

Improved vacuum tube models for SPICE simulations

Part 1: Models and example

by Norman L. Koren

Updated March 20, 2003

[On to Part 2: Listings and PSpice operation](#) | [Norman Koren Vacuum tube audio page](#)
[Finding SPICE tube model parameters](#)

[View image galleries](#)
[View or sign the guestbook](#)



Google Search

Search WWW Search www.normankoren.com

Table of contents

[Equations for vacuum tubes](#) | [New equations](#) | [SPICE](#) | [Parameters](#)
[Finding parameters by trial-and-error](#) | [Plate curves](#) | [Output transformers](#)
[Example: original PAS preamplifier](#) | [Lessons](#)
[Part 2](#) | [Appendix A: Listings](#) | [Appendix B: Operation with PSpice Schematics](#)

Much of the material in here originally appeared in [Glass Audio](#), Vol. 8, No. 5, 1996, p. 18.

As my principal hobby interest is now [photography](#) and I work crazy long hours at [lmatest](#), I won't have much time to update this article or answer difficult questions.

James E. Lanier's [PSpice Triode Calculator](#) is a promising program for finding tube model parameters. It's still under development; he plans to expand it to include tetrodes and pentodes.

[LTspice](#) can apparently be used for these models.

SPICE, an electronic circuit simulation program originally developed at the University of California at Berkeley, has found wide acceptance in the electronics and semiconductor industries. It can perform highly accurate time and frequency domain analysis of complex analog and digital circuits (including harmonic and IM distortion). Several commercial versions of SPICE run on personal computers, most notably [PSpice](#) from [OrCAD](#). The downloadable [PSpice student version \(version 9.1 at last look\)](#) has limited capabilities by standards of solid-state circuit design, but it can be amazingly useful in simulating vacuum tube amplifiers, which tend to have fewer components. It was used for all examples in this article. It's hard to find on the [OrCAD](#) website. I had to go to the [CadencePCB](#) home page and click on [All Downloads](#). Other well-known SPICE implementations include [Electronics Workbench](#), 800-263-5552, and ICAP/4, from [Intusoft](#), P.O. Box 710 San Pedro, CA 90733-0710; 310-833-0710. Duncan Munro's [list of SPICE vendors](#) may contain some hidden treasures. [WinSpice](#) is a free Spice3F4 Port for Windows 95, 98, 2000 & NT. *Very* intriguing. It will simulate much larger circuits than evaluation PSpice.

SPICE contains built-in models for passive devices (resistors, capacitors, inductors, etc.) and for most active semiconductor devices (bipolar transistors, FETs, Op Amps, etc.), but none for vacuum tubes. SPICE models and can be downloaded from manufacturer's Websites.

Equations for vacuum tubes

Scott Reynolds¹ and Marshall Leach² model vacuum tubes as voltage-controlled current sources whose output current is a weighted sum of controlling element voltages raised to the three-halves power. The controlling elements are different for triodes and pentodes. (Since the suppressor grid has little effect on the transfer characteristics, pentodes are modeled as tetrodes.) The plate current equation for triodes is

$$I_P = (E_G + E_P / \square)^{3/2} / k_{G1} \quad \text{for } E_G + E_P / \square \geq 0 \\ = 0 \quad \text{otherwise.}$$

(1)

The plate current equation for pentodes is

$$I_P = (2/k_{G1p})(E_G + E_{G2}/\mu)^{3/2} \arctan(E_P/k_{VB}) \quad \text{for } E_G + E_{G2}/\mu \geq 0$$

$$= 0 \quad \text{otherwise.} \quad (2)$$

where I_P is plate current, E_G is control grid voltage, E_P is plate voltage, E_{G2} is screen grid voltage (all voltages with respect to the cathode), μ is the amplification factor, and k_{G1} is a factor used to fit the equation to data. The pentode equation differs from the triode equation in that the screen grid replaces the plate as a controlling element, and an arctangent term (from Scott Reynolds' model) is added to model a response curve "knee" whose location is proportional to k_{VB} . Equations (1) and (2) are forms of the Langmuir-Childs law, which can be derived from fundamental physics.³ Plate curves for a 12AX7 triode with $\mu = 93$ (below the specified value but a good overall fit to equation (1)) are illustrated below in Figure 1.

The screen grid current equation for pentodes is

$$I_{G2} = (E_G + E_{G2}/\mu)^{3/2} / k_{G2} \quad \text{for } E_G + E_{G2}/\mu \geq 0$$

$$= 0 \quad \text{otherwise.} \quad (3)$$

where I_{G2} is the screen grid current and k_{G2} is a constant similar in function to k_{G1} .

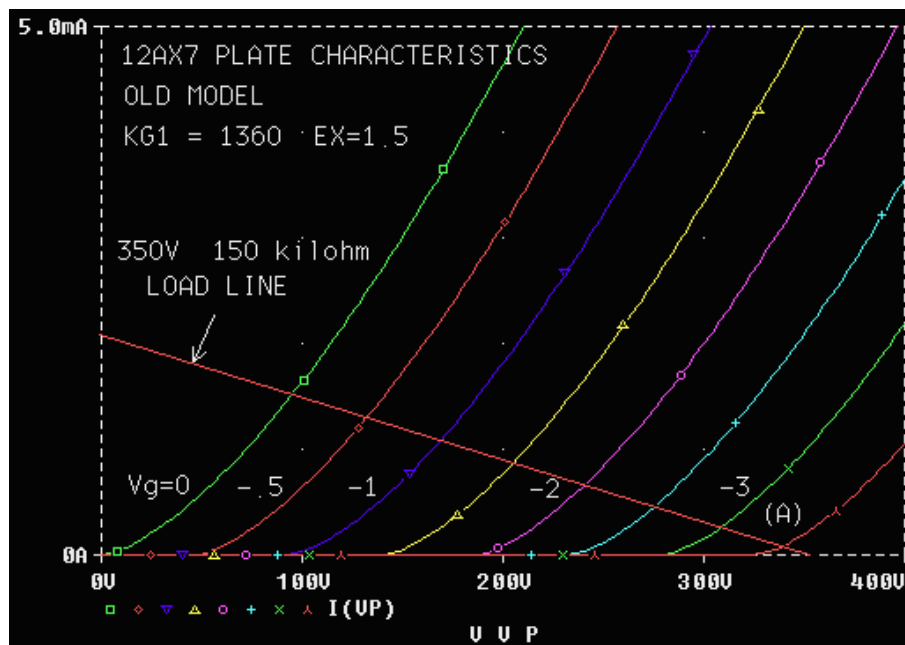


Figure 1. 12AX7 plate curve from the old model.

The difficulty with equations (1) and (2) is that they assume that the grid has perfect control over the plate current, i.e., that there is no leakage current. The real world is, alas, not so kind, and these equations give a poor estimate of plate current for large positive plate voltage and large negative grid voltage,⁵ region (A) in Figure 1, above. Compare this with region (A) in Figure 2, below, which is derived from the new model but matches published curves⁴ extremely well.

The modeling error would not be serious if tubes did not operate in the region of greatest error. Unfortunately they do, as illustrated in Figure 1 by a typical load line for a 12AX7 with a 350V plate supply and a 150kΩ plate resistor. The load line crosses region (A), and operation may extend into this region for large signals. This problem is exacerbated in class AB push-pull amplifiers, where the operation of each tube traverses the region of worst error as it moves from conduction to cutoff and back. An accurate estimate of distortion in push-pull amplifiers is impossible without an improved model.

New equations

The new equations for tube characteristics are *phenomenological* equations, that is, equations that model the behavior of physical phenomena using a reasonable number of parameters, but are not derived from fundamental physics. They have been designed so that plate current $I_P > 0$ whenever plate voltage $E_P > 0$ (for triodes) or screen grid voltage $E_{G2} > 0$ (for pentodes). Both equations take advantage of the fact that $\log(1+\exp(x))$ approaches x when $x \gg 1$ and 0 when $x \ll 1$.

The new plate current equation for triodes is

$$\begin{aligned} E_1 &= (E_P/k_P) \log(1 + \exp(k_P(1/\square + E_G/\sqrt{k_{VB} + E_P^2}))) \\ I_P &= (E_1^X/k_{G1})(1 + \text{sgn}(E_1)) \end{aligned} \quad (4)$$

The signum function, $\text{sgn}(x) = 1$ for $x \geq 0$; $\text{sgn}(x) = -1$ for $x < 0$, is used to prevent current flow when $E_1 < 0$. The new plate current equation for pentodes is similar to that of triodes:

$$\begin{aligned} E_1 &= (E_{G2}/k_P) \log(1 + \exp(k_P(1/\square + E_G/E_{G2}))) \\ I_P &= (E_1^X/k_{G1})(1 + \text{sgn}(E_1)) \arctan(E_P/k_{VB}) \end{aligned} \quad (5)$$

Equation (4) reduces to equation (1) when $E_P^2 \gg k_{VB}$ and $k_P(1/\square + E_G/E_P) \gg 1$. Similarly, equation (5) reduces to equation (2) when $k_P(1/\square + E_G/E_{G2}) \gg 1$. We continue to use equation (3) for pentode screen grid current for three reasons: 1. Screen current is not as critical to the tube performance as plate current. 2. Good data on screen current is scarce. 3. The model should be kept as simple as possible for use with evaluation version of PSpice.

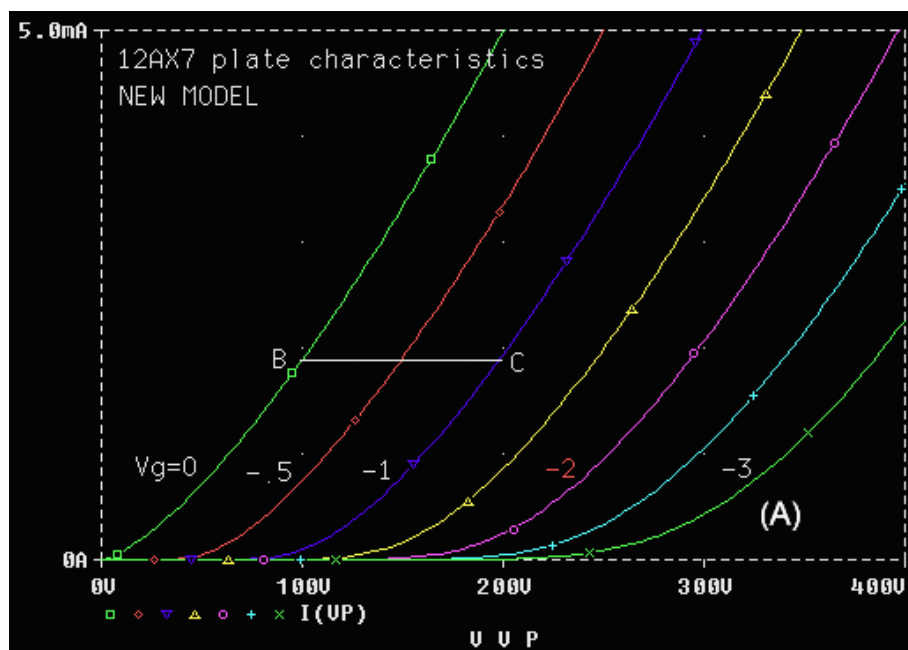


Figure 2. 12AX7 plate curve from the new model.

SPICE language and PSpice

The SPICE language is an industry standard for describing electronic circuits (called *netlists*) and instructions for modeling them. PSpice is a commercial implementation of SPICE that features convenient schematics entry through a graphic interface. PSpice actually consists of three separate programs: Schematics for data entry, PSpice for running the SPICE simulation, and Probe for displaying the results.

If you plan to modify or add tube models you'll need to know a bit about the SPICE language. [Appendix A](#) has listings of several SPICE files. The first line of a SPICE file contains a title. Data on this line is ignored. The last line consists of the .END statement. Netlist data lines have the general format,

```
PART node_1 node_2 {additional nodes if applicable} value(s) ; optional comments
```

Comments may appear on lines that start with an asterisk (*) or on data lines following a semicolon (;). Continuation lines begin with a plus sign (+), except for equations, where the portion to the right of the equal sign (enclosed in braces {...}) is restricted to one line. This limitation can present problems with long equations. To get around it, E_1 in Equations (4) and (5) has been modeled as a voltage-controlled voltage source that controls I_p , which is a voltage-controlled current source. This makes two relatively short equations out of one long one.

I strongly recommend the excellent textbook by Paul W. Tuinenga,¹⁰ "SPICE, A Guide to Circuit Simulation & Analysis using PSpice." It contains detailed descriptions SPICE commands and models, including how to perform frequency-domain analysis (.AC), time-domain analysis (.TRAN) and Fourier analysis (.FOUR), which produces a detailed harmonic distortion analysis. In the PSpice Schematics program these instructions are generated by clicking on **Analysis, Setup...**, then working with the appropriate boxes.

To perform a SPICE simulation with the MicroSim evaluation package, run the Schematics program, then click on **File, Open...** to open a Schematics data file using the usual Windows navigation techniques. (You can also create a new schematic.) We'll call it *filename.SCH* in this example. You may edit and save it if you wish. To perform the PSpice simulation, click on **Analysis, Simulate**. This creates two SPICE language files: *filename.CIR* containing simulation instructions and *filename.NET* containing the netlist. These files are then read into the PSpice simulator, which writes text results (voltages, currents, etc.) to *filename.OUT* and graphics data to *filename.DAT* for display with Probe, which is run automatically if you have the .PROBE statement in the data file. Probe displays time and/or frequency-domain plots of voltages at selected nodes or currents through selected components. The format of the display is quite versatile. If V(n) is the voltage at node n, VDB(n) is the voltage in dB and VP(n) is the phase. Current display is similarly flexible.

To use the Schematics program I had to create symbols for tubes: a difficult task you can avoid by [downloading the models](#) and following the instructions in [Appendix B](#) for loading the symbol and SPICE model libraries into PSpice.

Parameters

Table 1 gives parameters for some familiar tubes. These parameters are used in the tube model library listed in Appendix A3. The first six are equivalent to μ , X , k_{G1} , k_{G2} , k_p , and k_{VB} in equations (3)-(5).

TUBE	MU	EX	KG1	KG2	KP	KVB	CCG	CPG	CCP	RGI
6DJ8	28	1.3	330		320	300	2.3P	2.1P	0.7P	2k
6L6CG	8.7	1.35	1460	4500	48	12	14P	0.85P	12P	1k
12AX7	100	1.4	1060		600	300	2.3P	2.4P	0.9P	2k
12AU7	21.5	1.3	1180		84	300	2.3P	2.2P	1.0P	2k
6550	7.9	1.35	890	4800	60	24	14P	0.85P	12P	1k
KT88	8.8	1.35	730	4200	32	16	14P	0.85P	12P	1k

Table 1. Spice parameters for some popular tube models

Most parameters in Table 1 can be determined from published graphs and tables. Small-signal triode data was obtained from Tom Mitchell's book.⁴ EL34 data came from the [Audiomatica Sofia web page](#). Other sources were the Parts Connection catalog (1-800-769-0747), which has a large collection of reprints, old copies of the Sylvania and RCA tube manuals, and the pages of Glass Audio. The best places to find tube data on the Web are [Duncan's Amp Pages](#) and [Frank's Electron tube Pages](#). Published values for inter-electrode capacitance always give better high frequency response than experiment. I measured the capacitance of a tube socket with two inch leads, and found it to be around 0.7pF for adjacent pins and 0.5pF for others. I added these capacitances to the

published values. Even so, simulated frequency response may be somewhat optimistic. Tubes of different origin (e.g., Chinese, Russian, and old British 12AX7s) can differ markedly in their inter-electrode capacitances. This may explain some of the sonic differences. Of course, real circuits have stray capacitance, and you must specify it if you want to include it in SPICE simulations.

Finding parameters by trial and error

The trial and error procedure for finding tube parameters is **tedious**. My Matlab program for determining them is now available on [Finding SPICE tube model parameters](#). Two other people have relevant work.

- [Teodoro Marinucci](#) has written an optimization program in Excel. His website covering tube theory and modeling is mirrored on *two* URL's, <http://web.tiscali.it/teodorom/> and <http://digilander.iol.it/teodorom/>. Alas, they're in Italian. I don't think his program is online yet, but it's very appealing because almost everybody has Excel. You can encourage him by sending him an [e-mail](#). You can get a crude but readable translation by searching for "Valvole e Matematica" in <http://www.google.com>, then clicking on [Translate this page] next to his article. Amazing.
- Mithat F. Konar's article [Vacuum Tube Parameter Identification Using Computer Methods](#) describes a technique that uses the Matlab Optimization Toolbox.

The parameters in Table 1 can be found by a methodical trial and error procedure. To use this procedure, the programs in Appendix A1 through A3 must be employed. Listings A1 and A2 produce plate characteristic curves for triodes and pentodes, respectively. Listing A3 is a portion of the tube library referenced by the .LIB statement in listings A1 and A2. Tubes are modeled as subcircuits, starting with .SUBCKT and ending with .ENDS. .SUBCKT is followed by the model name and then the element node names. We have chosen the order the element nodes to be plate, control grid, cathode, and (in pentodes) screen grid.

To find the parameter values for a new tube, it is best to start with values for a similar existing tube, except for amplification factor μ (\square), which is generally close to the manufacturer's specification. μ is (minus) the change of plate voltage for a given change of grid voltage at constant plate current. This is illustrated by line segment BC in Figure 2, which is a good choice for determining μ because it is located in a region of tube operation far from region (A), where effective μ is reduced. In this case, plate voltage increases by 100V when grid voltage is decreased by 1V while plate current remains constant at 1.9 mA, resulting in $\mu = 100$.

To determine EX and KG1 (X and k_{G1}), run the appropriate plate curve program (Appendix A1 or A2), and observe plate current $I(VP)$ using Probe. Adjust EX and KG1 so that curves for relatively low negative grid voltages (e.g., $V_g = 0$ or -0.5 for the 12AX7 in Figure 2) match experimental data. Typical values of EX range from 1.3 to 1.4. Line curvature increases with EX. The textbook value of $3/2$ for EX is appropriate for equations (1) and (2), where it gives a reasonable average representation of tube operation, but is not really accurate in all regions. KG1 is inversely proportional to plate current for given grid and plate voltages. It usually requires several runs to get a really good match for EX and KG1. μ may need to be adjusted slightly in the process.

For pentodes, the best estimates of μ , EX, and KG1 are obtained from triode-mode curves, i.e., curves taken with the screen grid connected to the plate. Such curves were available for the pentodes listed in Table 1. It is very gratifying to see how parameters obtained from the triode curves (Figure 4) result in excellent fits to pentode and ultra-linear curves (Figures 3 and 5). In running the pentode plate curve program (Appendix A2), operating mode can be set by adjusting parameter TRIMODE (0 for pentode, 40 for ultra-linear (UL), and 100 for triode). VG2NOM (quiescent screen voltage) must also be set appropriately.

KG2 is inversely proportional to pentode screen grid current ($I(VG2)$ in Appendix A2) for given control and screen grid voltages. Curves for pentode screen grid current are rarely published, but screen grid current is usually included in tables of typical tube operation. It is adequate to match a point or two from these tables. KG2 does not need to be estimated with great accuracy to obtain good results in SPICE simulations.

KP (k_p) dominates the behavior of the new model in region (A), which is characterized by large negative grid voltage, large plate voltage, and low plate current. Plate current is inversely proportional to KP in this region. Determining KP for a given tube is done entirely by trial-and-error. Triode-mode curves must be used to obtain

an accurate estimate of KP for pentodes because published pentode curves tend to have insufficient resolution in region (A).

KVB (k_{VB}) relates to the "knee" of the characteristic curves, and is defined differently in the triode and pentode equations. For pentodes, the knee is proportional to KVB, and is most visible in the pentode-mode curves (Figure 3 for the 6550). Equation (5) does not give an accurate estimate of the knee for all levels of grid voltage, E_G . Fortunately, this is not a serious limitation because load lines for practical designs pass close to the knee for $E_G = 0$. (Otherwise there would be a serious impedance mismatch.) The location of the knee for $E_G = 0$ was used to determine KVB in Table 1. For triodes, the knee is proportional to the square root of KVB, and is only apparent when the tube is operated with positive grid voltage. Triode curves for positive grid voltage are published infrequently, for example, the 12AU7 in the Sylvania tube manual.

Figure 3. Circuit for obtaining triode characteristic curves.

RGI is the grid-to-cathode resistance that controls grid current when $E_G > 0$. (No grid current flows during normal operation, when $E_G < 0$.) The numbers in Table 1 are rough estimates based on measurements of similar tubes in old texts.³ Good up-to-date data is very scarce, and would be most welcome. Grid current flows in class AB₂ push-pull amplifiers, but doesn't need to be modeled precisely because it does not contribute to the output signal. It should, however, be modeled well enough to determine if the driver tubes can supply the necessary current. Grid current presents a special problem in capacitively-coupled circuits: Since it only flows in one direction, it charges the coupling capacitor, driving the tube closer to cutoff. This effect does not appear in SPICE small signal analysis or in transient (.TRAN) solutions for a single cycle of the input waveform: Many cycles are required. This effect can be very audibly jarring.

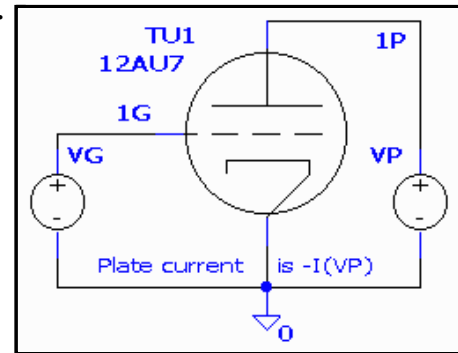
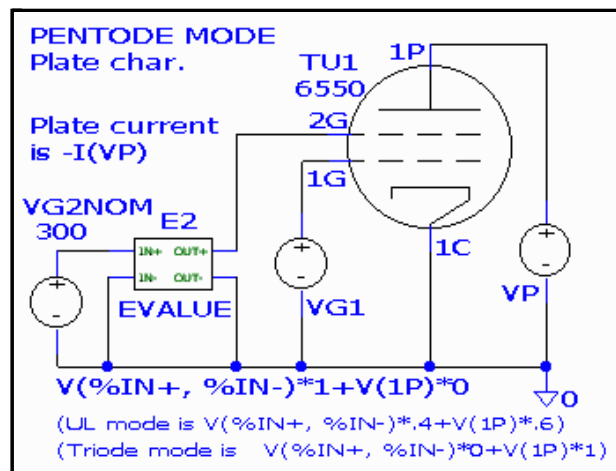


Plate curves

Figure 4. Circuit for obtaining triode characteristic curves.

Plate curves (plate current as a function of plate voltage for a set of grid voltages) may be obtained by running one of four files in the tube models file, which can be downloaded by shift-clicking on [Tubemods.zip](#). Schematics for Triotest.sch and Pent_P.sch are shown on the right.



1. Triotest.sch Figs. 2, 3. Triode curves.
2. Pent_P.sch Figs. 4, 5. Pentode curves in pentode mode (screen grid at a fixed voltage). VG2NOM should be set to the fixed screen voltage used in the data sheet. EVALUE E2 attribute EXPR is set to $V(\%IN+, \%IN-)*1+V(1P)*0$ (could be simply $V(\%IN+, \%IN-)$).
3. Pent_TR Fig. 6. Pentode curves in triode mode (screen grid connected to plate). VG2NOM is not used. EVALUE E2 attribute EXPR should be set to $V(\%IN+, \%IN-)*.0+V(1P)*1$ (or simply $V(1P)$).
4. Pent_UL.sch Fig. 7. Pentode curves in Ultra-Linear (UL) mode (screen grid on 40% output transformer tap). VG2NOM should be set to the quiescent plate voltage. EVALUE E2 attribute EXPR should be set to $V(\%IN+, \%IN-)*.4+V(1P)*.6$. Ultra-Linear mode is a compromise between pentode and triode mode that shares some of the best attributes of both: much lower output impedance than pentode mode and much greater power output than triode mode.

Control files and netlists for these files, as well as a earlier versions written in SPICE language, are shown in Appendices A1 and A2. Additional instructions on using these files are [near the end of Part 2](#). Pentode, triode, and UL mode plots for the GE 6550A are shown below. The [6550A data sheet](#) is available from [Frank's electron tube pages](#). It's gratifying to see the excellent agreement with data in all three modes.

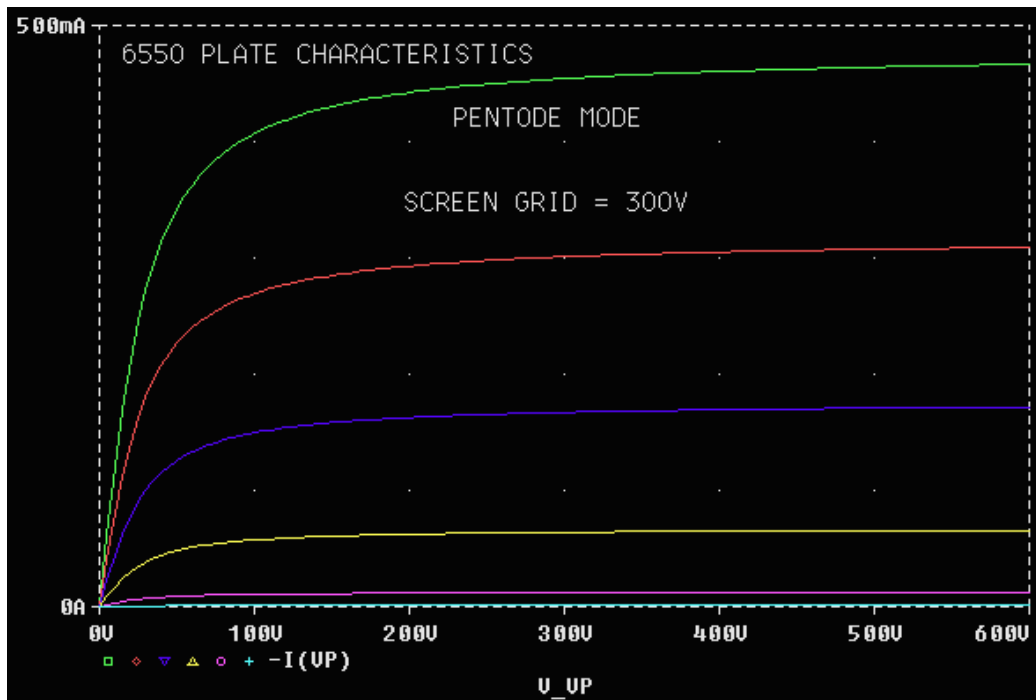


Figure 5. 6550A Pentode mode plate curves.

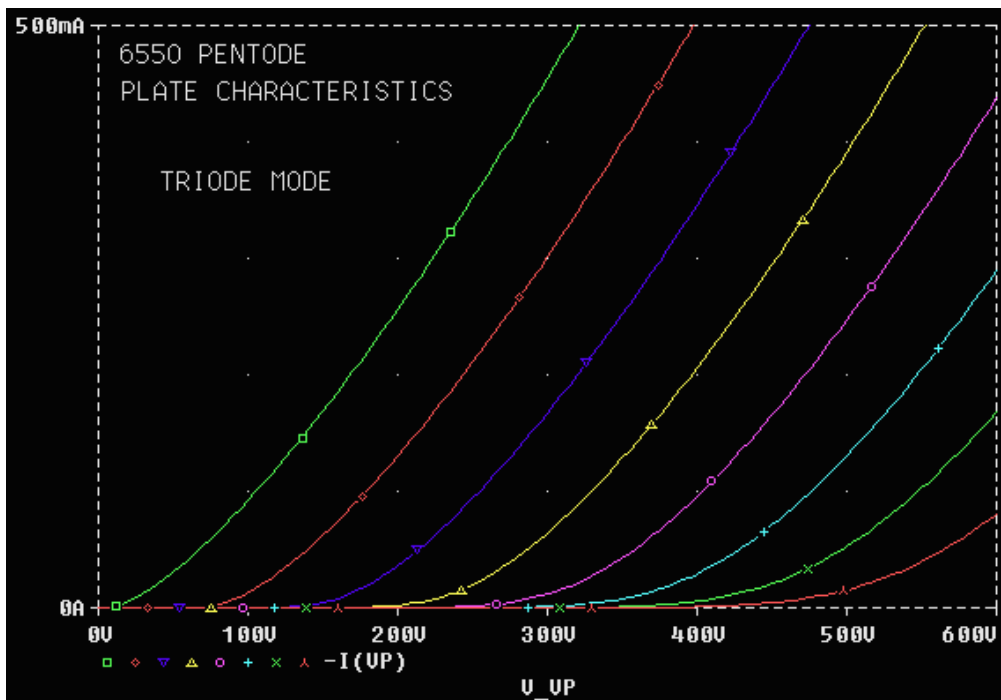


Figure 6. 6550A Pentode, Triode mode plate curves.

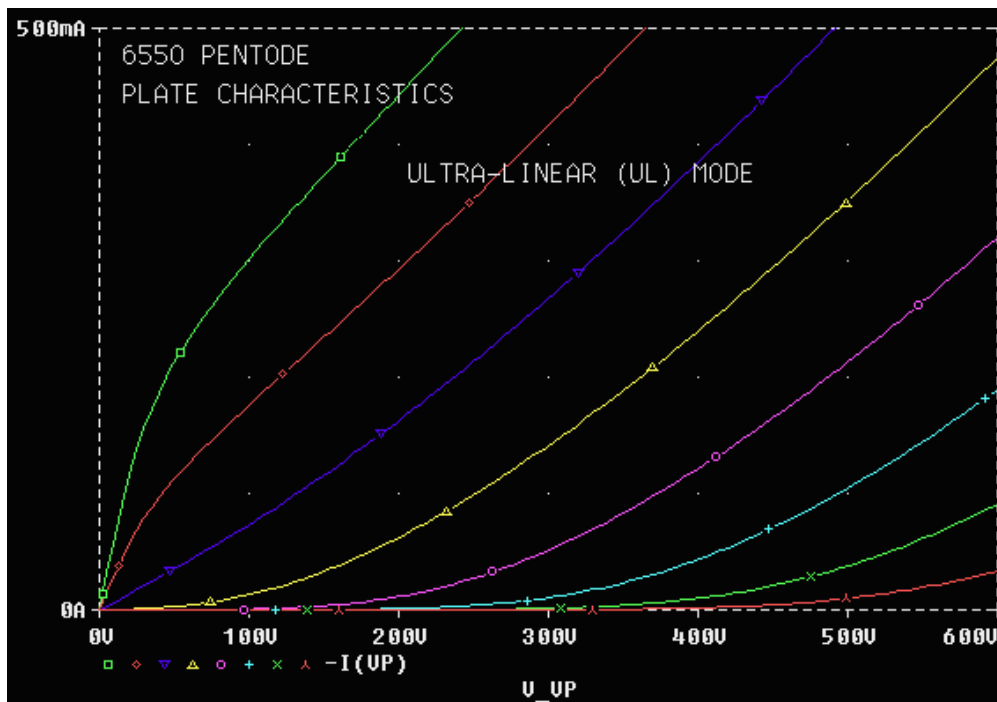


Figure 7. 6550A Pentode, Ultra-Linear (UL) mode plate curves.

Output transformers

Output transformers have traditionally been surrounded by an aura of mystery, and hard data has been hard to find. [Plitron Manufacturing Inc.](#) (601 Magnetic Drive #8, Toronto, Ontario, Canada M3J3J2, 416-667-9914) has improved this situation in a new line of toroidal output transformers that have an unusually complete [set of specifications](#).⁹ They have also published an excellent book, "Transformers and Tubes in Power Amplifiers", by Menno Vanderveen, that describes these specs in detail.

Toroidal transformers have much lower leakage flux than traditional "EI" transformers (so-named after their core geometry). This results in an extremely large bandwidth. Their only drawback is that their extremely low magnetic circuit reluctance makes it necessary to balance the dc current in the two halves of the primary winding extra care. Table 2 gives parameters for the PAT-4006-CFB output transformer relevant to SPICE simulation. The PAT-4006-CFB is a semi-custom version of the 100W PAT-4006, available on special order, designed for four 6550 or similar tubes in push-pull-parallel. It has a 2 kohm primary winding and two center-tapped secondary windings: a 5 ohm winding for the speaker and a 20 ohm winding for feedback.

PARAMETER	SYMBOL	VALUE
Total primary inductance	L_P	392.5H
Primary leakage inductance	L_{SP}	0.972mH
Quality factor = L_P / L_{SP}	Q	403600
Ultra-linear tap		40%
Turns ratio	N_P/N_S	20
Effective primary capacitance	C_{IP}	342pF
Total primary resistance	R_{IP}	56 ohms
Total secondary resistance	R_{IS}	0.1 ohms

Table 2. Parameters of the Plitron PAT-4006-CFB output transformer

To develop the SPICE model (Appendix A3), the center-tapped primary winding with ultra-linear taps is divided into four mutually-coupled segments that have $\{0.3, 0.2, 0.2, 0.3\}$ of the total number of windings. The inductance of each segment is proportional to the square of the number of windings: $\{.09L_P, .04L_P, .04L_P,$

.09LP} = {35.325H, 15.7H, 15.7H, 35.325H}. The ratio of primary to secondary inductance is $(N_P/N_S)^2$. The transformer's coefficient of mutual coupling, K_{ALL} , is entered into the SPICE data file:

$$K_{ALL} = \text{sqrt}(1-1/Q) \approx 1-1/(2Q) \quad (\text{The approximation holds for large } Q.)$$

(6)

The primary leakage inductance, L_{SP} , should not be entered into the SPICE model. Its effects are calculated from K_{ALL} . Similarly, the total primary inductance, L_P , is not entered. The effective primary capacitance, C_{IP} , is inserted between the end terminals of the primary winding. The SPICE model of the PAT-4006-CFB in Appendix A3 matches Plitron's published data very closely. Primary and secondary resistances have been omitted for the sake of simplicity (to avoid bumping into the limits of the PSpice evaluation package) and because they have very little effect on performance: They are only about 2.5% of their corresponding impedances. They can be easily added if required.

Example: The original PAS preamplifier

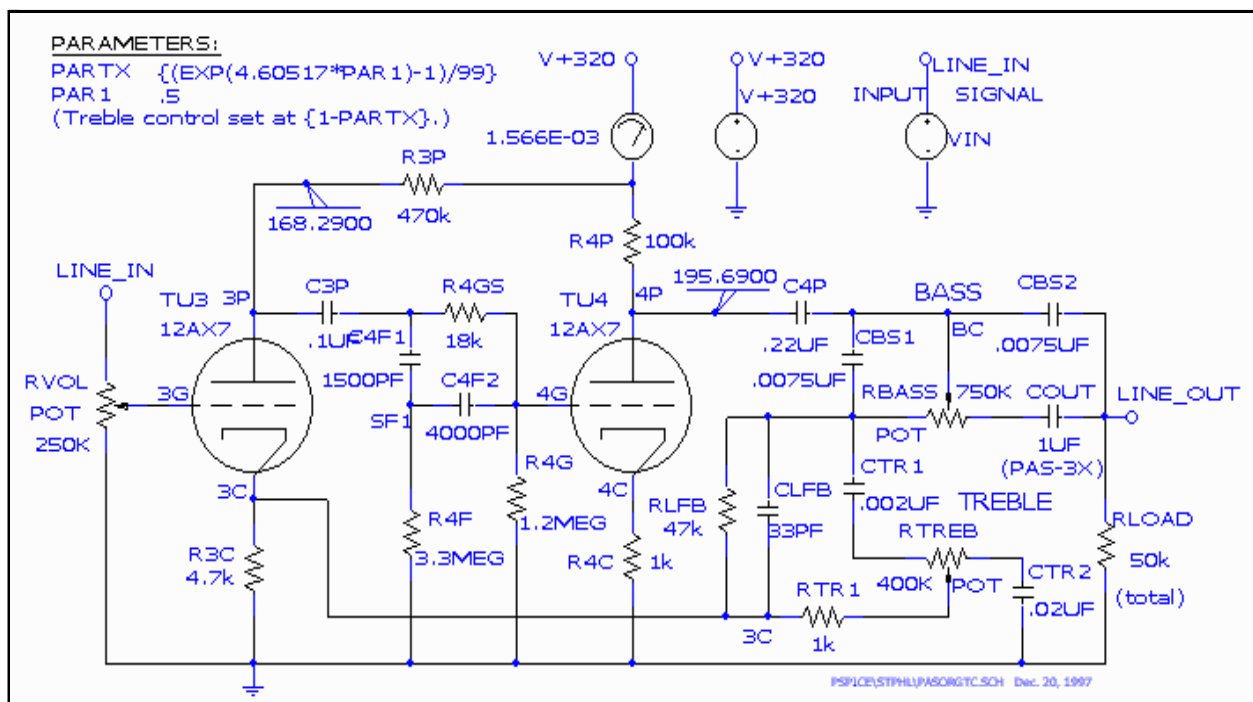


Figure 8. Original PAS-3 line amplifier schematic diagram.

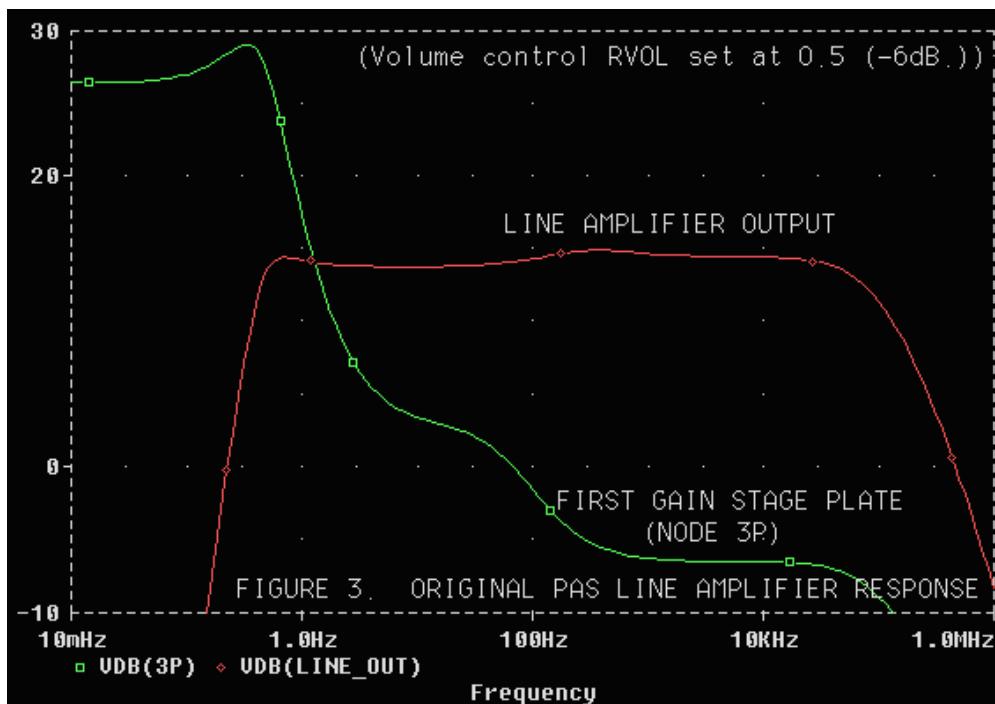


Figure 9. Original PAS-3 line amplifier frequency response at first gain stage plate (3P) and output (LINE_OUT). Generated by Probe.

The frequency response at the output meets the 20-20,000 Hz ± 1 dB specification with no problem, but there is a huge 0.5 Hz peak at the first gain stage plate. Although this peak is well outside the range of hearing it caused definite audible problems, especially on records, where the rotational frequency at 33.3 RPM is-- you guessed it-- almost exactly 0.5 Hz. It was a sort of grindy sound-- perhaps an intermodulation distortion component. A redesign, discussed in [Feedback and Fidelity](#) and [SPICE and the art of preamplifier design](#), fixed it beautifully. Listings and excerpts from the output files are in [Appendix A4](#) in part 2.

Output files PASorgTC.cir and PASorgTC.net were generated by schematics. Output file PASorgTC.out was generated by PSpice. Many of the lines in PASorgTC.cir are generated from Schematics by clicking on [Analysis, Setup...](#), then working with the appropriate boxes. .LIB is followed by the name of the library file that contains subcircuits referenced in the program. .PROBE instructs the program to generate data for the PROBE program, which displays graphs of time and/or frequency domain signals at each node. .ac calls for an ac (frequency-domain) analysis, i.e., a frequency sweep with 20 points per decade from 0.01 Hz to 1 MHz. If present, .TRAN calls for a transient (time-domain) analysis, which is required if a Fourier analysis is to be performed.

Appendix A4C shows some of the results of running PSpice, including the dc voltages at each node, the currents drawn by the two voltage sources, and the total power consumption. The first tube is labelled TU3 because TU1 and TU2 were in the phono preamplifier. For example, the plates of TU3 and TU4 are at 168 and 196 volts-- about where they should be (a little over half the supply voltage). A Fourier analysis can be performed for time domain simulations.

Lessons

Audio myths die hard, but several wither under the bright light of SPICE simulation. Feedback loops, which generate plenty of controversy in audio circles, are well-suited for SPICE analysis. We can only touch on them here- I discuss them in detail in [Feedback and Fidelity](#).

Let A be the open-loop (non-feedback) gain of an amplifier. Let B be the fraction of the output voltage that the feedback circuit subtracts from the input. The closed-loop gain (with feedback) of the amplifier is

$$G = A/(1+AB)$$

(7)

$T = AB$ is called loop gain. A 180 degree phase shift in open-loop gain A at any frequency where $T > 1$ will cause oscillation. An RC network has a maximum phase shift of 90 degrees at frequencies well beyond its corner frequency ($1/2\pi RC$).

For a feedback loop to be stable, rolloff at both low and high frequencies must be dominated by a single RC network (known to the mathematically literate as a pole on the complex plane) at frequencies where $T > 1$. Instabilities (near-oscillation situations) appear as peaks or irregularities in the frequency response curve. Such instabilities are shockingly common, even in very expensive high-end amplifiers. In *Stereophile* equipment reviews, they appear as supersonic peaks of up to 3dB between 50 and 150kHz in frequency response curves measured with 8 ohm resistive loads.⁶ In a particularly amusing case, an \$18,990 amplifier with such a peak received a mixed review, drawing an angry long-winded response explaining how every detail of the setup (down to the power cord) had to be perfect for these incredible amplifiers to sound decent. *Ridiculous!* (*It was a Jadis; I didn't want to mention the brand when I first published this article.*)

Response peaks are indicative of poor *phase margin*- the amount of additional phase shift that can be tolerated before oscillation begins. Since reactive loads tend to *increase* phase shift, poor phase margin can make an amplifier particularly sensitive to cables and loudspeakers. (Of course, some audiophiles might regard sensitivity to speaker cables as a *virtue!*) Phase margin should be a standard specification in audio amplifiers: It is one of those seldom-mentioned factors that affect sonic quality. One of the reasons for the current popularity of triode amplifiers without feedback may be that they have no phase margin problem.

Some instability is inevitable when feedback is applied to tube amplifiers with output transformers because the high frequency rolloff of transformers is dominated by a complex pole pair. This necessitates some form of compensation, i.e., an RC network that rolls off response at frequencies lower than the transformer cutoff. The most common technique is to place a capacitor in parallel with the global feedback resistor. Beware of this technique: It works in the near-audio range, but SPICE shows that it can cause response irregularities at extremely high frequencies (as well as inserting RF interference from speaker cables into the amplifier input).

A better approach to compensating feedback loops is to add a capacitor between the grid and plate of the tube to which the feedback returns (usually the input tube). The RC network formed by this capacitor and the input grid-stop resistor (a 2k to 10k resistor which should be added if not present) then becomes the dominant frequency-limiting network. The effective capacitance in this RC network is the sum of this capacitor and the tube inter-element capacitance multiplied by the gain of the stage (which is reduced by feedback). This is known as the Miller effect. The proper capacitor value (a few picofarads) must be determined by trial-and-error: The calculations are not simple. This is where SPICE really shines. You can try a new capacitor value and examine its effects in about a minute.

Tube response extends well beyond 1MHz. The .AC sweep in SPICE makes it easy to observe extremes of frequency: I routinely test 0.01Hz to 100MHz. To determine the correct compensation capacitor, observe the signals at the amplifier output and the plate of the input (feedback return) stage at frequencies up to 100MHz. Do this without and with capacitors across the load, which always degrade stability. (Instabilities are visible as high frequency peaks.) Increasing the compensation capacitor rolls off high frequencies but increases stability. If your circuit and output transformer are well-designed, you should be able to choose a value for the compensation capacitor that extends your amplifier's frequency response well-beyond the limits of audibility (20kHz) while maintaining stability with the largest shunt capacitance likely to arise from cables and speakers. If you plan to use electrostatic speakers, you will probably need extra compensation capacitance. The final value must be determined experimentally, but it should be close (within 30%) to the SPICE-determined value.

Few of us have signal generators that go down to 0.5Hz, but SPICE shows up a resonance at that frequency in the line stage of the venerable PAS⁷ (40 dB or more at the input stage plate), which can be quite serious because 0.5 Hz is close to the frequency of record warps. SPICE makes it easy to observe the response at intermediate gain stages as well as the output.

Using SPICE to investigate circuits has been nothing short of a revelation for me. I've been able to try out and rapidly debug new designs that I wouldn't have otherwise risked, and I've answered many nagging questions. I hear a strong correlation between SPICE results (in areas such as stability and RF susceptibility) and subjective sound quality. I find SPICE to be far more beneficial than "flobie dust" (extremely expensive premium components alleged to produce magical results).

[Part 2: Listings and PSpice operation](#) | [Norman Koren Vacuum tube audio page](#)
[Finding SPICE tube model parameters](#)



visits to this page
since December 8, 2003

Images and text copyright © 2001-2003 by [Norman Koren](#). Norman Koren lives in Boulder, Colorado, where he worked in developing magnetic recording technology for high capacity data storage systems through 2001. He has been involved with photography since 1964. Designing vacuum tube audio amplifiers was his passion between about 1990 to 1998.

