



Hello DIY Tube-Amplifier Friends

Sorry, my English is not the best, because my mother tongue is Swiss-German. This is a working paper and I will update my faults as soon as possible. From time to time I will expand the document with new examples and information.

I wrote this document mainly for myself, because I'm an amateur user of the nice LTspice software. In the past I always fail with the transformer simulation and was not sure how good the tube models are, which I found on the web.

- Only amplifiers calculated after the physical laws, well soldered, professional analyzed and at the end tested with the human ears, are audiophile tube amplifiers.
- I assume the readers are familiar with LTspice. This manual is a supplement on how simulate tube amplifiers with LTspice.
- LTspice is a very interesting, helpful and complex simulation program. But it's primarily a tool to simulate switching power supplies with the chips from the Analog Devices Company and with Mike Engelhardt as the author of the program.
- To simulate tube amplifiers with LTspice it's a compromise, because it is used for something, where it's not intended made for. Especially for the two most important parts topics in the tube amplifiers, tubes and wideband audio transformers.
- There are no standards for tube models incl. in the program. You have to find out for yourself, which tube models from the web works properly together with LTspice. If they are not precise enough for you, then go and create your own models.
- Wideband audio transformers modeling is a sad story for LTspice for tube amplifier enthusiasts:
 No audio transformer manufacture publishes or discloses all the parameters to create a good transformer spice model like the CHAN-Model, because they like to protect their know-how (and it's in some way understandable)
 - Under normal circumstance it is impossible to know the transformer data or measure the linear and nonlinear parameters of an existing audio transformer etc.
 - So be happy to get the winding ratio, something like primary inductance and DC-resistance of the windings and don't trust any data like impedance etc.
 - Transformers themselves, have no impedance, only inductance and so on !
 - Transformers only reflect impedance from the secondary to primary or the other way around !

Big thanks to my wife Debbie and my brother-in-law Charles Wong in Toronto for correcting my poor English

• If there are questions about to simulate tube amplifiers with LTspice, use the excellent and official LTspice platform https://groups.io/g/LTspice and you will get professional answers.

Contents

Keyboard Shortcuts Table	5
Simulation	6
Examples of Simulation	6
Preamplifier Examples	6
1.SE Preamplifier	6
2.PP Preamplifier	7
Phono Preamplifier Example	9
1.Phono Preamplifier with passive RIAA	9
Power Amplifier	10
1.PP Power Amplifier	10
SPICE Directives Examples	12
1. Simulation SPICE Directives	12
2.SPICE Dot Commandes	13
Tubes	17
Tube-Symbols and Tube-Models installation from the internet	17
Tube-Symbol, -Model and Program-Code	17
1. Tube-Models are split into three different sections	17
2.Tube-Symboles (Placeholder)	17
3.Tube-Model Code (Mathematical)	20
4. The Attributes (Connections)	23
Link Tube-Symbol and Tube-Model or -Library together	24
1.Tube-Symbol	24
2.Place Tube-Model Code into the Schema	24
3.Link Tube-Symbol and Tube-Library together	25
4. Check Tube-Model with LTspice	25
Tube-Model Creation-Software	33
1. Triode Tube-Models created with "Curve Captor" of (Andrei Frolov)	33
2.Triode Tube-Models created with "Model Paint Tools" (Dmitry Nizhegorodov)	45
3.Pentode Tube-Models created with "Model Paint Tools" (Dmitry Nizhegorodov)	51
4.Tube-Models created with μ Tracer and ExtractModel (Ronald Dekker, Derk Refman)	52
Tube-Curve Contest	58
1.Norman L. Koren	58
2.S. Bench/Ducan	59
3.Rydel	60
4. Dimitry Nizhegorodov	61
5.Ronald Dekker, Derk Refman	62
Transformers	63
1.General	63
2.Elements of a simple Audio Transformer	64
3. Measuring of an Audio Output Transformers	65

4. Calculation of Transformer Parameters	66
Real Transformer Parameters	68
1.Input Transformer	68
2. Interstage Transformer	68
3. Line Output Transformer	69
4. Power Output Transformer	70
Oh dear LTspice !	71
1.Oscillation !	71
2. Incorrect Information in the SPICE Error Log Table when calculating minus 3dB Bandwidth !	73
3.Additional Column in SPICE Error Log Table when calculating "Harmonic Distortion"!	74
4.LTspice Sinusoidal Signal V_{PP} , V_P and V_{RMS} and your Lab-Instruments !	74
5. Sinus Signal and the Measuring Pointer-Tool and nice to have V_{RMS} with and without V_{DC} !	75
Bibliography	76



_
_
_
-
-
\sim
~
_
<u> </u>
=
_
a
\sim

		LTspice H	otKeys		Simulator	Directives - D	ot Commands	
IJ	Schematic	Sumhol	Maveform	Natliet	Command	Short De:	cription	
ì		o longe		101101	.AC	Perform a Sr	1all Signal AC Analysis	
	ESC - Exit Mode	ESC - Exit Mode			.BACKANNO	Annotate the	Subcircuit Pin Names on Por	currents
	F3 – Draw Wire				.DC	Perform a D	Source Sweep Analysis	
S	F5 - Delete	F5 – Delete	F5 – Delete		.END	End of Netlis		
ə p	F6 – Duplicate	F6 – Duplicate			.ENDS	End of Subci	rcuit Definition	
o N	F7 – Move	F7 – Move			.FOUR	Compute a F	ourier Component	
I	F8 – Drag	F8 – Drag			.FUNC	User Defined	Functions	
	F9 – Undo	F9 – Undo	F9 – Undo	F9 – Undo	.FERRET	Download a	File Given the URL	
	Shift+F9 - Redo	Shift+F9 - Redo	Shift+F9 – Redo	Shift+F9 – Redo	.GLOBAL	Declare Glob	al Nodes	
	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area		.IC	Set Initial Co	nditions	
	Ctrl+B - Zoom Back	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back		.INCLUDE	Include anot	ier File	
	Space – Zoom Fit		Ctrl+E - Zoom Extents		.LIB	Include a Lib	rary	
M	Ctrl+G – Toggle Grid		Ctrl+G – Toggle Grid	Ctrl+G – Goto Line #	.LOADBIAS	Load a Previ	ously Solved DC Solution	
9i	U – Mark Unncon. Pins	Ctrl+W – Attribute Window	'0' - Clear		.MEASURE	Evaluate Use	r-Defined Electrical Quantities	
٨	A – Mark Text Anchors	Ctrl+A – Attribute Editor	Ctrl+A - Add Trace		.MODEL	Define a SPI	E Model	
	Atl+Click - Power		Ctrl+Y – Vertical Autorange	Ctrl+R – Run Simulation	.NET	Compute Ne	work Parameters in a .AC Ana	ysis
	Ctrl+Click - Attr. Edit		Ctrl+Click - Average		.NODESET	Supply Hints	for Initial DC Solution	
	Ctrl+H – Halt Simulation		Ctrl+H – Halt Simulation	Ctrl+H – Halt Simulation	.NOISE	Perform a N	vise Analysis	
	R - Resistor	R – Rectangle	Comman	d I ine Switches	-0P	Find the DC	Derating Point	
	C - Capacitor	C – Circle			OPTIONS.	Set Simulato	r Options	
	L – Inductor	L – Line	Flag Short Desc	sription	PARAM	User-Defined	Parameters	
	D – Diode	A – Arc	-ascii Use ASCII .r	aw files. (Degrades performance!)	SAVF	I imit the Out	Intity of Saved Data	
9	G – GND		-b Run in batcl	n mode.	SAVEBIAS	Save Operati	na Point to Disk	
9 E	S – Spice Directive		-big or -max Start as a m	aximized window.	STFP	Parameter S	Neens	
Ы	T – Text	T – Text	-encrypt Encrypt a m	odel library.	SUBCKT	Define a Sub	circuit	
	F2 – Component		-FastAccess Convert a bi	nary .raw file to Fast Access Format.	TEMP	Temperature	Sweeps	
	F4 – Label Net		-netlist Convert a so	chematic to a netlist.	±۲.	Find the DC	small-Signal Transfer Function	
	Ctrl+E – Mirror	Ctrl+E – Mirror	-nowine Prevent use	of WINE(Linux) workarounds.	.TRAN	Do a Nonline	ar Transient Analysis	
	Ctrl+R - Rotate	Ctrl+R - Rotate	-PCBnetlist Convert a so	chematic to a PCB netlist.	.WAVE	Write Selecte	d Nodes to a .WAV file	
			-registry Store user p	reterences in the registry.				
			-Run Start simula	ting the schematic on open.	Suffix	Suffix	Constants	
			-SOI Allow MOSFE	:T's to have up to 7 nodes in subcircuit.		f 1e-15	E 2.71828182845	0452354
		メジアション	-uninstall Executes on	e step of the uninstallation process.	T 1e12	p 1e-12	Pi 3.14159265358	79323846
		CHINOLOGY	-wine Force use of	WINE(Linux) workarounds.	G 1e9	n 1e-9	K 1.3806503e-23	
					Meg 1e6	u 1e-6	0 1.602176462e-1	6
					K 1e3	M 1e-3	TRUE 1	
I						Mil 25.4e-6	FALSE 0	

Keyboard Shortcuts Table

Simulation

Examples of Simulation

Examples to learn how LTspice is working with tube models and audio transformers Hmm, of course there is still room to improve these examples to turn it into the world's best amplifier out of it !

- Find more detail information about tube models, transformers and LTspice in the further chapters •
 - More or less the input sensitivity standards of preamps:
 - Phono Preamp: $0.2V_{RMS} = 0.566V_{pp} = 0.283V_{p}$ Tuner: $0.5V_{RMS} = 1.414V_{pp} = 0.707V_{p}$

 - CD + DAC: 2,0V_{RMS} = 5.657V_{pp} = 2.828_p

Preamplifier Examples

1. SE Preamplifier

Preamplifier example with low impedance output •



- Simulation: .ac Dec 50 1 500k
- Input: sinus 1Hz to 500kHz and 200mV_{RMS} = 0.283V_p
- Bandwidth: -3dB = 363.5kHz, from 1.07Hz to 363.5kHz !



- Simulation: .tran 10m
- Input: sinus 1kHz and 200mV_{RMS}, output 5.915V_{RMS},
- Gain: $20 \log \frac{V_{(out)}}{V_{(in)}} = 29,4$ dB

- Harmonic distortion: THD = 0.19%; THD+N = 0.202% !



2. PP Preamplifier

 Example of push pull preamplifier with tube 6N1P, volume control 0-50kΩ, zobel network input- LL1644A and output transformer LL2745



- Simulation: .ac dec 50 1 500k with zobel network
- Input: sinus 1Hz to 500kHz and 200mV_{RMS}
- Bandwidth: -3dB = 124.1kHz, from 4.94Hz to 124.1kHz !

▯◙▨▯ਸ਼ੑਖ਼ਖ਼ੑੑੑਖ਼ੑਖ਼ੑੑਖ਼ੑੑੑਖ਼	Հ≑ 3 幸⊅ॐ♡♡♡∈≣≘3Aa ∘p
🕻 Preamp_04.asc 🔛 Preamp_04.raw	
Preamp_04.raw	
10dB V(out) 10dB 6dB 6dB PSPICE Error Log: End 2dB Electronic\LTSpice\Doc -2dB bw=124089 FROM 4.94164 TO 124094 -6dB <	
-10dB 1Hz 10Hz 10Hz 1KHz 1	-330° -360° 0KHz 100KHz
-10dB 1Hz 10Hz 100Hz 1KHz 1 \$\$\$ Preamp_04.asc	
-10dB 1Hz 10Hz 10Hz 1KHz 1 Preamp_04.asc memory takes an anglobal memory takes an anglobal memory takes anglobal takes to Visit takes t	Image: Second
10dB 11dz 10Hz 10Hz 1 <	Image: Second
-10dB 1Hz 10Hz 100Hz 1KHz 1 Preamp_04.asc Preamp	WHz 100KHz Image: State St
10dB 1dHz 10Hz 10Hz 1KHz 1 Image: State Frequency State St	
• 10dB Hz 10Hz 10Hz 1KHz 1 • Preamp_04.asc Image: Bit of the second	
• 10dB Hz 10Hz 10Hz 1KHz 1 • Preamp_04.asc • ************************************	

- Simulation: .ac dec 50 1 500k without zobel network





- Simulation: .tran 10m
- Input: sinus 1kHz and 200mV_{RMS}, output 2.51V_{RMS},
- Gain: $20 \log \frac{V_{(out)}}{V_{(in)}} = 22,0$ dB

- Harmonic distortion: THD = 0.00449% !, THD+H = 0.054799% !



Phono Preamplifier Example

1. Phono Preamplifier with passive RIAA

- More or less about input sensitivity and gain standards of phono preamplifiers:
 - MM sensitivity: $3mV_{rms} = 8.49mV_{pp} = 4.24mV_{p}$
 - MM gain: 40dB
 - MC sensitivity: '0.3mV_{rms} = 0.849mV_{pp} = 0.424mV_p
 - MC gain: 60dB
- Example with tube EC95 / 6ER and ECC83 / 12AU7
 For measurement connect sinus- or RIAA-generator to phono preamp



- Simulation: .ac dec 50 10 100k



- Simulation: .tran 030m 20m
- Input: sinus 1kHz and 3mV_{\text{RMS}}, output 401.6V_{\text{RMS}},
- Gain: $20 \log \frac{V_{(out)}}{V_{(in)}} = 42,5$ dB
- Total harmonic distortion: 1.32%



Power Amplifier

1. PP Power Amplifier

- Example of push pull amplifier with tube ECC99 and 300b, input transformer LL1544A, interstage transformer LL1635 PP and output transformer LL1620 PP
- See simulation problem with this schematic in later chapter "Oh dear LTspice !" .measure tmp max mag(V(out))





- Simulation: .ac dec 50 1 500k, coupling factor of all transformers K = 1.0 - Input: sinus 1Hz to 500kHz and 200mV_{RMS}

C PP_Pov veramp_02.asc 🚩 PP Po

PP Poweramp 02 raw	1	7					X
19dB 18dB 17dB 16dB 15dB 14dB		SPICE Error Log: tmp: MAX (mag (v (c bw=351401 FROM 3 <	V(out)	1,0°) FROM 1 TO 500	×		200° 160° 120° 80° 40° - 0° - 40° 40°
12dB	1047	10047	14	101		100KHz	-120°
1112	10112	100112	IN	12 10	1112		
PP_Poweramp_02.asc ressure top intex in	10112 (v(o.t.))			12 10			×
PP_Poweramp_02.asc	10112 náj(v(to.t)) g(v(o.t/)=thiptor(2) rist=1 Larg mag(v(.tl.) (2.13-1	x(j=truptor(2) fall=last x2(141516171			Waveform: V(out)	- C	
PP_Poweramp_02.asc Adducte tray offer Adducte tray offer Adducte tray offer Adducte tray offer Adducte tray Adducte Adducte	10112 H(1/047)-Implot(2) Hits-1 Larg map() K1 (3 12-51 - 1000 			a 🔊	Waveform: V(out) Start Frequency:	× 1000mHz	
PP_Poweramp_02.asc desarray for the two desarray desarray	10112				Waveform: V(out) Start Frequency: End Frequency:	× 1000mHz 500KHz	
PP_Poweramp_02.asc Mediate the stem manuale the trips manuale the trips					Waveform: V(out) Start Frequency: End Frequency: Reference:	1000mHz 500KHz 1827dB@1047KHz	
Int Image: Control of the image of th					Waveform: V(out) Start Frequency: End Frequency: Reference: BW:	× 1000mHz 500KHz 18.27dB @ 1047KHz 004.52KHz	

- Simulation: .tran 10m
- Input: sinus 1kHz and 200mV_{RMS}, output $5.83V_{RMS}$,



SPICE Directives Examples

1. Simulation SPICE Directives

- Transient Simulation
 - This is the most direct simulation of a circuit. It basically computes what happens when the circuit is powered up
 Example Simulation: stops after 10 milliseconds

- Example Simulation: stops after 10 milliseconds	
😕 Edit Simulation Command	×
Transient AC Analysis DC sweep Noise DC Transfer DC op pnt	
Perform a non-linear, time-domain simulation.	
Stop time: 10m	
Time to start saving data:	
Maximum Timestep:	
Start external DC supply voltages at 0V:	
Stop simulating if steady state is detected:	
Don't reset T=0 when steady state is detected:	
Step the load current source:	
Skip initial operating point solution:	
Syntax: .tran <tstop> [<option> [<option>]]</option></option></tstop>	
.tran 10m	
Cancel OK	

- The active ".tran" simulation directive is in black color on the schematics

	•	• •	· · · · · · · · · · · · · · · · K1 L1 L2 L3 1 ·
			THE HER TO HER OVER
	•		
			LLID44A.
			tran 10m
		1	

- AC Analysis Simulation
 - AC analysis computes the AC complex node voltages as a function of frequency
 The frequency is swept between frequencies start frequency and end frequency

Compute the	small signal AC behavior of the cir point.	cuit linearized abo	out its DC operating	
	Type of sweep:	Decade	~	
	Number of points per decade:	50		
	Start frequency:	1		
	Stop frequency:	500k		
Syntax: .ac <oct,< td=""><td>dec, lin> <npoints> <startfreq> <e< td=""><td>ndFreq></td><td>-</td><td></td></e<></startfreq></npoints></td></oct,<>	dec, lin> <npoints> <startfreq> <e< td=""><td>ndFreq></td><td>-</td><td></td></e<></startfreq></npoints>	ndFreq>	-	

The active ".ac" simulation directive is in black color on the schematics

•		•	•		•	÷	÷	• •	÷	÷	÷	÷		•	÷	÷	•	÷	•	÷	÷	K1	-Li	1	_2	L3	1	-
											1	-	14	1.		-	12.											
	_		_	_				ac.	de	ec	.5	0	1		50	00	K	·	·	·		· ·	• •	T	14	544	17	
					1.		÷ .			11	÷.,				1.0		4									54	τn	
		•	·			÷	4	ra	n.	10	Jn	n		·		H	V)	┢						_		· r		_
								• •									-					· ·			- 10	السرزا		•

2. SPICE Dot Commandes

• Harmonic Distortion THD and THD+H Check



- This dot command is only computing a result after the ".tran" simulation - Please see also comments in the chapter "O dear LTspice"
- Compute a Fourier analysis after ".tran" simulation at the net label "V(out)" of frequency 1kHz and as default of 9 harmonics
- Select menu → View → SPICE Error Log to see analysis result
- Also result of total harmonic distortion

```
- Example: THD = 0.093731% and THD+H = 0.1056221
```

SPICE Erro	r Log:					Х
Fourier co	mponents of v	V(out)	I			^
De compone						
Harmonic	Frequency	Fourier	Normalized	Phase	Normalized	
Number	[Hz]	Component	Component	[degree]	Phase [deg]	
1	1.000e+03	8.244e+00	1.000e+00	0.08°	0.00°	
2	2.000e+03	1.390e-03	1.685e-04	-1.92°	-1.99°	
3	3.000e+03	7.510e-03	9.110e-04	0.59°	0.51°	
4	4.000e+03	8.436e-04	1.023e-04	5.58°	5.50°	
5	5.000e+03	7.422e-05	9.003e-06	24.86°	24.78°	
6	6.000e+03	4.315e-04	5.234e-05	-174.07°	-174.15°	
7	7.000e+03	5.770e-04	6.999e-05	-170.60°	-170.67°	
8	8.000e+03	3.737e-04	4.533e-05	-169.06°	-169.13°	
9	9.000e+03	4.016e-06	4.871e-07	9.11°	9.03°	
Total Harm	onic Distort	ion: 0.09373	1%(0.105621%)			\checkmark
<						>

• Bandwidth minus 3dB Check

- Find the dB peak response and call it "temp" at the net label "V(out)
- measure BW trig mag(V(out))=tmp/sqrt(2) rise=1 targ mag(V(out))=tmp/sqrt(2) fall=last
- "BW trig mag" computes the difference on frequency between the two points 3dB down from the dB peak response at the net label "V(out)
- "trig mag and targ mag" compute after the ".ac" simulation the difference in frequency between the two points 3dB down from dB peak response.

- Example: max dB = 18.36 dB and minus 3dB bandwidth = 351.4kHz and from 3.27Hz to 351.4kHz

AC 0.00424 SINE(0 0.00424 1k)

F R10

R1 47k

plus

V1



RIAA Forward Curve of Phono Preamp Check

Only calculates the dB values of the RIAA curve after ".ac" simulation - ".ac" simulation: 20Hz to 20kHz

plus

- Sinus input level AC: 3mV_{RMS} = 4.24mV_P

meas ac point 01 find V(out) at 20
.meas ac point_02 find V(out) at 50.5
.meas ac point_03 find V(out) at 500.5
.meas ac point_04 find V(out) at 1000
.meas ac point_05 find V(out) at 2121.5
.meas ac point_06 find V(out) at 10000
.meas ac point_07 find V(out) at 20000
and a second second second second second second	







- Open SPICE Error Log file

SPICE Error Log: E:\2019_06_15_Archiv_Kurt_Daten\Electronic\LTSpice\Doku_LTspi	\times
point_01: v(out)=(14.547dB,-11.1827°) at 20 +19.6dB	^
point_02: v(out)=(12.128dB,-38.0359°) at 50.5 +17.2dB	
point_03: v(out)=(-2.37446dB,-52.7731°) at 500.5 +2.7dB	
point_04: v(out)=(-5.06261dB,-48.5922°) at 1000 OdB	
point_05: v(out)=(-7.83538dB,-55.874°) at 2121.5 -2.8dB	
point_06: v(out)=(-18.5073dB,-80.1912°) at 10000 -13.4dB	
point_07: v(out)=(-24.3786dB,-85.2032°) at 20000 -19.3dB	~
<	>

- The standard forward RIAA filter curve



- RIAA Inverse Curve Generator check
 - The RIAA Generator is the combination of two components: $$\mathbf{v}_3$$

"normal voltage source" voltage controlled voltage source" (VCVS)

E1

- ".ac" simulation: 20Hz to 20kHz
- Sinus level at output VCVS: 3m_{VRMS} = 4.24mV_P
- Output level at the voltage source: 1.2 VPP



- "Laplace" = Multiplier coefficient between command and output transfer function of VCVS

Laplace=0.7m [*]	1+3.18m*s)*(1+75u*s))/((1+318u*s)*(1+3.18u*	s))
		pius

".ac oct" simulation

				١	ac oct 500 20 20k
					.tran 0 30m 20m
					.four 1K V(out)
1		1	1		

amp_02_01.asc 🛛 🔛 Phonopreamp_02_01.raw 📢 Phono 122 1p_02_01.a 🕻 Pł -28dB -32dB -36dB 40dE 44dB 48dE -52dB -56dB -60dE -64dE 68dl 1KHz 10KHz 100Hz

- Open SPICE Error Log file

SPICE Error Log: E:\2019_06_15_Archiv_Kurt_Daten\Electronic\LTSpice\Doku_LTspi	>	<
point_01: v(in)=(-66.815&B,20.0109°) at 20 -19.1dB		^
point_02: v(in)=(-64.4448dB,40.8008°) at 50.5 -16.6dB		
point_03: v(in)=(-50.185dB,51.9868°) at 500.5 -2.5dB		
point_04: v(in)=(-47.5434dB,47.8092°) at 1000 OdB		
point_05: v(in)=(-44.6841dB,54.4878°) at 2121.5 +3.0dB		
point_06: v(in)=(-33.9773dB,69.2984°) at 10000 +13.7dB		
point_07: v(in)=(-28.5647dB,63.4513°) at 20000 +19.1dB		~
<	>	





Tubes

The most data-specifications of the tube-producer are "**com si com sa**" Best is to create real tube-curves with an curve-tracer (especially for high μ -Tubes) Also it's possible to create the mathematical program-code in different ways but not all are excellent (see chapter "Tube-Model Program-Code")

Tube-Symbols and Tube-Models installation from the internet

Save the tube-symbols into LTspice \rightarrow lib \rightarrow sym \rightarrow misc. folder and the tube-models into a separate folder

Find tube-symbols on the internet, here are some different sources:

- <u>http://www.intactaudio.com/forum/viewtopic.php?t=24</u> tube-symbols and tube-models from Stephie Bench and/or Duncan's
- <u>http://www.dmitrynizh.com/dmitry_composites.zip</u> tube-models from Dmitry Nizehegorodev save only the tube-model library: dmitry_composites.lib
- <u>http://www.normankoren.com/Audio/Tubemods.zip</u> tube-models from Norman Koren save only the tube-model library: Tube97.lib; tube1.lib; Tube.lib
- <u>https://www.dos4ever.com/uTracer3/TubeLib.inc</u> tube-models are created with the "ExtractModel" program of Derk Reefman and are based on real tube data measured with "µTracer" curve tracer from Ronald Dekker <u>https://www.dos4ever.com</u>
- <u>http://ayumi.cava.jp/audio/tubemodel_3.20_win.zip</u> save ZIP-file with Explorer, maybe the best Rydel tube model on the web, have to be modified for use with LTspice (see preamplifier example)

Tube-Symbol, -Model and Program-Code

1. Tube-Models are split into three different sections

• Workflow



- The symbol (placeholder of the tube-model on the schema)
- The model (mathematical description of the component)
- The attributes and connection-number (LTspice instruction and link between tube-symbol and tube-model)

2. Tube-Symboles (Placeholder)

• Workflow

Draw new or chenge Tube-Symbol		Add or change Pin/Port Netlist Order Digits	
--------------------------------------	--	--	--

- Open File → Open.. → "Existing Symbol"



📜 lib 🚽		D- pentode.asy	25.02.2019 14:42	LTspice Symbol	2 KB
📜 cmp		D- PIGBT.asy	25.02.2019 14:42	LTspice Symbol	1 KB
📕 sub 🥒		D- SCR.asy	25.02.2019 14:42	LTspice Symbol	1 KB
📕 sym		D- signal.asy	25.02.2019 14:42	LTspice Symbol	1 KB
ADC		D- tetrode.asy	25.02.2019 14:42	LTspice Symbol	2 KB
Comparators		D- TowTom2.asy	25.02.2019 14:42	LTspice Symbol	1 KB
		D- TRIAC.asy	25.02.2019 14:42	LTspice Symbol	1 KB
DAC		D- triode.asy	25.02.2019 14:42	LTspice Symbol	1 KB
Digital		D- urc.asy	25.02.2019 14:42	LTspice Symbol	1 KB
Filt roducts		D- urc2.asy	25.02.2019 14:42	LTspice Symbol	1 KB
Misc		D- xtal.asy	25.02.2019 14:42	LTspice Symbol	1 KB
Cnamps	~	⊕ xvaristor.asv	25.02.2019 14:42	Tspice Symbol	1 KB



Open → Draw and change tube-symbol
 D⁻ Eile Edit Hierarchy Draw View Iools Windon



Open → Edit → Add Pin/Port for add a new or extra Pin/Port
 D Eile Edit Hierarchy Draw !



- Change Pin/Port numbering, click with right mouse button on Pin/Port
- Netlist Order "digit" has to correspond with the tube-model
- Example: Label: "Plate" → Netlist Order: "1"



• Correspondent connection for the grid



• Correspondent connection for the cathode



• Example: the correspondent link from the tube-symbol to the tube-model (Library "Koren_Tubes.inc")



• Example: the link between tube-symbol and tube-model will not work, because there are two additional tubeheater Pin/Port (H1; H2) and an error will appear (Library "dmtriodep.inc).



3. Tube-Model Code (Mathematical)

Tube-Model mathematical code includes all the information of the specified tube curves and specification values. ٠

IMPROVED VT

SIMULATIONS

MODELS FOR SPICE

SPICE, an electronic-circuit simula-tion program developed at the Berkeley, has found wide acceptance in industries. It can perform highly accu-rate time and frequency-domain analy-sis of complex analog and digital circuits (including barronici and M distortion). Several commercial versions of SPICE notably Pspice, from Microism, and ICAP/A, from Intusoft. Both programs supply limited evaluation versions that



18 GLASS AUDIO 5/96

•

•

FIGURE 2: 12AX7 plate curve from new model. overall 1 in Fig. 1

of greatest error. Unfortu as illustrated in *Fig. 1* by line for a 12AX7 with a 35-and a 150kW plate resisto crosses region (A), and a extend into this region for This problem is exacerbat push-null amplifiers, wher

plate and a ractangent factor unon ds' model) is inserted to error as n non-exponse-curve 'knce,' on is proportional to kp-ind [2] are forms of the nilds law, which can be na LaXZ feiche with µ or a laXZ feiche with µ or but a good e curves for a 12AX7 triode with μ = below the specified value but a good all fit to equation [1]) are illustrated in *I*.

Reference: part of the excellent article out of Glass Audio 5/96

Plate curves for a 12AX7 trie 93 (below the specified valu

12AX7 Spice-Example out of Norman L. Koren Library "Tube1.lib" ٠

Looks like a nice piece of program code, but see the result of the Tube-Curve Contest SUBCKT 12AX7 1 2 3 ; P G C; NEW MODEL + PARAMS: MU=100 EX=1.4 KG1=1060 KP=600 KVB=300 RGI=2000 + CCG=2.3P CGP=2.4P CCP=.9P ; ADD .7PF TO ADJACENT PINS; .5 TO OTHERS. E1 7 0 VALUE= +{V(1,3)/KP*LOG(1+EXP(KP*(1/MU+V(2,3)/SQRT(KVB+V(1,3)*V(1,3)))))} RE1 7 0 1G G1 1 3 VALUE={(PWR(V(7),EX)+PWRS(V(7),EX))/KG1} RCP 1 3 1G ; TO AVOID FLOATING NODES IN MU-FOLLOWER C1 2 3 {CCG} ; CATHODE-GRID C2 2 1 {CGP} ; GRID=PLATE C3 1 3 {CCP} ; CATHODE-PLATE D3 5 3 DX ; FOR GRID CURRENT R1 2 5 {RGI} ; FOR GRID CURRENT.MODEL DX D(IS=1N RS=1 CJO=10PF TT=1N) .ENDS

12AX7 Spice-Example out of S. Bench and/or Duncan's Library "dmtriodep.inc" SUBCKT TRIODENH A G K +PARAMS: LIP=1 LIF=3.7E-3 RAF=18E-3 RAS=1 CDO=0 RAP=4E-3 + ERP=1.5 + MU0=17.3 MUR=19E-3 EMC=9.6E-6 GCO=0 GCF=213E-6 + CGA=3.9p CGK=2.4p CAK=0.7p Elim LI 0 VALUE {PWR(LIMIT(V(A,K),0,1E6),{LIP})*{LIF}} Egg GG 0 VALUE {V(G,K)-{CDO}} Erpf RP 0 VALUE {1-PWR(LIMIT(-V(GG)*{RAF},0,0.999),{RAS})+LIMIT(V(GG),0,1E6)*{RAP}} Egr GR 0 VALUE {LIMIT(V(GG),0,1E6)+LIMIT((V(GG))*(1+V(GG)*{MUR}),0,-1E6)} Eem EM 0 VALUE {LIMIT(V(A,K)+V(GR)*{MU0},0,1E6)} Eep EP 0 VALUE {PWR(V(EM),ERP)*{EMC}*V(RP)} Eel EL 0 VALUE {LIMIT(V(EP),0,V(LI))} Eld LD 0 VALUE {LIMIT(V(EP)-V(LI),0,1E6)}

20

Ga A I Eqf GE	<pre> VALUE VALUE </pre>	{V(EL)}	
Egi Gr Ga G	K VALUE	{(V(GF)+V(L)	(V(G,K)-{GCO},0,1E0),1.3) {GCF}}
CM1	G	K	{CGK}
CM2	А	G	{CGA}
CM3	А	К	{CAK}
RF1	А	0	1000MEG
RF2	G	0	1000MEG
RF3	K	0	1000MEG
.ENDS			
*********	************	- /	***************************************
* GENEF	RIC: 12AX	77ECC83	
	L: NHIZA	tor model	
NOTES	b. NUTIEA	ter moder	
*******	*********	******	**********
.SUBCK	T NH12AX	7 A G K	
XV1 A G	K TRIODI	ENH	
+PARAN	1S: LIP= 1	.5 LIF= 0.000	016 RAF= 0.076498 RAS= 1 CDO=-0.53056
+ RAP=	0.18 ERP=	= 1.5	
+ MU0=	87.302 ML	JR=-0.013621	1 EMC= 0.00000111
+ GCO=-	-0.2 GCF=	0.00001	
+ CGA=3	3.90E-12 C	CGK=2.40E-1	2 CAK=7.00E-13
.ENDS	له بان		
*******	*********		
12AX7 S	pice Exam	ple out of Ro	nald Dekker / Rolf Refman Library "TubeLib.inc
SUBCK	τ 12ΔΥ7 1	23.460	
X1 1 2 3	ECC83	2 J, A G C,	
FNDS	20000		
******	**********	*****	*****
*******	**********	*****	******
.SUBCK	T ECC83 ⁻	1 2 3; A G C;	
* Extract	Model V .9	98	
* Model of	created: 0	9-Dec-13	
X1 1 2 3	TriodeK M	1U=108.93 E>	X= .988 KG1= 389.8 KP= 677.7 KVB=10751. RGI=2000
+ CCG='	1.6P CGP	=1.6P CCP=0	0.33P;
.ENDS	****		
*******	***********	*****	****
SUBCK	T Triadak	1 2 2· A C C	
.3000CK	1G	123, AGC	
F170V	ALUE=		
+{V(1.3)/	KP*LOG(^	+EXP(KP*(1)	/MU+V(2.3)/SQRT(KVB+V(1.3)*V(1.3)))))}
G1 1 3 V	ALUE={0.	5*(PWR(V(7)	,EX)+PWRS(V(7),EX))/KG1}
RCP 1 3	1G ; TO	AVOID FLO	ATING NODES IN MU-FOLLOWER
C1 2 3 {(CCG};CA	THODE-GRI	D
C2 2 1 {(CGP};GF	RID-PLATE	
C3 1 3 {(CCP};CA	THODE-PLA	TE
D3 5 3 D	X ; FOF	R GRID CURF	RENT
R1 2 5 {F	RGI} ; FOI	R GRID CURI	RENT
.MODEL	DX D(IS=	1N RS=1 CJO	D=10PF TT=1N)
.ENDS T	riodeK	• • • • • • • • • • • • • • • • • • •	
*******	**********		*****
Cimple -	nalvaia of	the Denald D	alder / Dalf Dafman program and

- Simple analysis of the Ronald Dekker / Rolf Refman program code
- Sorry, I'm not a program code hero!

•

.SUBCKT 12AX7 1 2 3; A G C; X1 1 2 3 ECC83 .ENDS

.SUBCKT	= define a sub circuit (Modell)				
12AX7	= name of the model				
123	= number of the or Pin/Port				
AGC	= term of the Pin/Port numbers: A=Anode, G=Grid, C=Cathode				
X1 1 2 3 ECC83	= call of the identical tube-model ECC83				
.ENDS	= end				

.SUBCKT ECC83 1	2 3; A G C;
* ExtractModel V .99	8
* Model created: 09-	Dec-13
X1 1 2 3 TriodeK	= call of the subcircuit "TriodeK"
MU=108.93 EX= .98	8 KG1= 389.8 KP= 677.7 KVB=10751. RGI=2000
	= the "Phenomenological Equations Factors" are the most important part of the
tube-model, with the	nis factors LTspice calculates the tube-curves
MU=108.93	= amplification factor (μ)
EX= .988	= calculated factor out of tube data
KG1= 389.8	= out of grid- and anode-voltage calculated factor
KG2=	= additional factor only for pentode-model
KP= 677.7	= calculated factor by high negative grid-voltage and small plate voltage
KVB=10751.	= different twist factor for triode- or pentode-curves
RGI=2000	= resistor factor between grid and cathode calculate grid-current
+ CCG=1.6P	= cathode-grid capacity in pF
CGP=1.6P	= grid-plate capacity in pF
CCP=0.33P	= cathode–plate capacities in pF
.ENDS	= end of sub circuit

• To calculate the different tube-types (Diode, Triode, Pentode etc.) of the Tube-Model of Ronald Dekker / Rolf Refman the "Generic-Model Part" is needed

.SUBCKT TriodeK 1 2 3; A G C RE1 7 0 1G E1 7 0 VALUE= +{V(1,3)/KP*LOG(1+EXP(KP*(1/MU+V(2,3)/SQRT(KVB+V(1,3)*V(1,3)))))} G1 1 3 VALUE={0.5*(PWR(V(7),EX)+PWRS(V(7),EX))/KG1} RCP 1 3 1G ; TO AVOID FLOATING NODES IN MU-FOLLOWER C1 2 3 {CCG} ; CATHODE-GRID C2 2 1 {CGP} ; GRID-PLATE C3 1 3 {CCP} ; CATHODE-PLATE D3 5 3 DX ; FOR GRID CURRENT R1 2 5 {RGI} ; FOR GRID CURRENT .MODEL DX D(IS=1N RS=1 CJO=10PF TT=1N) .ENDS TriodeK

4. The Attributes (Connections)

- The attributes includes the work information for the symbol (placeholder).
- Open \rightarrow Edit \rightarrow Attributes \rightarrow Edit Attributes



• Edit → Prefix: "X"

Symbol Typ	e: Cell V	
attribute	value	
Prefix	X	
SpiceModel		
Value	Triode	
Value2		
SpiceLine		
SpiceLine2		
SpiceLine2 Description		

Open → Edit → Add Pin/Port → Attribute Window
 J→ Eile | Edit Hjerarchy Draw View Iools Window Help



- Click → "InstName" and "OK" and place "Unnn" to the tube-symbol (this will be the placeholder for the LTspice "Netlist Number"
- Click → "Value" and "OK" and place "Triode" to the tube-symbol (this will be the placeholder for the link between "Tube-Number" and "Tube-Model"



📕 sym	^ Name	Änderu
ADC	D-battery.asy	25.02.20
Comparators	℃ cell.asy	25.02.20
DAC	D- DIAC.asy	25.02.20
📙 Digital	D- DIP8.asy	25.02.20
FilterProducts	D- DIP10.asy	25.02.20
📜 Misc	DIP14.asy	25.02.20
Opamps	D- DIP1/ asy	25.02.2(
	:D- D 20.asy	25.02.20
BowerBroducts	Epoly.asy	25.02.20
PowerFloducts	 EuropeanCap.asy 	25.02.20
Dateiname: triode.asy		
Dateityp: Schematic Symbol Files(*.as	y)	
Dateityp: Schematic Symbol Files(*.as	y)	

Link Tube-Symbol and Tube-Model or -Library together

1. Tube-Symbol

- Place tube-symbol on the desktop
- Click name "Triode" and enter "Tube-Number"
 Click name "Triode" and enter "Tube-Number"
 Lie Edit Hjerarchy View Simulate Iools Window Help

🖻 😂 🖬 😭 🛪 🕘 🔍 🔍	<	1 1 2 % 1 2 1 2	M 86 /
U1			
	Enter new Value for U1	FontSize	×
	Left ~	1.5(default) V	Cancel
	12AX7		
Triode			

2. Place Tube-Model Code into the Schema

- When copy schema, all the information are included
- Perhaps too much code on the schema
 File View Plot Settings Simulation Tools Window Help



3. Link Tube-Symbol and Tube-Library together

- Copy tube-model library into folder of LTspice schema
- Easy to work with different tube-number in the same schema



4. Check Tube-Model with LTspice

Workflow



- Open → New Schematic
- Open → Component

📭 🐸 🖬 😭 🛪 🕘 🔍 🔍	₹ 💐 🖄 !	2 2 % %	5 X 🖻 I	a M 2 4	ji <mark>2</mark> → @	$2 \odot \odot 4$
	Select Compo	nent Symbol C:\Users\Kurt\Docur	nenta\LTopiceXVII\lt	× vaym v	/	1
	C:\Users\Kut [ACC] Comparators] [DAC] [DAC] [Data] [Reference] [SpecialFunctions] < Ca	Documenta'L Tapice [Switches] Tubes] bi bi bi cap cap caw caw caw caw caw caw caw caw	Open this macr XVII'Nb kym\ f FenteBead FenteBead2 g g h nd ISO16750-2 ISO16750-2 ISO7837-2 LED	lead load2 load2 load2 load2 load2 load2 load2 load2 load2 mmcs4 mmcs4 mmcs4 mpn3 mpn3 V OK		

- Select → Component Symbol
- Select → Triode

9 Select Component Symbol	×
Top Directory: C:\Users\Kurt\Documents\LTspiceXVII\lib\sym	\sim
This symbol is for use with a subcircuit macromodel that you supply.	
C:\Users\Kut\Documents\LTspiceXVII\bi\sym\Misc\	
Image: Second	
Cancel OK	

- Place "Triode Symbol" on new schema



- Write into the field Prefix: x
- Write into the field Value: the tube number
 Component Attribute Editor
 × ×



• Now the tube symbol has an number



- Open → Component Symbol
- Select → \lib\sim
- Select → voltage
 Select Component Symbol



- ٠
- Place two "voltage symbol on the schema Click on \rightarrow V2 symbol tap two times \rightarrow Contrl+R (negative connector is now above) •



Click \rightarrow right mouse button on symbol V2 Click \rightarrow Advanced Button

Click → Advanced Button	
Voltage Source - V2	×
DC value[1]:	OK
Series Resistance[Ω]:	Advanced

- Select → button (none) DC Value: write 5V •
- •

Independent Voltage Source - V2	×
Functions	DC Value
(none)	DC value: 5V
O PULSE, / 1 P Tdelay Trise Tfall Ton Period Ncycles)	Make this information visible schematic:
SINE(Voffset Vamp , a Td Theta Phi Ncycles)	
O EXP(V1 V2 Td1 Tau1 Td2 + 9)	Small signal analysis(.AC)
◯ SFFM(Voff Vamp Fcar MDI Fsig)	AC Amplitude:
OPWL(t1v1t2v2)	AC Phase:
O PWL FILE: Browse	Make , ils information visible on schematic: 🗹
	Parasitic Properties
	Series Resistance[Ω]:
	Parallel Capacitance[F]:
	Make this information visible on schematic: 🗹
Addational DW/I Detate	
Make this information visible on schematic:	Cancel OK

- Click \rightarrow right mouse button on symbol V1 Click \rightarrow Advanced Button •
- .

Voltage Source - V1	×
DC value[V]:	OK Cancel Advanced

- Select → button (none)
- DC Value: write 500V

Independent Voltage Source - V1	×
anctions	DC Value
(inque)	DC value: 500V
O PUL V1 V2 Tdelay Trise Tfall Ton Period Ncycles)	Make this information visible on matic: 🗹
SINE(Voh. Vamp Freq Td Theta Phi Ncycles)	
O EXP(V1 V2 To Tau1 Td2 Tau2)	Small signal ACatysis(.AC)
◯ SFFM(Voff Vamp H, KMDI Fsig)	- Amplitude:
○ PWL(t1 v1 t2 v2)	AC Phase:
O PWL FILE: Browse	Makes information visible on schematic: 🗹
	Parasitic Properties
	Series Resistance[Ω]:
	Parallel Capacitance[F]:
	Make this information visible on schematic: 🗹
Additional PWL Points	
Make this information visible on schematic: 🗹	Cancel OK

- Select → Wire
- Design wiring like on the schema
- Select → Ground and place ground symbol (without Ground = LTspice can't simulate)
 Give Let Herearchy Year Simulate Tools Window Help



- Select → Simulate
- Select → Edit Simulation Cmd



- Edit command for dynamic simulation ٠
- •
- Select \rightarrow DC sweep \rightarrow Select \rightarrow 1st Source Name of 1st source to sweep: enter V1 (anode power supply)
- Type of sweep: enter Linear •
- Start value: enter 0V •
- Stop Value: enter 500V •
- Increment: enter 10V •



- Select $\rightarrow 2^{st}$ Source .
- Name of 1st source to sweep: enter V2 (grid power supply)
- Type of sweep: enter Linear
- Start value: enter 0V
- Stop value: enter 5V
- Increment: enter 0.5V
- Click \rightarrow OK button .



Place the "Syntax Command" into the schema •



- Create the "link to the tube library"

_ 8 ×

• Write the SPICE directive: inc. Koren_Tubes.inc

🝠 Edit Text on the Schema	tic:		×
How to netlist this text O Comment O SPICE directive	Justifier 	Font Size	OK Cancel
inc Koren_Tubes.inc			^ ~



- Now save the schema to a separate folder
- Use for each new project a new folder to minimise problems of incorrectly link data files



• The saved file looks like this

Test_Triode_Curves_01.asc

30.06.2019 16:36 LTspice Schematic

1 KB

- To simulate the schema, LTspice needs the tube model information
- Find the tube library file on the web: <u>http://www.intactaudio.com/forum/download.php?id=1905</u>

Save the file as "Koren_Tubes.inc" into same folder as the schema
 phys.ufl.edu/~majewski/nqr/Si × +



46 KB 1 KB

• The desktop looks like this

•

.

V1

5V

Touch the anode wire with the curser, then it will change into
 ダー "voltage probe"
 ビビ は Hierdry Yew Simulate Tools Window Hep
 ビビ 目 マ オ 色 ヘ ヘ 気 脳口 三 雪 智 よ 知 向 例 合 当 ℓ 之 切 く キ 3 文 ひ ひ つ ご 品 4 **

500V



12AX7

.dc V2 0V 500V 50V V1 0V 5V .5V

• <u>Click on the anode wire, the simulation plotting</u> of the anode change voltage appears

🚊 T st_Triode_Cu	rves_01.ra	N					-		×
5001				V(n001					
450V								/	
400V									
350V									
300V									
250V				/					
200V									
150V									
100V									
50V-									
0V 50V	100V	150V	200V	250V	300V	350V	400V	450V	500\

• Click on the grid wire, the simulation plotting of the grid step voltage appears

💮 Test_Tri	ode_Cur	ves_01.ra	N				5	C		8
0.01					V(n002))				
0.61										
-0.5V										
-1.0V										
-1.5V-										
-2.0V-	-					_				_
-2.5V	-				-	_				_
-3.0V										_
-3.5V-		_		-			_			
4.0V										
4 5V										
5.01										
ov	50V	100V	150V	200V	250V	300V	350V	400V	450V	500

- Touch the tube anode connector the curser will change into
 "current probe"
- Click on the anode connector, the simulation plotting of the anode curves appears



- If this error appears
- Check connections of the tube-symbol numbering with the numbering of tube
 LTspice XVII
 X

Port(pin) count mismatch between the definition of subci instance: "xu1" The instance has fewer connection terminals than the def	rcuit "12ax7" and inition.
	OK

- If this error appears
- Check if the number of the tube-model is included in the tube library
 LTspice XVII
 X

Unknown subcir	cuit called in:
xu1 n001 n00	2 0 12ax7
	ОК

Tube-Model Creation-Software

1. Triode Tube-Models created with "Curve Captor" of (Andrei Frolov)

- This program creates in the most simplest way excellent tube models incl. additional information, but is still the beta version since 2013 and has some stumbling blocks.
- On YouTube you will find also a Film how to use this program https://www.youtube.com/watch?v=tGPOx9GtXIM

•	First download the free runtime software "ActiveTcl" from ActiveState
	https://www.activestate.com/products/tcl/downloads/
	Active Qtate:

ACUVESIGIE		=
Do	wnload Tcl: Activ	eTcl
The tri	usted Tcl distribution for Windows, Linux and Mac, pre-bur top Tcl/Tk modules – free for development use.	ndled with
Get ActiveTcl Cor ActiveTcl is free to use for de development, see our plans quote. Download for Windows L ▲ ActiveTcl 8.6 By downloading ActiveTcl Commun ActiveState Community License. Ne	mmunity Edition For Free welopment purposes. For use beyond and pricing details or contact us for a custom inux Mac:	proprint and a second s
マ Branch main 〜	Manage	
Windows	Build Status 🕑 COMPLETE	1 Packages Built
🗯 Mac	> Package build status	
👌 Linux	Download Installer	
	Cownload .exe ActiveTcI-8.6.9.8609.2- x64-5ccbd9ac8.exe	MSWin32-
ActiveState ActiveState ActiveState ActiveState ActiveState ActiveState ActiveState ActiveState ActiveState	H-bit) Setup	
Choose Setup Type Choose the setup type that best	suits your needs	
Image: Second	e most common program features. Recommended for rs. ers to choose which program features will be installed e they will be installed. Recommended for advanced e m features will be installed. (Requires most disk space)	
	< Back Next > Cancel	



 Download the open source program "Curve Captor" from SOURCDFORGE https://sourceforge.net/projects/curvecaptor/

SOURCE FORGE

Home / Browse / Science & Engineering / Simula Curve Ca Status: Beta Brought to you by	ations / Curve Captor ptor : afrolov			
$\star \star \star \star \star$ 1 Review		Downloads: 7 This Weel		Last Update: 2013-0
Sector Secto	Get Updates	Share This		
Öffnen von curvecaptor-0.9.1-winxp.zip		×		
Sie möchten folgende Datei öffnen: Curvecaptor-0.9.1-winxp.zip Vom Typ: ZIP-Archiv (116 KB) Von: https://versaweb.dl.sourceforge.net Wie soll Firefox mit dieser Datei verfahren?	¢.			
Öffnen mit SpeedCommander (Standare)	~ (b			
O Datei <u>s</u> peichern	OK Abbrechen			
🔋 curvecaptor-0.9.1-winxp.zip	19.	.04.2021 17:12	117 KB	

Open zip-folder

curvecaptor	40 KB	19 KB	54%	02.06.2005 15:18
📧 m4.exe	141 KB	67 KB	53%	31.03.2005 13:15
models.m4	3 KB	1 KB	61%	30.05.2005 21:15
README.txt	3 KB	2 KB	55%	13.05.2005 23:49
📧 tubefit.exe	64 KB	29 KB	55%	02.06.2005 15:21

• Copy files into \rightarrow bin folder

	bitmap-editor.tcl	25.04.2015 01:15	TCL-Datei	22 KB
	critcl	07.03.2019 20:22	Datei	1 KB
💺 Acer (C:)	critcl.tcl	07.03.2019 20:22	TCL-Datei	1 KB
ActiveTcl	curvecaptor.tcl	19.04.2021 19:33	TCL-Datei	40 KB
📜 bin	diagram-viewer.tcl	03.12.2014 19:34	TCL-Datei	2 KB
doc	dtplite.tcl	04.02.2016 06:02	TCL-Datei	1 KB
include	🅐 m4.exe	19.04.2021 19:33	Anwendung	141 KB
lib	models.m4	19.04.2021 19:33	M4-Datei	3 KB
	nns.tcl	04.02.2016 06:02	TCL-Datei	8 KB
L licenses	nnsd.tcl	04.02.2016 06:02	TCL-Datei	4 KB
L man	nnslog.tcl	04.02.2016 06:02	TCL-Datei	5 KB
share	pt.tcl	04.02.2016 06:02	TCL-Datei	5 KB
	README.txt	19.04.2021 19:33	Textdokument	3 KB
	tcl86t.dll	07.03.2019 20:07	Anwendungserwei	1.676 KB
	tcldocstrip.tcl	04.02.2016 06:02	TCL-Datei	14 KB
	🚳 tclsh.exe	07.03.2019 20:07	Anwendung	68 KB
	🚳 tclsh86t.exe	07.03.2019 20:07	Anwendung	68 KB
	🚳 tclsht.exe	07.03.2019 20:07	Anwendung	68 KB
	tk86t.dll	07.03.2019 20:09	Anwendungserwei	1.436 KB
	tkcon.tcl	07.03.2019 20:22	TCL-Datei	194 KB
	👹 tubefit.exe	19.04.2021 19:33	Anwendung	64 KB
	🚳 wish.exe	07.03.2019 20:09	Anwendung	69 KB
	🚳 wish86t.exe	07.03.2019 20:09	Anwendung	69 KB
	🞯 wisht.exe	07.03.2019 20:09	Anwendung	69 KB

• Open Settings → Apps

=	Apps
	Deinstallieren, Standardwerte,
	optionale Funktionen

• Connect "file type" ".tcl" with "Wish Application"

.tak	JRiver Tclsh Application
Media Center file .tar	Tclsh Application
TAR-Archiv .tbc	Wish Application
TBC-Datei	Suchen Sie nach einer App im Microsoft Store
TBZ-Archiv	
.tcl TCL-Datei	Wish Application

• Create of the file "curvecaptor.tcl" a "Shurtcut"

🐛 Acer (C:)	🔞 bitmap-editor.tcl	25.04.2015 01:15	Т
ActiveTcl	Critcl	07.03.2019 20:22	0
📙 bin	😵 critcl.tcl	07.03.2019 20:22	Т
📙 doc	🔞 curvecaptor.tcl	19.04.2021 19:33	T
📜 include	curvecaptor.tcl - Shurtcut	21.04.2021 17:12	V
📕 lib	😡 diagram-viewer.tcl	03.12.2014 19:34	t T
licenses	😵 dtplite.tcl	04.02.2016 06:02	Т
	💜 m4.exe	19.04.2021 19:33	A
man			

• Tag and trop the file "curvecaptor.tcl" on the desktop

	🐛 Acer (C:)	😵 bitmap-editor.tcl	25.04.2015 01:15
curvecaptoritel -	ActiveTcl	Critcl	07.03.2019 20:22
	📜 bin	🞯 critcl.tcl	07.03.2019 20:22
	100	curvecaptor.tcl	19.04.2021 19:33
	include	😵 diagram-viewer.tcl	03.12.2014 19:34
	lib	😵 dtplite.tcl	04.02.2016 06:02
	licenses	💖 m4.exe	19.04.2021 19:33
	_ neenses	models m4	10.07.2021.10.22

- Now you can start "Curve Captor" from the desktop
- Prepare a tube curve copy from the manufacture data sheet max. 750 x 600 pix and as XXX.gif format (only this format accepts Curve Capture) Example below: 12AX7


• Save the created file into the path of ActiveTcl



Start Curve Capture and load the prepared curve copy

. **
curvecaptor.tcl - Shurtcut

urve Tracer	Device Data Device Model Spice	Code About						
riode 🔟	Vp axis Vp = 0	@ 100.0 [V] ir	ncrement			_		
	🦁 Öffnen							×
	\leftarrow \rightarrow \checkmark \uparrow \blacksquare « ActiveTcl >	12AX7	~ (U	Q			
	Organisieren 🔻 Neuer Ordner					-		?
	Lecer (C:)		<i>311.</i>					
	12AX7		11/1/1.					
	l bin	12	XXXXXX _					
	doc		12AX7.gif					
	lib							
	licenses							
	man	~						
	Dateiname: 12A	X7.gif						~
		_			Öffner	1	Abbrech	en

- Never enlarge the window with the left handle or you will start again
- With the bottom handle you can enlarge the window, so you can see the buttons (do not forget; this is still a Beta-Version ©).



- Set start point for "Volt-Plate Axis (V_P)" "Example below: V_P = 0 @ 50[V] increment"
- Start click with mouse pointer on Plate Voltage = 0 Volt,
- after marker placed to 0 Volt, value V_P = 0 jumps automatically to value V_P = $50 \rightarrow 100 \rightarrow 150 \rightarrow \text{etc.}$
- Place the marker precis, if not, no chance to delete the wrong marker and start the Curve Captor program again :-(
- · Help: to minimise errors marks, reduce mouse speed in the mouse settings



- Set start point for "Current-Plate Axis (I_P)" "Example below: I_P = 0 @ 5[mA] increment"
- Start click with mouse pointer on Plate Current = 0 mA,

after marker placed to 0 mA, value $I_P = 0$ jumps automatically to value $I_P = 0.5 \rightarrow 1.0 \rightarrow 1.5 \rightarrow \text{etc.}$



- Set start point for "Plate Curve (Vg)" "Example below: Vg = 0 @ -0.5[V] increment"
- Start click with mouse pointer on Plate Curve = 0 V, after marks placed on plate curve 0V until the end,

click on "increment" for the next plate curve of V_g = 0 jumps to value V_g = $-0.5 \rightarrow -1.0 \rightarrow -1.5 \rightarrow -2.0 \rightarrow$ etc.



Click "Save Markers" button (creates a file with all the markers
 Gurve Captor -- 12AX7
 Curve Captor -- 12AX7





Curve Capture displays the saved data points

•

Cllick "Build Model >>>" button Curve Captor -- 12AX7 _ × Curve Tracer Device Data Device Model Spice Code About 12AX7 0 -0.2862282951 0.01500923778 0 3.493303188 0.2552787391 5.493305100 0.253270391 8.02207036 0.4505189337 17.06507987 0.6383181463 29.11740501 0.8186544674 41.1702681 1.006497499 52.47335758 1.231863127 64.52783453 1.442226289 72.3266465 1.70514737 0 I 0 0 0 77.33854056 87.88755278 102.2026476 1.705147377 1.90798192 2.163418207 0 0 2.163418207 2.388772881 2.67419219 2.937157098 3.237578873 3.515513382 112.7532737 124.0606668 139.8812264 0 0 0 0 150.437232 163.249014 0 163.249014 3.515513382 -0.5 -0.285152386 0.03002265827 -0.5 17.78311419 0.1578996453 -0.5 37.35953528 0.3308388032 -0.5 55.43479526 0.5563030234 -0.5 96.86278009 1.149935641 -0.5 96.86278009 1.149935641 -0.5 115.7001866 -0.5 132.2753614 -0.5 148.8548398 -0.5 165.4327044 -0.5 184.2738766 -0.5 210.6520557 1.510531601 1.803534304 2.156590689 2.487126943 2.900269874 3.486176688 210.6520557 3.48617688 44.10964967 0.02316227551 64.43584437 0.1585788369 86.27019358 0.3390575692 105.8519942 0.5870638295 126.9387216 0.835091993 -1 -1 -1 -1 -1 Build Model >> Load Data Save Data Dismiss

Click "rydel5" button and check which "Device Model fits your original curves the best



Click button "Redraw" and see "bad fit" of model "korean6"



Get additional information, when enter: max. plat power, working point, load and AC input signal 🔞 Curve Captor -- 12AX7 Curve Tracer Device Data Device Model Spice Code About 12AX7 1.6 mA load 400000 R 🔽 AC signal input Rated po 1 W Working point: 250 V, (5 parameters): mean fit e 133945113,0.000103250243 0.0385833 mA Rydel mo 12AX7 ydel5(0.001 4 -2.5 ng poin -1.77 V 2198 W) 3.6 50db 00db 17db 51db 36db 73db 11db 04db 14db 3.2 -58 -68 -81 -90 Vg = -1.77V 2.8 -3.5 2.4 9 10 2 13 14 15 16 2 1.6 1.2 **400** 0.8 0.4 0 100 150 200 250 300 350 400 450 500 rvdel5 0.001133945113.0.0001032502434.91.73445445.54.19977619.2.336947589

.

 Get additional calculated information of desired working point, load and input signal <u>THD, Vg, Rk, AC output signal, power</u>

Dismiss

See Spice Code >>

Wor	Working point: 250 V, 1In fpontA1.5 V													
Bias	Bias: -1.77 V (Rk = 1103)utput: 133 V (0.02198 W)													
0.135% (0.123% second, 0.0531% odd, 0.00859%														
		0	24	19.1	8258	348V	1	88.2	2860	688		5	. 50	db
		1	13	32.0	6841	192V	1	00.0	0000	0008		0	.000	db
1		2		0.3	1637	71V		0.1	1234	1299		-58	.170	db
1		3		0.0	0703	322V		0.0	0529	9998		-65	.51	db
		4		0.0	0113	845V		0.0	0085	5518		-81	.360	db
58.2		5		0.0	0038	357V		0.0	0029	9078		-90	.73	db
•	35.5	6		0.0	0010)41V		0.0	0007	7849	-	-102	.110	db
		. 7		0.0	0002	296V		0.0	0002	234		-113	.040	db
		816		0.0	0000)82V		0.0	0000	624	-	-124	.14	db
		4	90.7	0.0	0000)23V		0.0	0000	178		-135	.26	db
{				102										
					113									
						124	135	146	158	169	180	192	203	215
2	3	4	5	6	7	8	9	10	11	12	13	14	15	16

Print

• Best fit of curves is model "rydel5"

Redraw



ľ	r 12AX7		-	
Curve Tracer 1 * 12AX7 mac .subckt 12A rydel5(Device Data Device Mode ro model X7 P G K 0.01203500602,0.002	Spice Code About	1712,48.377096	12AX
99635) .ends 12AX7				
* 12AX7 Spi .subckt 12A Bp P K 7021712)*V(35)) .ends 12AX7	ce 3F4 model X7 P G K I=((0.0120350060) G,K)+V(P,K)+(48.37)	2m)+(0.00121762801 709694))^1.5 * V(F	lm) *V(G,K)) *u1 P,K) / (V(P,K) + +	camp((90.5 (2.7419996
	Caise 2E4		Diamias	
	Spice 3F4		Dismiss	
	OrCAD PSpice			
	LT SwitcherCAD			
	Altium CircuitMaker			
Curve Tracer Curve	Altium CircuitMaker - Tspice Model a 12AX7 Vevice Data Device Model model (7 P G K 0.001133945113,0.000	Spice Code About	t editor pro 	ngram × 12AX7 19,2.33694
Curve Captor Curve Tracer [] * 12AX7 mac1 .subckt 12A3 ryde15 ((7589) .ends 12AX7 * 12AX7 LTS; .subckt 12A3 Bp P K 445445) *V (G, .ends 12AX7	Altium CircuitMaker TSpice Model a 12AX7 Pevice Data Device Model to model (7 P G K 0.001133945113,0.000 Pice model (7 P G K I= (0.001133945113 K) +V (P, K) + (54.19977)	nnd copy to tex Spice Code About 1032502434,91.7344 m)+(0.000103250243 619)) **1.5 * V(P,K	t editor pro 	egram × 12AX7 19,2.33694 amp ((91.73 36947589))
Curve Captor Curve Tracer [] * 12AX7 mac1 .subckt 12A2 .ryde15 ((7589) .ends 12AX7 * 12AX7 LTSF .subckt 12AX Bp P K 445445) *V (G, .ends 12AX7	Altium CircuitMaker Tspice Model a 12AX7 vevice Data Device Model co model (7 P G K).001133945113,0.000 cice model (7 P G K I = (0.001133945113 K)+V(P,K)+(54.19977)	nnd copy to tex Spice Code About 1032502434, 91.7344 m) + (0.000103250243 619)) **1.5 * V (P, K	tt editor pro 	bgram
Curve Tracer C Curve Tracer C * 12AX7 macr .subckt 12A3 .subckt 12A3 .subckt 12A3 * 12AX7 LTSp .subckt 12A3 Bp P K 445445)*V(G .ends 12AX7	Altium CircuitMaker Tspice Model a - 12AX7 vevice Data Device Model co model (7 P G K .001133945113,0.000 cice model (7 P G K T=(0.001133945113 K)+V(P,K)+(54.19977	nd copy to tex Spice Code About 1032502434, 91.7344 m) + (0.000103250243 619)) * * 1.5 * V (P, K	t editor pro → 5445,54.199776 4m) *V (G,K)) *ur)/ (V (P,K) + (2.3)	egram × 12AX7 19,2.33694 amp((91.73 36947589))
Curve Tracer C Curve Tracer C * 12AX7 mac1 .subckt 12A3 .subckt 12A3 .subckt 12A3 * 12AX7 LTSF .subckt 12A3 Bp F K 445445 * VA .ends 12AX7	Altium CircuitMaker Tspice Model a - 12AX7 vevice Data Device Model (7 P G K).001133945113,0.000 C P G K T=(0.001133945113 K)+V(P,K)+(54.19977)	and copy to tex Spice Code About 1032502434, 91.7344 m) + (0.000103250243 619)) **1.5 * V(P, K	t editor pro ↓ 5445, 54.199776 4m) *V (G, K)) *ur)/ (V (F, K) + (2.3)	gram × 12AX7 19,2.33694 amp((91.73 36947589))
Curve Tracer C Curve Tracer C * 12AX7 mac1 .subckt 12A3 .subckt 12A3 .subckt 12A3 * 12AX7 LTSF .subckt 12A3 Bp F K 4454451 *V.	Altium CircuitMaker Tspice Model a - 12AX7 vevice Data Device Model (7 P G K .001133945113,0.000 C P G K T=(0.001133945113 K)+V(P,K)+(54.19977	And copy to tex Spice Code About 1032502434, 91.7344 m) + (0.000103250243 619)) **1.5 * V (P, K	t editor pro ↓ 5445,54.199776 4m) *V (G,K)) *ur)/ (V (F,K) + (2.3)	gram × 12AX7 19,2.33694 amp((91.73 36947589))
Curve Tracer L Curve Tracer L * 12AX7 mac1 .subckt 12A3 .subckt 12A3 .ends 12AX7 * 12AX7 LTSSF .subckt 12A3 Bp P K 445445) *V(G, .ends 12AX7	Altium CircuitMaker Tspice Model a 12AX7 vevice Data Device Model co model (7 P G K).001133945113,0.000 cice model (7 P G K I = (0.001133945113 K) + V(P,K) + (54.19977)	and copy to tex spice Code About 1032502434,91.7344 m) + (0.000103250243 619)) **1.5 * V(P,K	t editor pro ↓ 5445, 54.199776 4m) *V (G, K)) *ur)/ (V (F, K) + (2.3)	gram × 12AX7 19,2.33694 amp((91.73 36947589))
Curve Tracer [Curve Tracer] * 12AX7 mac1 .subckt 12A3 .subckt 12A3 .ends 12AX7 * 12AX7 LTSS .subckt 12A3 Bp P K 445445) *V(G, .ends 12AX7	Altium CircuitMaker Tspice Model a 12AX7 vevice Data Device Model co model (7 P G K).001133945113,0.000 cice model (7 P G K I = (0.001133945113 K) + V(P,K) + (54.19977)	and copy to tex spice Code About 1032502434,91.7344 m) + (0.000103250243 619)) **1.5 * V(P,K	t editor pro ↓ 5445,54.199776 4m) *V (G,K)) *ur)/ (V (F,K) + (2.3)	gram × 12AX7 19,2.33694 amp((91.73 36947589))
Curve Tracer L Curve Tracer L * 12AX7 mac1 .subckt 12A3 .subckt 12A3	Altium CircuitMaker -TSpice Model a 12AX7 tevice Data Device Model co model (7) P G K 0.001133945113,0.000 cice model (7) P G K 1= (10.001133945113 K) +V(P,K) + (54.19977)	and copy to tex spice Code About 1032502434,91.7344 m)+(0.000103250243 619))**1.5 * V(P,K	t editor pro → 5445, 54.199776 4m) *V(G, K)) *ur)/ (V(P, K) + (2.3) Dismiss	gram □ × 12AX 19,2.33694 amp((91.73 36947589))

- Save tube model file as example XXXX.inc ٠ / *Uni Date: Bearbeiten Format Ansicht Hilfe * 12AX7 LTSpice model .subckt 12AX7 P G K Bp P K I=((0.001133945113m)+(0.0001032502434m)*V(G,K))*uramp((91.73445445)*V(G,K)+V(P,K)+(54.19977619))**1.5 * V(P,K)/(V(P,K)+(2.336947589)) .ends 12AX7 Speichern unter × $\leftarrow \rightarrow \checkmark \uparrow |$ ActiveTcl > 12AX7 ・ ひ P "12AX7" durchsuchen Organisieren 👻 Neuer Ordner -? ActiveTcl 12AX7 📕 bin 📕 doc 📕 include 12AX7.crv 12AX7.gif 📕 lib Zeile 4, Spalte 12 100% Windows (CRLF) UTF-8 licenses 📜 man Dateiname: 12AX7.inc Dateityp: Alle Dateien (*.*) Codierung: UTF-8 Speichern
 Abbrechen Ordner ausblenden
- Compare original tube corves and tube model of Curve Creator
 Eile Edit Hierarchy View Simulate Tools Window Help



2. Triode Tube-Models created with "Model Paint Tools" (Dmitry Nizhegorodov)

• Download-link to the triode program: http://www.dmitrynizh.com/paint_kit.jar

Sie möchten folgende Datei öffnen:			
🕌 paint_kit.jar			
Vom Typ: JAR-Datei (190 KB) Von: http://www.dmitrynizh.com			
Möchten Sie diese Datei speichern?			
	Datei speichern	Abbrechen	

• Find program file in "Downloads-Folder" (if possible, virus check !)

🛃 paint_kit.jar 15.02.2020 14:02 Executable Jar File 190 KB

• Drag file on desktop for easy start up



Start program



• Tube specification (KR 300BXLS) Power Amplifier class A1

Maximum DC Plate Voltage Dissipation DC Plate Current	600 V 70 W 160 mA
DC Plate Voltage	450 V
DC Plate Current	100 mA
Grid Voltage	-94 V
Plate Resistance	650 ohm
Transconductance	5,7 mA/V



• Capture tube curves from datasheet



• Enter tube curve datasheet values into the boxes





- Open "Control, Options, Settings" page •
- Zoom in the past curves with "Zoom Less Button" •

Model	DHT	✓ XY Grid	
Vg from	-20 decr	20 ount 6 Vp max 600 lp max 250	Scontrols, Options, Settings X
✓ Plate 0	Current A	2 Grid Current 🗌 lg Level 2	Help About Paste BG Image, import Model text
MU	4.2 I		
KG1	1500		
KP	200		Bacground Image Colors:
KVB	300		brightness: Less More
VCT	0.2		
EX	1.4		contrast: Less More
RGI	2000	▲	Invert Colors Revert
VGOFF	-0.6	4	
IGA	0.001	∢	
	0.3	∢ ▶	Bacground Image Geometry:
	8	< ►	Zoom Less More
IGEX	2		Datata 00 dagaOlastaviasDask
Dissip	ation		Rotate 90 degr Clockwise Back
Results			Rotate 1/10 degr Clockwise Back
Pmax W	40		Shear Left Right
Rp ohm	4000	4	Devert geometry to existing
Vin ampl	1 55	4	Revent geometry to original
Vg qui	-48	4	
Vp qui	300	4	Plot lines Narrower Wider Revert to deault
Asym	1	4	
CCG 3	CGP	1.4 CCP 1.9 RFIL 10	
Script	600 200 1	2	

Move and resize the two curves until they match each other • Paint KIT: Sync Image and Model



Try with the three most important slide controls to match the curves (it's not an easy task !) • Paint KIT: Sync Image and Model Model DHT V XY Grid



_

×

_

- Slip the original tube curves out of the view (I didn't find out how to delete the original curve picture)
- Enter value of the max. plate dissipation (Pmax W)
- Enter the chosen value of the primary transformer impedance (or plate resister of the small signal tube)
- Enter the working point of the tube ("Vg qui" and "Vp qui") and the input AC-signal ("Vin ampl")
- If necessary change colors of the curves
- Check the working point, check input- and output-signal
- Enable "XYProj-Box to see the input- and output-span



· Check the calculated tube specification, working point and distortion parameters



- Create the LTspice "tube model" of the tube KR 300B XLS
- Enter the tube system capacities "CCG", "CGP" and "CCP" if necessary

🛓 Paint KIT: Syn	ic Image and Model					_		×
		Model	DHT	⊭ XY	Grid			Ξ
250mA 240mA	-20 40 -60 -80 +100	Vg from	-20 secr	20 0	ount 6	Vp max 60	0 Ip max	250
220mA		Plate (Current 🖌	42	Grid C	urrent [] Ig Level 2	
200mA	SPICE Model TEST - X / -120	MU	3.696	•				
100m 179.5		KG1	1515	•				
	••••• TEST ••••••	KVB	200	4				
16UMA	* Created on (using paint_kit.jar 3.1	VCT	0.204					- P
140mA	Plate Curves image file:	EX	1.4	4				•
120mA	Data source link: -70	RGI	2000	•				
100mA 99.3	SUBCKT TRIODE_TEST 1 2 3; Plate Grid Cathode	VGOFF	-0.6					
80mA	+ MU=3.696 KG1=1515 KP=200 KVB=300 VCT=0.204 EX=1.4	IGA	0.001	•				•
60må	* Vp_MAX=600 lp_MAX=250 Vg_step=20 Vg_start=-20 Vg_count=6 * Rp=2000 Vg_ac=55 P_max=70 Vg_gui=-60 2 Vp_gui=302 26	IGB	0.3	•				
	* X_MIN=53 Y_MIN=47 X_SIZE=762 Y_SIZE=346 FSZ_X=1480 FSZ_Y=682 XYGrid=true	IGC	8	4				> ·
40mA	* showLoadLine=y showip=y isDH1=h isPP=h isAsymPP=h showDissipLimit=y * showlg1=h gridLevel2=h isInputShapped=h	Dissin	ation			_ unio	ock SPICE n	arams
20mA	* XYProjections=y harmonicPlot=n dissipPlot=n	Loadli	ne for loa	d Rp	Vg sr	oan 🗌 Diss	ipation Plot	
50	E1 7 0 VALUE={V(1,3)/KP*LOG(1+EXP(KP*(1/MU+(VCT+V(2,3))/SQRT(KVB+V(1,3)*V(1,3)))))}	Results	× ×	(YProj	HPlot	PP	Asym	
mouse no	RE17.0.1G ; TO AVOID FLOATING NODES G1.1.2.VALUE=//PIMPO/(7) EYV/PIMPSO/(7) EYV/KG1)	Pmax W	70	•				Þ
modoc po	RCP 1 3 1G ; TO AVOID FLOATING NODES	Rp ohm	2000	4				•
	C1 2 3 (CCG); CATHODE-GRID C2 2 1 (CGP): GRID=PLATE	Vin ampl	55	•			Loc I	•
	C3 1 3 (CCP); CATHODE-PLATE	Vg qui	-60.2	4		Lac.		•
	R125 (RGI) ; POSITIVE GRID CURRENT	Asym	1					
	MODEL DX D(IS=1N RS=1 CJ0=10PF TT=1N) ENDS **	CCG 0	CGP	0	CCP 0		RFIL 1	0
	* < >					_		
	Update and create LTSpice 'JIG' Update PARAMS	Script	600 200 1	12				

- Copy the "tube model" text into a "editor program"
- Enter the correct tube model name "300BXLS", so LTspice can find it when simulate the amplifier

- * Created on XX/XX/XX XX:XX using paint kit.jar 3.1
- * http://www.dmitrynizh.com/tubeparams_image.htm
- * Plate Curves image file:
- * Data source link:

*_____.SUBCKT 300BXLS 1 2 3 ; Plate Grid Cathode

- + PARAMS: CCG=0P CGP=0P CCP=0P RGI=2000
- + MU=3.696 KG1=1515 KP=200 KVB=300 VCT=0.204 EX=1.4
- * Vp_MAX=600 lp_MAX=250 Vg_step=20 Vg_start=-20 Vg_count=6
- * Rp=2000 Vg_ac=55 P_max=70 Vg_qui=-60.2 Vp_qui=302.26
- * X_MIN=53 Y_MIN=47 X_SIZE=762 Y_SIZE=346 FSZ_X=1480 FSZ_Y=682 XYGrid=true
- * showLoadLine=y showIp=y isDHT=n isPP=n isAsymPP=n showDissipLimit=y
- * showlg1=n gridLevel2=n isInputSnapped=n
- * XYProjections=y harmonicPlot=n dissipPlot=n

E1 7 0 VALUE={V(1,3)/KP*LOG(1+EXP(KP*(1/MU+(VCT+V(2,3))/SQRT(KVB+V(1,3)*V(1,3)))))} RE1 7 0 1G ; TO AVOID FLOATING NODES G1 1 3 VALUE={(PWR(V(7),EX)+PWRS(V(7),EX))/KG1} RCP 1 3 1G ; TO AVOID FLOATING NODES C1 2 3 {CCG} ; CATHODE-GRID C2 2 1 {CGP} ; GRID=PLATE C3 1 3 {CCP} ; CATHODE-PLATE D3 5 3 DX ; POSITIVE GRID CURRENT R1 2 5 {RGI} ; POSITIVE GRID CURRENT .MODEL DX D(IS=1N RS=1 CJO=10PF TT=1N) .ENDS *\$

• Save the tube model into your tube amplifier project folder

*300BXLS.ini - Editor Datei Bearbeiten Format Ansicht Hilfe * Created on XX/XX/XXXX XX:XX using paint kit.jar 3.1 www.dmitrynizh.com/tubeparams_image.htm * Plate Curves image file: * Data source link: .SUBCKT 300BXLS 1 2 3 ; Plate Grid Cathode + PARAMS: CCG=0P CGP=0P CCP=0P RGI=2000 + MU=3.696 KG1=1515 KP=200 KVB=300 VCT=0.204 EX=1.4 Vp_MAX=600 Ip_MAX=250 Vg_step=20 Vg_start=-20 Vg_cou Rp=2000 Vg_ac=55 P_max=70 Vg_qui=-60.2 Vp_qui=302.2(X_MIN=53 Y_MIN=47 X_SIZE=762 Y_SIZE=346 FSZ_X=1480 F showLoadLine=y showIp=y isDHT=n isPP=n isAsymPP=n sH showIg1=n gridLevel2=n isInputSnapped=n * XYProjections=y harmonicPlot=n dissipPlot=n E1 7 0 VALUE={V(1,3)/KP*LOG(1+EXP(KP*(1/MU+(VCT+V(2,3) RE1 7 0 1G ; TO AVOID FLOATING NODES

- Push button "Update and create LTspice "JIG"
- Creates an icon on the desktop
- Click the icon and program "Explorer" open automatically and click the file "transient_jig.asc



LTspice open and presents a test schematic with the new created 300BXLS tube model

伊 LTspice XVII - [transient_jig.asc]	-]	\times
🔸 Eile Edit Hjerarchy View Simulate Iools Window Help			-	ъ×
(1) (1) (1) (1) (1) (1) (1) (1) (1) (1)	0	C É	-	Aa
		- L.		
.param Vp_MAX=600 lp_MAX=225 Vg_step=20 Vg_start=0 Vg_count=7 Rp=3000 Vg_ac=50 P_max=65 Vg_qui=-60 Vp_qui=315		1.1		
param tick=0 ac=1 B+ Out Distortion measuring 'mobe'	1 -	• •	• • •	
param bpLs=f(ac,Vp_qu)3*Vp_qu) [] jparam (b=100H KLLL2] [] jparam (b=100H KLL12] [] [] jparam (b=100H KLL12] [] [] [] [] [] [] [] [] [] [] [] [] [
$ \begin{array}{c} \downarrow \downarrow \{\mathfrak{n}(\mathfrak{a},\mathfrak{c},\mathfrak{l},\mathfrak{m})\} \\ & \qquad \qquad$	1.1	• •		
(intersection) → Step param ac list 0.1		1.1		
	1.1	• •	• •	
		1.1	111	
		• •	• • •	
tran 0.5ms 100µ 20µ		1.1	111	
Comment out the step commands'		• •		
when swtching to AC or DC analyss Vg SINE(/if(ac,Vg_qui,Vg_start-tick*Vg_step)) {Vac} 1khz)				
Ac Vbp 083 Vbp 0-120-12 AC (Vac) Abs the following is not possible because A	ep con count}	trols pa {-vo_s	rms, r tep}	iot vise v
the following absides not work in spice		1.1		
		· · ·	• • •	
		÷ ; ;		
TEST TEST		• •	• • •	
 Created on using paint_wttpi 3.1. * www.dmttynizh.com/tubeparams_mage.htm 		11		
* Plate Curves image file:			• • •	
The second			111	
.SUBKATINUOUE_IESTIZZ; Pate ond Cathode + PARAMS:CCG=0P:CGP=0P CCP=2R20;		• •	• • •	
+ MU=3.99 KG1=1290 KP=430 KVB=420 VCT=0.1019 EX=1.47		1.1		
* Rp=3000 Vg_ac=50 P_max=65 Vg_qui=-60 Vp_qui=315				
A multipol t_multipol x_JL2=7800 J_SL2=7810 FS_X=1311+52_1=061 AY000=000 * showLoad(lne=y showLp=y) s0HT=n is BPP=n isAymPP=n showDsspLimt=y * showLoad(lne=y) showLp=y) s0HT=n isAymPP=n showDsspLimt=y * showLoad(lne=y) showLp=y) * showLp=n showLp=n showLp=n showDsspLimt=y * showLp=n showLp=n showLp=n showLp=n showLp=n showLp=n * showLp=n showLp=n showLp=n showLp=n showLp=n * showLp=n showLp=n showLp=n showLp=n * showLp=n showLp=n * showLp=n showLp=n * show				
* showlg1=n gridLevel2=n isInputSnapped=n * YVDroist-ficesna harmonicPlate_n diseblate_n				
	• •	• •	• • •	
E1 7 0 VALDE={V(1,3)/KV*LOG(1+EXP(KV*1/MU+V(1+V(2,3))/SQR((KVB+V(1,3)*V(1,3))))} REI 7 0 1G (* TO AVOID FLOATING NODES			1.1.1	
6.1.3 VALUE={(PWR{V(7),EX)+PWRS(V(7),EX)}}/KG1}- PCP.1.3.16 - TO AVDD F10.4TM/S NOPES		• •		
CI 2 3 (CCG) ; CATHODE GRID				
C2 21 (CGP); GRID=PLATE C3 13 (CCP); CATHODE-PLATE	• •		• • •	
D3 5 a DX; POSITIVE GRID CURRENT B1 25 4 GGID - POSITIVE GRID CURRENT	1.1	1.1		
	• •			
CENDS **		1.1		

- Click "Simulation Button" and click the out coil with the mouse •
- See the stepped output-signal •



3. Pentode Tube-Models created with "Model Paint Tools" (Dmitry Nizhegorodov)



Download-link to the triode program: http://www.dmitrynizh.com/paint_kip.jar ٠

- Sorry, I tried many times to create pentode curves for KT88, EL84, EL34 etc. but didn't get it, maybe you do can !

4. Tube-Models created with µTracer and ExtractModel (Ronald Dekker, Derk Refman)

- × 🖏 u-Tracer (V3.11) Measurement Set-Up ----Curve Output Select Measurement type: I (Va=Vs , Vg) with Vh constant • la (mA) 12AX7_Sylvania_04 chruesy.chu-Tracer V3.11 4
 Start
 Stop
 Nintervals

 Va = Vs
 2
 400
 30
 □
 log
 3.5 Vg 0 -0.5 3 Stepping Variable (e.g: 50 100 150 200): 2.5 Vg 0-0.5-1-1.5-2-2.5-3-3.5-4-4.5-5 -1.5 2 Constan 1.5 6.3 Vh [pins 1 Range Preax (W) 0.5 la Automatic - Is Automatic -0 Compliance Average Anode and Screen [V] 4× • 200 mA • Axis Scale - Style -Min. Max Ticks Delay: Manual ▼ 0 400 ∨ 8 X Va=Vs Abort ! Measure Curve Y1 la V Solid V Menual V 4 mA 8 0 □ Beep Quick Test • × • Dots ▼ Manual ▼ Y2 none 8 Miscellaneou Keep Plot Distortion Clear Pmax: 1 W PolyDegree: 3 Debug Cal. Save Data Save Plot Open Setup Save Setup Ext. heater
- 12AX7 measured with µTracer, only one tube-systems measured ٠

12AX7 very bad tube example, the two systems create different curves ! •



12AX7 measured Data of µTracer ٠

12AX7_Sylvania_04_single.utd - Edbor _ Datei Bearbeiten Format Ansicht ? Point Curve Va (V) Ia (mA) Is (mA) Vg (V) Vs (V) Vf (V) 0.11 0.01 0 2.91 3.07 6.3 1 1 0.01 13.71 13.95 2 1 0.26 0 6.3 0.01 0.48 0 25.82 27.02 3 1 6.3 4 0.73 0.01 39.22 40.51 1 0 6.3 5 0.99 0.01 0 52.62 53.58 6.3 1 6 1.27 0.01 0 65.16 66.2 6.3 1 7 0.01 0 79.27 1.58 78.56 6.3 1 8 1.89 0.01 0 91.96 93.2 1 6.3 9 2.18 0.01 0 104.49 105.83 1 6.3 10 2.54 0.01 0 117.89 119.32 1 6.3 11 2.84 0.01 0 131.72 132.39 1 6.3 12 1 3.21 0.01 0 144.69 145.88 6.3 13 1 3.6 0.01 0 158.52 159.38 6.3

X

- ExtractModel installation
- Program "Gnuplot" is necessarily to create visual control of the tube-model <u>https://sourceforge.net/projects/gnuplot/files/gnuplot/</u>

SOURCE F	ORGE				lelp
Open Source Software	Business Software	Services	Resources	y	f

Home / Browse / O	Graphics / Gra gr A port Broug	aphics/gnuplot/F TUPIC table, multi-pl tht to you by: I	iles) t latform, com broeker, cga	nmand-line drivy ylord, lhecking,	en graphing utili Sfeam	ty	
Summary	Files	Reviews	Support	Tickets •	gnuplot-main	Mailing Lists	•••
Home / gnut	wnload La ?7-win64-min plot	test Version gw.exe (27.5 MB)		Get Updates)
Name 🖨				Modified	🗧 Size 🖨	Downloads / '	Week 🗢
🕽 Parent fo	lder						
5.2.7				2019-05-3	D	3,	875 🖬
testing				2019-05-2	2	15	
5.2.6				2019-02-2	1	94 📃	
5.2.5				2018-10-0	в	20	

Save into separate folder

ExtractModel_3p0.zip

∽ 📙 gnuplot	📜 bin	01.07.2016 18:22
> 📜 bin	docs	01.07.2016 18:22
docs	📜 etc	01.07.2016 18:22
> 📜 etc	license	01.07.2016 18:22
license	share	01.07.2016 18:22
> 📙 share	NEWS	22.02.2016 17:01
	README-Windows.txt	14.02.2016 01:51
	RELEASE_NOTES	22.02.2016 17:01
	 unins000.dat 	01.07.2016 18:22
	unins000.exe	01.07.2016 18:21

Open ExtractModel webpage from Derk Reefman
 <u>https://www.dos4ever.com/uTracer3/uTracer3_pag14.html</u>





My friend and colleague <u>Derk Reefman</u> has spent a lot of effort on improving the modeling of especially pentodes and beam pentodes. His improved models include secondary emission, modeling of beam pentodes, and improved screen and anode current modeling. Next to that he wrote an intelligent parameter extraction program (https://www.screen.org anode current modeling. Next to that he wrote an intelligent parameter extraction program (screen.org anode current modeling. Next to that he wrote an intelligent parameter extraction program (screen.org anode current modeling. Next to that he wrote an intelligent parameter extraction program (screen.org anode current modeling. Next to that he wrote an intelligent parameter extraction program (screen.org anode current modeling. Next to that he wrote an intelligent parameter extraction program (screen.org anode current modeling. Next to that he wrote an intelligent parameter extraction program (screen.org anode current modeling. Next to that he worde anote he program uses elaborate algorithms to find the best initial parameter values to obtain the best and also physical model fits. The program directly interfaces with the data files generated by the uTracer, and happily Derk was kind enough to share the model and the parameter extraction is an anotation of the screen.

Download ExtractModel . Click Here to Download ExtractModel, Examples and References D The download contains the latest version of ExtractModel version 3.0 (February-2016). Save into separate folder ExtractMode Nan Änderungsdatum Examples ExtractModel.exe 23.05.2017 13:09 ExtractModel_3p0 23.05.2017 13:09 Theory.pdf References

- Create new folder for tube-module 12AX7
- Copy tube-curve µTracer data into folder (XXXXXX.utd)
- Copy Extract Model program into folder (ExtractModel.exe)
- Copy model.ini from other tube examples into folder (Model.ini)



- Open Model.ini file with editor
- Change all the data like in the screen shot below
- For more information see ExtractModel manual



Click on file ExtractModel.exe to start program

•

Examples	^	Name	Änderungsdatum	Тур
12AX7_Sylvania		12AX7.utd	16.08.2016 15:48	UTD-Dat
ECC81 example		ExtractModel.exe	09.08.2016 12:50	Anwendu
📜 EF85 example		🚮 Model.ini	23.05.2017 17:20	Konfigur
El 500 evample		_		

• After start of program press "Return"

E\Electronic\Computer\Programme\Curve Captor\ExtractModel\Examples\12AX7_Sylvania\ExtractModel.exe	-	×
*******		^
Extract V3.000 Jan 23, 2016		

Initialization file is:Model.ini		
Data will be read from file 12AX7(.utd) Tube model is triode.		
Using Koren model		
Fitting model to max power data < 1.000000 W. Vg offset equals 0.000000E+00		

Total number of datapoints in the fit equals: 283. This is 82.99% of the total number 341 of datapoints.		
11111		
For a triode, Ia(Va=0)/Ia(Va=Va,max)should be less than 5.000000E-03 For a good fit of the cut-off region.		
Press return to continue.		
		\sim



• Program-Information -→ just press "Return"

$\blacksquare E\Electronic\Computer\Programme\Curve\Captor\ExtractModel\Examples\12AX7_Sylvania\ExtractModel.exe$	-	\times
*********		^
Extract V3.000		
Jan 23, 2016		
Initialization file is:Model.ini		
Data will be read from file 12AX7(.utd)		
Tube model is triode.		
Fitting model to max power data < 1.000000 W.		
Vg offset equals 0.000000E+00		

Total number of datapoints in the fit equals: 283.		
!!!!!! For a triode Ta(Va=0)/Ta(Va=Va max)should be less than 5 000000E-03		
For a good fit of the cut-off region.		
Press return to continue.		
Saturation effects detected for datafile 1 for Vg = .0.		
Saturation effects detected.		
Fit will continue - but results may be untrustworthy. Hit return to continue.		
		\sim

• ExtractProgram calculate the Korean LTspice code

Extract V3.000
Jan 23, 2016
Initialization file is:Model.ini
Data will be read from file 12AX7(.utd)
Tube model is triode.
Using Koren model
Fitting model to max power data < 1.000000 w. Vg offset equals 0.000000E+00

Total number of datapoints in the fit equals: 283.
This is 82.99% of the total number 341 of datapoints.
$\frac{1}{1}$
For a good fit of the cut-off region.
Press return to continue.
Saturation effects detected for datafile 1 for Vg = .0.
Saturation effects detected.
Fit will continue - but results may be untrustworthy.
Hit return to continue.
ana cara contante conta
kVB = 8.468101
kG1 = 562.428500
kp = 50.00000
ex = 1.226632
mu = 89.1/8250
Before refinement $R_2 = 1.058203E-02$
Iter: 1, 2, 3, 4, 5, 6, 7, 8, 9,10,11,
After refinement R2 = 1.689793E-07
Spice Triode subcircuit written to file 12AX7.cir
Please aut capacitances yoursell?
Elapsed time: 5.0s
Final estimates:
RVB = 2278.915000
kg = 568.71400
ex = 1.193594
mu = 93.103400
Standard variances (in %) for each parameter:
kVB = 5.452899
kG1 = 3.963895E-01
Rp = 1,198544
$ex = 3.05276^{-01}$ $m_1 = 1.493413^{-01}$



- · At the same time ExactModel Program saves different files into the folder
- 12AX7.cir contains the Tube-Model for LTspice



Open 12AX7.cir file and if necessarily add the Tube-Capacities

all 12AX7.cir - Editor - E) ×
Datei Bearbeiten Format Ansicht ?	

.SUBCKT 12AX7 1 2 3; A G C;	
* Extract V3.000	
* Model created: 24-May-2017	
*	2000
AT I Z 3 THIOUEK MUE 92.90 EAEL.180 KGIE 524.4 KPE 890.3 KVBE 2114. KGIE	2000
ENDS	

.SUBCKT TriodeK 1 2 3; A G C	
* $ 0TE_1 0C(y) $ is base a $ 0C $ on natural loganithm	
* For some Spice versions, e.g. MicroCap, this has to be changed to IN(x)	5
*	<i>)</i> .
E1 7 0 VALUE=	
+{V(1,3)/KP*LOG(1+EXP(KP*(1/MU+V(2,3)/SQRT(KVB+V(1,3)*V(1,3)))))}	
RE1 7 0 1G	
G1 1 3 VALUE={0.5*(PWR(V(7),EX)+PWRS(V(7),EX))/KG1}	
RCP 1 3 1G ; TO AVOID FLOATING NODES IN MU-FOLLOWER	
$(2 2 3 \{CCG\}, CATHODE-GRID$	
C3 1 3 {CCP} : CATHODE-PLATE	
D3 5 3 DX ; FOR GRID CURRENT	
R1 2 5 {RGI} ; FOR GRID CURRENT	
.MODEL DX D(IS=1N RS=1 CJO=10PF TT=1N)	
.ENDS TriodeK	



Tube-Curve Contest

1. Norman L. Koren

Curves above: 12AX7 Library "Tube1.lib" (Curves created with LT-Spice)
Curves below: original Data-Sheet of 12AX7 Sylvania



2. S. Bench/Ducan

Curves up: 12AX7 Library "triode_nh.inc" (Curves created with LT-Spice)
Curves bellow: original Data-Sheet of 12AX7 Sylvania



3. Rydel

Curves up: 12AX7 Tube-Model created with "Curve Captor Program" (Andrei Frolov) according to the "Rydel5 Tube-Model" and original Data-Sheet of 12AX7 Sylvania (Curves created with LT-Spice)
 Curves bellow: original Data-Sheet of 12AX7 Sylvania



4. Dimitry Nizhegorodov

Curves up: 12AX7 Library "Dmitry_composites.lib" (Curves created with LT-Spice)
Curves bellow: original Data-Sheet of 12AX7 Sylvania



5. Ronald Dekker, Derk Refman

- ٠
- Curves up: original Data-Sheet of 12AX7 Sylvania Curves middle: 12AX7 "Tube-Model created out of original "µTracer-Curves" data (Curves created with LT-Spice) •
- Curves bellow: original screen shot of measured "µTracer-Curves" of 12AX7 Sylvania tube •



Conclusion: Hmmm..... which tube model is the best ?

Transformers

1. General

- The producer publishes normally only the value of the primary and secondary impedance but not how they have measured or calculated the specification.
- It exists no specification standard of audio transformers, like by tubes.
- By itself, the output transformer has no impedance. Simply it reflects the secondary impedance back to the primary. This means the impedance between the loudspeaker and the tube(s) and vice versa. So the most important parameter is the Turns Ratio *N*.
- The primary inductance L_p value has a direct effect on the low frequency response of the transformer. If you increase the primary inductance, you will get a better low frequency response. That means larger core and/or more turns of the primary winding = heavier transformer. But more windings means also higher primary interwinding capacitance C_p and leakage inductance L_{pf}.
- On the other hand, higher winding capacitance *C_p* and leakage inductance *L_{pf}* will adversely affect the high frequency response.
- The minimum parameter to simulate a transformer with LTspice:
 - Turns Ratio N (this value is very important to calculate L_p / L_s ratio)
 - **Prim./sec. Inductance** L_p / L_s (the exact values are not so important but the ratio, if available, use the manufacture specification value, if not, use the value of about an equal transformer)
 - **Coupling Coefficient** *K* (the closer the coefficient is to "1", the more efficient or better the transformer is)
 - **Prim. DC-Resistor** *R*_p (important for the anode voltage level)
 - Sec. DC-Resistor R_s
- The high quality LTspice transformer model (CHAN-Model):
 - This Model includes all non- and linear-values of the transformer parameters: leakage capacity, magnetic saturation and hysteresis etc. The audio transformer manufactures don't publish all the required transformer parameter to create a real audio-transformer CHAN-Model.
- Conclusion:
 - The simulation of an audio transformers will be always "so so la la" because of the missing data.
 - I guess you might seen this article on transformer modeling http://www.beigebag.com/case_xfrmer_1.htm

2. Elements of a simple Audio Transformer



- **Z**_p = Primary Impedance (Ω) normally get this value from the producer to hit the tube working curve.
- L_p = Primary Inductance (H) is an important value for the low frequency response and is a must for the LTspice simulation.
- V_p = Primary Voltage (V) value can be used to calculate turns ratio quotient or primary inductance (see "Measuring of an Audio Output Transformer").
- R_p = Primary DC-Resister (Ω) of the winding is a must to simulate the value of the anode voltage.
- N_p = Primary number of windings loops and is necessary to calculate turns ratio quotient.
- **C**_p = Primary inter winding capacitance (F) is not easy to measure and dependent much on which winding technic is being used.
- I_p = Primary current (A) can be used to calculate primary inductance (see: "Measuring of an Audio Output Transformer").
- L_{pf} = Primary Leakage Inductance (H) shows the primary winding loss of magnetic energy which never will be transferred to the secondary side of the transformer. A must value to calculate the coupling coefficient K for the LTspice simulation.
- $Z_s =$ Secondary Impedance (Ω) is normally the same value like the loudspeaker value.
- L_s = Secondary Inductance (H) is a must value for the LTspice simulation.
- V_s = Secondary Voltage (V) value can be used to calculate turns ratio quotient or secondary inductance (see: "Measuring of an Audio Output Transformer").
- R_s = Secondary DC Resistor (Ω) of the windings and is not important for the LTspice simulation.
- N_s = Secondary number of windings loops is necessary to calculate turns ratio quotient.
- **C**_s = Secondary inter winding capacitance (F) is not easy to measure and depends much on which winding technic is used.
- Is = Secondary current (A) is not important for LTspice simulation.
- L_{sf} = Secondary Leakage Inductance (Henry) is usually a very small value and not important for the LTspice simulation.
- \mathbf{R}_{s} = Secondary Resistance (Ω)
- C_{ps} = Leakage Capacitance (F) of primary to secondary windings, measured between shorted primary and shorted secondary windings with an 4-wire bridge
 If there is a shield between primary and secondary windings, there exists also no C_{ps} leakage capacitance.
 - Important value resulted, when high frequency bandwidth of the transformer is simulated.
- **Prim./Sec. Shield** = Shield between the primary and secondary windings connected to ground, eliminates the leakage capacitive coupling
- Magnetic Flux (Φ) = is the magnetic flux density (Tesla) which runs through an imagined area
- **Transformer Core** = is a piece of magnetic material with a high magnetic permeability, used to guide magnetic fields and is made of ferromagnetic metal or ferrites
- Core Air Cap = single ended transformer need a large air cap to avoid core saturation, because of the high DC current flow. Push pull transformer have also a small air cap because of tubes current imbalance

3. Measuring of an Audio Output Transformers

Attention: This way of measuring Output Transformers can kill you! You have to know what you are doing. Do it on your own risk !



- Measuring Inductance with an LCR-Bridges generates very little magnetic flux in the core of the transformer and the measured values are not correct
- The most regular, simple but danger method to measure the value of an unknown output transformer is:
 - As source, use a variable transformer (normally: 50 to 100VAC).
 - Connect Meters V_p , I_p and V_s like in the schema.
 - V_p and I_p = measure V_p and I_p and V_s (measure I_s it's not necessary).
 - Short the secondary side of the transformer.
 - V_p and I_p = measure again V_p and I_p .
 - R_p = measure primary winding DC-resistor if possible with a 4-wire bridge-meter.
 - *C_{ps}* = measure the leakage capacitance with a 4-wire bridge, primary and secondary winding are separately shorted.
- Calculate Specifications:
 - Z_{mo} = Impedance (Ω) primary winding measured with open secondary winding.

$$Z_{mo} = \frac{V_p}{I}$$

- \mathbf{Z}_{mf} = Leakage Impedance (Ω) primary winding measured with short circuit secondary winding(s).

 $Z_{mf} = \frac{V_p}{I}$ (this value is usually 100th to 1000th of Z_{mo})

- N = Turns Ratio = the quotient of the primary and secondary windings

- K = Coupling Coefficient from primary to secondary windings

$$K = \sqrt{1 - \frac{Z_{mf}}{Z_{mo}}}$$

 V_p

- \mathbf{Z}_{pf} = primary leakage Impedance (Ω)

$$Z_{pf} = (1 - K) \cdot Z_m$$

- $\mathbf{Z}_{\mathbf{p}}$ = primary Impedance (Ω)
- $Z_p = K \cdot Z_{mo}$
- $Z_s = secondary Impedance (\Omega)$

$$Z_{\rm s} = K \cdot Z_{mo} \cdot N^2$$

 $L_p = \frac{Z_p}{2\pi \cdot f}$ (use measured and calculated Z_p and 50Hz (60Hz)

$$L_s = \frac{L_p}{N^2}$$

4. Calculation of Transformer Parameters



- N Turns Ratio Quotient
 - Turns ratio is the quotient of the primary and secondary windings.
 - The calculation of the ratio quotient on the base of the loops windings is more precise then on the base of impedance or inductance.

 N_p = Number of loops of the primary windings

 N_s = Number of loops of the secondary windings

$$N = \frac{V_p}{V_s} = \frac{N_p}{N_s} = \frac{I_s}{I_p} \text{ or } N = \sqrt{\left(\frac{L_p}{L_s}\right)} \text{ or } N = \sqrt{\left(\frac{Z_p}{Z_s}\right)}$$
$$L_s = \frac{L_p}{N^2}$$
$$L_p = N^2 \cdot L_s$$

- $\mathbf{Z}_{\mathbf{P}}$ Primary Impedance (Ω)
 - The primary impedance should have the same value as the selected working straight line of the tube
 All the audio-transformer producer publish the primary impedance Z_P, but they don't tell how they have measured the impedance.

$$Z_P = \frac{V_p}{I_p}$$
$$Z_P = N^2 \cdot Z_s$$

- LP Primary Inductance (Henry)
 - The primary inductance is determined by the core, the number of primary turns and the degree of magnetization of the core.
 - No correct value when measured with an LCR bridge.

$$L_P = N^2 \cdot L_s$$

- L_{pf} Primary Leakage Inductance (Henry)
 - Many of the audio transformer producers publish the L_{pf} value.
 - The leakage inductance is the electric property of an imperfectly couplet transformer. Each primary winding should be "completely seen" by the secondary windings. The inactive windings form a (not real) serial coil and affect the high frequencies.
 - The value of the primary leakage inductance is directly related to the coupling factor between the primary and secondary windings.
 - If you add to your LTspice circuit design the leakage inductance $L_{\rho f}$, then your coupling factor has to be K = 1 (without the "leakage losses").
- **R**_p Primary DC Resistance (Ω)
 - Correspond the primary copper resistance and influence the loss of the DC voltage value at the tube anode.
 - Has to be measured with a 4-wire bridge.
- C_{pf} Primary Leakage Inter-Winding Capacitance
 - It's not easy to measure.
 - The value varies very much, depending on which winding technic is used.

- K Coupling Coefficient
 - The coupling coefficient defines the efficiency of which the transformer transmits energy from the primary winding to the secondary winding energy.
 - The closer the coefficient is to "1", the more efficient is the transformer.
 - The K factor in LTspice adds all the transformer losses together.
 - K is frequency neutral, which means it is not frequency related.

$$K = \sqrt{1 - \binom{L_{mf}}{L_{mo}}} \text{ or } \sqrt{1 - \binom{L_{pf}}{L_p}}$$
$$L_{pf} = (1 \cdot K^2) \cdot L_p$$

K = Coupling Coefficient

 L_{mo} = measured primary inductance (open secondary winding)

 L_{mf} = measured leakage inductance (short circuit secondary winding)

 L_{ρ} = primary inductivity

 L_{pf} = primary leakage inductivity

Examples of real Coupling Coefficients:

- K = 0.90 0.95 this value is too low for audio transformer coupling
- K = 0.95 0.98 this is the minimum of coupling
- K = 0.98 0.995 this is a good value of an audio transformer
- K = 0.995 0.999 only excellent audio transformer will reach this value
- K = 1 this value includes no transformer losses
- C_{ps} Primary to Secondary Windings Leakage Capacitance
 Capacitance of primary to secondary windings, measured between shorted primary and shorted secondary windings with a 4-wire bridge.
 - If there is shield between primary and secondary windings, then it exists also no C_{ps} leakage capacitance.
- **Z**_s Secondary Impedance (Ω)
 - The secondary impedance should be equal to the loudspeaker impedance.

$$Z_s = \frac{V_s}{I_s}$$
$$Z_s = \frac{Z_p}{N^2}$$

- Ls Secondary Inductance (Henry)
 - The secondary inductance is determined by the core, the number of secondary turns and the degree of magnetization of the core.

- No correct value when measure with an LCR bridge.

 $L_s = \frac{L_p}{N^2}$

- L_{sf} = Secondary Leakage Inductance (Henry)
 It's a very small value and not very important for the simulation
- **R**_s Secondary DC Resistance (Ω)
 - Correspond to the secondary copper resistance.
 - To be measured with a 4-wire bridge.
- C_{sf} Secondary Inter-Winding Leakage Capacitance
 - It's not easy to measure.
 - It's a very small value and not very important for the simulation.

Real Transformer Parameters

• LTspice needs three values to emulation an simple transformer L_P = primary Inductance, L_S = secondary Inductance and K = Coupling Coefficient

1. Input Transformer

- Please read first the chapter "Calculation of Transformer Parameters"
- Example: Lundahl LL1544

Specification out of the data sheet									
Termination Alternative Turns		Copper	Idle impedance	Suggested Use	THD < 0.5%@50 Hz				
	ratio	Resistance	@40 Hz, 0dBU		primary level / real				
		Prim/sec	-		source impedance				
R4B / R4U : L4B / L4U	1:1	520Ω / 520Ω	$80 \mathrm{k}\Omega$ / $80 \mathrm{k}\Omega$	$10~\mathrm{k}\Omega$ / $10~\mathrm{k}\Omega$	$20~dBU$ / 600Ω				

• Calculation:

 $N = \frac{N_p}{N_s} = \frac{1}{1} = 1$

 $L_s = \frac{L_P}{N^2} = \frac{34}{1^2} = 34$ H no manufacture specification of the primary inductance (L_P) available,

measured L_P with simple LCR-Meter = 34H

K = no manufacture specification available (see chapter "Calculation of Transformer Parameters" - Analyse in the real amplifier, if zobel-network is necessary

• Connection of the Windings (R4U:L4B)



2. Interstage Transformer

- Please read first the chapter ""Calculation of Transformer Parameters"
- Example: Lundahl LL1635 P-P
- Specification out of the data sheet

	-
Primary DC current, primaries in series (for $B_0 = 0.9T$)	
Maximum DC current before saturation, primaries in series	
Primary inductance (primaries in series)	> 300 H
Freque by response, primaries in series	5 Hz - 60 kHz
(Source 4 k Ω for PP and 5mA, 2 k Ω for 20 mA. Load 68 pF)	+/ - 1 dB
Group delay @ 20 kHz (Source and load as above)	0.5µs
Max. output voltage @ 30 Hz	2x220 V peak
	(tot. 310Vrms)
Recommended max DC current through any primary section	40mA

Turns ratio	Static resistance, each primary	Static resistance, each secondary
1+1:1+1	500 Ω	500 Ω

Calculation: •

$$N = \frac{N_p}{N_s} = \frac{2}{2} = 1$$
$$L_s = \frac{L_p}{N^2} = \frac{300}{1^2} = 300$$
H

K = no manufacture specification available (see chapter "Calculation of Transformer Parameters" - Analyse in the real amplifier, if zobel-network is necessary



3. Line Output Transformer

Please read first the chapter ""Calculation of Transformer Parameters" •

Example: Lundahl LL2745/PF Specification out of the data s	sheet		
Туре	LL2745/PP		
Connection	Alt N		
	PP to Line Out.		
	2.8+2.8 : 2		
Primary DC current for 0.9	-		
Tesla			
Primary Inductance	290 H		
Freq. Response (+/-1dB) @			
source impedance (*)	15kΩ		
Secondaries open			
Max sec. voltage	190V r.m.s.		
@ 30 Hz			

Calculation: • $N = \frac{N_p}{N} = \frac{2 \cdot 2.8}{2 \cdot 2.8} = 2.8$

$$L_s = \frac{L_P}{N_s^2} = \frac{290}{20^2} = 36.9$$

$$s = \frac{L_P}{N^2} = \frac{290}{2.8^2} = 36.9$$
 Henry

K = no manufacture specification available (see chapter "Calculation of Transformer Parameters"

Connection of the Windings Alt N .



4. Power Output Transformer

- Please read first the chapter ""Calculation of Transformer Parameters"
- Example: Lundahl LL1620 P-P Specification out of the data sheet
 LL1620
 Turns ratio: 4 x 19.2 : 8 x 1 Static resistance of primary (all in series)
 Static resistance of each secondary (average)
 Primary leakage inductance (all in series)
 11 mH

- 1	, ,				-7 D/C	DICID	CDL		
[Max. recommended primary								
	DC current (heat dissip. 7W)	150) mA		Primar	y Load Imp	edance	DC current fo	r 0.9 Tesla (rec. operating point)
					(transformer	copper resista	nce included)	F	Primary Inductance
	Max. primary signal voltage	Push-Pull	Single End	LL1620	11.5 kΩ	6.0 kΩ	3.3 kΩ	Push-Pull	
	r.m.s. at 30 Hz (all in series)	860V	380V					300 H	
									-

Calculation:

$$N = \frac{N_p}{N_s} = \frac{4 \cdot 19.2}{3} = 25.6$$

$$L_s = \frac{L_p}{N^2} = \frac{300}{25.6^2} = 0,46 \text{ Henry}$$

$$K = \sqrt{1 - \left(\frac{L_p f}{L_p}\right)} = \sqrt{1 - \left(\frac{0.011}{300}\right)} = 0.99998$$

Connection of the Windings Alt C (6.0kΩ / 8Ω)
 Lundahl



Oh dear LTspice !

1. Oscillation !

- LTspice don't like this PP Amplifier with three transformers in the row •
- It is oscillating with SPICE directive ".tran 20m" and coupling factor "K = 0.999 (also with 0.9998)" ! • ᄙ▤窄淋๏९९९๕|๊๊ํํํ!▤ቘቘゞゞங๏๗๏ํํ๏ๅํੑ∠◡ๅฃํํ÷冫ҳъ७७๓๓๘๚๛๛ K PP_Poweramp_02.asc K PP_Poweramp_02.raw



Change coupling factor "K to 1.0" and there is not oscillation anymore !



Same PP Amplifier with SPICE directive from ".tran 5m until .tran 36m" and •



Change SPICE directive to ".tran 40m" and higher and same coupling factor "K = 0.999", the amplifier is not oscillating anymore !



Same PP Amplifier with SPICE directive ".ac dec 50 1 500k" and coupling factor "K = 0.9999" • simulation of bandwidth curve is strange !




2. Incorrect Information in the SPICE Error Log Table when calculating minus 3dB Bandwidth !

- The calculated dB values in "Error Log Table" are correct, but the value above 1kHz for the signal phase angle are not correct !
- I was asking Mike Engelhardt at his "2019 Safari Tour" in Switzerland about this problem?
 His answer: see the curves ! (hmm, maybe not so important but.....)



3. Additional Column in SPICE Error Log Table when calculating "Harmonic Distortion"!

• I ask Mike Engelhard at the "2019 Safari Tour" in Switzerland: if it's possible to add an additional column when using the SPICE directive ".four" to calculate the FFT harmonics values? His answer: this is the industry standard ! (hmm, maybe this is not so important for the industry,



4. LTspice Sinusoidal Signal $V_{PP},\,V_P$ and V_{RMS} and your Lab-Instruments !

- Lots of internet-examples are not correct interpreted, because of the misunderstanding of the voltage terms: V_{PP} or V_P or V_{RMS} and the calculation out of it.
- It's easy to oversee the voltage terms at the LTspice input screen of the signal source is always V_p, but at the output signal is V_{PP} !
- Keep in mind, the output meter of your sinus lab-generator shows normally always the V_{RMS} value and not V_{P} !
- I asked Mike Engelhard at the 2019 Safari Tour" in Switzerland, if it's possible to add the information about V_P, V_{PP} and V_{RMS} on the screen, and his answer was: Vp, is the industry standard (hmm, maybe it is the industry standard, but it would prevents lot of mistakes incl. myself)
- Convert examples below: " $V_{RMS} \rightarrow V_P$ " = $V_{RMS} * \sqrt{2} = 0.2V_{RMS} * 1.414 = 0.2828V_P$ " $V_{RMS} \rightarrow V_{PP}$ " = $V_{RMS} * 2\sqrt{2} = 0.2V_{RMS} * 2.828 = 0.566V_{PP}$

"V_P → V_{RMS}" = V_P
$$*\frac{1}{\sqrt{2}}$$
 = 0.2828V_P * 0.707 = 0.199V_{RMS}



5. Sinus Signal and the Measuring Pointer-Tool and nice to have V_{RMS} with and without V_{DC} !

- Attention: The pointer tool 🖋 shows always V_{pp} curves and on the scale the V_{pp} values incl. the V_{DC} component
- In addition it is possible to get the information of V_{RMS} including V_{DC} with the mouse pointer + click left mouse button on the curve notation letter and
- The value of measured V_{RMS} incl. V_{DC} is correct !
- But there is not an additional tool in LTspice to measure direct V_{RMS} excluding the V_{DC} component !
- Example below: V(n002) has a $95.672V_{RMS}$ sinusoidal signal incl. V_{DC}
- The V(in): has an V_{RMS} 199.86m V_{RMS} sinusoidal signal incl. V_{DC}
- When I ask Mike Engelhard: if it's possible to, his answer was, this the industry standard. (hmm, but maybe not so important because is an industry standard but for others it would be nice to have)
- Everyone has on his lab-scope a button and can switch between incl. or excl. V_{DC} and so has the possibility to measures very easy V_{PP} with or without the V_{DC} component !
 Define a set of the set



Bibliography

- Gills Brocard; SIMULATION IN LTSPISE (Würth Elektronik)
- Menno van der Veen; Moderne High-End Röhrenverstärker (Elektor)
- Menno van der Veen; High-End-Röhrenverstärker (Elektor)
- Menno van der Veen; Measuring Output Transformer Performance (Glass Audio 5/97)
- Alexander Potchinkov; Simulation von Röhrenverstärkern mit SPICE (Vieweg+Teubner)
- Diverse; Elektronik Industrie-, Rundfunk-, Fernsehelektronik; (Europa Lehrmittel)
- Colonel Wm T.Mclyman; Transformer and Inductor Design Handbook, Chapter 17 (Marcel Dekker)
- Georg Rose; Grosse Elektronik Formelsammlung (Franzis-Verlag)
- G. Barrere; Measuring Transformer Coupling Coupling Factor K, (Exality Corporation)
- Andy Grove and Peter Qvortrupp; The Audio Note Transformer Design Philosophy
- Gerd Reinhöfer; Ausganstransformer Daten
- C.A.V. Sowter; Soft Magnetic Material for Audio Transformers
- Sowter Transformer; Transformer Specification
- Lundahl; Transformer Specification
- Norman L. Koren; Improved VT Models for Spice Simulationn (Glass Audio 5/96)
- Nathan R. Grossner; Transformer for electronic circuits
- Derk Reefman; ExtractModel: Examples
- Stephie Bench; LTspice Tube Models (Forum Intact Audio)
- Dmitry Nizhegorodov; Tube Simulation with PSPICE: Tips, Tricks, Techniques
- Duncanamps.com; LTspice and vacuum tube models
- Ronald Dekker; µTracer Tube Tester
- EMAG Harvey Morehouse; Transformer Modeling
- Antique Sound Lab
- groups.io/g/LTspice
- Mike Engelhardt Safari Tour 11.11.2019 in Switzerland
- EXCEM; Vacuum Tube Modeling Package
- ZHAW H. H. Hochreutener; LTspice, Einführung ins Simulations-Tool
- Erhard Schemainda, Eine Spice-Simulation des Grundig Hi-Fi Verstärkerbaustein NF1
- Robert G. Wolpert; Transformer Design and Manufacturing Manual
- Rod Elliott (ESP); Transformers The Basics (Best explanation of Audio Transformers!)
- N. H. Crowhurst; Measuring up an Audio Transformer
- William Walther, Gérard Chrétien; Amplificateurs monotriodes de puissance, Les transformateurs de sortie