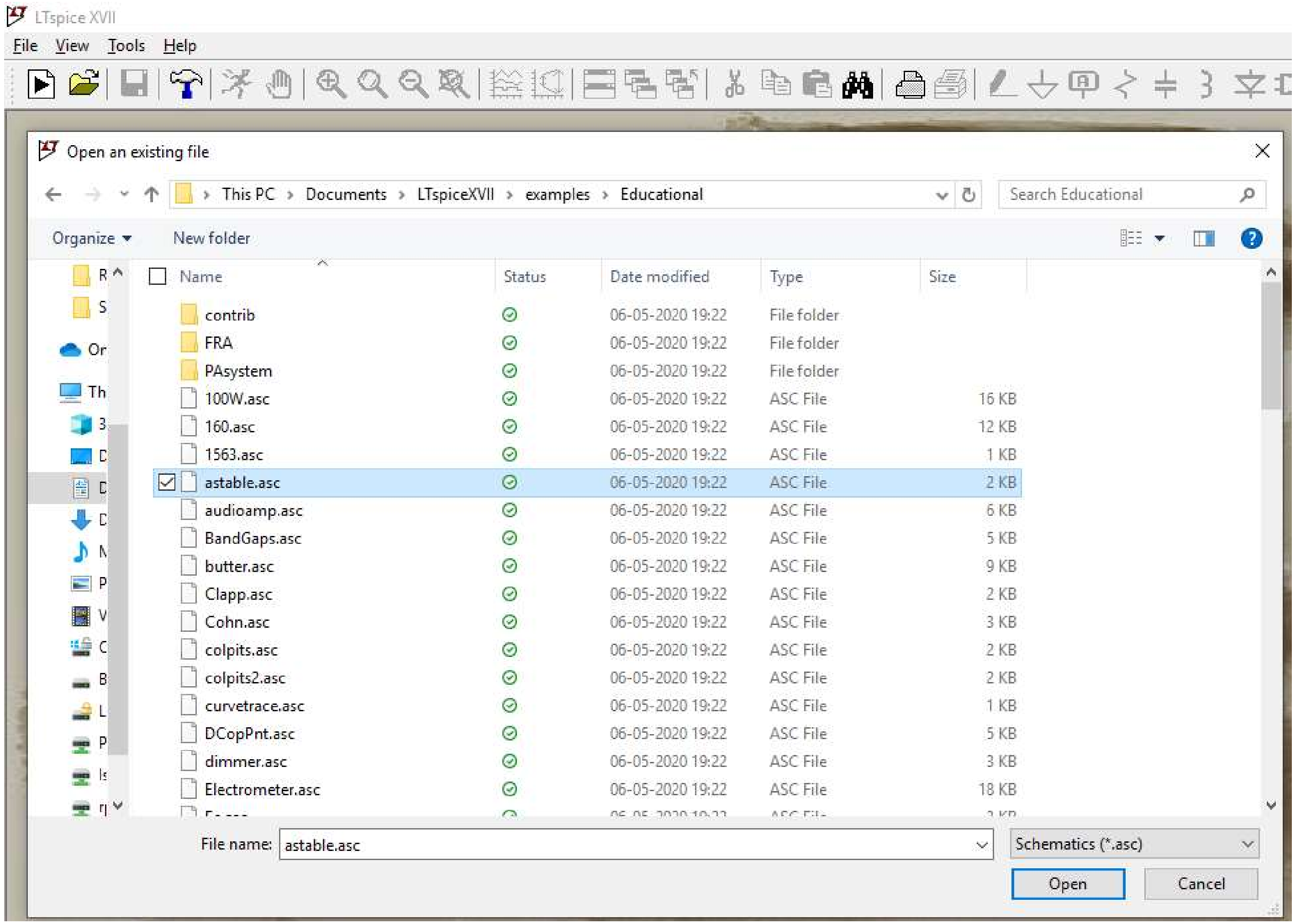
Educational examples provided with the software

- Very helpful to get acquainted with the software.
- Automatically installed in your "My Documents" directory.
- Also: a library with electronic symbols, including tubes and RF-stuff like Ferrite Beads, Transmission lines, Coils, etc.

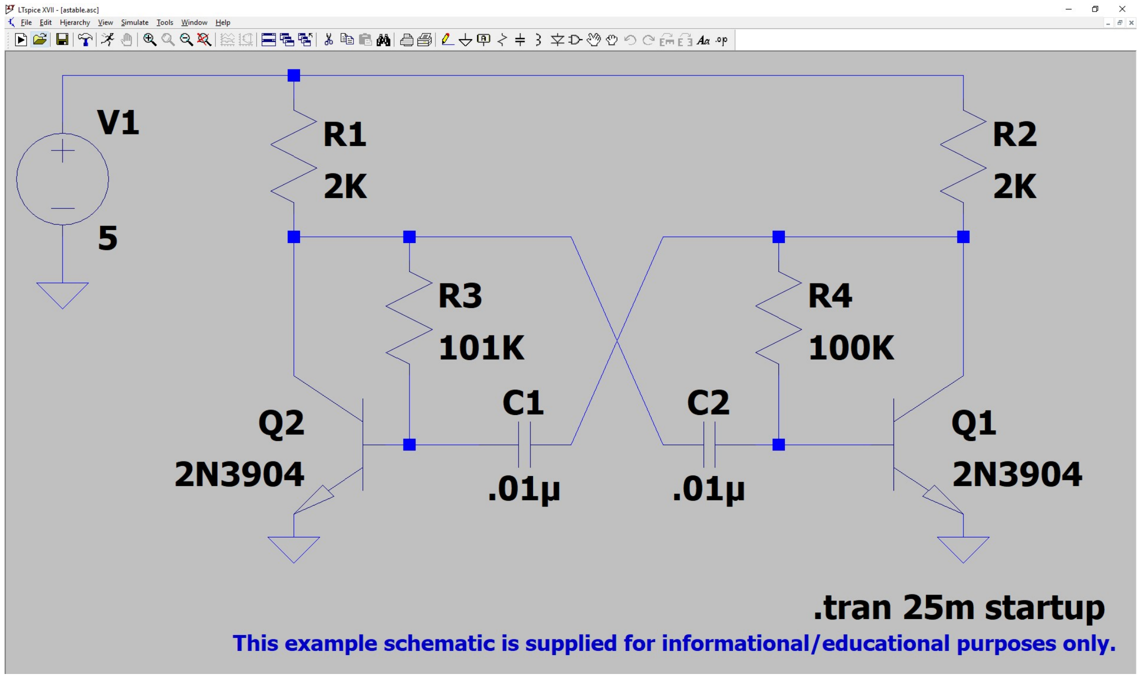


Getting started

- Start LTspice
- Open "astable"



Then you should see this:



"**.tran 25m startup**" is an important simulation command: Move your mouse over it and right click. A menu appears where you can set the way your creation will be simulated. This one is set for a simulation in "time domain" (like an oscilloscope) and runs for 25 milliseconds.
Five other modes are available which will be explained later.

- What you see is the schematic diagram.
- E.g. V1 is the power supply set at 5 V DC.
- Move your mouse to either component and "right click":
  - A menu appears and you can change the values.
  - At V1: click the box "advanced": a menu appears where you can set virtually any type of power or signal output.
  - <u>Don't change things yet!</u>

Now, click on , and you will see this:

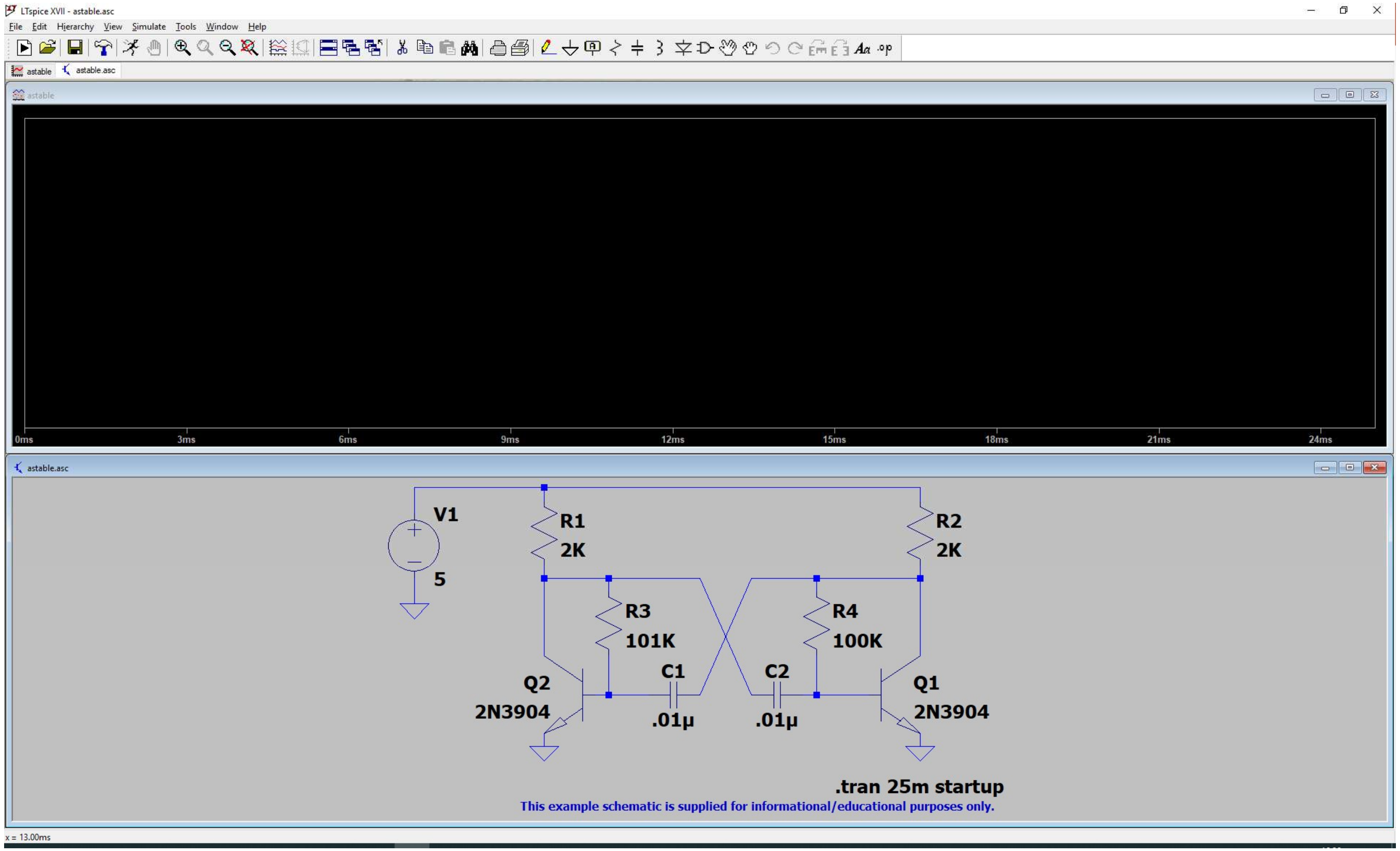
- An empty output window appears.
- Move your mouse to a wire or junction and "left-click". A voltage waveform should appear.
- Move your mouse over a component and "left-click". A current waveform should appear.

Values on axis are automatic. To set it manually: move mouse over it, and "right-click".

General rule of thumb: If you want to change something: move mouse over it and "right-click".

F9 = "UNDO"

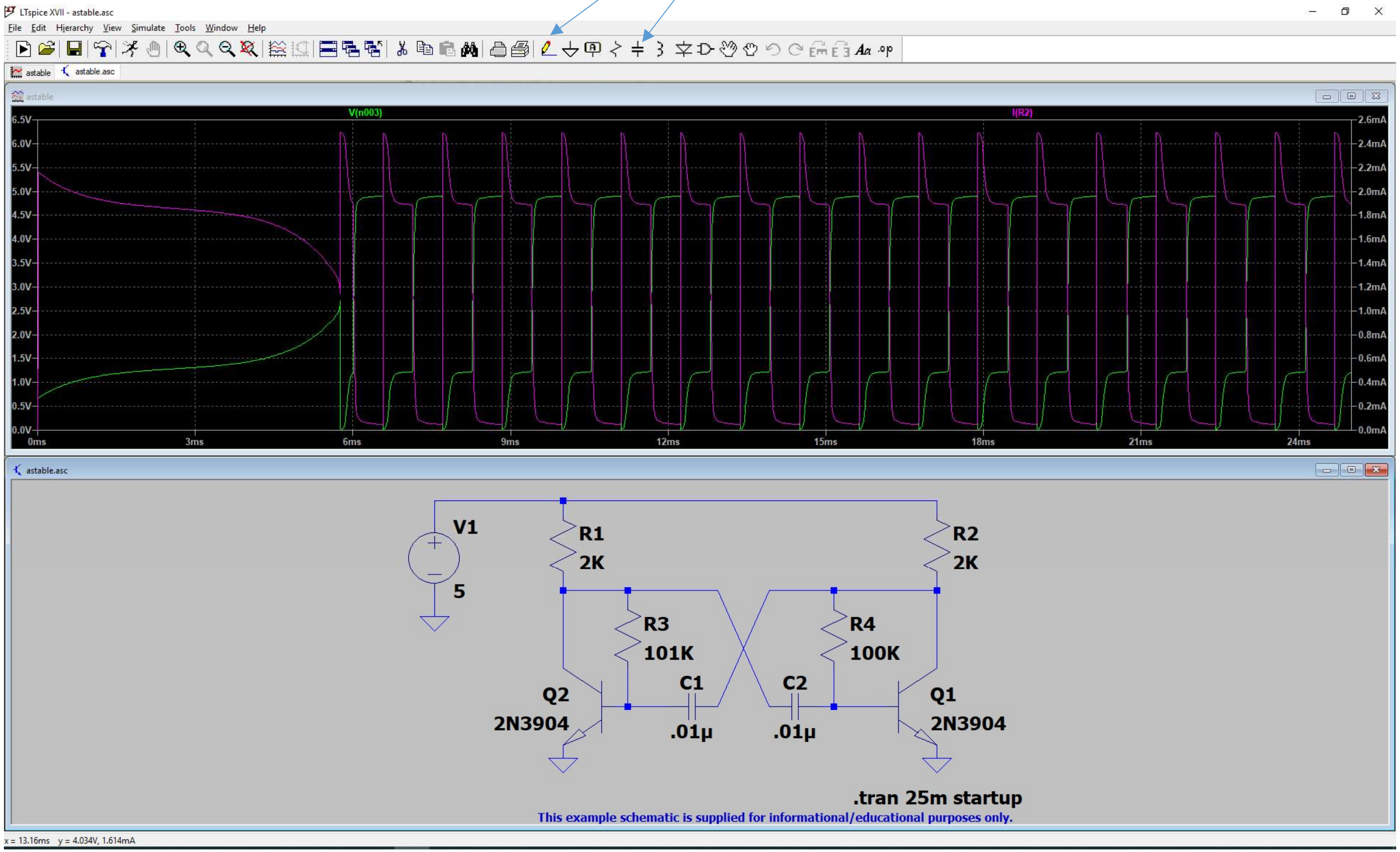
SPACE = "un-zoom"



Example "astable" waveforms

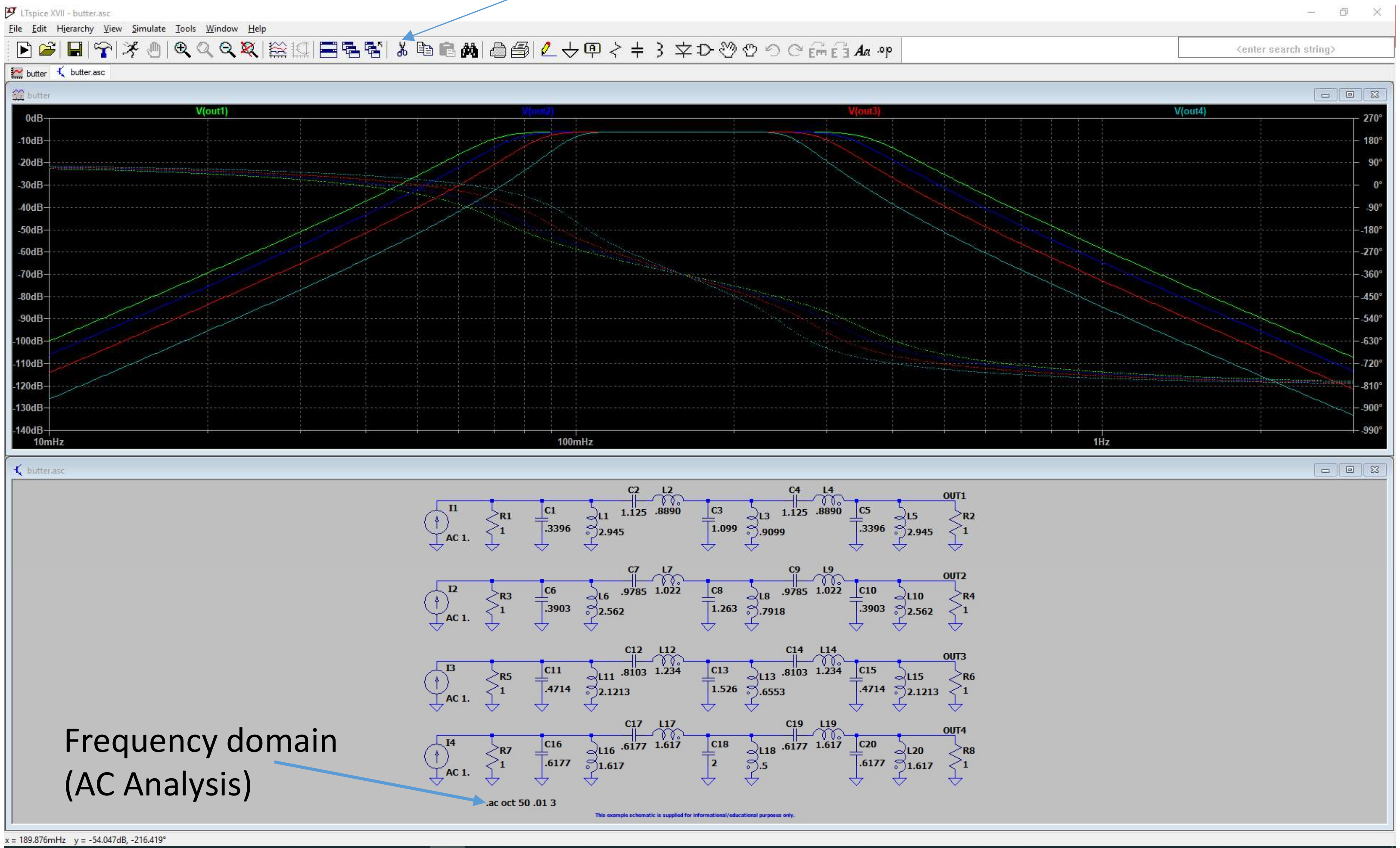
Click here to add a wire
Click here to add a capacitor

- Voltage at junction R2-R4-C1-Q1
- Current through R2
- Try this:
    - Change R4 to 200k.
    - Add C3=10uF parallel to R2.



Example “butter” waveforms (1)

Click here to cut a wire / remove a component



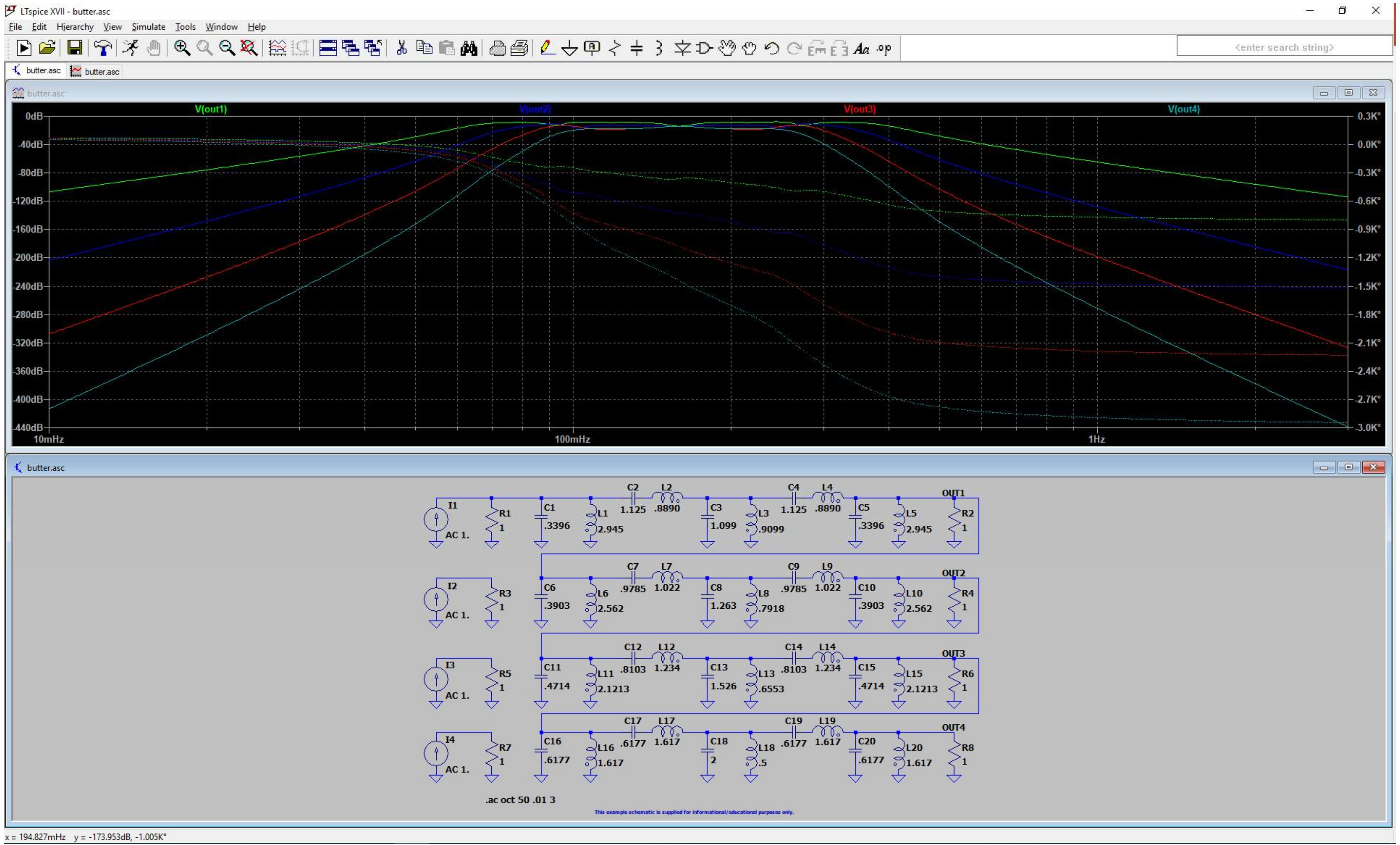
- Play around with this simulation

- Try this:
  - Disconnect I2-I3-I4
  - Connect all filters in series
  - Observe simulation

  (next page)

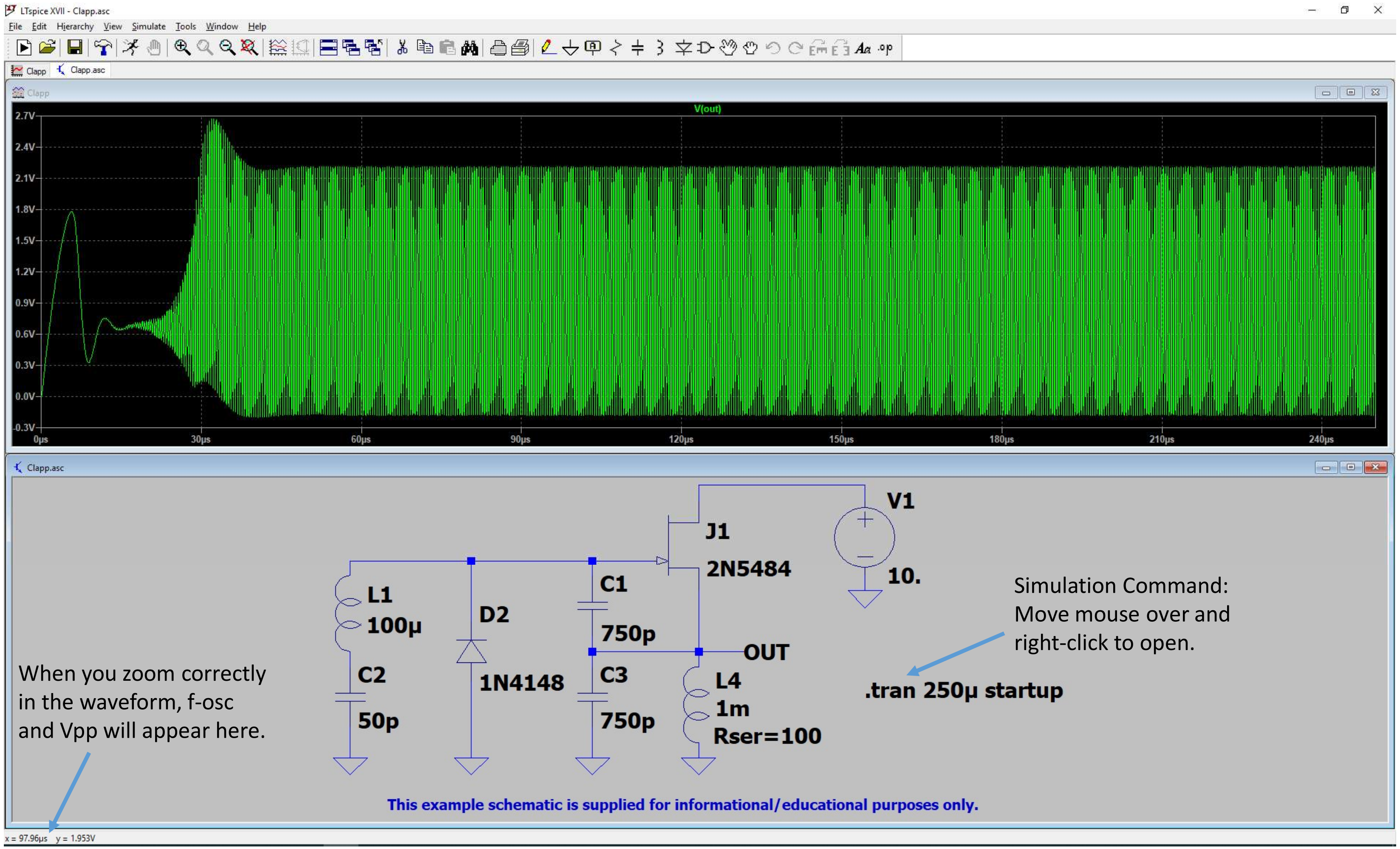
Example “butter” waveforms (2)

- Result should look like this.
- Try to adjust y-axis to 0 to -140dB.
- For the die hard’s:
  - Transform this filter to e.g. 3650kHz +/- 150kHz.



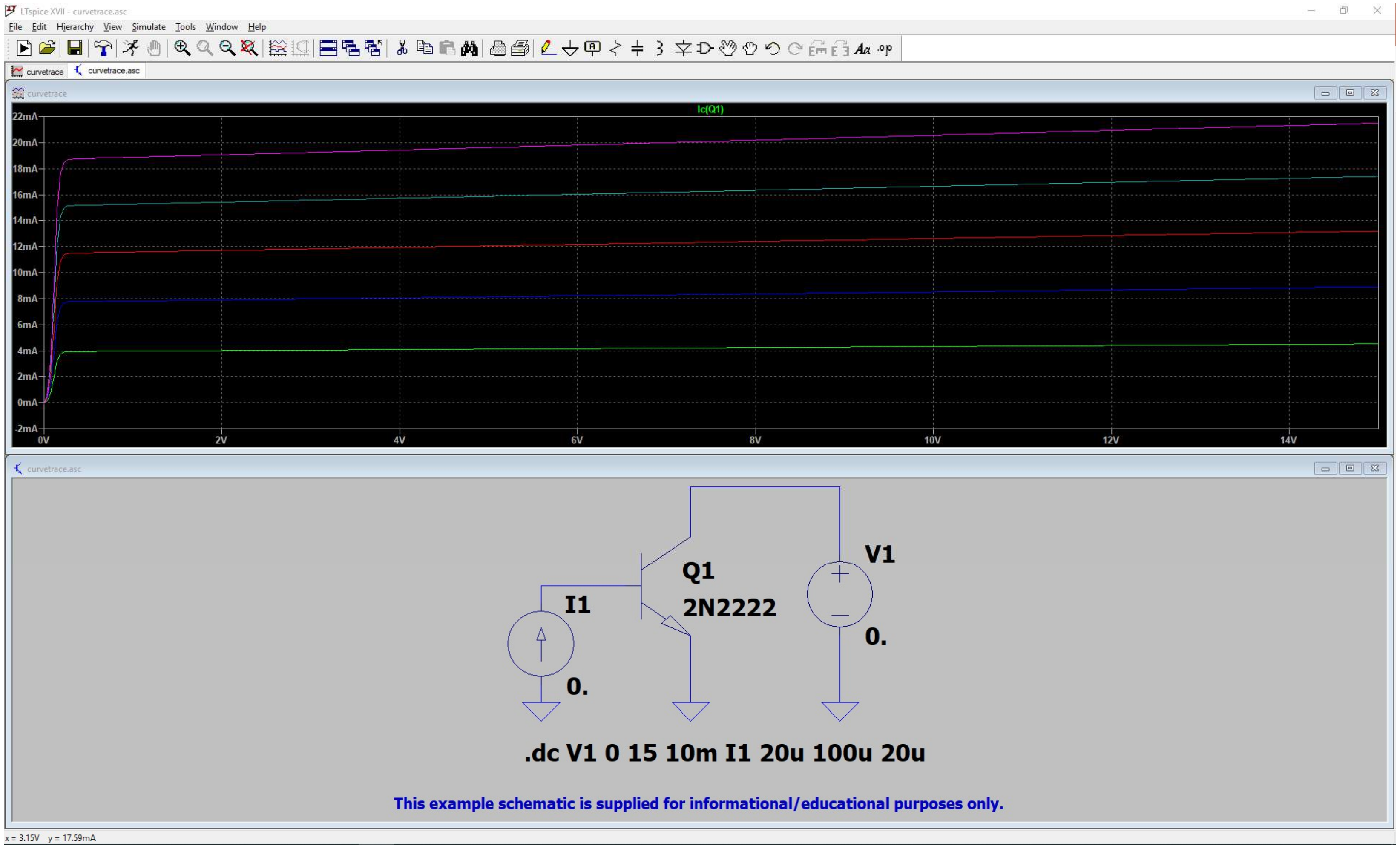
Example “Clapp” waveforms

- Try:
  - To zoom into waveform and determine f-osc and Vpp.
  - Switch off “startup” in “simulation command” and run simulation again.
  - Switch on “Skip initial operating point solution” in “simulation command”.
  - *Observe and figure out the differences. These are important settings to let oscillators start.*



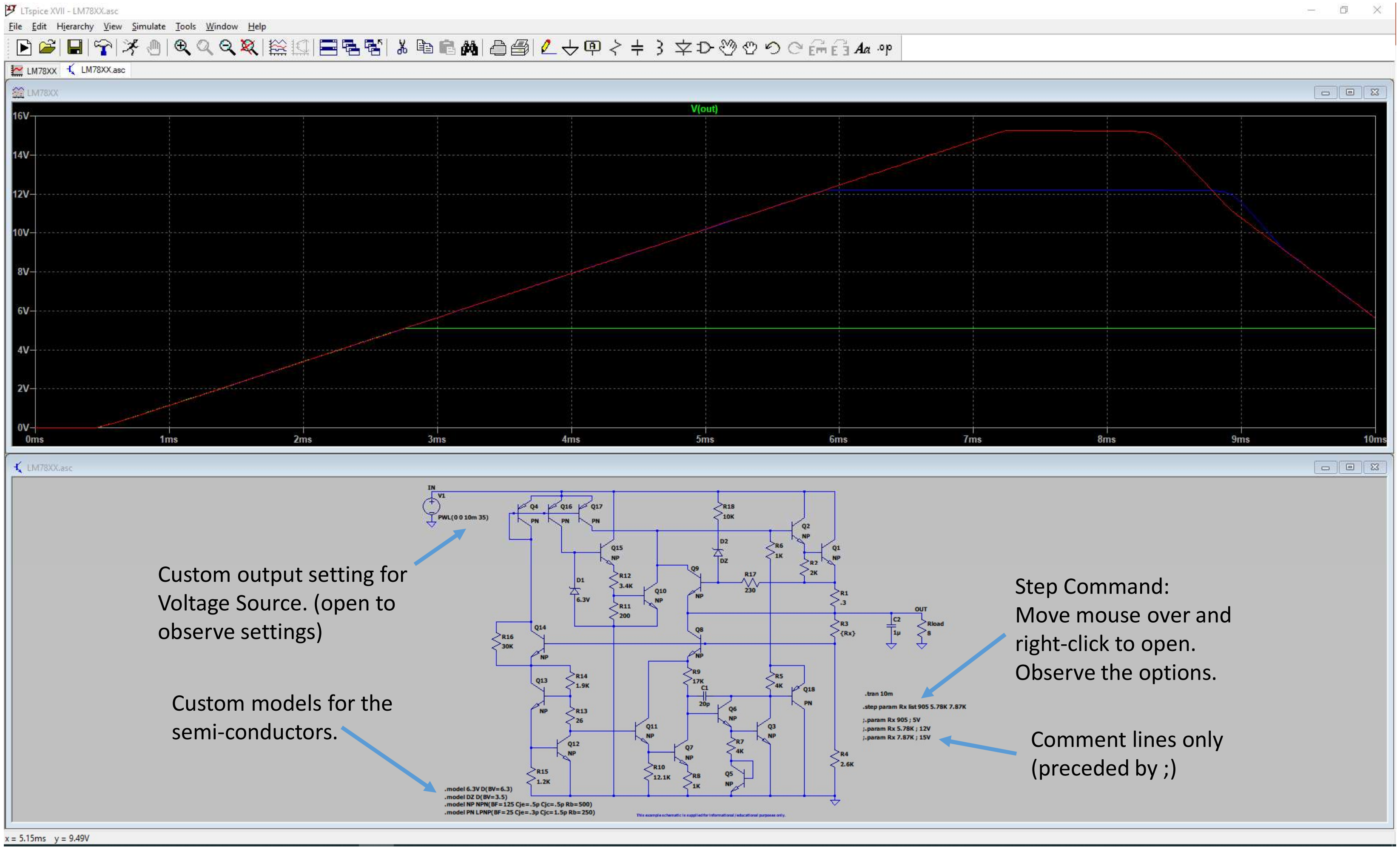
Example “curvetrace” waveforms

- Try:
  - Open “simulation command” and change settings.
  - *Observe and figure out the differences.*
  - *This option is very handy to make models yourself of non-linear components which are not provided by LTspice.*



Example “LM78XX” waveforms

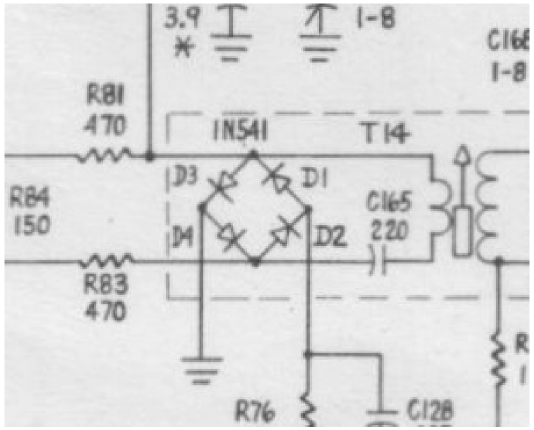
- Try:
  - Change settings and observe results.
  - *As in the real world: many ways to get the same result.*



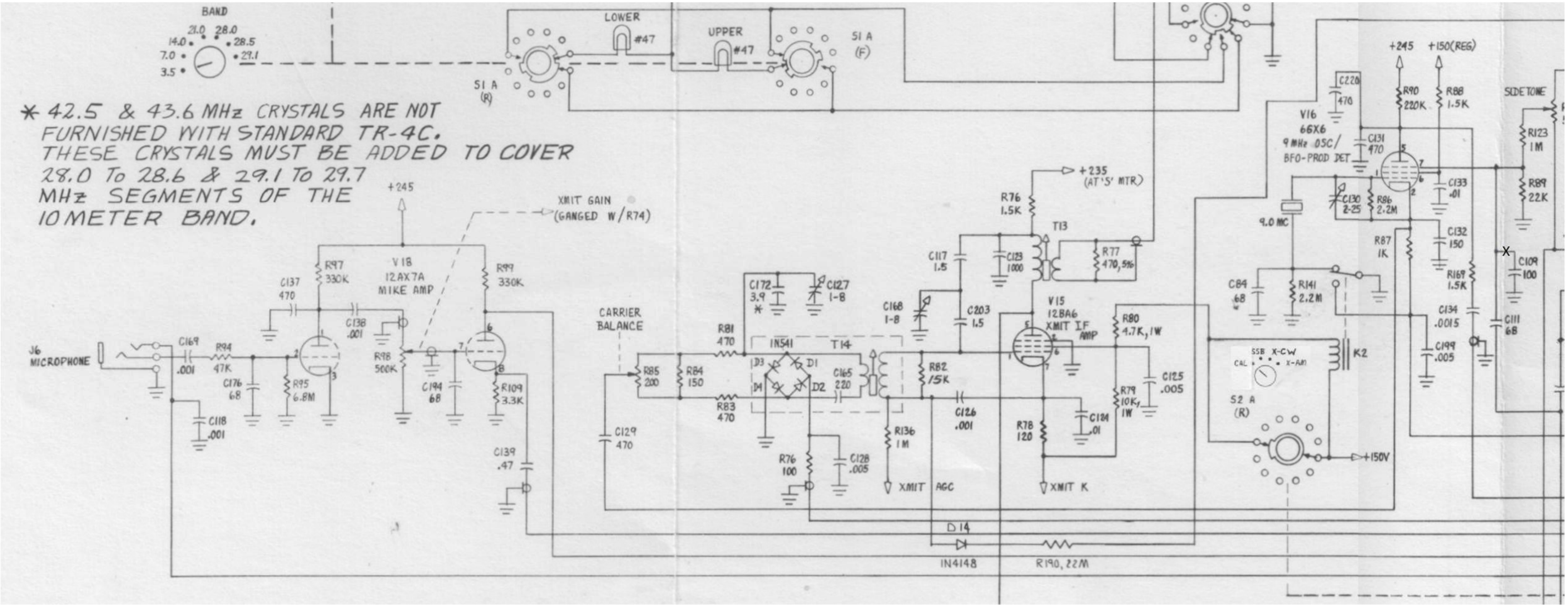
The DRAKE TR4-C, HF HAM-transceiver

Modulator section:

- Schematic
- Simulation
- Guessing Drake’s design considerations
- Worth to try modifications?



TR4-C modulator section, schematic diagram

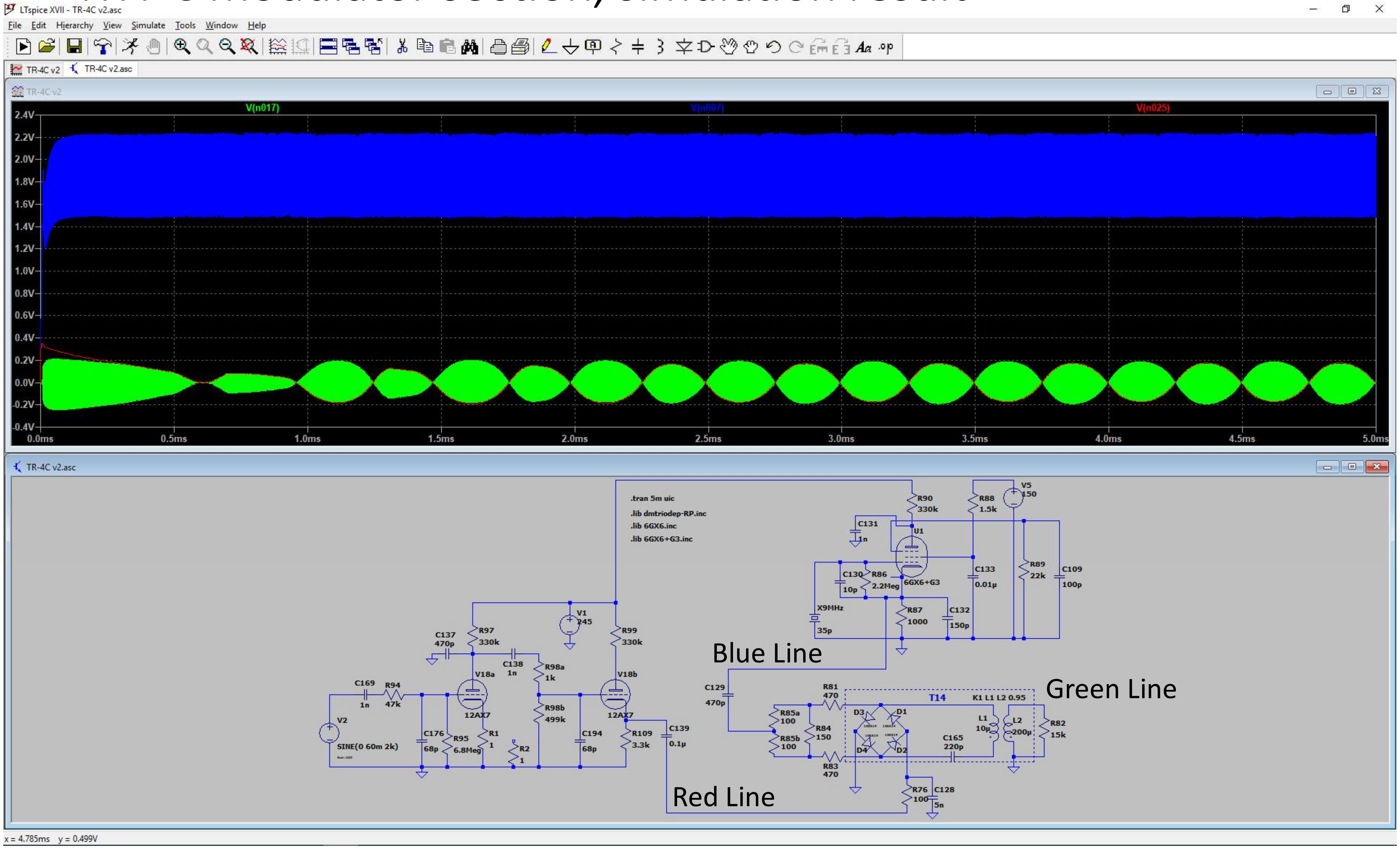


AF-stages **Ring modulator** **9 MHz Oscillator**

TR4-C modulator section, setting up simulation

1. Put the provided .asc and .inc files in a separate folder: E.g. \My documents\Simulations
2. Start Ltspice and open TR-4C v1.asc
3. Run simulation

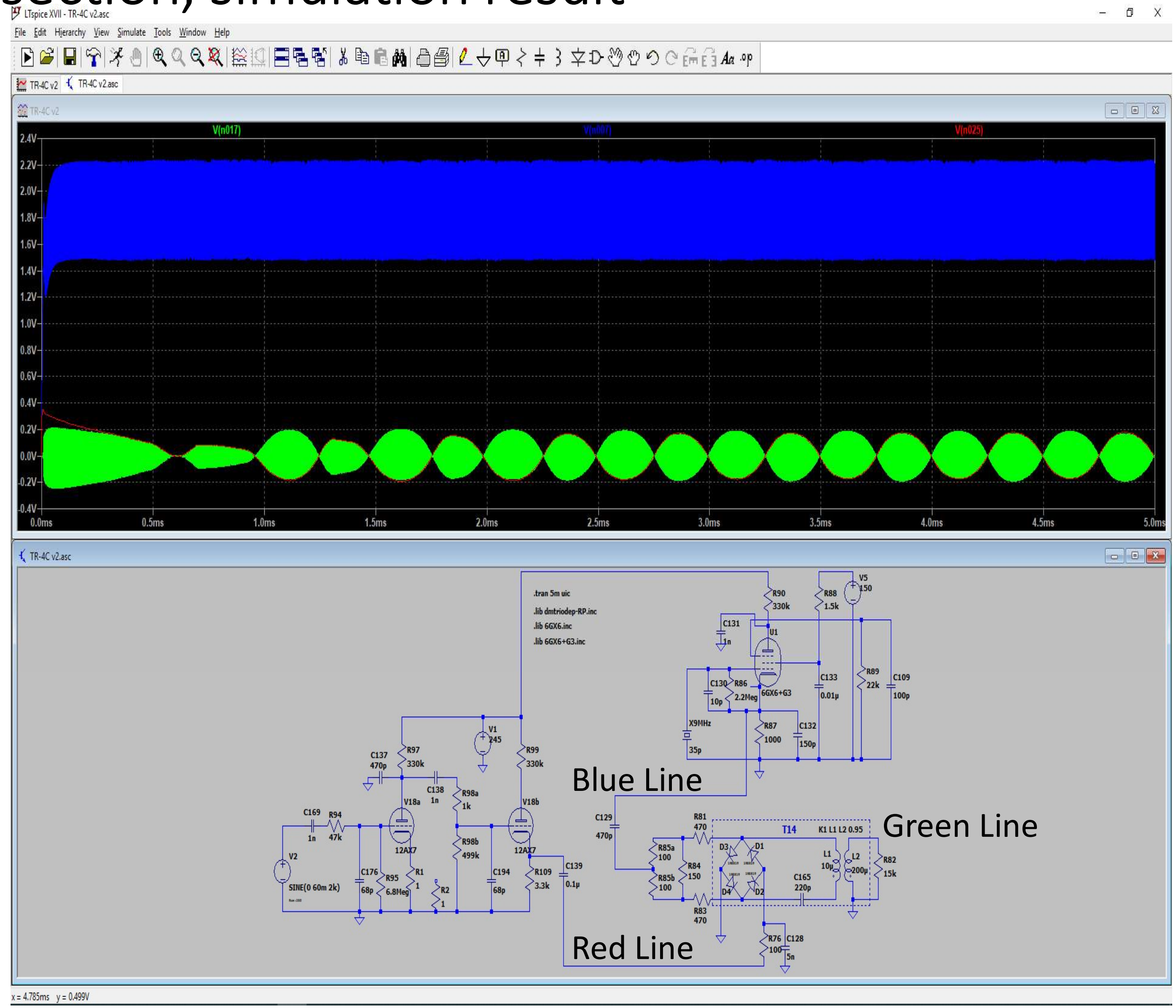
TR4-C modulator section, simulation result



TR4-C modulator section, simulation result

Try these:

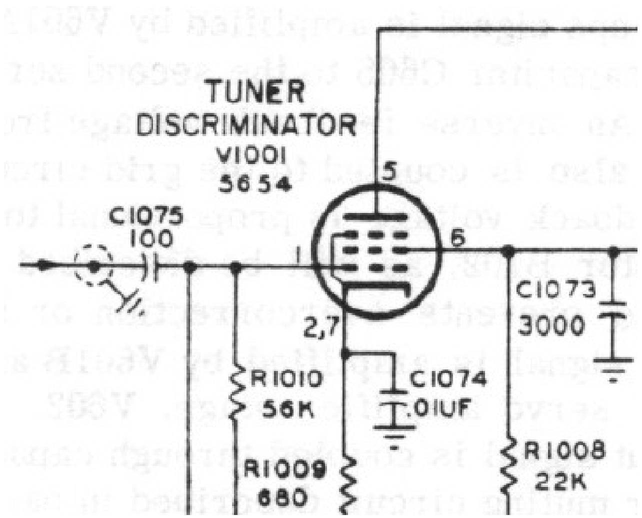
- Adjust C130, and discover why this is a trimmer in reality.
- Over-modulate in steps of 10mV until 150mV. Observe that the distortion is caused by the AF-stages.
- Change the following components:
    - U2: 12AU7 (ECC82)
    - R99: 33k
    - R109: 3.3k
    - R1: 2.2k
- Results:
    - More smooth over-modulation.
    - Less distortion.
- Disclaimer:
    - Not yet put into on-air practice.



The Collins 618S-1, HF Aircraft-transceiver

Tuner-discriminator section:

- Schematic
- Simulation
- Possible issues?
- Worth to try modifications?



618S-1 tuner-discriminator, schematic diagram

- The schematic:
  - The reference signal enters left and is amplified by V1001. The Anode circuit is tuned by a servo motor, bringing the tuned circuit at the same frequency as the incoming reference signal.
  - *L1001B is loosely coupled to L1001A. The task is to find the right inductances and coupling factor which match the original design. At first start to take values which give highest output within a bandwidth of 5 to 10 kHz.*

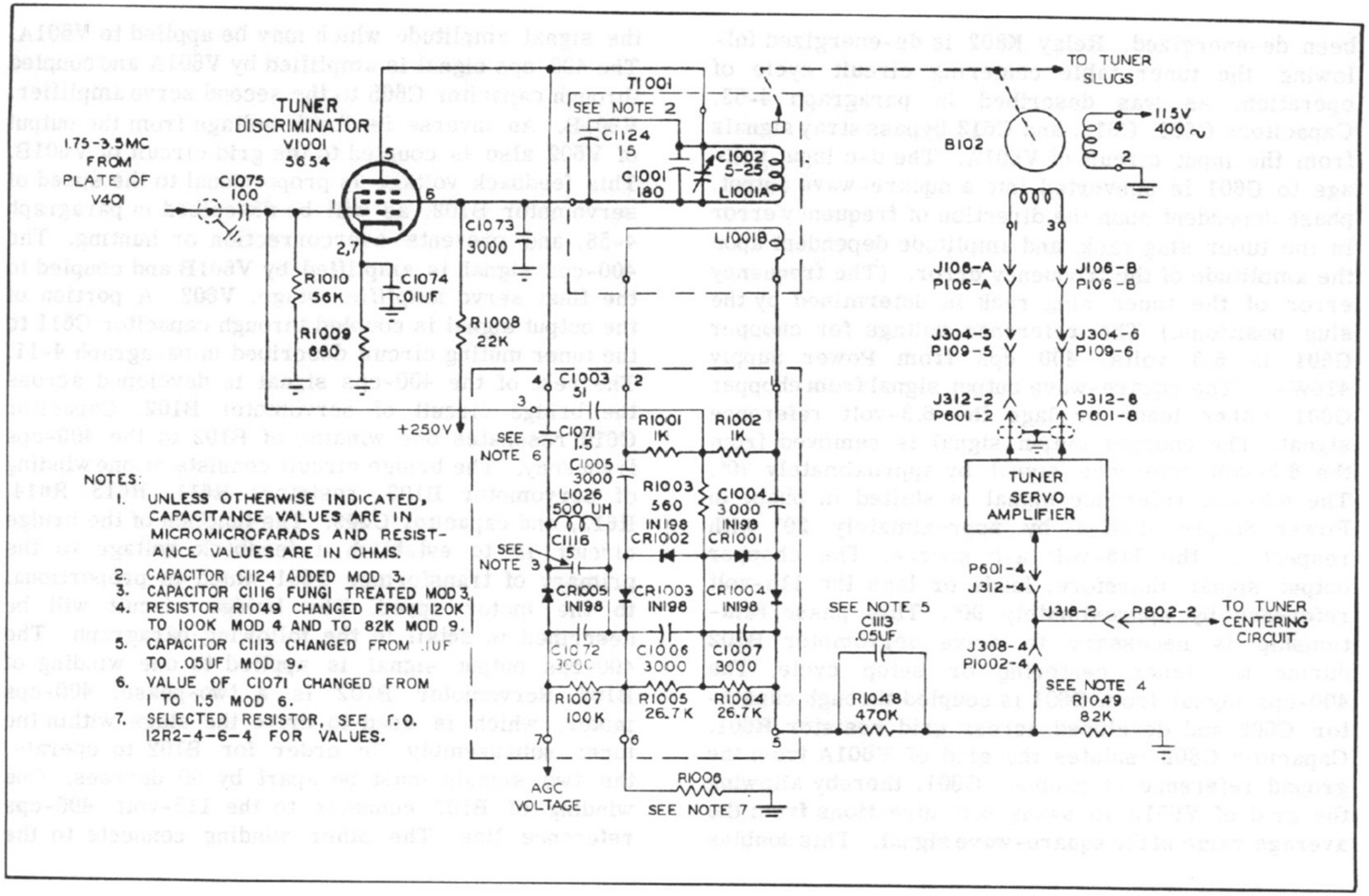
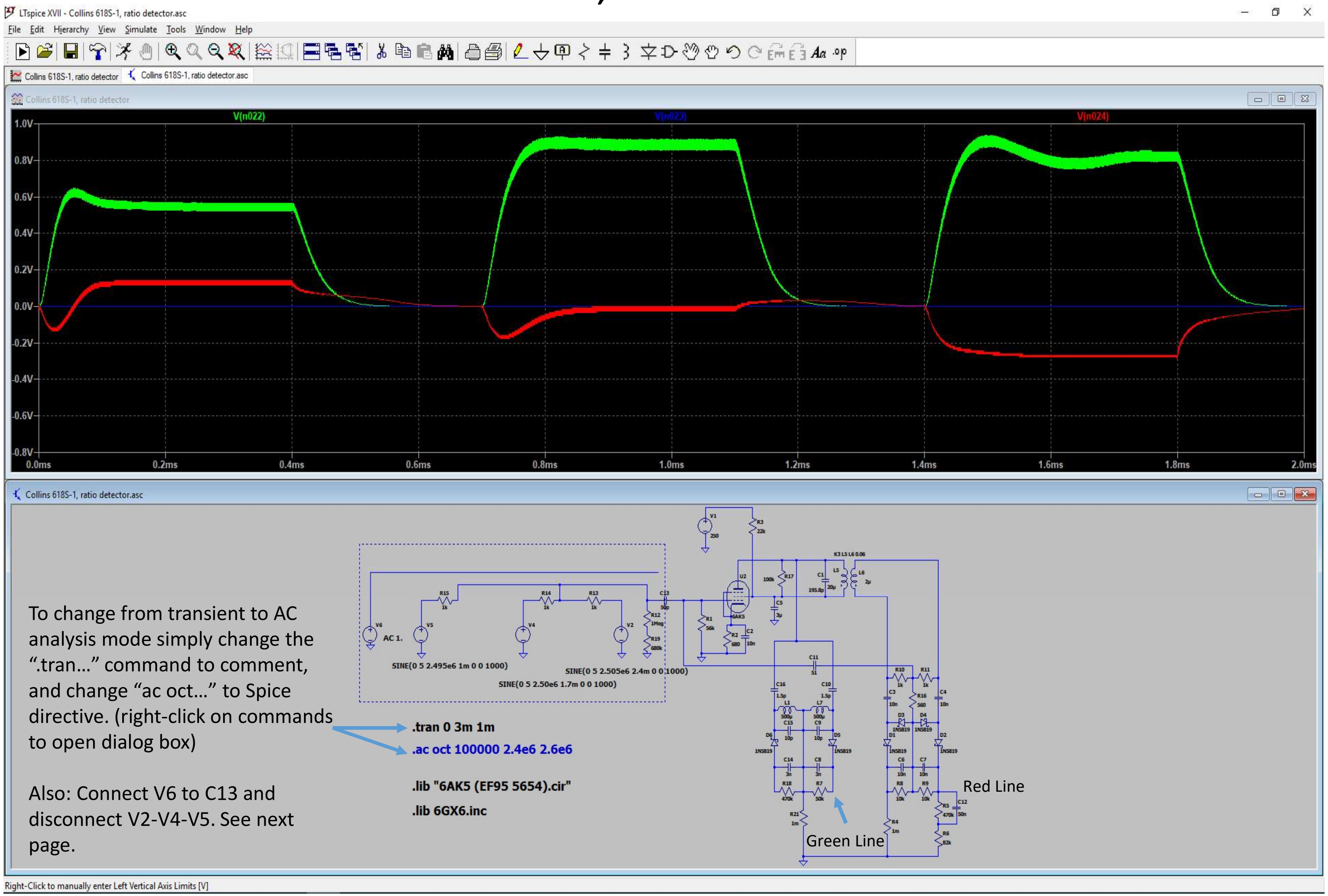


Figure 4-25. Tuner Discriminator and Servomotor B102, Simplified Schematic Diagram

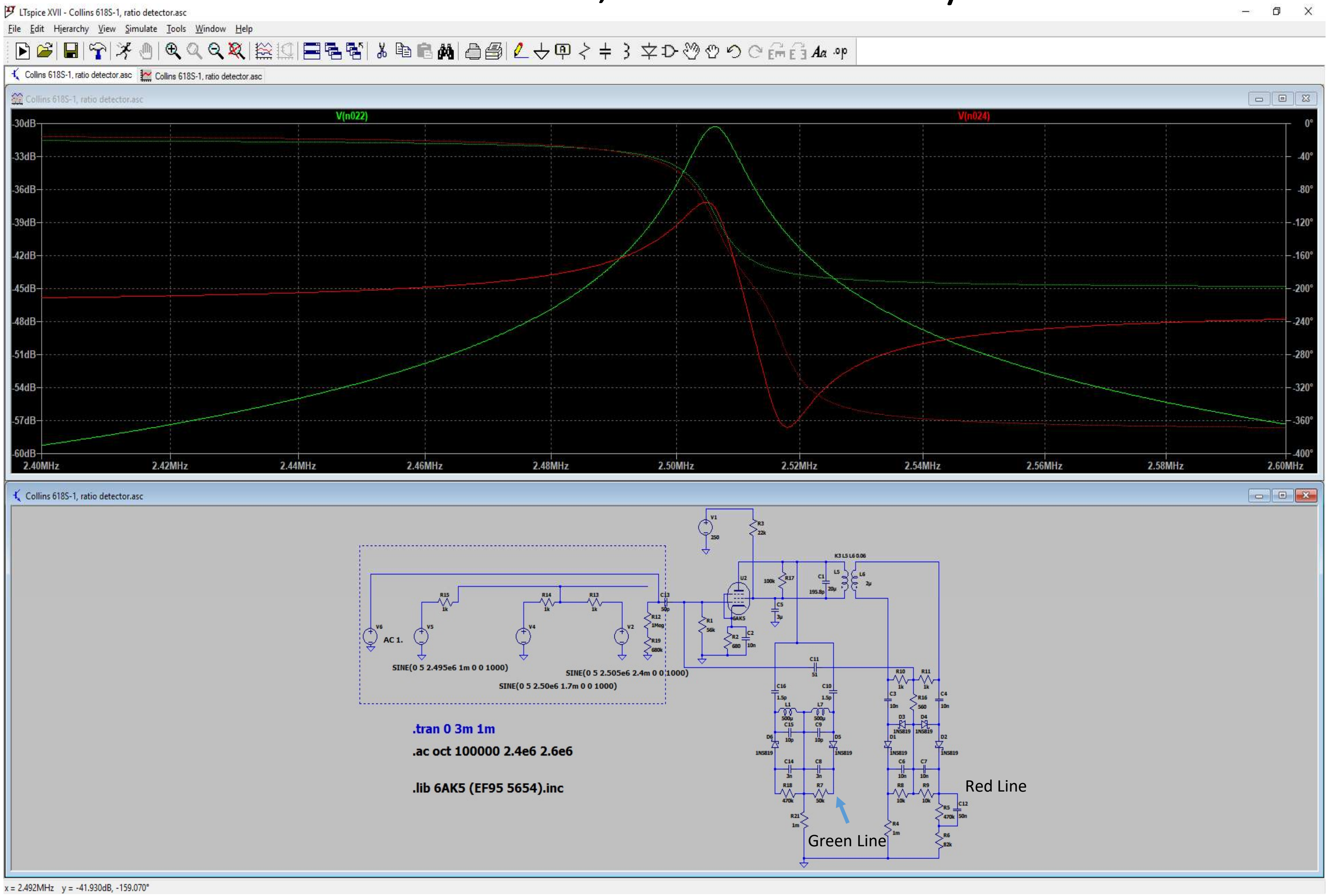
618S-1 tuner-discriminator section, simulation result

- Observations:
  - Output is somewhat delayed compared to the input. This could lead to instability of the servo motor.
- *Try this:*
  - *Change values of the RF-transformer and observe the results. You should adjust C1 also to tune to center frequency, in very small steps of 0.5 pF.*
  - *Short circuit C13. Guess what is going happen and then run the simulation. (Beware that node numbers may be changed, so you have to click again on the desired nodes)*



618S-1 tuner-discriminator section, result AC Analysis

- **Observations:**
  - Output and phase are given compared to the generator output.
  - The bandwidth can be determined with graph.
- *Try this:*
  - *Change values of the R17 and observe bandwidth and center frequency.*



Some words about non-ideal components

Most components in LTspice behave like ideal components, or incorporate only a limited amount of parasitic parameters. Obviously, in reality the components must be suited for e.g. the desired frequency range and maximum voltage/current/power. Especially RF-designs are sensitive for parasitic capacitances and inductances. Also, skin effects and magnetic effects increase the resistance significantly, e.g. in coils and parallel lines. A proper simulator is therefore necessary to identify their effects in each component, before you start building. Many others developed already models for Spice based simulators which incorporate non ideal characteristics, and are somewhere available on internet. It’s a matter how to find them. E.g. not using a EL84 tube in an UHF-amplifier is just common sense. Most spice models of tubes don’t incorporate internal inductances. A good idea is to make models yourself. Although it can be quite time consuming and the need for more fundamental knowledge, it keep the accuracy in your own hand.

In conclusion, a basic simulation is just a good starting point. Also, when making a physical sketch or printed circuit board design, try to estimate parasitic parameters between components and enclosure as good as possible, and incorporate them into your simulation. Thus, making a simulation and designing the physical layout is a parallel process. For HF and lower, you can rely on your experience and some Googling to get things on air, but for VHF and higher this might not be sufficient. E.g. unwanted coupling between coils need to be taken into account.

Remember that long before the digital era engineers were already capable to build terrific working RF equipment, so life today with things like LTspice must be more easy. Whatever, always give a suspicious look the beautiful graphical output of any simulator. Success!