

SPICE PRIMER: GETTING STARTED WITH FREE ICAP/4 DEMO

APPENDIX J

Free SPICE is easy, and fun. It's available for several platforms (Macs, Linux, PCs) Here's how to get started in Windows.

J.1 Setting up ICAP SPICE

1. Download "ICAP/4Windows Demo" zip file (ICAP4Demo.zip) from intusoft.com.
2. Extract zip file to some temporary folder.
3. In that folder, double click (abbreviated hereafter as "cc") on setup.exe.
4. Launch ICAP/4:
(Programs → ICAP_4 Demo → Start ICAPS).

J.2 Entering a Diagram

- Add standard components with
 - R resistor
 - C capacitor
 - L inductor
 - o ground (the letter "oh," not the number)
 - v signal or voltage source
 - Y voltage test point.(These are case-insensitive: r = R, etc.)
- For a library component, Parts → Parts Browser,
 - browse by type, and subtype, or use Find,
 - then Place.
- Moving and orienting:
 - highlight component by click-hold, then can drag;
 - when highlighted, type + to rotate (90°CW), or – to flip 180°.
- Editing a component label:
 - highlight label by click-hold, drag to relocate;
 - when label highlighted, cc on label to edit values.
 - * Note units multipliers: f, p, n, u, m, k, meg (not M).
 - * Note that SPICE only needs the units *multiplier*, not the units themselves. For example, for a capacitor "1u" is OK for SPICE; but you can type "1.0uF" for clarity on the schematic, which SPICE will treat the same.

- * You can cc on the component, instead, to bring up the value dialog.
- You can add parameters by entering the value and clicking Apply; this is necessary, e.g., for a voltage source (add dc or ac, give amplitude). Or you can click >>Add>>, which also adds the parameter to the schematic drawing.
- Wires:
 - type w, then drag a wire between points;
 - you can drag multiple separate wires; ESC when done.
- Settings:
 - set "Rubberbanding" in Options, to maintain connections when dragging components.

J.3 Running a simulation

Can a passive circuit consisting only of resistors and capacitors have *voltage gain*? The surprising answer is yes. Let's use this simple example to illustrate an ICAP/4 SPICE entry and simulation.

J.3.1 Schematic entry

We launch ICAPS, and place two resistors (type the letter "r"), two capacitors (type the letter "c"), and a voltage source (type the letter "v") with the bottom terminal grounded (type the letter "o" right after typing "v"). If the pieces are too small to see on your screen, you can re-size them by using Options→Zoom (or the function keys F6, F7, and F8). We drag them around to connect them as shown in Figure J.1, and add an output-voltage test point (type the letter "y" and change the label to Vout). We assign values by double clicking on each component, then entering the value in the value field (where "???" appears initially); note that pressing the Enter key after each value moves you along to another field in the dialog – you have to click OK to complete the operation.

For the capacitors we could have used microfarads (e.g., 0.01uF instead of 10nF); note that the "F" can be omitted, but including it makes the schematic marking clearer. For the voltage source we enter the value 1V (i.e., 1 volt) in the

“AC” field, and then we click on the ADD button so that the value shows on the schematic (SPICE uses the value, even if we do not ADD it; but it’s nice to show it on the circuit diagram). As before, the unit symbol “V” can be omitted.¹

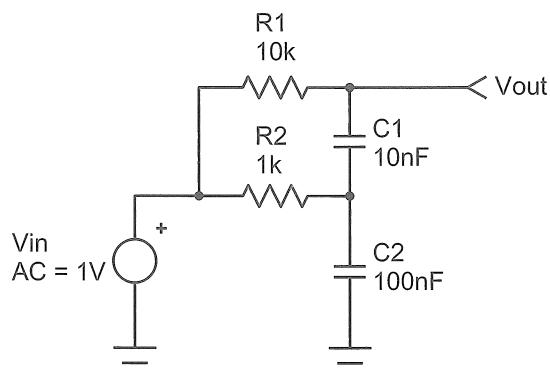


Figure J.1. An RC circuit with voltage gain! Schematic entry in Intusoft ICAP/4 is swift, taking no more than a couple of minutes. Printing the captured schematic to “Adobe PDF” produced this figure.

J.3.2 Simulation: frequency sweep

Now for the fun! We begin by saving the project to a folder of our choosing, using the File→Save As menu. Then we set up the simulation by Actions→ICAPS/Simulation Control (or clicking on the button with a pencil over a wavy line). Choose and click “AC Analysis,” then enter 20 points per octave, start at 100 Hz, end at 10 kHz (type 10k in the box), and click OK. Then click DONE. Now run the simulation by clicking on the little running person (or Actions→Simulate). This launches the Spice engine, with a small window (“IsSpice4”) showing status and errors, and (if successful) a little graph of the result. Running the simulation also launches the “IntuScope” display program (which you can launch manually by clicking on the ‘scope icon next to the running person, or by Actions→Scope). At this point there are multiple windows piled up, which you should resize and drag to convenient locations on the screen and then do Options→Save Preferences to stick them down. Now click the “Test Pts Only” box in the “Add Waveform” window that belongs to IntuScope, then highlight V_{out} ² and click “Add.” Voilà – this produces a nice

plot of V_{out} versus frequency (see Figure J.2), with default axis labels, grids, and scaling. You can change the axis labels by double clicking on the label; and you can fiddle with the scaling in the “Scaling” window.

To add a plot of phase shift versus frequency, highlight “ph_ V_{out} ” in the Add Waveform window, click on the box labeled “With Like Traces” (unless you want a separate graph), and click “Add.” We’ve done that to make Figure J.2, where we’ve also used the cursor tools on the IntuScope window to mark the point of maximum voltage gain.

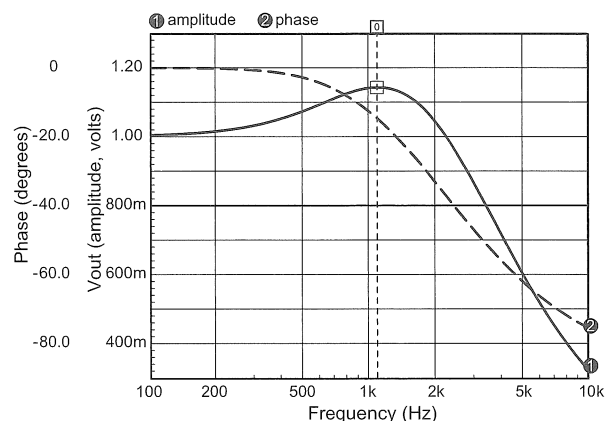


Figure J.2. “Scope” output of the RC gain circuit, showing amplitude and phase (“AC Analysis”) for the circuit of Figure J.1. The cursor indicates the peak of the voltage gain, $V_{out}/V_{in}=1.142$ at $f=1.096$ kHz. The original IntuScope output plotted phase in blue, which we’ve converted to a dashed black line; we enlarged the text, also, for readability.

J.3.3 Simulation: input and output waveforms

Do we really believe this gain-versus-frequency plot? Everyone knows that a simple RC network just mashes things out – shifted phase and reduced amplitude. We won’t believe this thing has voltage gain unless SPICE shows us the actual input and output signal waveforms for a sinewave near the frequency of maximum voltage gain (and maybe not even then!).

Here’s how you do it: first, add another voltage test point at the input (“y” again), and change the label to V_{in} . Then double click on the input voltage source and click on the field labeled “Tran Generators” (SPICE calls any old waveform, including a sinewave, a “transient”). Select SIN, set the peak amplitude to 1, the offset to 0, and the frequency to 1.096 kHz. Click OK, and OK. Then go to Simulation Setup (pencil over wavy line), deselect AC Analysis, and select Transient. In the Transient Analysis

¹ For SPICE’s ac analysis, the frequency-sweep gain computation is performed with infinitesimal small-signal amplitudes, which are then normalized to the signal amplitude you specified. Here, for example, SPICE uses a signal amplitude much smaller than 1 V.

² Or, if you have not renamed the output to V_{out} , select from among the signals with names like v2, etc., in the Y Axis output list.

window that pops up, set Data Step Time to 1 μ s, Total Analysis Time to 105ms, and Time to Start Recording Data at 100ms (this gives the circuit time to settle into its steady state). Leave the Maximum Time Step field blank (i.e., accept default value). Click the runner icon to run the simulation. At this point the IsSpice4 window should show a wiggly trace. Now go to the IntuScope window, clear the old graph by pressing the DEL key for each selected trace (you can save it, if you want, or start a new plot with File→New Graph), and Add “vin” and “vout” in the Add Waveform subwindow of IntuScope, with the “With Like Traces” box checked as before.

Not bad... but the waveforms look a bit jagged. If you zoom in (drag the box boundaries in the “Scaling” window) you’ll see that there are only about 20 points plotted per cycle. We asked for 1 μ s steps, but we’re getting output at $\sim 50\mu$ s intervals! That’s because we didn’t constrain the Maximum Time Step in the Transient Analysis window. Go back now (Simulation Setup→Transient) and set Maximum Time Step to 1 μ s. Then click on the running person (the simulation will take slightly longer, maybe a full second this time); then clear the old graph and Add the two waveforms in IntuScope. You should see silky-smooth plots, as in Figure J.3. If you still have doubts about an RC circuit with voltage gain, you can hook up the circuit and check it out on a real-live oscilloscope, as we did (Figure J.4). Evidently this RC voltage-gain stuff is real!

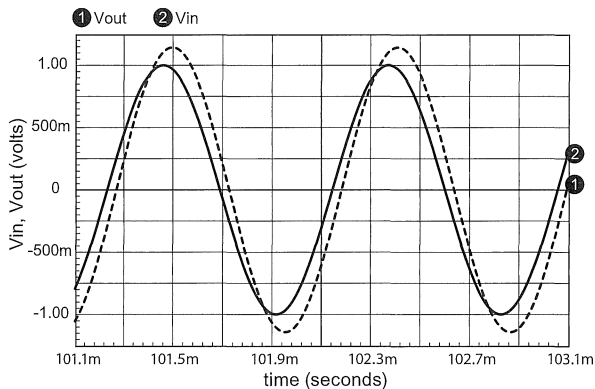


Figure J.3. “Scope” output of the RC gain circuit, showing sinewave response (“Transient Analysis”) at the frequency of maximum gain for the circuit of Figure J.1. The original IntuScope output plotted V_{out} in blue, which we’ve converted to a dashed black line; we enlarged the text, also, for readability.

J.4 Some final points

The ICAP/4 demo program includes ten “tutorial movies” (in the Help menu), which are mercifully short (about

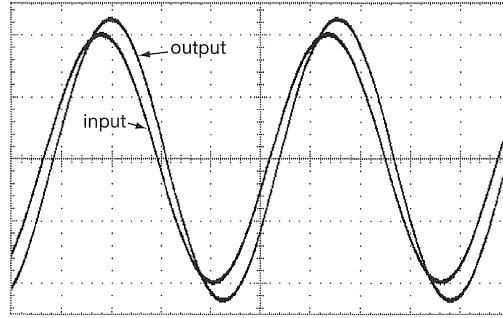


Figure J.4. A real ‘scope (Tek TDS3044B) validates that crazy SPICE stuff! Horizontal; 200 μ s/div; Vertical; 0.5 V/div.

a minute each) and quite helpful for the beginner. Take ten minutes to view them! You’ll learn, for example, that you can create many signal sources by going directly to Parts→Parts Browser→!Generators. You’ll also learn some tricks to simplify schematic entry (copying, zooming), and ways to make the parts labels more readable (“tall,” “wide,” and “split” options). And by experimenting a bit you can discover that both the schematic and the simulation graphs can be copied (“Print”) to the clipboard and then pasted into Word or Wordpad, where they exist as scalable graphics. If you Print to Adobe PDF you’ll get a scalable graphic that can be cleaned up in Adobe Illustrator.

We have been using the Intusoft “ICAP/4” full version of SPICE now for many years; it provides a convenient Windows-based schematic entry and simulation environment, which we use to explore circuit configurations. Typically we begin with a starting circuit design, making explanatory notations in a Word document as we go. We copy and paste the schematics and simulation graphics into the running text, making a “lab notebook” of sorts; see next.

J.5 A detailed example: exploring amplifier distortion

In Chapter 2x (“BJT Amplifier Distortion: a SPICE exploration”) we do some serious work with SPICE, in the form of an exploration of distortion in discrete BJT amplifier designs.³ The amplifier stuff is lots of fun; and there are some nice SPICE tricks there revealed. Take a look!

³ When we do this in real life, we like to paste circuit diagrams and screen shots of simulated performance into a Word document, annotating as we go. We’ve kept that spirit alive in the discussion in Chapter 2x, albeit in the form of a \LaTeX conversion of the original Word document.

J.6 Expanding the parts database

You can freely add SPICE models to the ICAP/4 parts library. An easy way to start the process is

to download the folder AoE-PR from our website (www.artofelectronics.com), and place it under the default PR folder. Then do File→Update Part Database